VSim Examples

Release 9.0.2-r2448

Tech-X Corporation

Nov 29, 2018

CONTENTS

CHAPTER ONE

OVERVIEW

These are examples for illustrating the capabilities of VSim.

VSim *[VSi]* is an arbitrary dimensional, electromagnetics and plasma simulation code consisting of two major components:

- VSimComposer, the graphical user interface.
- Vorpal *[NC04]*, the VSim Computational Engine.

VSim also includes many more items such as Python, MPI, data analyzers, and a set of input simplifying macros.

CHAPTER

TWO

VSIM FOR BASIC SIMULATIONS EXAMPLES

These examples demonstrate the basic solvers for simple, grid-aligned boundary conditions.

These examples can be run with any license.

2.1 Basic Examples

2.1.1 Cylindrical Capacitor (cylindricalCapacitor.sdf)

Keywords:

`cylindrical, capacitor, electrostatic`

Problem description

The Cylindrical Capacitor simulation solves for the potential between two cylinders with a ring of charge.

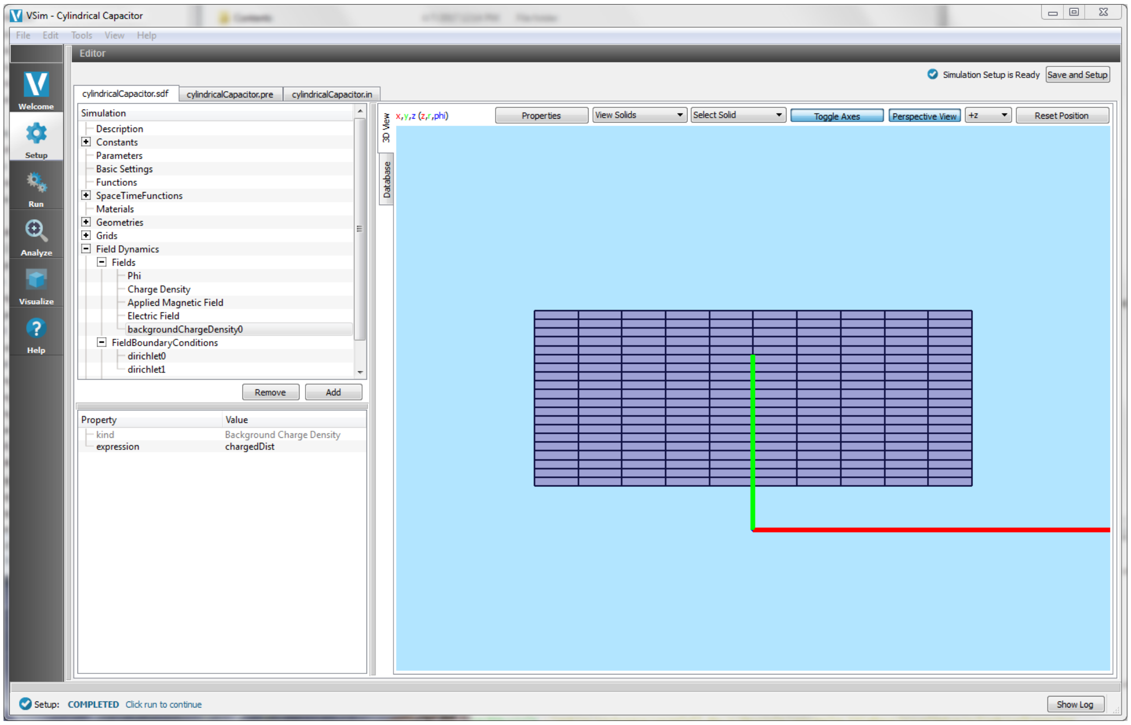
This simulation can be performed with a VSimBase license.

Opening the Simulation

The Cylindrical Capacitor example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples* option.
- Select *Cylindrical Capacitor* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 2.1. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



Simulation Setup is Ready

Property	Value
kind	Background Charge Density
expression	chargedDist

Setup: COMPLETED Click run to continue

Fig. 2.1: Setup Window for the Cylindrical Capacitor example.

Simulation Properties

In this simulation there is a backgroundChargeDensity0 field which is given by an expression, chargedDist. That expression is defined as a SpaceTimeFunction. The variables x and y in the expression are place holders for the actual variables, Z and R, in the simulation. So this is a ring of charge, centered at R = 0.3, with a Gaussian fall off.

There are Dirichlet boundary conditions on the lower and upper R boundaries, with the lower bound set to 10 volts.

Running the simulation

After performing the above actions, continue as follows:

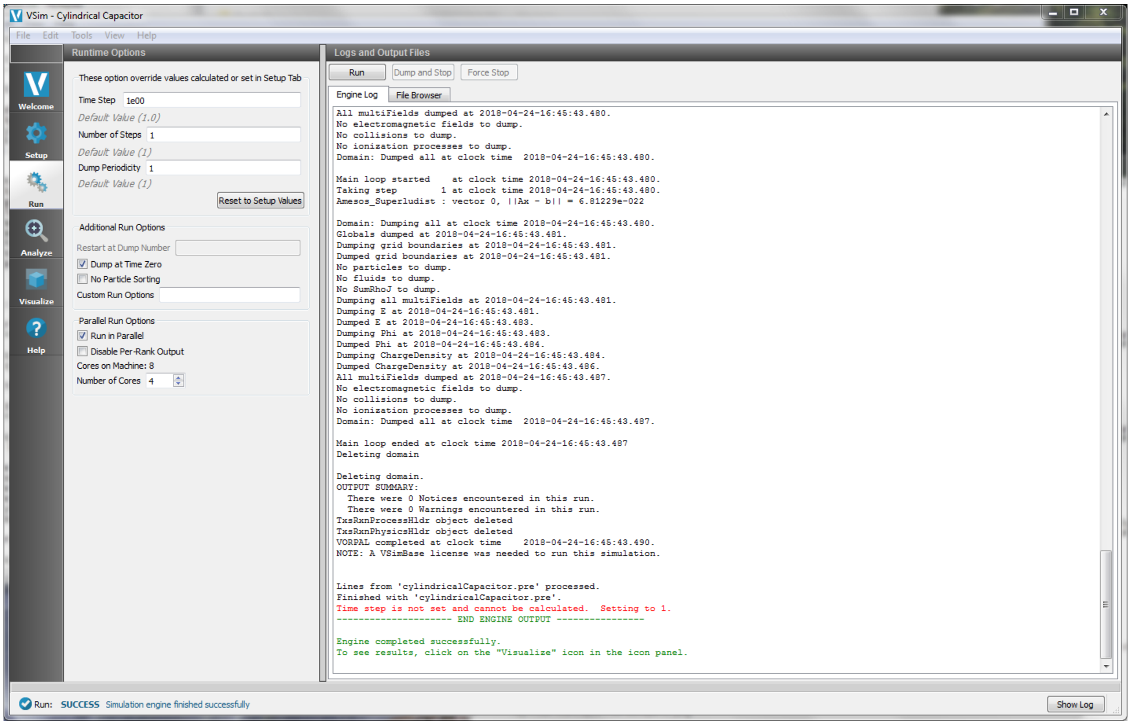
- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- Here you can set run parameters, including how many cores to run with (under the Parallel Run Options).
- When you are finished setting run parameters, click on the *Run* button in the upper left corner of the window. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 2.2.

Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

This particular run computes the electrostatic potential, which we see by opening the scalar data and checking the box next to Phi, which is shown in the right of the visualization tab. See Fig. 2.3.



These option override values calculated or set in Setup Tab

Time Step 1e00

Default Value (1.0)

Number of Steps 1

Default Value (1)

Dump Periodicity 1

Default Value (1)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

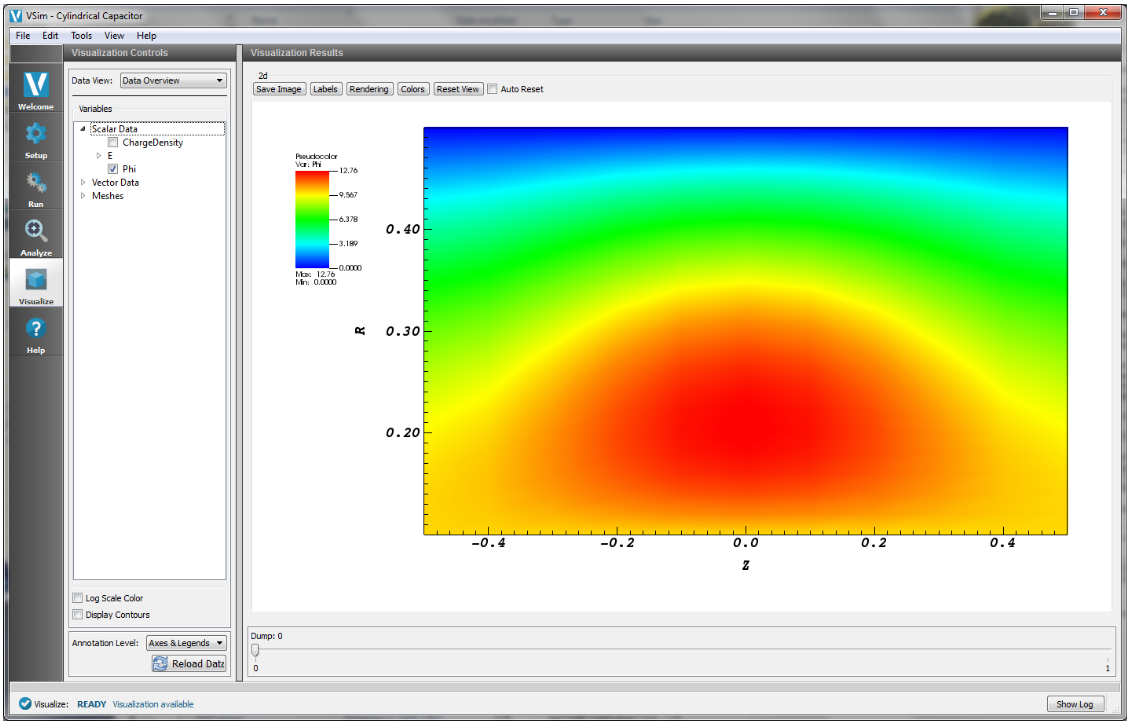
Cores on Machine: 8

Number of Cores 4

```
All multiFields dumped at 2018-04-24-16:45:43.480.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-16:45:43.480.

Main loop started    at clock time 2018-04-24-16:45:43.480.
Taking step        1 at clock time 2018-04-24-16:45:43.480.
Amesos_Superludist : vector 0, ||Ax - b|| = 6.81229e-022

Domain: Dumping all at clock time 2018-04-24-16:45:43.480.
Globals dumped at 2018-04-24-16:45:43.481.
Dumping grid boundaries at 2018-04-24-16:45:43.481.
Dumped grid boundaries at 2018-04-24-16:45:43.481.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-16:45:43.481.
Dumping E at 2018-04-24-16:45:43.481.
Dumped E at 2018-04-24-16:45:43.483.
Dumping Phi at 2018-04-24-16:45:43.483.
Dumped Phi at 2018-04-24-16:45:43.484.
Dumping ChargeDensity at 2018-04-24-16:45:43.484.
Dumped ChargeDensity at 2018-04-24-16:45:43.486.
All multiFields dumped at 2018-04-24-16:45:43.487.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-16:45:43.487.

Main loop ended at clock time 2018-04-24-16:45:43.487
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-16:45:43.490.
NOTE: A VSimBase license was needed to run this simulation.


Lines from 'cylindricalCapacitor.pre' processed.
Finished with 'cylindricalCapacitor.pre'.
Time step is not set and cannot be calculated.  Setting to 1.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 2.2: The Run Window at the end of execution.



Data View: Data Overview

Variables

- Scalar Data
  - □ ChargeDensity
  - E
  - ☑ Phi
- Vector Data
- Meshes

□ Log Scale Color

□ Display Contours

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 0

Visualize: READY Visualization available

Fig. 2.3: Visualization of the electrostatic potential

Further Experiments

Looking inside the field boundary conditions, and highlighting dirichlet0, you can see that 10V was put on the lower R boundary. You can try experimenting with this, going to run and vizualize with each change. The more voltage, the less the background charge should matter.

You can take the charge out of the system. Highlight the backgroundChargeDensity0 label. In the property editor below, double click on chargedDist, hit delete, and type 0.0. Then run and viz, and you will see a potential that is independent of Z.

In Data View, choose Field Analysis and Field E_r with the Vertical Lineout Settings, then hit “Perform Lineout”. You will see, as expected, that the radial electric field is positive (pointing outward) and falling off with the expected 1/r behavior.

2.1.2 Oscillating Dipole Above Conducting Plane (emOscDipoleAboveConductor.sdf)

Keywords:

`emOscDipoleAboveConductor, radiation`

Problem Description

This problem consists of an infintesimally short dipole located a variable height and orientation above a conducting plane. This simulation consists of electric and magnetic fields that can be visualized to see how the distance between, and orientation of the dipole relative to the antenna effects these fields.

This simulation can be performed with a VSimBase license.

Opening the Simulation

The Dipole Above Conducting Plane example is accessed from within VSimComposer by the following actions:

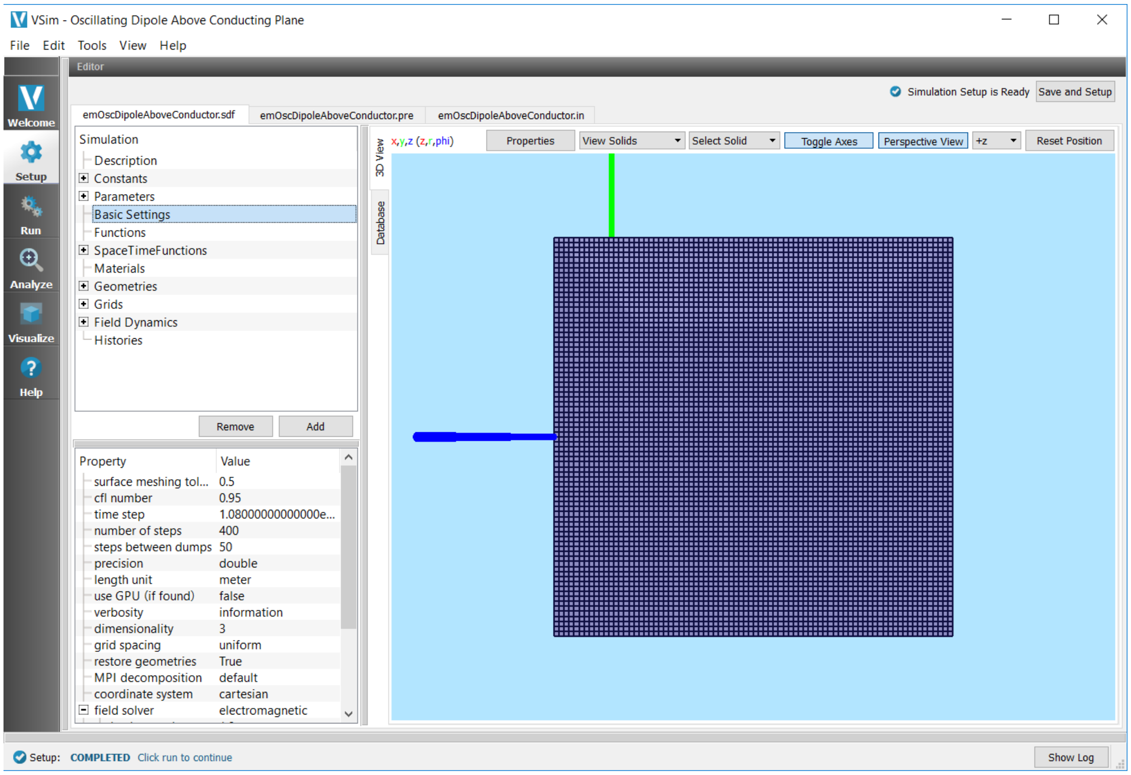
- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples* option.
- Select *Dipole Above Conducting Plane* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 2.4. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.

Simulation Properties

This example includes several constants for easy adjustment of simulation properties, Including:

- AMPLITUDE: The amplitude of the dipole current
- FREQUENCY: The operating frequency



Property	Value
surface meshing tol...	0.5
cfl number	0.95
time step	1.08000000000000e...
number of steps	400
steps between dumps	50
precision	double
length unit	meter
use GPU (if found)	false
verbosity	information
dimensionality	3
grid spacing	uniform
restore geometries	True
MPI decomposition	default
coordinate system	cartesian
field solver	electromagnetic

Setup: COMPLETED Click run to continue

Fig. 2.4: Setup Window for the Dipole Above Conducting Plane example.

There is also a SpaceTimeFunction to define the current driver of the dipole source

Other properties of the simulation include open boundaries on all sides except for the lower x boundary, which is a perfect electric conductor. A *Dipole Current* source is used to set the location of the dipole source.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 2.5.

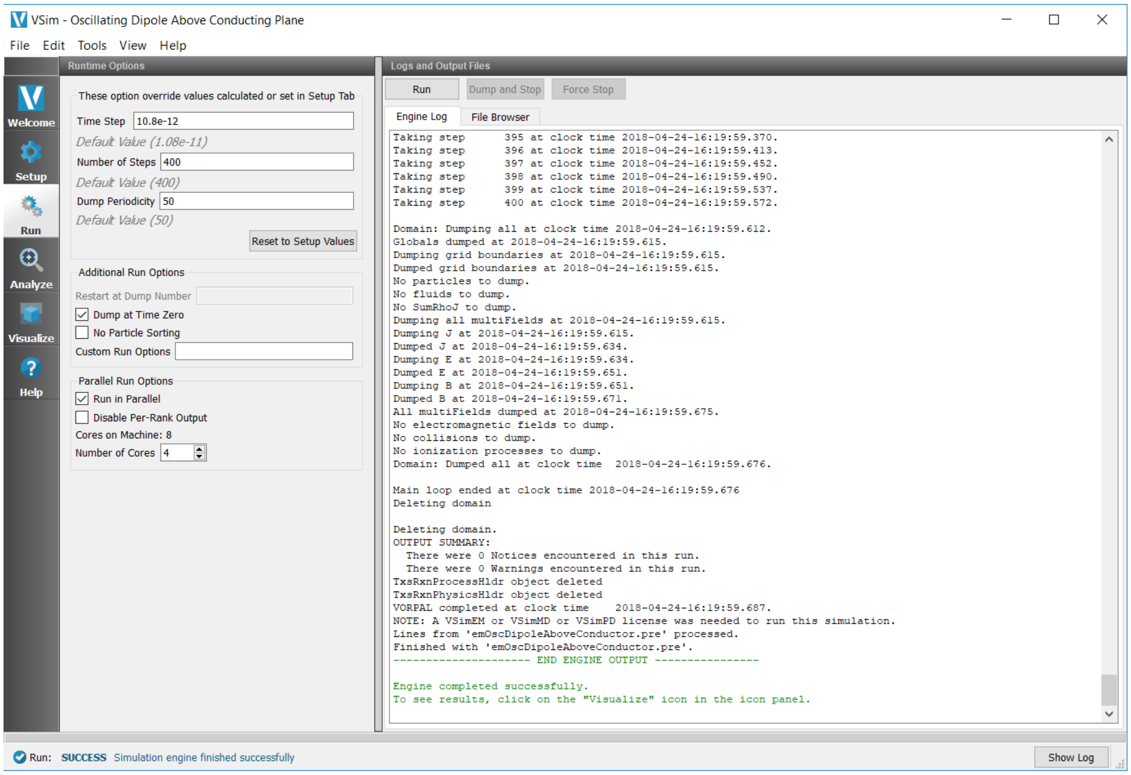
Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The electric and magnetic field components can be found in the scalar data variables of the data overview tab. To create the plot shown in Fig. 2.6 do the following:

- Expand *Scalar Data*
- Expand *E*
- Select *E_y*



These option override values calculated or set in Setup Tab

Time Step 10.8e-12

Default Value (1.08e-11)

Number of Steps 400

Default Value (400)

Dump Periodicity 50

Default Value (50)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Cores on Machine: 8

Number of Cores 4

```
Taking step      395 at clock time 2018-04-24-16:19:59.370.
Taking step      396 at clock time 2018-04-24-16:19:59.413.
Taking step      397 at clock time 2018-04-24-16:19:59.452.
Taking step      398 at clock time 2018-04-24-16:19:59.490.
Taking step      399 at clock time 2018-04-24-16:19:59.537.
Taking step      400 at clock time 2018-04-24-16:19:59.572.

Domain: Dumping all at clock time 2018-04-24-16:19:59.612.
Globals dumped at 2018-04-24-16:19:59.615.
Dumping grid boundaries at 2018-04-24-16:19:59.615.
Dumped grid boundaries at 2018-04-24-16:19:59.615.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-16:19:59.615.
Dumping J at 2018-04-24-16:19:59.615.
Dumped J at 2018-04-24-16:19:59.634.
Dumping E at 2018-04-24-16:19:59.634.
Dumped E at 2018-04-24-16:19:59.651.
Dumping B at 2018-04-24-16:19:59.651.
Dumped B at 2018-04-24-16:19:59.671.
All multiFields dumped at 2018-04-24-16:19:59.675.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-16:19:59.676.

Main loop ended at clock time 2018-04-24-16:19:59.676
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-16:19:59.687.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.
Lines from 'emOscDipoleAboveConductor.pre' processed.
Finished with 'emOscDipoleAboveConductor.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 2.5: The Run Window at the end of execution.

- Select the box next to *Display Contours* and set the # of contours to 12
- Select the box next to *Clip All Plots*
- Rotate the plot by clicking and dragging your mouse

Further Experiments

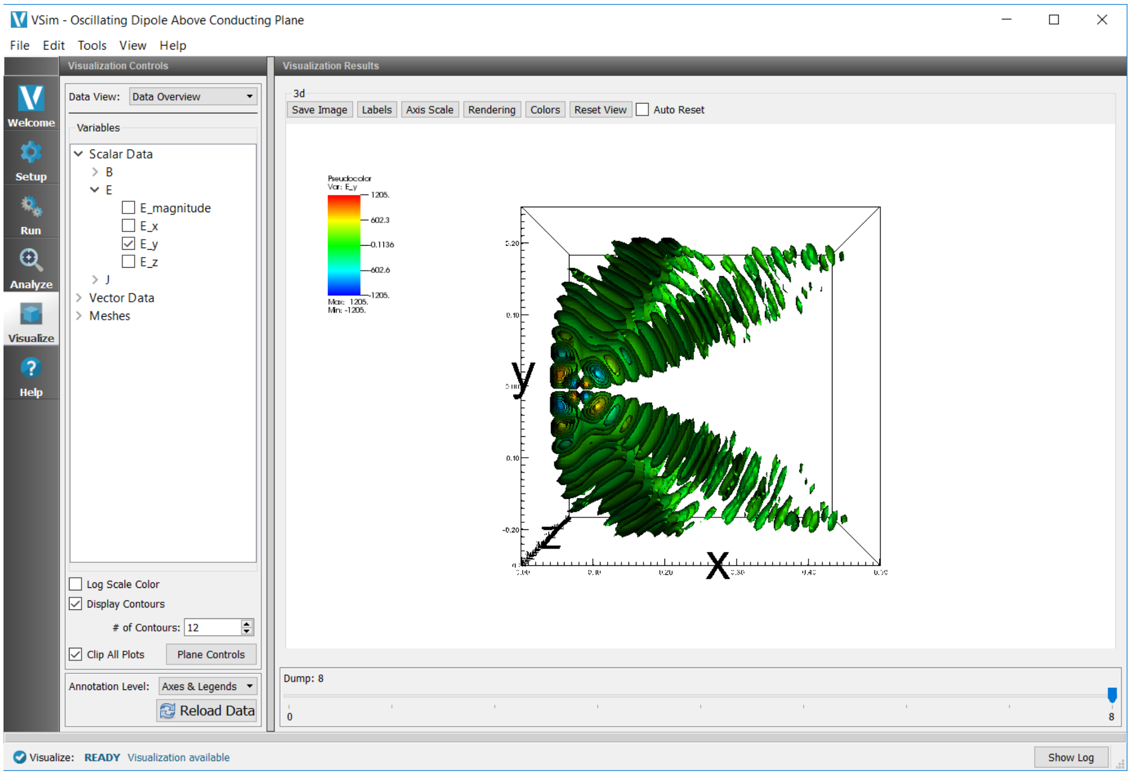
In this example the “infinite” electric conductor is simulated by a physical conducting boundary at the bottom of the simulation. It would be possible to achieve the same results by having a second, equal infintesimal dipole placed the same height “below” the conducting plane.

The number of “lobes” visible in the far field is dependent on Antenna Orientation and height. If vertically oriented there will be 2*Height/Wavelength +1 lobes.A horizontally oriented dipole will produce 2*Height/Wavelength lobes. This can be a bit difficult to visualize using just E-field data as it must be properly thresholded. The lobes will be easier to see in the example Advanced Dipole Above Conductor, a part of the VSimEM package.

2.1.3 Electromagnetic Plane Wave (emPlaneWave.sdf)

Keywords:

electromagnetics, plane wave, periodic boundary conditions, wave launcher



Data View: Data Overview

□ E_magnitude
□ E_x
☑ E_y
□ E_z

□ Log Scale Color
☑ Display Contours
# of Contours: 12
☑ Clip All Plots
Annotation Level: Axes & Legends
□ Auto Reset
Dump: 8

Visualize: READY Visualization available

Fig. 2.6: The electric field

Problem Description

A linearly-polarized (with electric field in the z-direction) electromagnetic pulse with a sinusoidal amplitude on a plane wave is launched from the left side (x=0) to propagate in the x-direction. The transverse (y,z) boundary conditions are periodic.

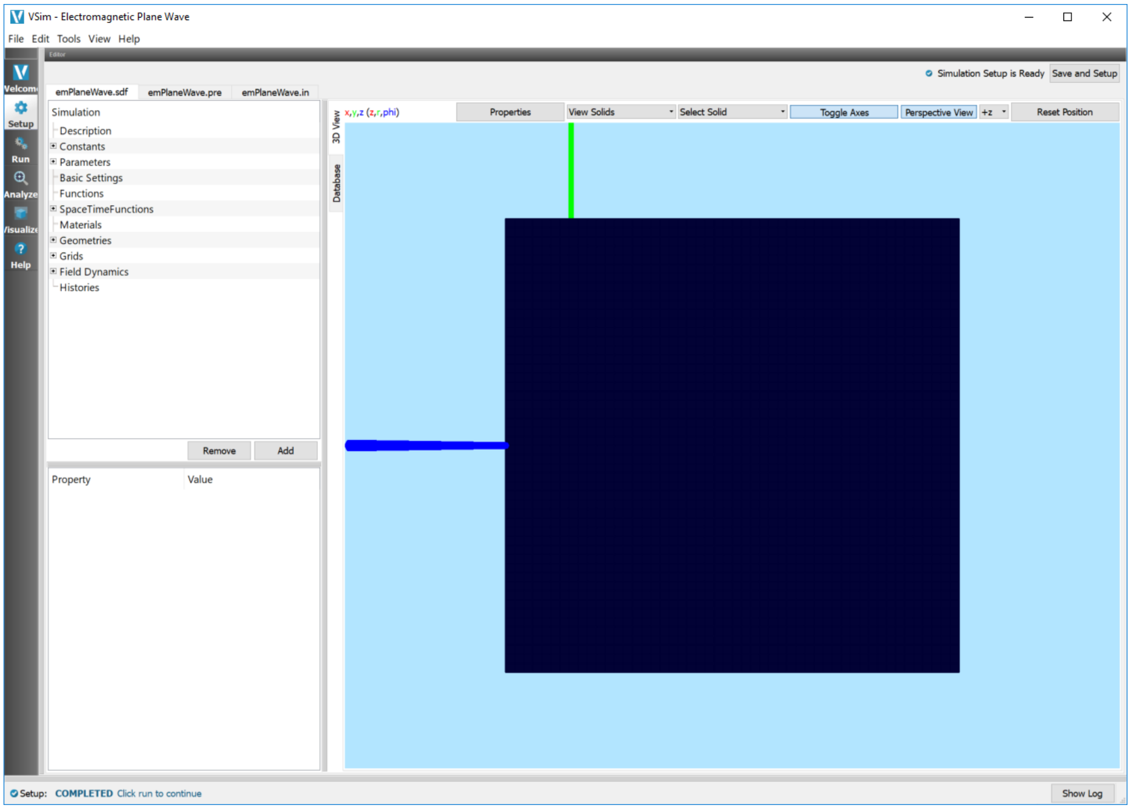
This simulation can be performed with a VSimBase license.

Opening the Simulation

The Electromagnetic Plane Wave example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples* option.
- Select *Electromagnetic Plane Wave* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 2.7. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



Simulation Setup is Ready

Setup: COMPLETED Click run to continue

Fig. 2.7: Setup Window for the Electromagnetic Plane Wave example.

Simulation Properties

This example includes several constants for easy adjustment of simulation properties. Those include:

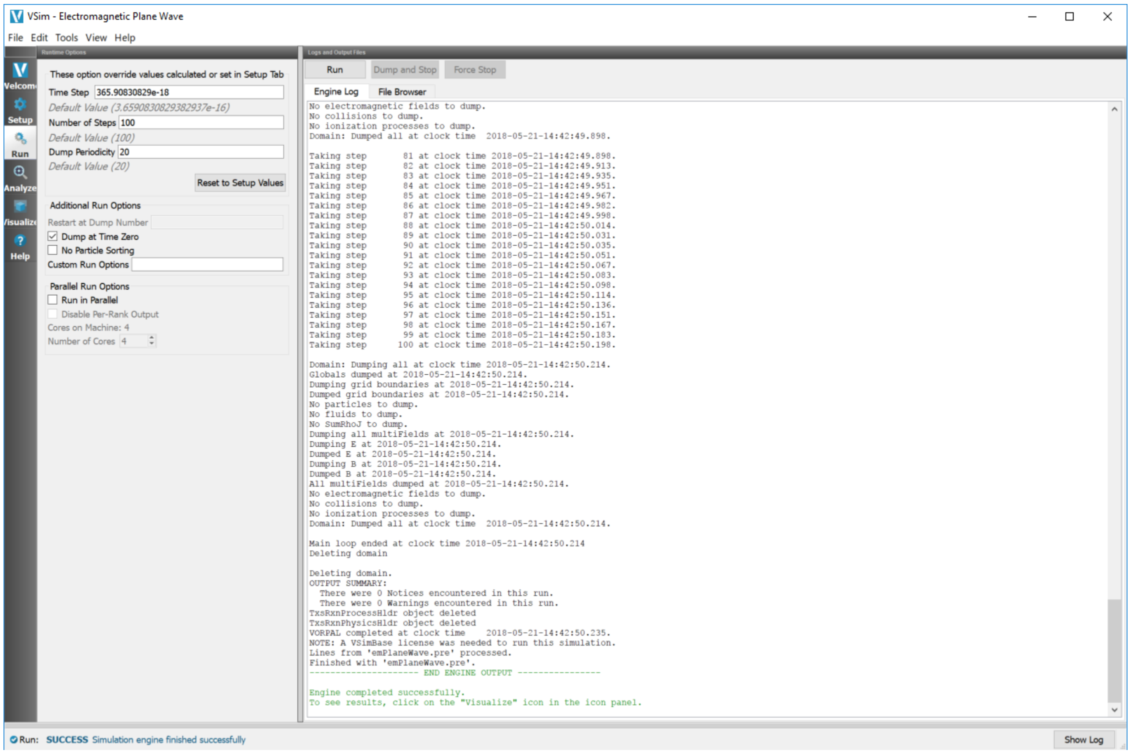
- AMPLITUDE: The amplutide of the plane wave
- WAVELENGTHS: The number of wavelengths inside the domain

There is a SpaceTimeFunction to define the plane wave that is launched with a *Port Launcher* boundary condition.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 2.8.



These option override values calculated or set in Setup Tab

Time Step 365.90830829e-18
Default Value (3.6590830829382937e-16)
Number of Steps 100
Default Value (100)
Dump Periodicity 20
Default Value (20)

Additional Run Options
Restart at Dump Number
☑ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options
☐ Run in Parallel
☐ Disable Per-Rank Output
Cores on Machine: 4
Number of Cores 4

```
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-21-14:42:49.898.

Taking step      81 at clock time 2018-05-21-14:42:49.898.
Taking step      82 at clock time 2018-05-21-14:42:49.913.
Taking step      83 at clock time 2018-05-21-14:42:49.935.
Taking step      84 at clock time 2018-05-21-14:42:49.951.
Taking step      85 at clock time 2018-05-21-14:42:49.967.
Taking step      86 at clock time 2018-05-21-14:42:49.982.
Taking step      87 at clock time 2018-05-21-14:42:49.998.
Taking step      88 at clock time 2018-05-21-14:42:50.014.
Taking step      89 at clock time 2018-05-21-14:42:50.031.
Taking step      90 at clock time 2018-05-21-14:42:50.035.
Taking step      91 at clock time 2018-05-21-14:42:50.051.
Taking step      92 at clock time 2018-05-21-14:42:50.067.
Taking step      93 at clock time 2018-05-21-14:42:50.083.
Taking step      94 at clock time 2018-05-21-14:42:50.098.
Taking step      95 at clock time 2018-05-21-14:42:50.114.
Taking step      96 at clock time 2018-05-21-14:42:50.136.
Taking step      97 at clock time 2018-05-21-14:42:50.151.
Taking step      98 at clock time 2018-05-21-14:42:50.167.
Taking step      99 at clock time 2018-05-21-14:42:50.183.
Taking step     100 at clock time 2018-05-21-14:42:50.198.

Domain: Dumping all at clock time 2018-05-21-14:42:50.214.
Globals dumped at 2018-05-21-14:42:50.214.
Dumping grid boundaries at 2018-05-21-14:42:50.214.
Dumped grid boundaries at 2018-05-21-14:42:50.214.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-21-14:42:50.214.
Dumping E at 2018-05-21-14:42:50.214.
Dumped E at 2018-05-21-14:42:50.214.
Dumping B at 2018-05-21-14:42:50.214.
Dumped B at 2018-05-21-14:42:50.214.
All multiFields dumped at 2018-05-21-14:42:50.214.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-21-14:42:50.214.

Main loop ended at clock time 2018-05-21-14:42:50.214
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-05-21-14:42:50.235.
NOTE: A VSimBase license was needed to run this simulation.
Lines from 'emPlaneWave.pre' processed.
Finished with 'emPlaneWave.pre'.
-------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 2.8: The Run Window at the end of execution.

Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The electric and magnetic field components can be found in the scalar data variables of the data overview tab.

- Make sure the *Data View* drop down is set to *Data Overview*.

- Here you can see *Variables*. Expand the *Scalar Data*.
- Expand E
- Select E_z

Initially, no field will be seen, as one is looking at Dump 0, the initial dump, when no fields are yet in the simulation. Move the slider at the bottom of the right pane to see the electric field at different times. The final time is shown in Fig. 2.9.

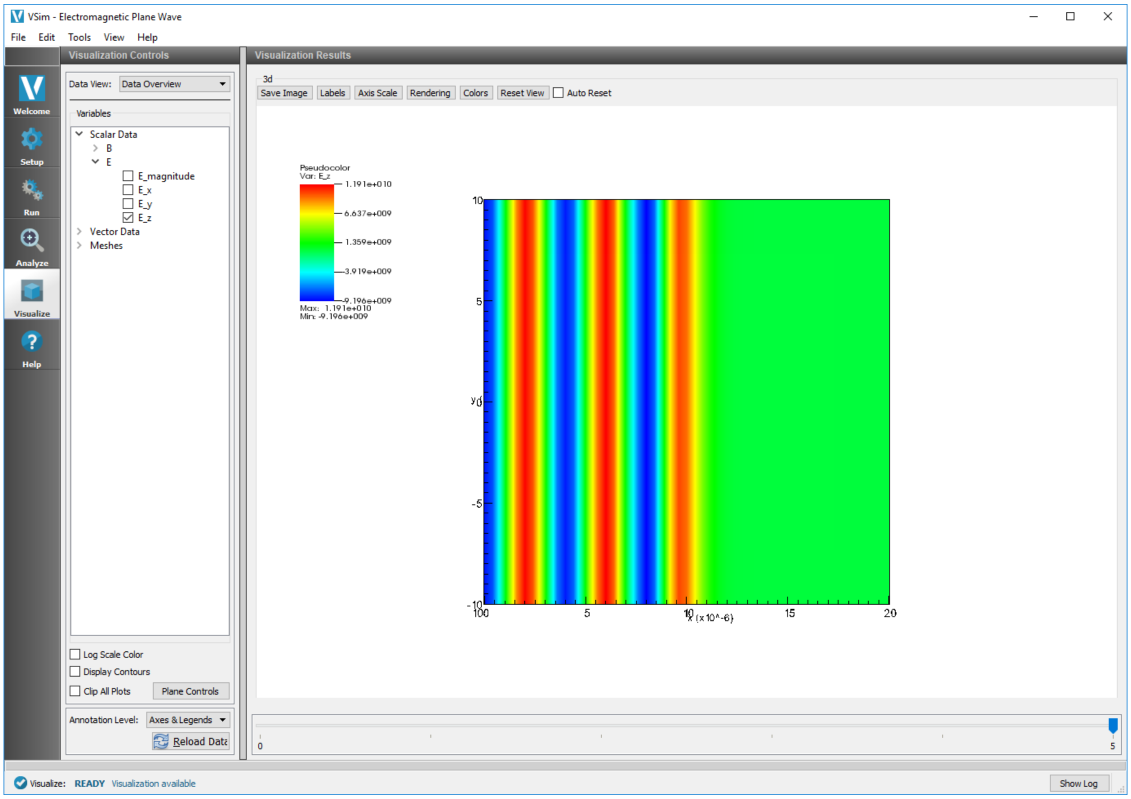


Fig. 2.9: Visualization of plane wave as a color contour plot.

Further Experiments

To see more wavelengths, change the value of the WAVELENGTHS variable. What happens to the waves when there are very few cells in a wavelength?

See the wave reflect off the right boundary by running for more time steps.

Try changing NDIM to 3 to see the how the simulation is uniform across the z- dimension.

2.1.4 Electromagnetic Particle In Cell (emPtclInCell.sdf)

Keywords:

`electromagnetics, particle in cell, sheath`

Problem description

A dipole antenna launches a wave from a point that is midway in x and y. The simulation is periodic in y and open in x. The electromagnetic field and plasma respond self consistently to the antenna current.

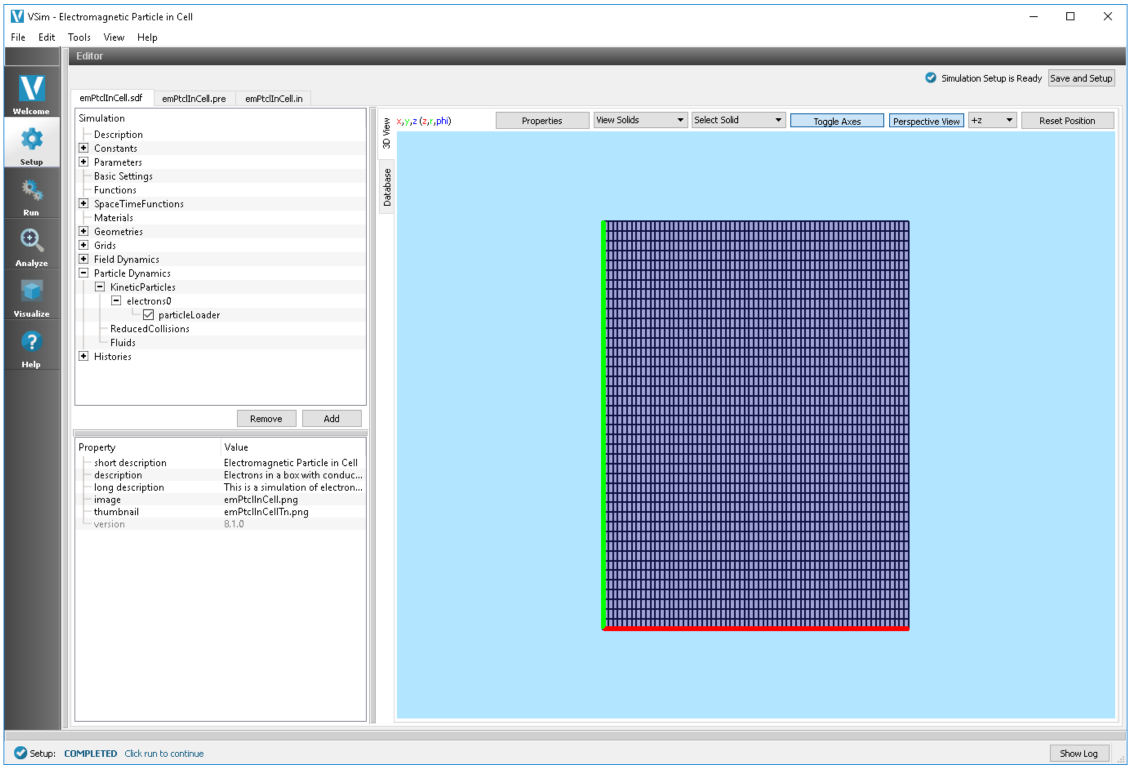
This simulation can be performed with a VSimBase license.

Opening the Simulation

The Electromagnetic Particle In Cell example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples* option.
- Select "Electromagnetic Particle In Cell" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 2.10. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



Simulation Setup is Ready

Property	Value
short description	Electromagnetic Particle in Cell
description	Electrons in a box with conduc…
long description	This is a simulation of electron…
image	emPtclInCell.png
thumbnail	emPtclInCellTn.png
version	8.1.0

Setup: COMPLETED Click run to continue

Fig. 2.10: Setup Window for Electromagnetic Particle in Cell.

Simulation Properties

This simulation includes several constants for easy adjustment of simulation properties including:

- N_X, N_Y: The number of cells in each direction
- LEN_X, LEN_Y: The length of the domain in each direction
- PPC: The number of macroparticles per cell
- FREQUENCY: The frequency of the dipole antenna

The *Parameters* element contains several parameters useful for calculating basic plasma physics properties such as the plasma frequency and Debye length.

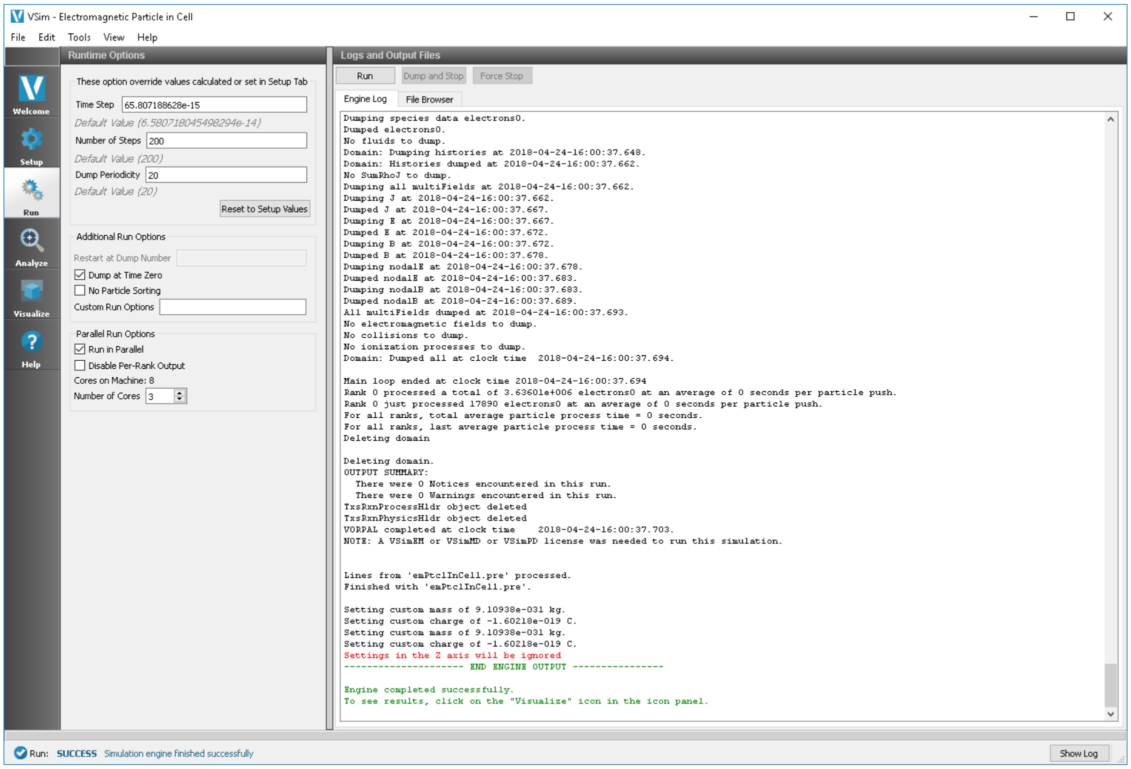
There are 2 SpaceTimeFunctions that are used later in the setup to describe the thermal velocity of the electrons and the annena current profile.

The simulation has open boundary conditions in x, and periodic in y.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 2.11.



These option override values calculated or set in Setup Tab

Time Step: 65.807188628e-15
Default Value (6.580718045498294e-14)
Number of Steps: 200
Default Value (200)
Dump Periodicity: 20
Default Value (20)

Additional Run Options

Restart at Dump Number:
☑ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options:

Parallel Run Options

☑ Run in Parallel
☐ Disable Per-Rank Output
Cores on Machine: 8
Number of Cores: 3

```
Dumping species data electrons0.
Dumped electrons0.
No fluids to dump.
Domain: Dumping histories at 2018-04-24-16:00:37.648.
Domain: Histories dumped at 2018-04-24-16:00:37.662.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-16:00:37.662.
Dumping J at 2018-04-24-16:00:37.662.
Dumped J at 2018-04-24-16:00:37.667.
Dumping E at 2018-04-24-16:00:37.667.
Dumped E at 2018-04-24-16:00:37.672.
Dumping B at 2018-04-24-16:00:37.672.
Dumped B at 2018-04-24-16:00:37.678.
Dumping nodalE at 2018-04-24-16:00:37.678.
Dumped nodalE at 2018-04-24-16:00:37.683.
Dumping nodalB at 2018-04-24-16:00:37.683.
Dumped nodalB at 2018-04-24-16:00:37.689.
All multiFields dumped at 2018-04-24-16:00:37.693.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-16:00:37.694.

Main loop ended at clock time 2018-04-24-16:00:37.694
Rank 0 processed a total of 3.63601e+006 electrons0 at an average of 0 seconds per particle push.
Rank 0 just processed 17890 electrons0 at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-16:00:37.703.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.


Lines from 'emPtclInCell.pre' processed.
Finished with 'emPtclInCell.pre'.

Setting custom mass of 9.10938e-031 kg.
Setting custom charge of -1.60218e-019 C.
Setting custom mass of 9.10938e-031 kg.
Setting custom charge of -1.60218e-019 C.
Settings in the Z axis will be ignored
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 2.11: The Run Window at the end of execution of Electromagnetic Particle in Cell.

Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the electric field and particles as shown in Fig. 2.12, do the following:

- Expand *Particle Data*
- Expand *electrons0*
- Select *electrons0*
- Expand *Scalar Data*
- Expand *E*
- Select *E_z*

Initially the field is at zero and particles are evenly distributed throughout the simulation. Move the dump slider forward in time to view the results.

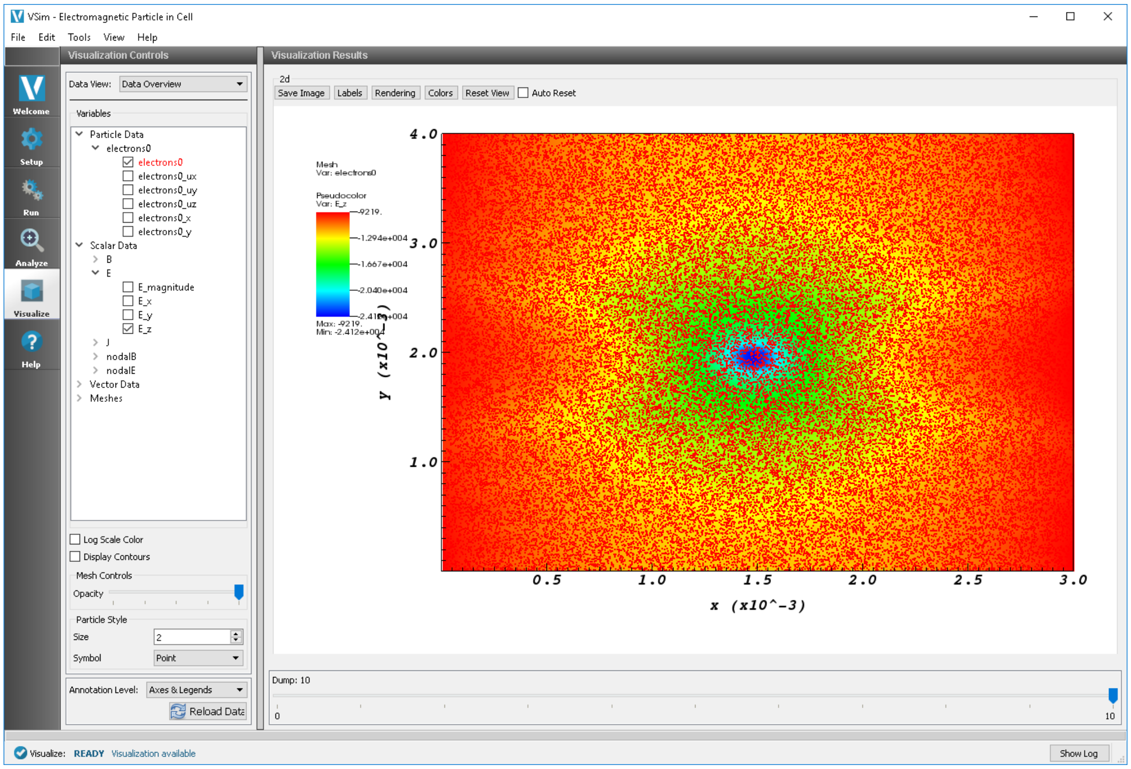


Fig. 2.12: Visualize Window with electric field and particles

Further Experiments

Vary the antenna amplitude, to find out how low it can be before the signal is swamped by the plasma noise.

Add in a magnetic field in the plane.

2.1.5 Vacuum Electromagnetic Pulse (emPulseInVacuum.sdf)

Keywords:

`electromagnetics, laser, plane wave pulse, field energy monitoring`

Problem description

A linearly-polarized (with electric field in the z-direction) electromagnetic pulse with a sinusoidal amplitude on a plane wave is launched from the left side (x=0). The transverse (y, z) boundary conditions are periodic, but the pulse has finite transverse extent.

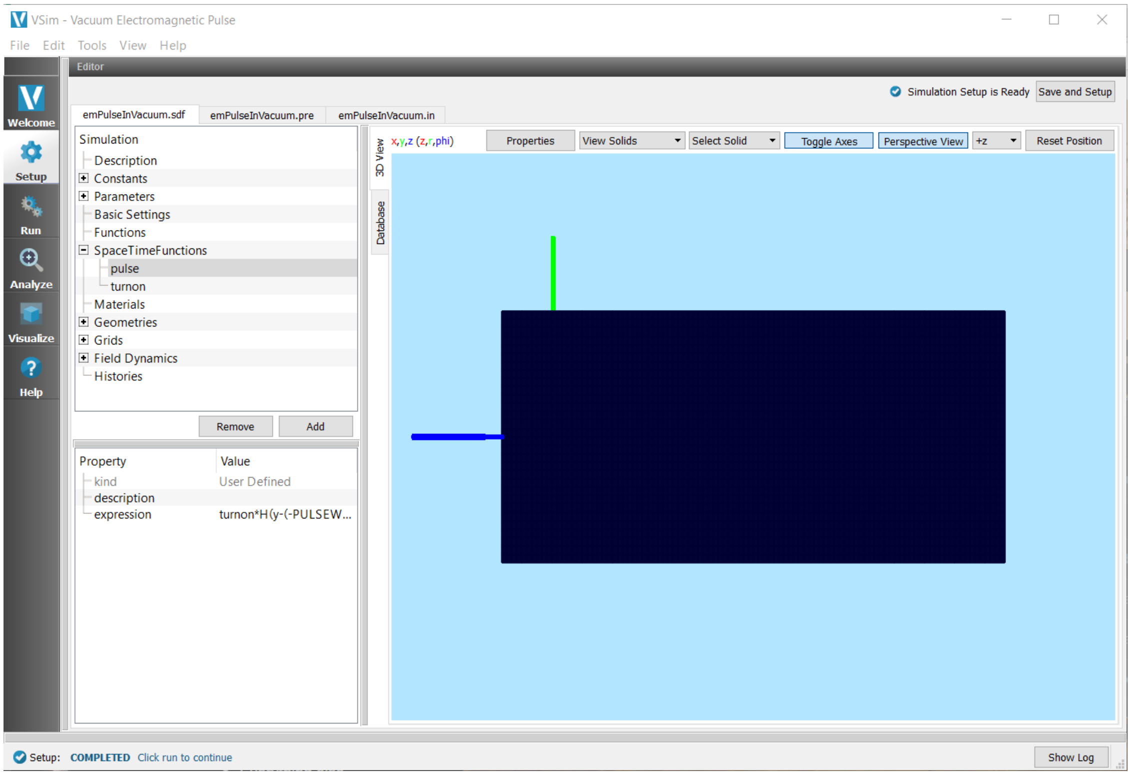
This simulation can be performed with a VSimBase license.

Opening the Simulation

The Vacuum Electromagnetic Pulse example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples* option.
- Select *Vacuum Electromagnetic Pulse* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 2.13. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



Property	Value
kind	User Defined
description	
expression	turnon*H(y-(-PULSEW...

Setup: COMPLETED Click run to continue

Fig. 2.13: The Setup Window for the electromagnetic pulse.

Simulation Properties

The Vacuum Electromagnetic Pulse example includes several constants for easy adjustment of simulation properties. Those include:

- AMPLITUDE: The amplitude of the pulse
- WAVELENGTH: The wavelength of the pulse
- PULSELENGTH: The length of the pulse in the propagation direction
- PULSEWIDTH: The width of the pulse in the transverse direction

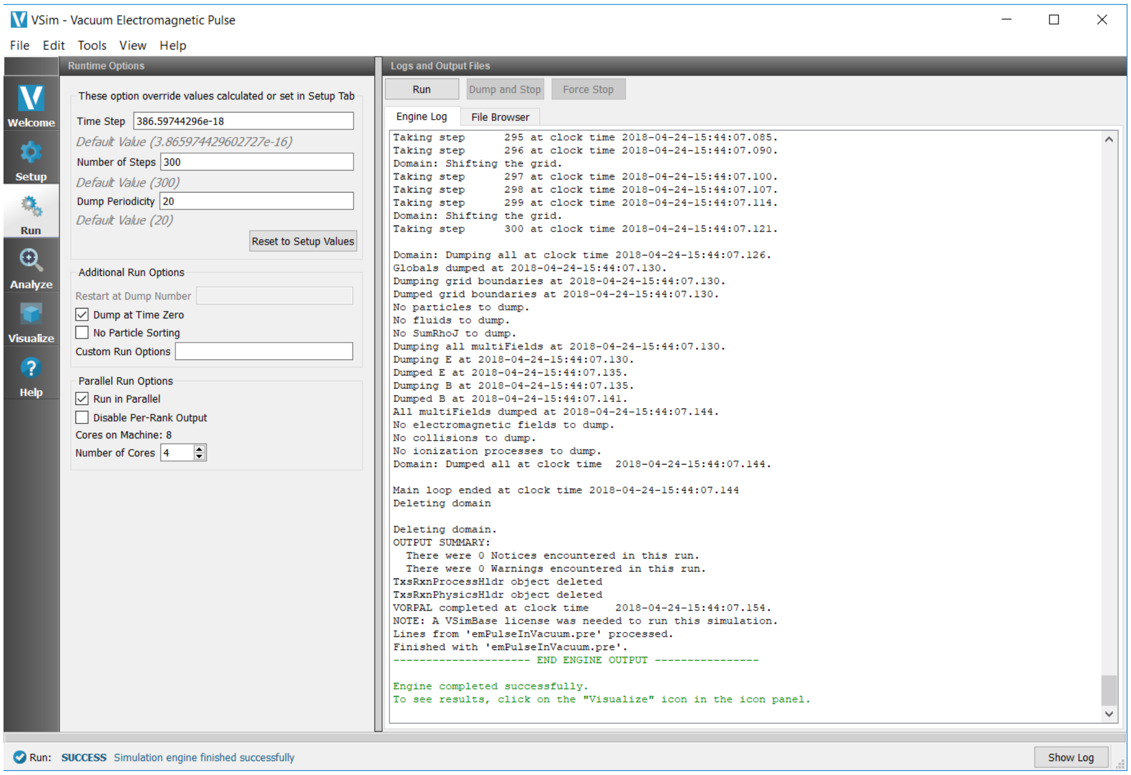
There is also a SpaceTimeFunction defined for the pulse shape and is used in the *Port Launcher* boundary condition.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully."

The Run Window, showing settable parameters with the engine output in the right pane, is shown in Fig. 2.14.



These option override values calculated or set in Setup Tab

Time Step 386.59744296e-18

Default Value (3.865974429602727e-16)

Number of Steps 300

Default Value (300)

Dump Periodicity 20

Default Value (20)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Cores on Machine: 8

Number of Cores 4

```
Taking step      295 at clock time 2018-04-24-15:44:07.085.
Taking step      296 at clock time 2018-04-24-15:44:07.090.
Domain: Shifting the grid.
Taking step      297 at clock time 2018-04-24-15:44:07.100.
Taking step      298 at clock time 2018-04-24-15:44:07.107.
Taking step      299 at clock time 2018-04-24-15:44:07.114.
Domain: Shifting the grid.
Taking step      300 at clock time 2018-04-24-15:44:07.121.

Domain: Dumping all at clock time 2018-04-24-15:44:07.126.
Globals dumped at 2018-04-24-15:44:07.130.
Dumping grid boundaries at 2018-04-24-15:44:07.130.
Dumped grid boundaries at 2018-04-24-15:44:07.130.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-15:44:07.130.
Dumping E at 2018-04-24-15:44:07.130.
Dumped E at 2018-04-24-15:44:07.135.
Dumping B at 2018-04-24-15:44:07.135.
Dumped B at 2018-04-24-15:44:07.141.
All multiFields dumped at 2018-04-24-15:44:07.144.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-15:44:07.144.

Main loop ended at clock time 2018-04-24-15:44:07.144
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-15:44:07.154.
NOTE: A VSimBase license was needed to run this simulation.
Lines from 'emPulseInVacuum.pre' processed.
Finished with 'emPulseInVacuum.pre'.
---------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 2.14: The Run Window for the electromagnetic pulse.

Visualizing the Results

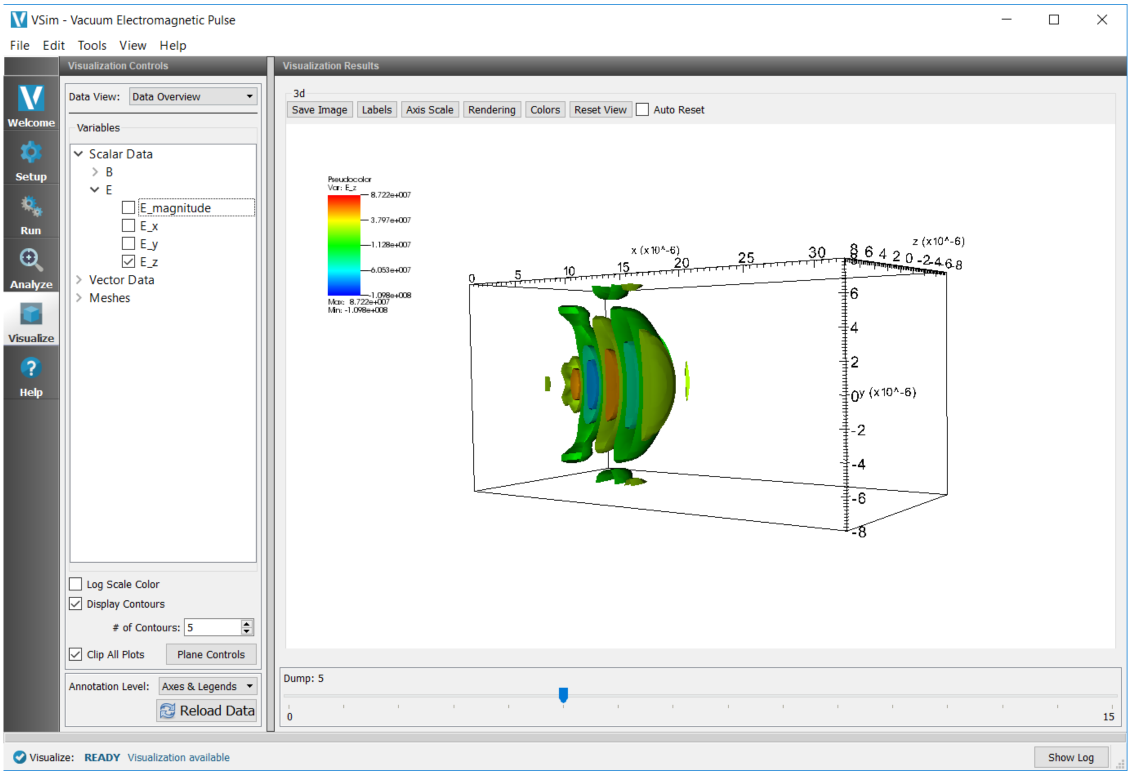
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The electric and magnetic field components can be found in the scalar data variables of the data overview tab.

- Make sure the *Data View* drop down is set to *Data Overview*.
- Here you can see *Variables*. Expand the *Scalar Data*.
- Expand E
- Select E_z
- Check the box next to *Clip All Plots*
- Check the box next to *Display Contours* and set the # of contours to 5
- Click and drag with your mouse to rotate the view

Initially, no field will be seen, as one is looking at Dump 0, the initial dump, when no fields are yet in the simulation. Move the slider at the bottom of the right pane to see the magnetic field at different times.



Data View: Data Overview

Variables

- Scalar Data
  - B
  - E
    - □ E_magnitude
    - □ E_x
    - □ E_y
    - ☑ E_z
- Vector Data
- Meshes

□ Log Scale Color

☑ Display Contours

# of Contours: 5

☑ Clip All Plots

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 5

Visualize: READY Visualization available

Fig. 2.15: Ez field at Dump 5.

Further Experiments

Increase NX to better resolve the wave and see whether it slips less with respect to the box.

Increase the pulse and box widths (you will also need to increase the number of cells in the transverse directions) to reduce diffraction.

2.1.6 Electrostatic Particle In Cell (esPtclInCell.sdf)

Keywords:

electrostatics, particle in cell, sheath

Problem description

This Electrostatic Particle in Cell example computes the electrostatic potential and field in a box with conducting walls and particle absorbers and with an immobile, background neutralizing charge density. The electrons move to the wall by the potential, creating a sheath.

This simulation can be performed with a VSimBase license.

Opening the Simulation

The electrostatic particle in cell example is accessed from within VSimComposer by the following actions:

- Select the *New → From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples* option.
- Select *Electrostatic Particle in Cell* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 2.16. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.

Simulation Properties

This simulation includes several constants for easy adjustment of simulation properties including:

- N_X, N_Y: The number of cells in each direction
- W_X, W_Y: The length of the domain in each direction
- PPC: The number of macroparticles per cell

The Parameters element contains several parameters useful for calculating basic plasma physics properties such as the plasma frequency and Debye length.

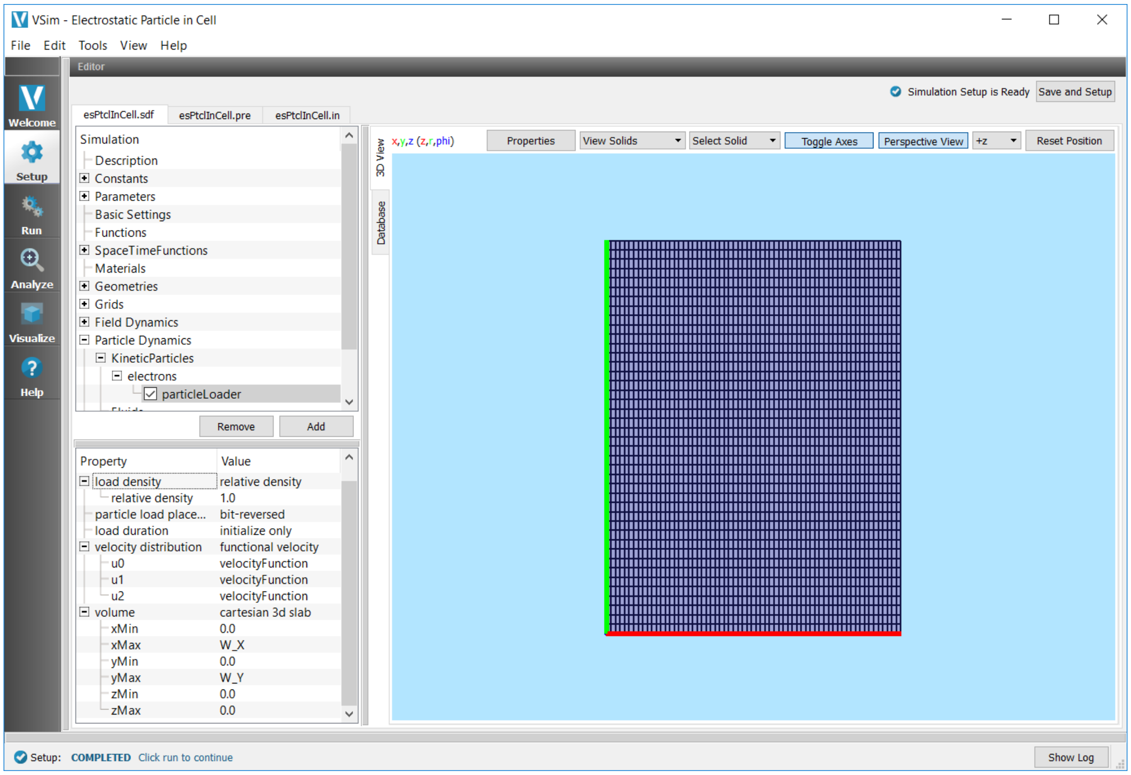
There is a SpaceTimeFunction used later in the setup to describe the thermal velocity of the electrons.

The simulation is periodic in y with Dirichlet boundary conditions in X set to zero.

Running the Simulation

After performing the above actions, continue as follows:

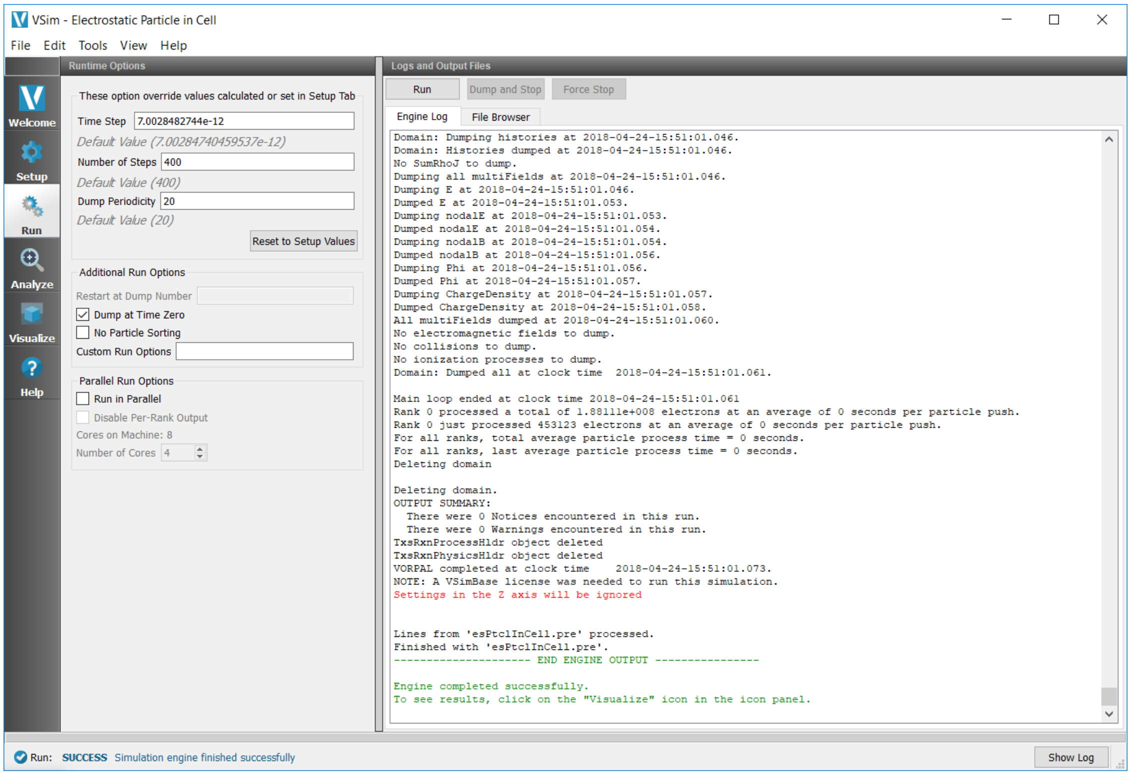
- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” A snapshot of the simulation run completion is shown in Fig. 2.17.



Property	Value
load density	relative density
relative density	1.0
particle load place...	bit-reversed
load duration	initialize only
velocity distribution	functional velocity
u0	velocityFunction
u1	velocityFunction
u2	velocityFunction
volume	cartesian 3d slab
xMin	0.0
xMax	W_X
yMin	0.0
yMax	W_Y
zMin	0.0
zMax	0.0

Setup: COMPLETED Click run to continue

Fig. 2.16: Setup Window for the electrostatic particle in cell example.



These option override values calculated or set in Setup Tab

Time Step 7.0028482744e-12

Default Value (7.00284740459537e-12)

Number of Steps 400

Default Value (400)

Dump Periodicity 20

Default Value (20)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☐ Run in Parallel

☐ Disable Per-Rank Output

Cores on Machine: 8

Number of Cores 4

```
Domain: Dumping histories at 2018-04-24-15:51:01.046.
Domain: Histories dumped at 2018-04-24-15:51:01.046.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-15:51:01.046.
Dumping E at 2018-04-24-15:51:01.046.
Dumped E at 2018-04-24-15:51:01.053.
Dumping nodalE at 2018-04-24-15:51:01.053.
Dumped nodalE at 2018-04-24-15:51:01.054.
Dumping nodalB at 2018-04-24-15:51:01.054.
Dumped nodalB at 2018-04-24-15:51:01.056.
Dumping Phi at 2018-04-24-15:51:01.056.
Dumped Phi at 2018-04-24-15:51:01.057.
Dumping ChargeDensity at 2018-04-24-15:51:01.057.
Dumped ChargeDensity at 2018-04-24-15:51:01.058.
All multiFields dumped at 2018-04-24-15:51:01.060.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-15:51:01.061.

Main loop ended at clock time 2018-04-24-15:51:01.061
Rank 0 processed a total of 1.88111e+008 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 453123 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-15:51:01.073.
NOTE: A VSimBase license was needed to run this simulation.
Settings in the Z axis will be ignored


Lines from 'esPtclInCell.pre' processed.
Finished with 'esPtclInCell.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 2.17: The Run Window at the end of execution.

Visualizing the Results

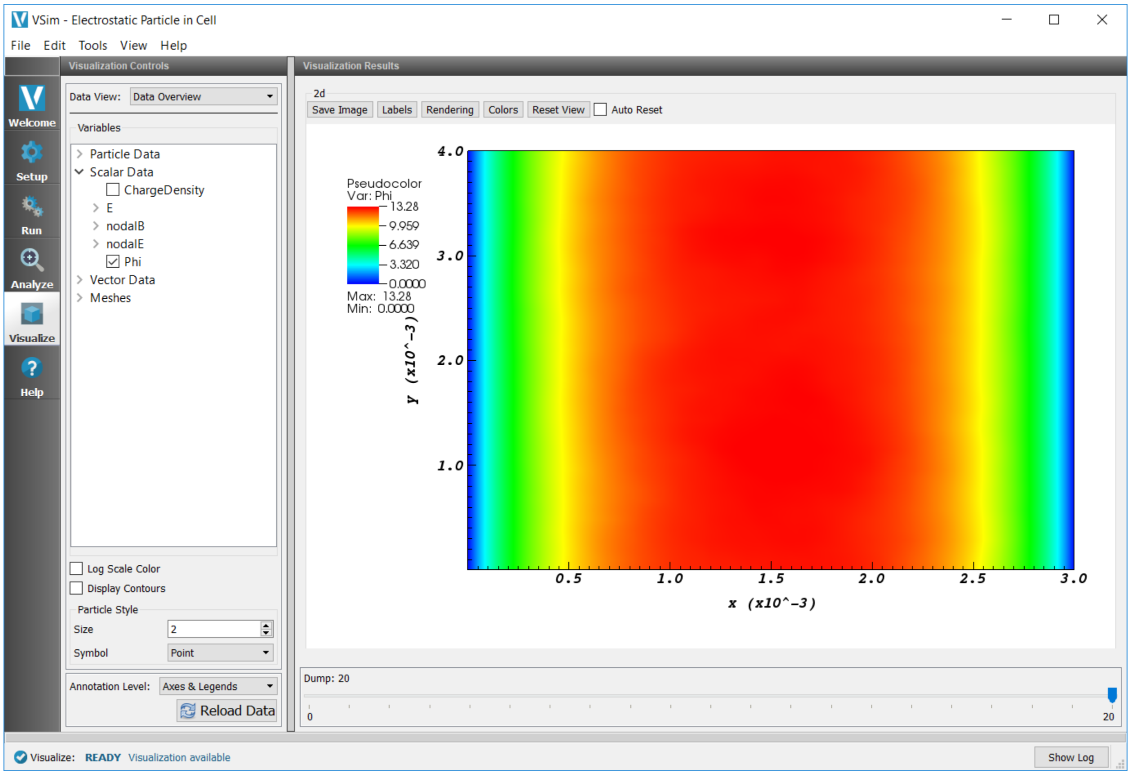
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the electric potential as shown in Fig. 2.18, do the following:

- Expand *Scalar Data*
- Select *Phi*

Move the dump slider forward in time to see the evolution of the field.



Data View: Data Overview

☐ ChargeDensity
☑ Phi

☐ Log Scale Color
☐ Display Contours

Size: 2
Symbol: Point

Annotation Level: Axes & Legends

☐ Auto Reset

Dump: 20

Visualize: READY Visualization available

Fig. 2.18: The Visualize Window showing the electric potential, *Phi*, at dump 20.

Further Experiments

Change the plasma density and see whether the frequency in the histories changes.

Use the computePtclNumDensity analysis script in the *Analyze* Tab to calculate the electron density at each dump and view the sheath formation.

2.1.7 Half-wave Antenna (halfWaveAntenna.sdf)

Keywords:

electromagnetics, antennas

Problem Description

The half wave antenna example describes a simple box source in a vacuum.

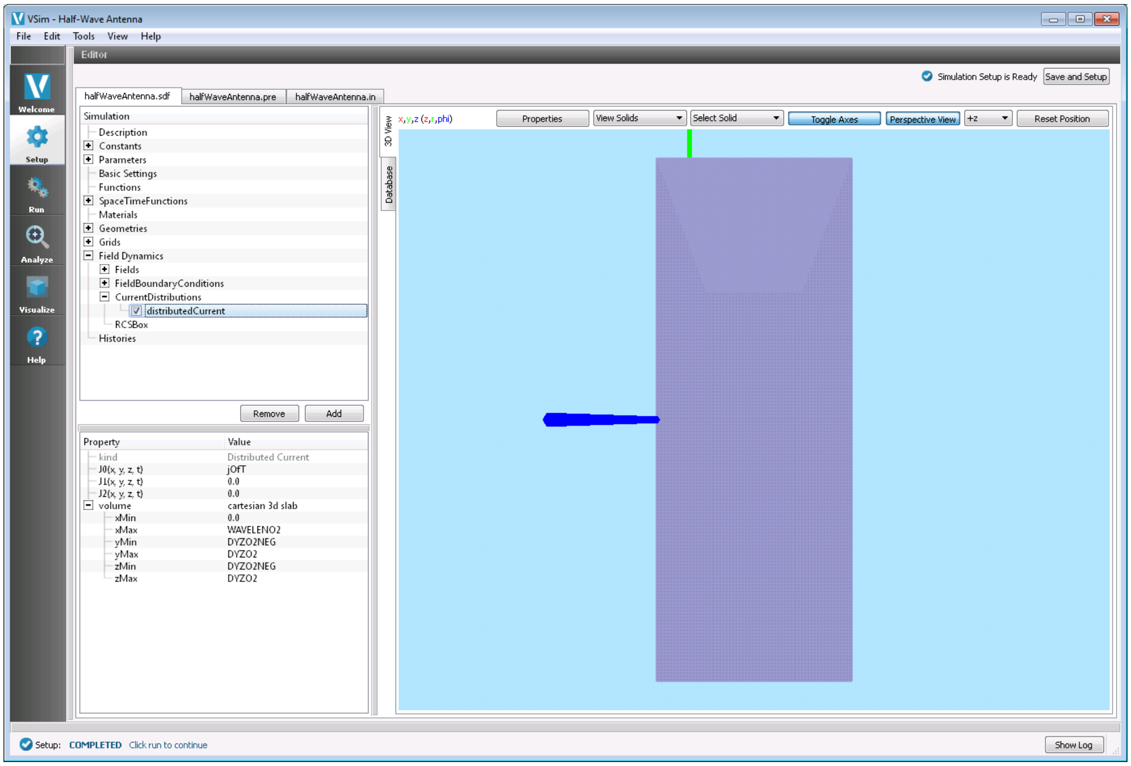
This simulation can be performed with any license.

Opening the Simulation

The Half Wave Antenna example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples* option.
- Select *Half Wave Antenna* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 2.19. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



Property	Value
kind	Distributed Current
J0(x, y, z, t)	jOfT
J1(x, y, z, t)	0.0
J2(x, y, z, t)	0.0
volume	cartesian 3d slab
xMin	0.0
xMax	WAVELENO2
yMin	DYZO2NEG
yMax	DYZO2
zMin	DYZO2NEG
zMax	DYZO2

Setup: COMPLETED Click run to continue

Fig. 2.19: Setup Window for the Half Wave Antenna example.

Simulation Properties

This example includes several constants for easy adjustment of simulation properties, Including:

- WAVELENGTH: The wavelength of the antenna

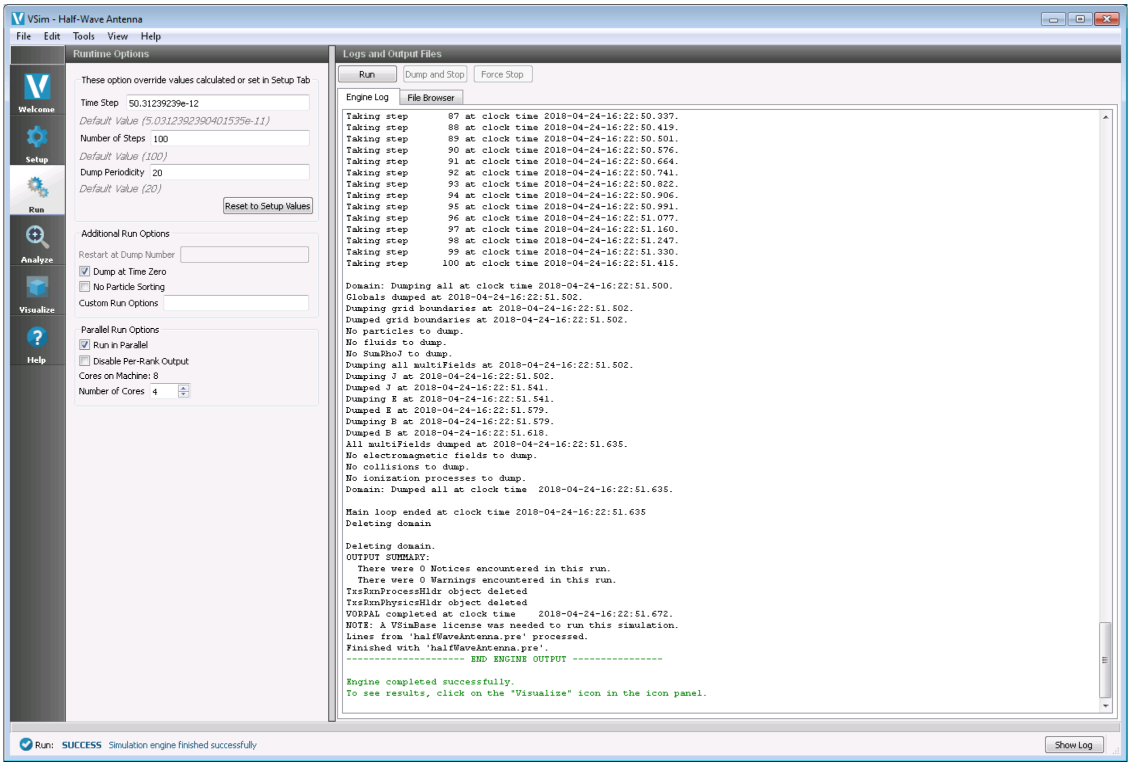
There is also a SpaceTimeFunction to define the current driver of the half wavelength source.

Other properties of the simulation include port boundaries on all sides except for the lower x boundary, which is a perfect electric conductor. A Distributed Current source is used to set the current of the half wavelegth source.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. of the window. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 2.20.



These option override values calculated or set in Setup Tab

Time Step 50.31239239e-12
Default Value (5.0312392390401535e-11)
Number of Steps 100
Default Value (100)
Dump Periodicity 20
Default Value (20)
Reset to Setup Values

Additional Run Options

Restart at Dump Number
☑ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
☐ Disable Per-Rank Output
Cores on Machine: 8
Number of Cores 4

```
Taking step      87 at clock time 2018-04-24-16:22:50.337.
Taking step      88 at clock time 2018-04-24-16:22:50.419.
Taking step      89 at clock time 2018-04-24-16:22:50.501.
Taking step      90 at clock time 2018-04-24-16:22:50.576.
Taking step      91 at clock time 2018-04-24-16:22:50.664.
Taking step      92 at clock time 2018-04-24-16:22:50.741.
Taking step      93 at clock time 2018-04-24-16:22:50.822.
Taking step      94 at clock time 2018-04-24-16:22:50.906.
Taking step      95 at clock time 2018-04-24-16:22:50.991.
Taking step      96 at clock time 2018-04-24-16:22:51.077.
Taking step      97 at clock time 2018-04-24-16:22:51.160.
Taking step      98 at clock time 2018-04-24-16:22:51.247.
Taking step      99 at clock time 2018-04-24-16:22:51.330.
Taking step     100 at clock time 2018-04-24-16:22:51.415.

Domain: Dumping all at clock time 2018-04-24-16:22:51.500.
Globals dumped at 2018-04-24-16:22:51.502.
Dumping grid boundaries at 2018-04-24-16:22:51.502.
Dumped grid boundaries at 2018-04-24-16:22:51.502.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-16:22:51.502.
Dumping J at 2018-04-24-16:22:51.502.
Dumped J at 2018-04-24-16:22:51.541.
Dumping E at 2018-04-24-16:22:51.541.
Dumped E at 2018-04-24-16:22:51.579.
Dumping B at 2018-04-24-16:22:51.579.
Dumped B at 2018-04-24-16:22:51.618.
All multiFields dumped at 2018-04-24-16:22:51.635.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-16:22:51.635.

Main loop ended at clock time 2018-04-24-16:22:51.635
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-16:22:51.672.
NOTE: A VSimBase license was needed to run this simulation.
Lines from 'halfWaveAntenna.pre' processed.
Finished with 'halfWaveAntenna.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 2.20: The Run Window at the end of execution.

Visualizing the Results

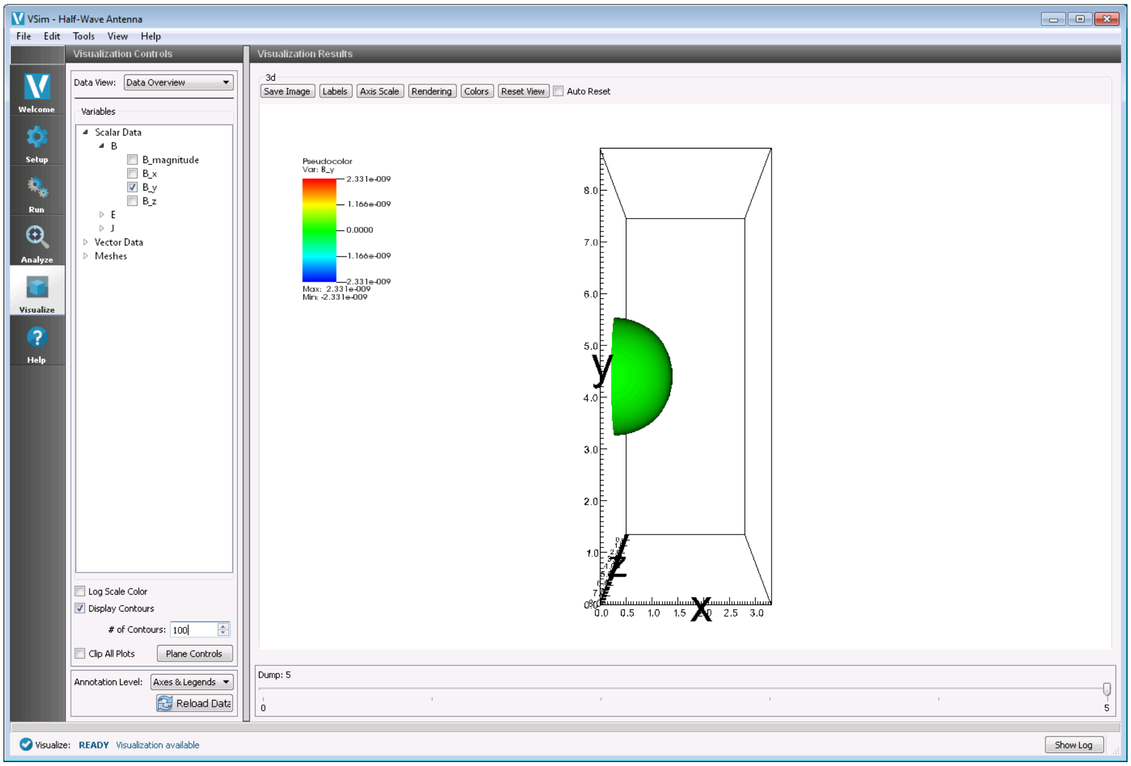
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the field pattern as shown in Fig. 2.21, do the following:

- Expand *Scalar Data*
- Expand *B*
- Select *B_y*

- Select *Display Contours* and set the # of Contours to 100
- Move the dump slider forward in time
- Rotate the plot by clicking and dragging with your mouse.



VSim - Half-Wave Antenna

Data View: Data Overview

Variables

Scalar Data
B
□ B_magnitude
□ B_x
☑ B_y
□ B_z
E
J
Vector Data
Meshes

□ Log Scale Color
☑ Display Contours
# of Contours: 100
□ Clip All Plots

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 5

Visualize: READY Visualization available

Fig. 2.21: Visualization of the wave pattern as a color contour plot.

Further Experiments

Additional experiments worth investigating are:

- Change the frequency of the source.

2.1.8 Parallel Plate Capacitor (parPlateCapacitor.sdf)

Keywords:

`electrostatics, parallel plate capacitor`

Problem description

This Parallel Plate Capacitor simulation computes the electrostatic potential and field for a parallel plate capacitor. It can be run in any number of dimensions. It is periodic in the y and z directions when they are present.

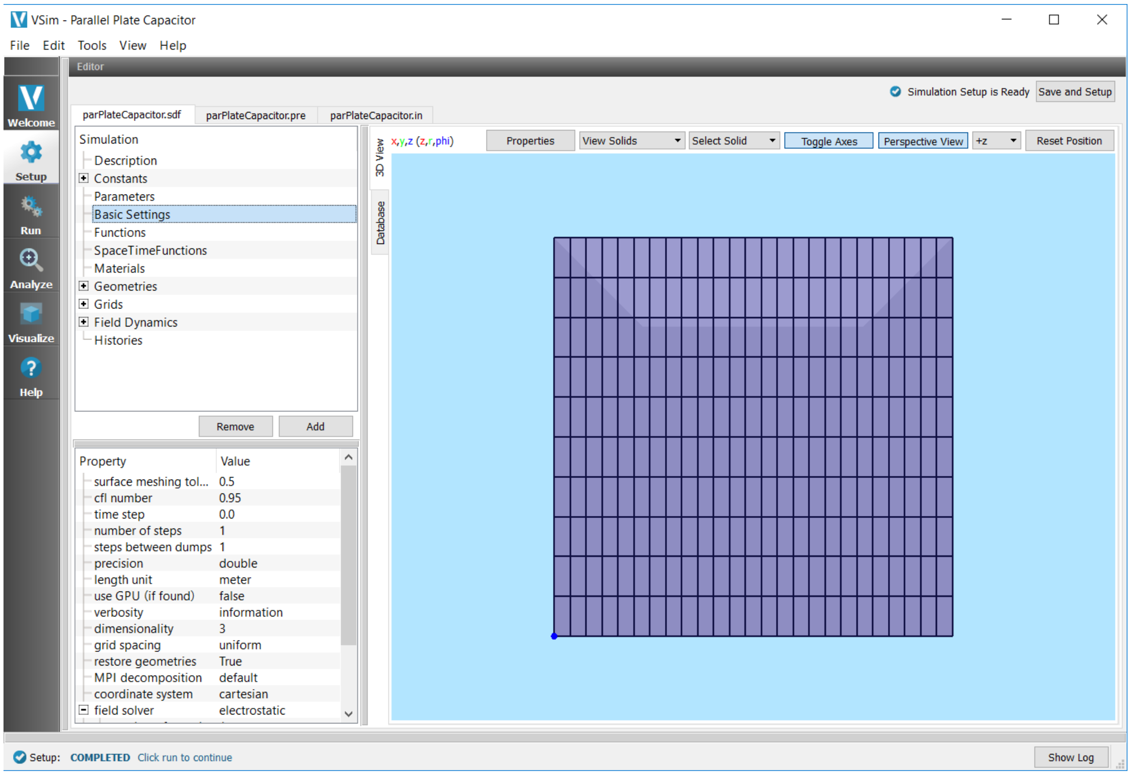
This simulation can be performed with a VSimBase license.

Opening the Simulation

The Parallel Plate Capacitor example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples* option.
- Select *Parallel Plate Capacitor* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 2.22. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`



Simulation Setup is Ready

Simulation
- Description
- ⊞ Constants
- Parameters
- Basic Settings
- Functions
- SpaceTimeFunctions
- Materials
- ⊞ Geometries
- ⊞ Grids
- ⊞ Field Dynamics
- Histories

Property	Value
surface meshing tol...	0.5
cfl number	0.95
time step	0.0
number of steps	1
steps between dumps	1
precision	double
length unit	meter
use GPU (if found)	false
verbosity	information
dimensionality	3
grid spacing	uniform
restore geometries	True
MPI decomposition	default
coordinate system	cartesian
⊟ field solver	electrostatic

Setup: COMPLETED Click run to continue

Fig. 2.22: Setup Window for the Parallel Plate Capacitor example.

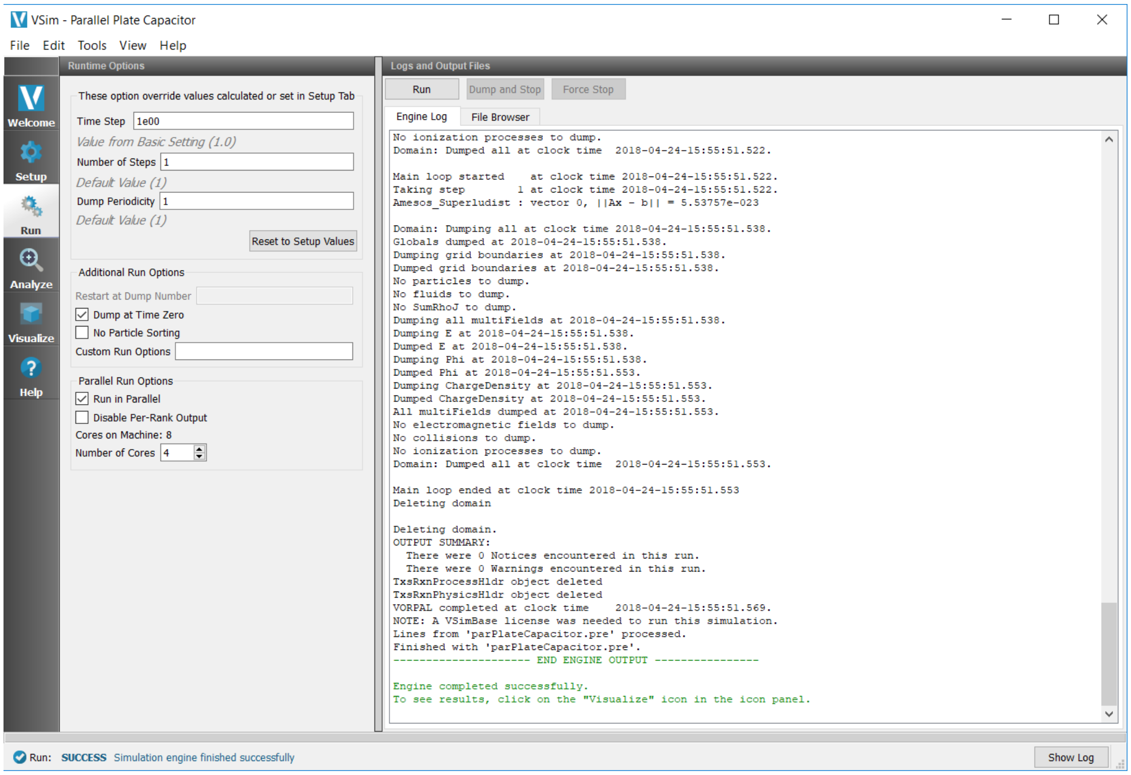
Simulation Properties

The Simulation Elements Tree and Property Editor allow one to choose the distance between the plates, width of the plates, voltage of the positive plate and the length of a time step (which is irrelevant as this is an electrostatic simulation)

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 2.23.



These option override values calculated or set in Setup Tab

Time Step 1e00

Value from Basic Setting (1.0)

Number of Steps 1

Default Value (1)

Dump Periodicity 1

Default Value (1)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Cores on Machine: 8

Number of Cores 4

```
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-15:55:51.522.

Main loop started    at clock time 2018-04-24-15:55:51.522.
Taking step        1 at clock time 2018-04-24-15:55:51.522.
Amesos_Superludist : vector 0, ||Ax - b|| = 5.53757e-023

Domain: Dumping all at clock time 2018-04-24-15:55:51.538.
Globals dumped at 2018-04-24-15:55:51.538.
Dumping grid boundaries at 2018-04-24-15:55:51.538.
Dumped grid boundaries at 2018-04-24-15:55:51.538.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-15:55:51.538.
Dumping E at 2018-04-24-15:55:51.538.
Dumped E at 2018-04-24-15:55:51.538.
Dumping Phi at 2018-04-24-15:55:51.538.
Dumped Phi at 2018-04-24-15:55:51.553.
Dumping ChargeDensity at 2018-04-24-15:55:51.553.
Dumped ChargeDensity at 2018-04-24-15:55:51.553.
All multiFields dumped at 2018-04-24-15:55:51.553.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-15:55:51.553.

Main loop ended at clock time 2018-04-24-15:55:51.553
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-15:55:51.569.
NOTE: A VSimBase license was needed to run this simulation.
Lines from 'parPlateCapacitor.pre' processed.
Finished with 'parPlateCapacitor.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 2.23: The Run Window at the end of execution.

Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To create the image shown in Fig. 2.24, proceed as follows:

- Expand *Scalar Data* and select the box next to *Phi*

Further Experiments

Change the gap between the plates by changing the length of the grid in the z direction and see how the electric field changes.

Change a width, e.g., LY and see whether it has an effect on the electric field.

Change the voltage on the right plate and see if it affects the electric field.

Set the periodic directions under *BasicSettings* to *none*, add boundary conditions on those directions and see how the field changes.

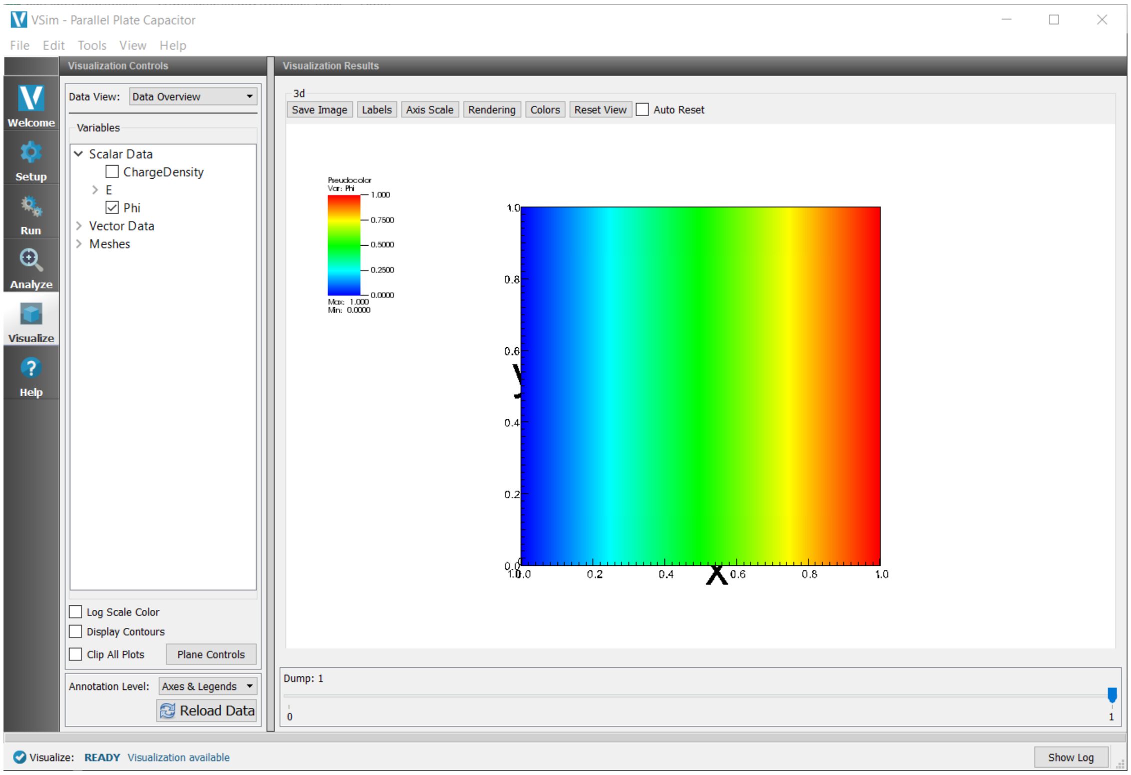


Fig. 2.24: Visualization of plane wave as a color contour plot.

2.1.9 Two-Stream Instability (twoStream.sdf)

Keywords:

`electromagnetics, two-stream instability`

Problem Description

The two-stream instability is a rapidly growing collision-less plasma instability arising from small charge imbalances. A local imbalance leads to the acceleration or deceleration of particles in its vicinity, which in turn leads to an even stronger imbalance. One setup that allows to easily observe the instability is two counter-streaming beams of identical charge in a periodic system. The advantage of this configuration is that the generated plasma wave becomes a standing wave, thus allowing to easily observe the formation of the phase space vortices.

In this example, we use two electron streams. At $t = 0$ the streams have drift velocities of magnitude 7.78×10^6 m/s. In order to accelerate the onset of the instability, the two particle beams are given a small sinusoidal perturbation in velocity space.

This simulation can be performed with a VSimBase license.

Opening the Simulation

The Two-Stream Instability example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.

- Expand the *Basic Examples* option.
- Select *Two-Stream Instability* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 2.25. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.

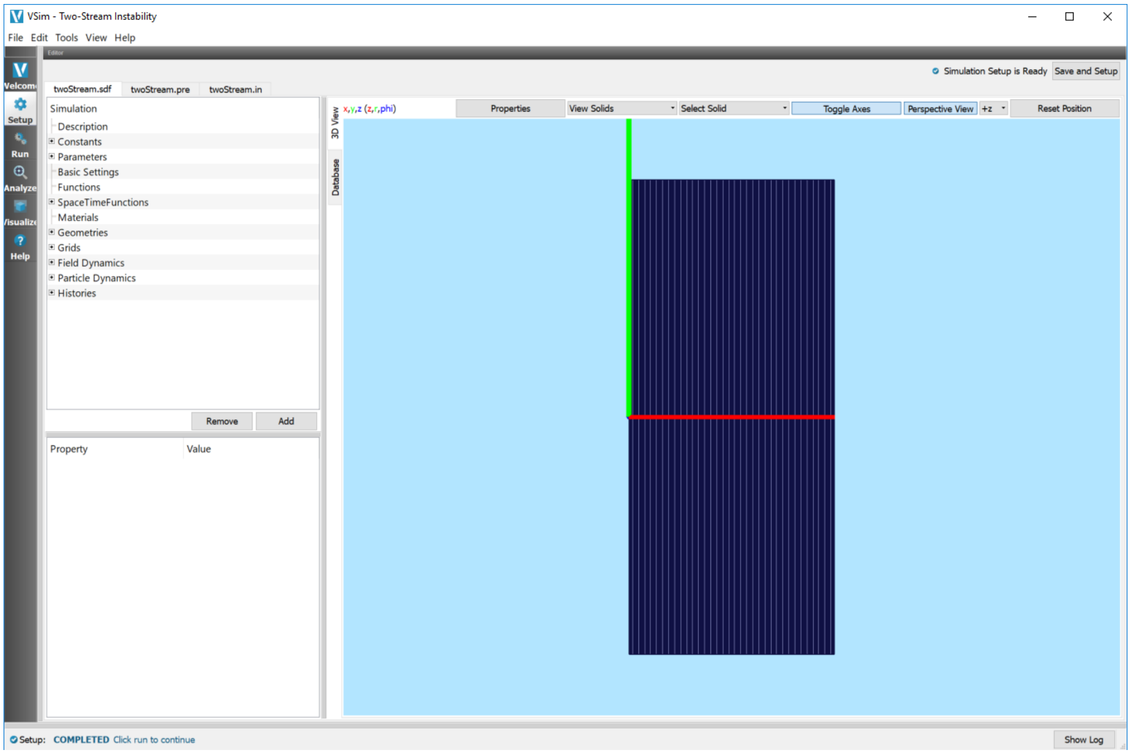


Fig. 2.25: Setup Window for the Two-Stream Instability example.

Simulation Properties

There are a number of *Constants* in this simulation to help make modifying the simulation even easier. Those include:

- XCELLS: The number of cells
- NOM_DEN_E: The electron density
- VBAR: The average velocity
- WAVELENGTHS: The number of wavelengths in the domain to simulate
- PPC: The number of particles per cell.

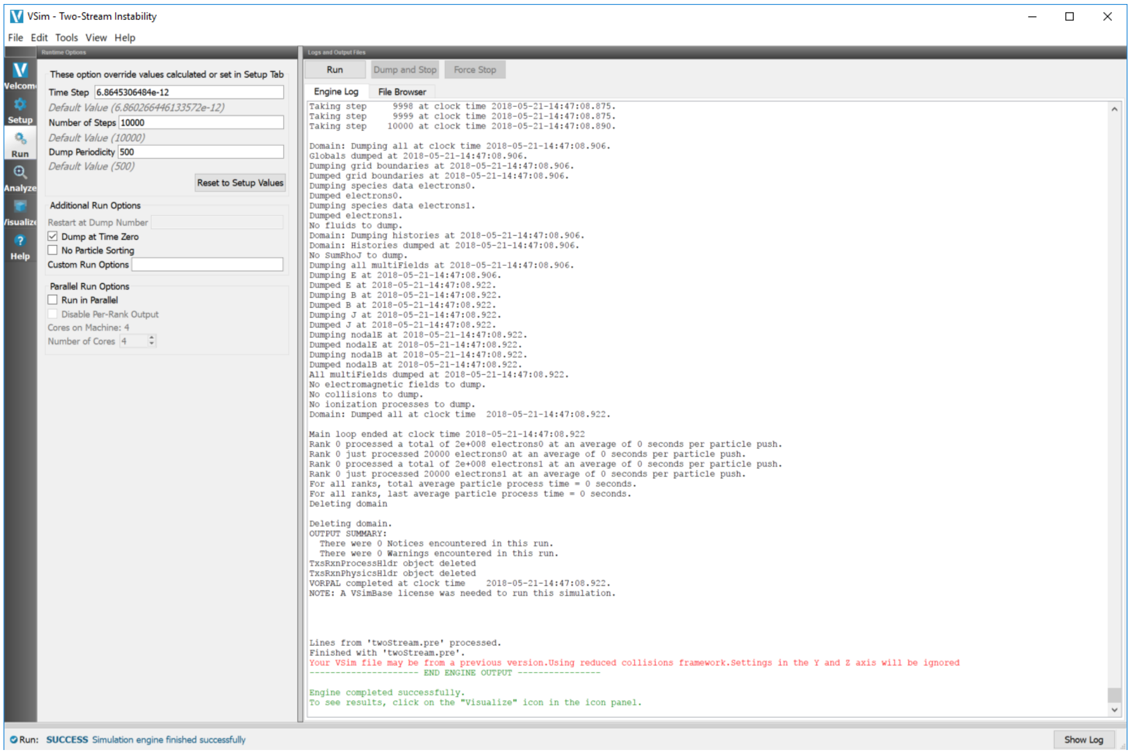
SpaceTimeFunctions are used to set the velocities of each particle stream.

The simulation is 1 dimensional and periodic in x.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper right corner of the window. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 2.26.



These option override values calculated or set in Setup Tab

Time Step 6.8645306484e-12
Default Value (6.860266446133572e-12)
Number of Steps 10000
Default Value (10000)
Dump Periodicity 500
Default Value (500)

Additional Run Options
Restart at Dump Number
☑ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options
☐ Run in Parallel
☐ Disable Per-Rank Output
Cores on Machine: 4
Number of Cores 4

```
Taking step     9998 at clock time 2018-05-21-14:47:08.875.
Taking step     9999 at clock time 2018-05-21-14:47:08.875.
Taking step    10000 at clock time 2018-05-21-14:47:08.890.

Domain: Dumping all at clock time 2018-05-21-14:47:08.906.
Globals dumped at 2018-05-21-14:47:08.906.
Dumping grid boundaries at 2018-05-21-14:47:08.906.
Dumped grid boundaries at 2018-05-21-14:47:08.906.
Dumping species data electrons0.
Dumped electrons0.
Dumping species data electrons1.
Dumped electrons1.
No fluids to dump.
Domain: Dumping histories at 2018-05-21-14:47:08.906.
Domain: Histories dumped at 2018-05-21-14:47:08.906.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-21-14:47:08.906.
Dumping E at 2018-05-21-14:47:08.906.
Dumped E at 2018-05-21-14:47:08.922.
Dumping B at 2018-05-21-14:47:08.922.
Dumped B at 2018-05-21-14:47:08.922.
Dumping J at 2018-05-21-14:47:08.922.
Dumped J at 2018-05-21-14:47:08.922.
Dumping nodalE at 2018-05-21-14:47:08.922.
Dumped nodalE at 2018-05-21-14:47:08.922.
Dumping nodalB at 2018-05-21-14:47:08.922.
Dumped nodalB at 2018-05-21-14:47:08.922.
All multiFields dumped at 2018-05-21-14:47:08.922.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-21-14:47:08.922.

Main loop ended at clock time 2018-05-21-14:47:08.922
Rank 0 processed a total of 2e+008 electrons0 at an average of 0 seconds per particle push.
Rank 0 just processed 20000 electrons0 at an average of 0 seconds per particle push.
Rank 0 processed a total of 2e+008 electrons1 at an average of 0 seconds per particle push.
Rank 0 just processed 20000 electrons1 at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-05-21-14:47:08.922.
NOTE: A VSimBase license was needed to run this simulation.




Lines from 'twoStream.pre' processed.
Finished with 'twoStream.pre'.
Your VSim file may be from a previous version.Using reduced collisions framework.Settings in the Y and Z axis will be ignored
-------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 2.26: The Run Window at the end of execution.

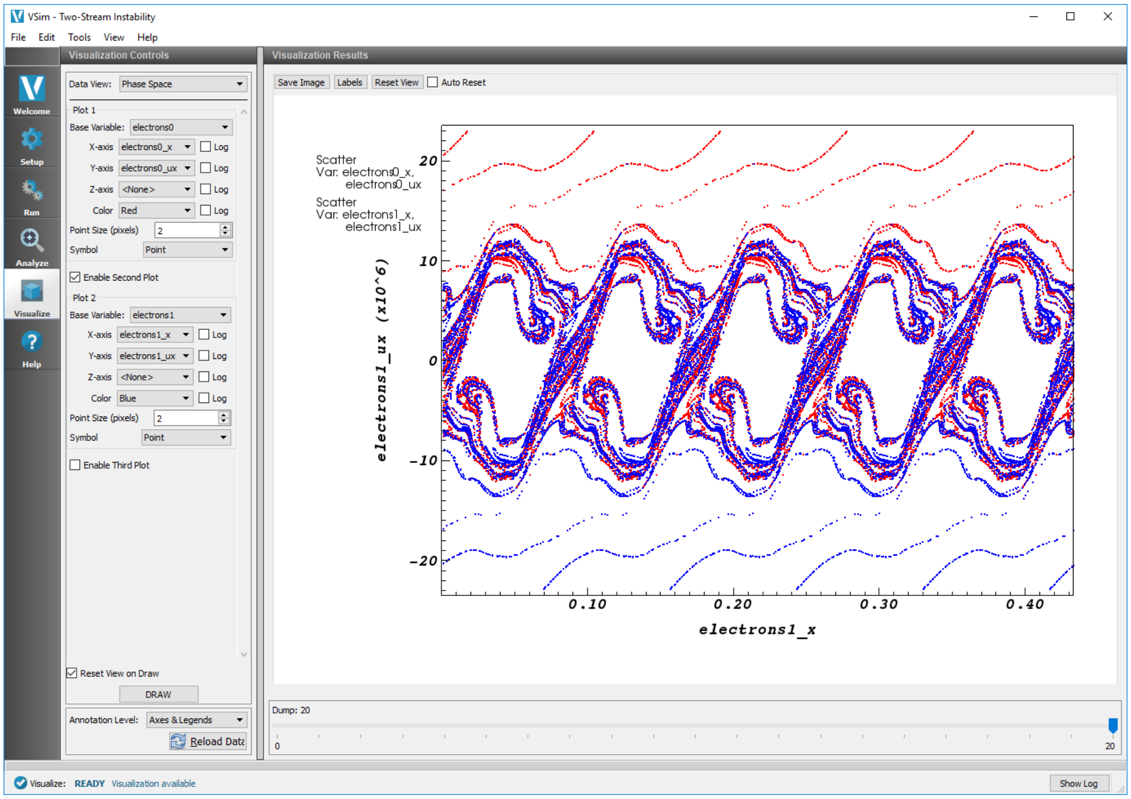
Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the instability as shown in Fig. 2.27, do the following:

- Select the *Phase Space* option from the *Data View* menu
- In the Plot 1 box, change the *X-axis* to *electrons0_x*, and the *Y-axis* to *electrons0_ux*
- Click the *Enable Second Plot* box
- In the Plot 2 box, change the *Base Variable* to *electrons1*, the *X-axis* to *electrons1_x*, and the *Y-axis* to *electrons1_ux*
- Click the *DRAW* button at the bottom, then move the *Dump* slider forward in time.



Data View: Phase Space

Plot 1
Base Variable: electrons0
X-axis electrons0_x □ Log
Y-axis electrons0_ux □ Log
Z-axis <None> □ Log
Color Red □ Log
Point Size (pixels) 2
Symbol Point

☑ Enable Second Plot

Plot 2
Base Variable: electrons1
X-axis electrons1_x □ Log
Y-axis electrons1_ux □ Log
Z-axis <None> □ Log
Color Blue □ Log
Point Size (pixels) 2
Symbol Point

□ Enable Third Plot

☑ Reset View on Draw

Annotation Level: Axes & Legends

Fig. 2.27: Visualization of the two-stream instability developing in particle phase space.

Further Experiments

Change the average velocity and velocity modulation and see how the speed at which the instability sets in depends on the modulation.

View the particle density by using the *computePtclNumDensity* script in the Analyze Window.

2.2 Basic Examples (text-based setup)

2.2.1 Magnetic Fields of Wire (bFieldByJT.pre)

Keywords:

Calculating A vector by a current carrying long linear wire.

Problem description

This simulation illustrates how to model magnetostatics. A straight current, J_o, is directed along the z-axis. The example solves Poisson's equation for the vector potential, A. The 0th dump of the simulation is the analytical solution for the purpose of comparison.

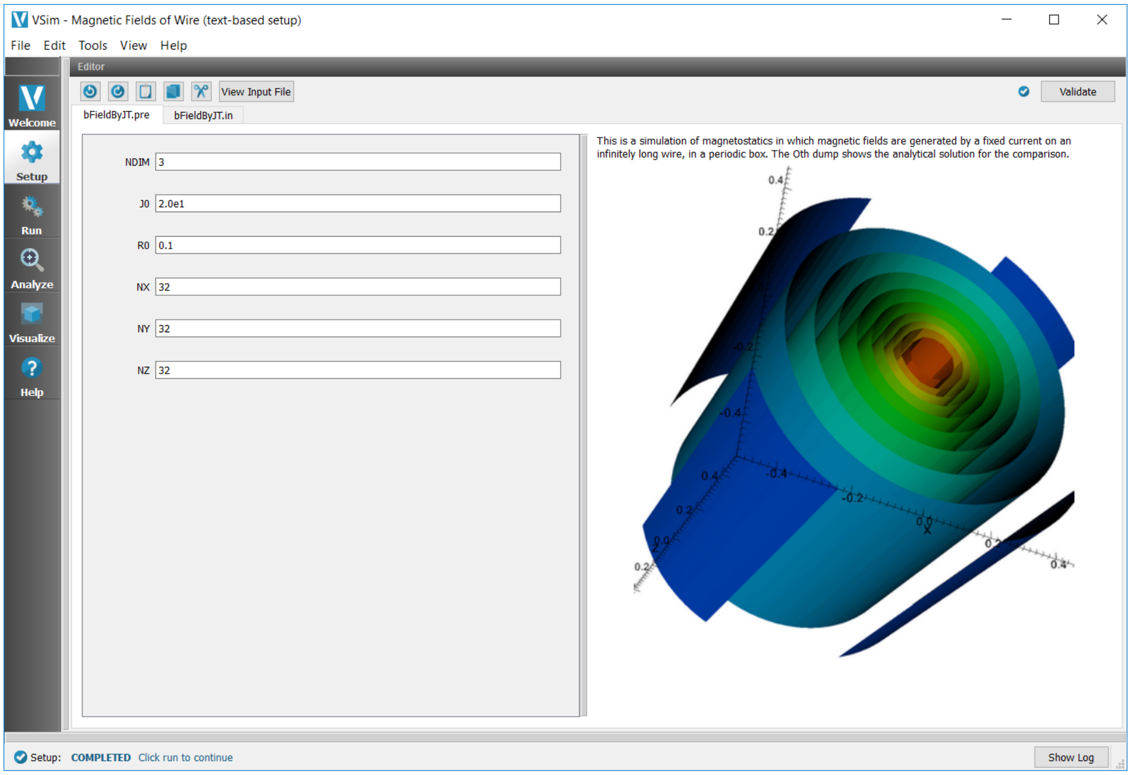
This simulation can be performed with a VSimBase license.

Opening the Simulation

The magnetic field by a current example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples (text-based setup)* option.
- Select "Magnetic Fields of Wire (text-based setup)" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 2.28.



NDIM 3

J0 2.0e1

R0 0.1

NX 32

NY 32

NZ 32

This is a simulation of magnetostatics in which magnetic fields are generated by a fixed current on an infinitely long wire, in a periodic box. The 0th dump shows the analytical solution for the comparison.

Setup: COMPLETED Click run to continue

Fig. 2.28: Setup Window for the magnetic field by a current.

Input File Features

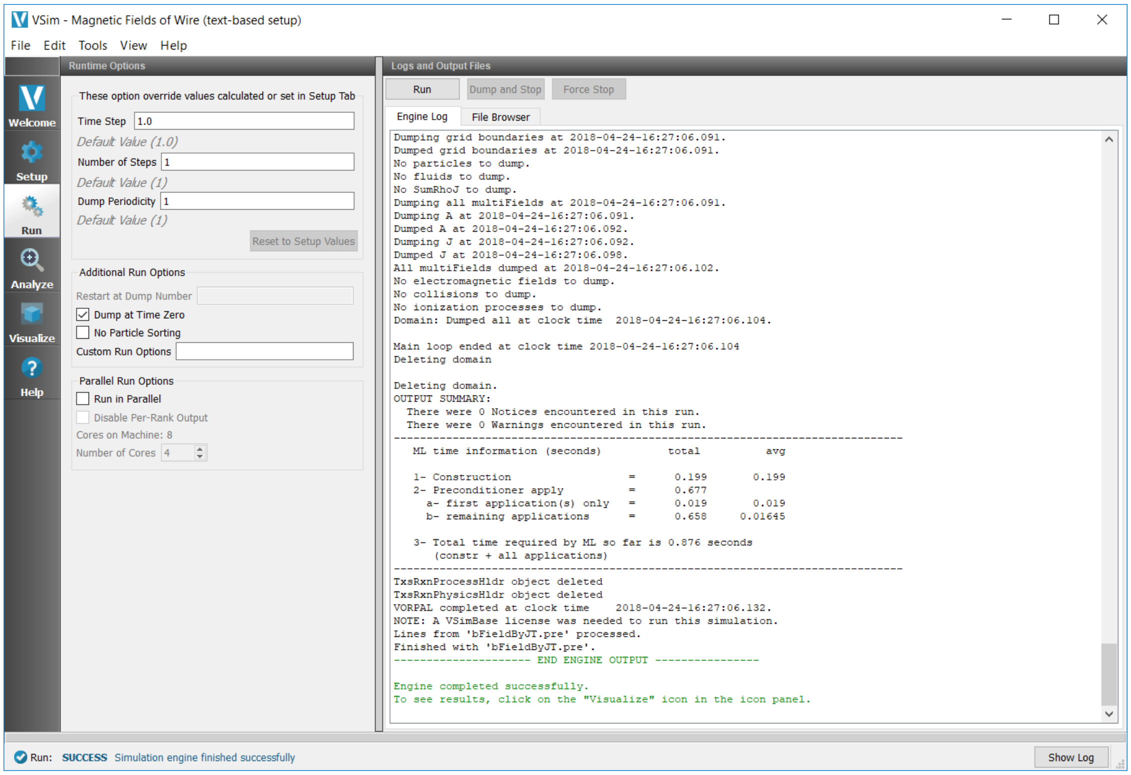
VSimComposer allows the user to vary the applied current and a radius of the wire. These are the main parameters that affect the A vector. By changing these parameters, a user can run simulations to explore how the A vector depends on the applied current and radius. The input file, when viewed in the editor, also exposes all the grid sizes of the simulation that are used by the implemented models.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.

- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 2.29.



These option override values calculated or set in Setup Tab

Time Step 1.0
Default Value (1.0)
Number of Steps 1
Default Value (1)
Dump Periodicity 1
Default Value (1)

Additional Run Options
Restart at Dump Number
☑ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options
□ Run in Parallel
□ Disable Per-Rank Output
Cores on Machine: 8
Number of Cores 4

```
Dumping grid boundaries at 2018-04-24-16:27:06.091.
Dumped grid boundaries at 2018-04-24-16:27:06.091.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-16:27:06.091.
Dumping A at 2018-04-24-16:27:06.091.
Dumped A at 2018-04-24-16:27:06.092.
Dumping J at 2018-04-24-16:27:06.092.
Dumped J at 2018-04-24-16:27:06.098.
All multiFields dumped at 2018-04-24-16:27:06.102.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-16:27:06.104.

Main loop ended at clock time 2018-04-24-16:27:06.104
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
------------------------------------------------------------------------------
   ML time information (seconds)          total          avg

   1- Construction                  =      0.199       0.199
   2- Preconditioner apply          =      0.677
     a- first application(s) only   =      0.019       0.019
     b- remaining applications      =      0.658     0.01645

   3- Total time required by ML so far is 0.876 seconds
      (constr + all applications)
------------------------------------------------------------------------------
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-16:27:06.132.
NOTE: A VSimBase license was needed to run this simulation.
Lines from 'bFieldByJT.pre' processed.
Finished with 'bFieldByJT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 2.29: The Run Window at the end of execution.

Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

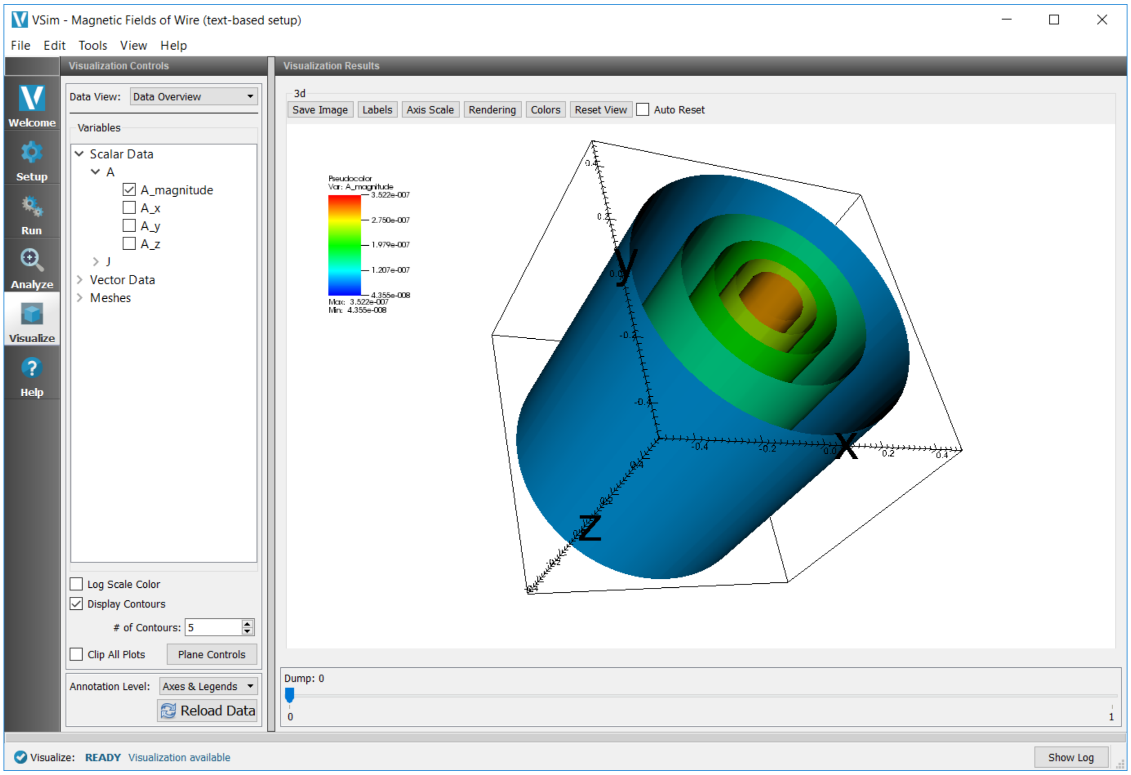
For this example, one can see the A vector fields. To see the A vector, continue as follows:

- Make sure the *Data View* drop down is set to *Data Overview*.
- Here you can see *Variables*. Expand the *Scalar Data*. Then expand *A*
- Four variables are available, the three components of *A* and the magnitude of the vector. Select A_magnitude.
- Check the box next to *Display Contours*
- Rotate the view by clicking and dragging your mouse.

Fig. 2.30 shows the visualization seen for A_magnitude.

Further Experiments

This input file can be modified to test different current, and wire radius. This will allow users to study how to use the magnetostatics capability in Vorpal.



Data View: Data Overview

☑ A_magnitude

□ Log Scale Color

☑ Display Contours

# of Contours: 5

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 2.30: Visualization of magnetic fields by the current as a color contour plot.

2.2.2 Oscillating Dipole Above Conducting Plane (emOscDipoleAboveConductorT.pre)

Keywords:

`emOscDipoleAboveConductorT, radiation`

Problem Description

This problem consists of an infintesimally short dipole located a variable height and orientation above a conducting plane. This simulation sets up a multifield, which can than have its electric and magnetic fields visualized to see how the distance between, and orientation of the dipole relative to the antenna effects these fields.

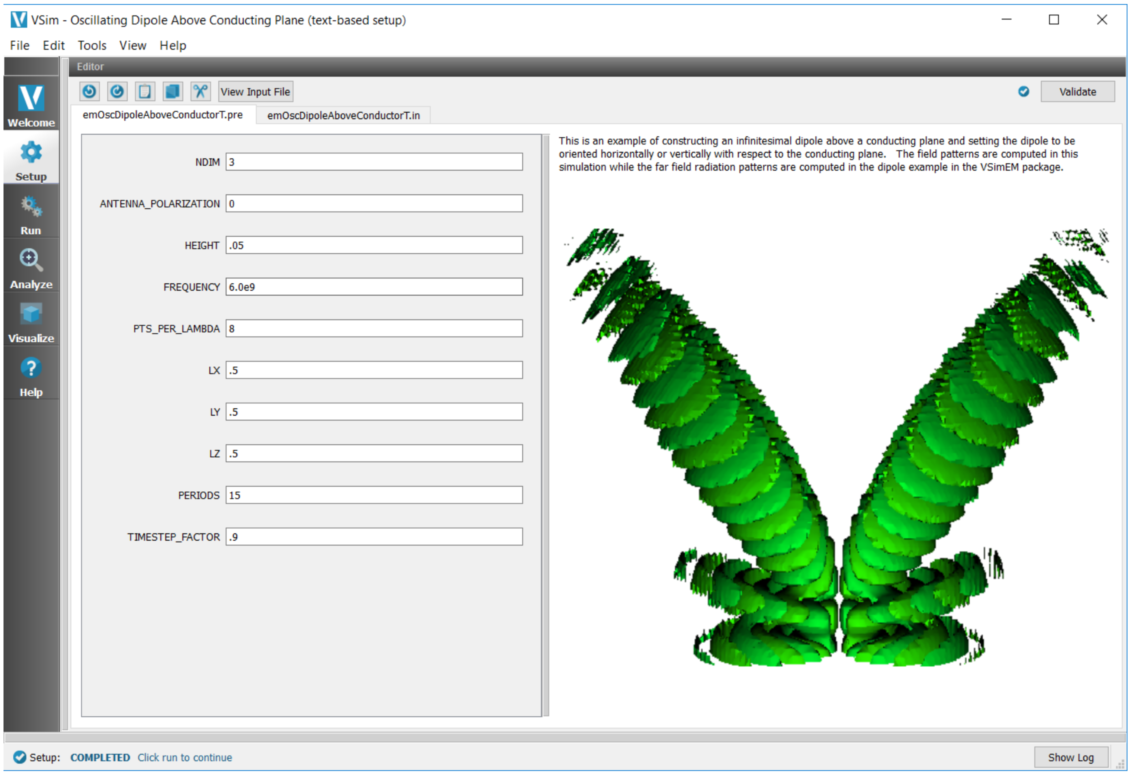
This simulation can be performed with a VSimBase license.

Opening the Simulation

The Dipole Above Conducting Plane example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples (text-based setup)* option.
- Select *Dipole Above Conducting Plane (text-based setup)* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 2.31.


VSim - Oscillating Dipole Above Conducting Plane (text-based setup)
File Edit Tools View Help
Editor
View Input File
Validate
emOscDipoleAboveConductorT.pre
emOscDipoleAboveConductorT.in
Welcome
Setup
Run
Analyze
Visualize
Help
Show Log

NDIM 3

ANTENNA_POLARIZATION 0

HEIGHT .05

FREQUENCY 6.0e9

PTS_PER_LAMBDA 8

LX .5

LY .5

LZ .5

PERIODS 15

TIMESTEP_FACTOR .9

This is an example of constructing an infinitesimal dipole above a conducting plane and setting the dipole to be oriented horizontally or vertically with respect to the conducting plane. The field patterns are computed in this simulation while the far field radiation patterns are computed in the dipole example in the VSimEM package.

Setup: COMPLETED Click run to continue

Fig. 2.31: Setup Window for the Dipole Above Conducting Plane example.

Input File Features

This file has key parameters to adjust the antenna polarization, height and operating frequency. In addition the number of points per wavelength as well as simulation domain size and runtime can be adjusted. Finally the length of one timestep can be adjusted slightly. The input file itself consists of a multifield with 5 open boundary conditions, and one conducting boundary condition to simulate the ground plane.

Running the Simulation

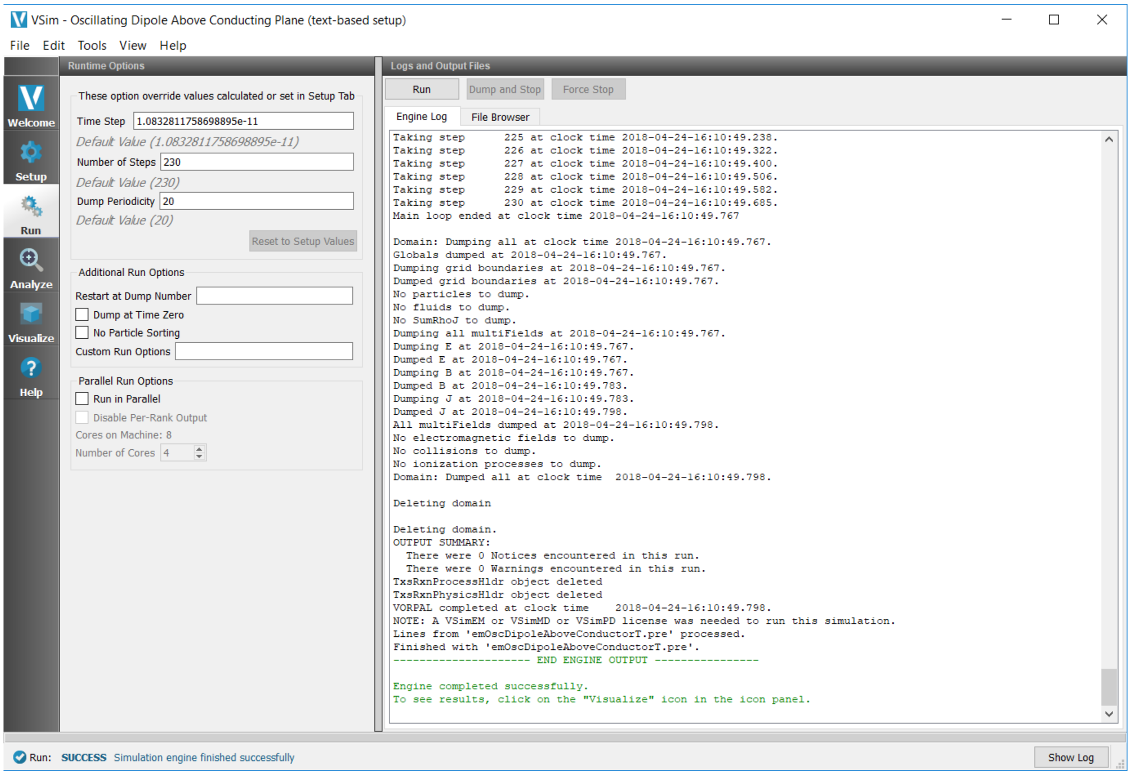
After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner. of the *Logs and output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in the window below.

Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.



These option override values calculated or set in Setup Tab

Time Step 1.0832811758698895e-11

Default Value (1.0832811758698895e-11)

Number of Steps 230

Default Value (230)

Dump Periodicity 20

Default Value (20)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

□ Run in Parallel

□ Disable Per-Rank Output

Cores on Machine: 8

Number of Cores 4

```
Taking step      225 at clock time 2018-04-24-16:10:49.238.
Taking step      226 at clock time 2018-04-24-16:10:49.322.
Taking step      227 at clock time 2018-04-24-16:10:49.400.
Taking step      228 at clock time 2018-04-24-16:10:49.506.
Taking step      229 at clock time 2018-04-24-16:10:49.582.
Taking step      230 at clock time 2018-04-24-16:10:49.685.
Main loop ended at clock time 2018-04-24-16:10:49.767

Domain: Dumping all at clock time 2018-04-24-16:10:49.767.
Globals dumped at 2018-04-24-16:10:49.767.
Dumping grid boundaries at 2018-04-24-16:10:49.767.
Dumped grid boundaries at 2018-04-24-16:10:49.767.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-16:10:49.767.
Dumping E at 2018-04-24-16:10:49.767.
Dumped E at 2018-04-24-16:10:49.767.
Dumping B at 2018-04-24-16:10:49.767.
Dumped B at 2018-04-24-16:10:49.783.
Dumping J at 2018-04-24-16:10:49.783.
Dumped J at 2018-04-24-16:10:49.798.
All multiFields dumped at 2018-04-24-16:10:49.798.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-16:10:49.798.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-16:10:49.798.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.
Lines from 'emOscDipoleAboveConductorT.pre' processed.
Finished with 'emOscDipoleAboveConductorT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 2.32: The Run Window at the end of execution.

The electric and magnetic field components can be found in the scalar data variables of the data overview tab.

- Make sure the *Data View* drop down is set to *Data Overview*.
- Here you can see *Variables*. Expand the *Scalar Data*.
- Expand E
- Select E_y
- Check the box next to *Display Contours* and set the # of contours to 10

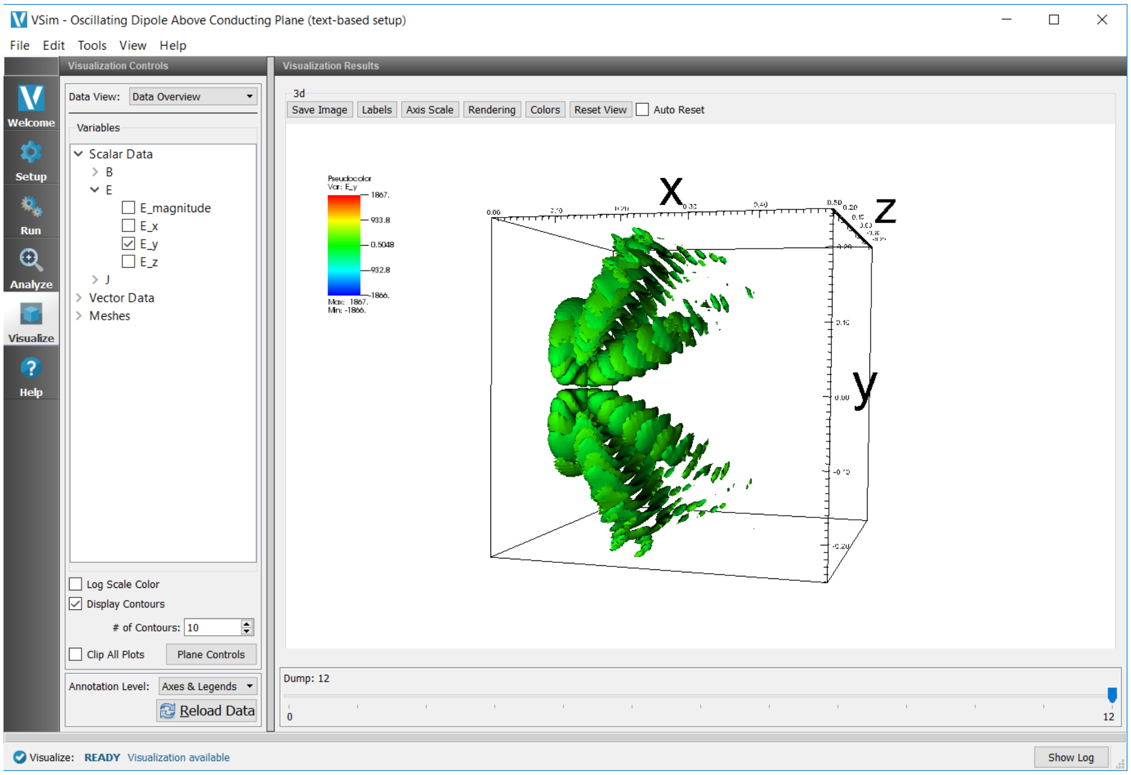
Initially, no field will be seen, as one is looking at Dump 0, the initial dump, when no fields are yet in the simulation. Move the slider at the bottom of the right pane to see the electric field at different times.

Further Experiments

In this example the "infinite" electric conductor is simulated by a physical conducting boundary at the bottom of the simulation. It would be possible to achieve the same results by having a second, equal infintesimal dipole placed the same height "below" the conducting plane.

The number of "lobes" visible in the far field is dependent on Antenna Orientation and height. If vertically oriented there will be 2*Height/Wavelength +1 lobes.A horizontally oriented dipole will produce 2*Height/Wavelength lobes. This can be a bit difficult to visualize using just E-field data as it must be properly thresholded. The lobes will be easier to see in the example Advanced Dipole Above Conductor, a part of the VSimEM package.

By adjusting the TIMESTEP_FACTOR timesteps can be made larger. If they get to large the simulation will become unstable.



Data View: Data Overview

☐ E_magnitude
☐ E_x
☑ E_y
☐ E_z

☐ Log Scale Color
☑ Display Contours
# of Contours: 10
☐ Clip All Plots

Annotation Level: Axes & Legends

Dump: 12

Visualize: READY Visualization available

Fig. 2.33: The electric field

To improve computational speed the size of the simulation domain can be optimized by adjusting LX/LY/LZ and PTS_PER_LAMBDA.

2.2.3 Electromagnetic Plane Wave (emPlaneWaveT.pre)

Keywords:

`electromagnetics, plane wave, periodic boundary conditions, wave launcher`

Problem Description

A linearly-polarized (with electric field in the y-direction) electromagnetic pulse with a sinusoidal amplitude on a plane wave is launched from the left side (x=0) to propagate in the x-direction. The transverse (y,z) boundary conditions are periodic.

This simulation can be performed with a VSimBase license.

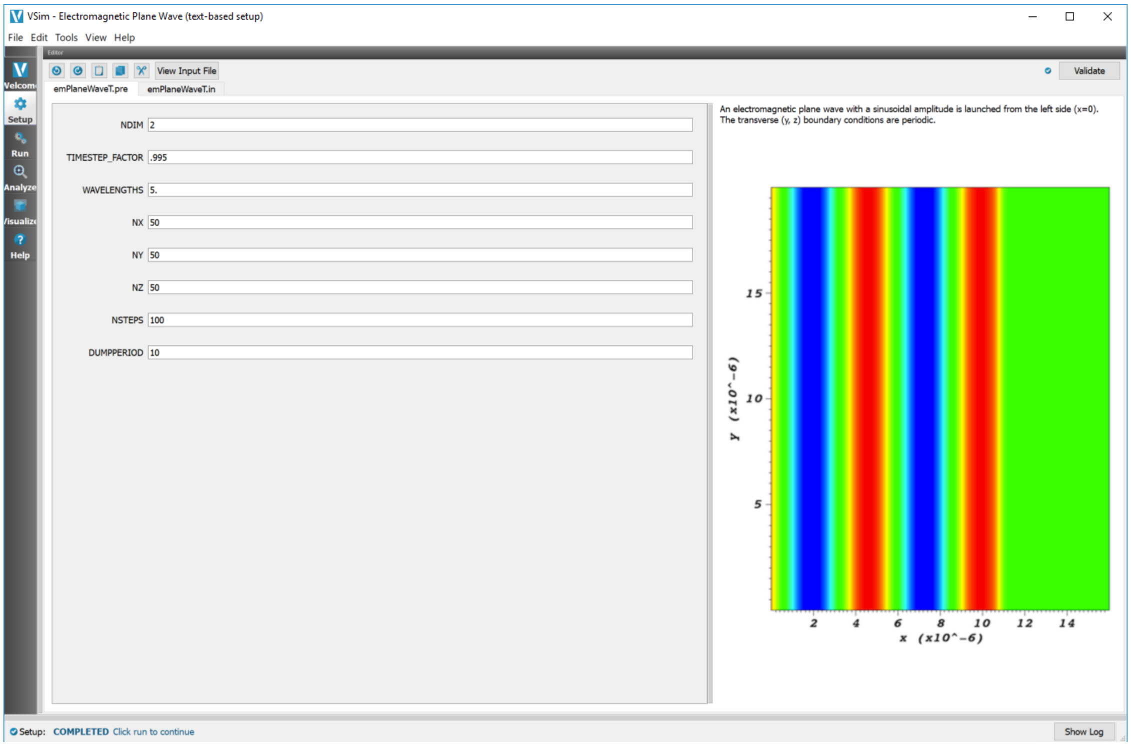
Opening the Simulation

The Electromagnetic Plane Wave example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples (text-based setup)* option.

- Select *Electromagnetic Plane Wave (text-based setup)* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 2.34.



VSim - Electromagnetic Plane Wave (text-based setup)

NDIM 2

TIMESTEP_FACTOR .995

WAVELENGTHS 5.

NX 50

NY 50

NZ 50

NSTEPS 100

DUMPPERIOD 10

An electromagnetic plane wave with a sinusoidal amplitude is launched from the left side (x=0).
The transverse (y, z) boundary conditions are periodic.

Setup: COMPLETED Click run to continue

Fig. 2.34: Setup Window for the Electromagnetic Plane Wave example.

Input File Features

The input file sets the wavelength to be one-third of the box length. It computes the TIMESTEP stability limit for the time step and sets the actual time step to TIMESTEP_FACTOR times the stability limit for the time step.

Running the Simulation

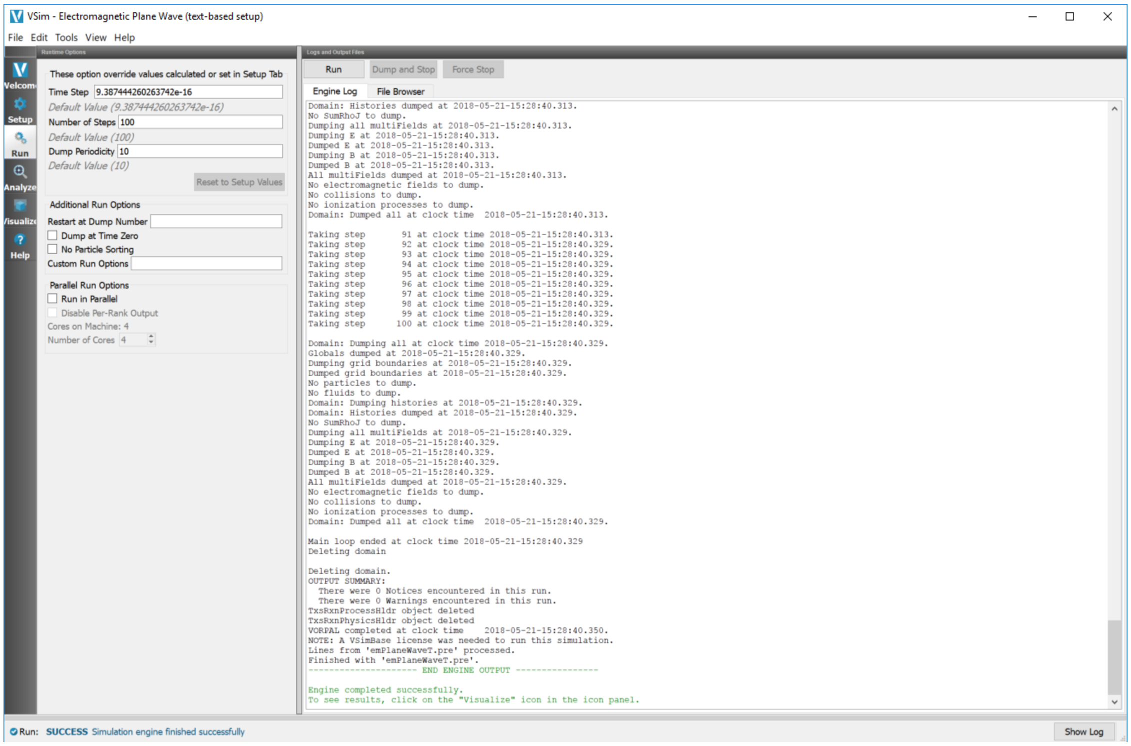
After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper right corner. of the window. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 2.35.

Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.



These option override values calculated or set in Setup Tab

Time Step 9.387444260263742e-16
Default Value (9.387444260263742e-16)
Number of Steps 100
Default Value (100)
Dump Periodicity 10
Default Value (10)

Additional Run Options

Restart at Dump Number
□ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options

□ Run in Parallel
□ Disable Per-Rank Output
Cores on Machine: 4
Number of Cores 4

```
Domain: Histories dumped at 2018-05-21-15:28:40.313.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-21-15:28:40.313.
Dumping E at 2018-05-21-15:28:40.313.
Dumped E at 2018-05-21-15:28:40.313.
Dumping B at 2018-05-21-15:28:40.313.
Dumped B at 2018-05-21-15:28:40.313.
All multiFields dumped at 2018-05-21-15:28:40.313.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-21-15:28:40.313.

Taking step       91 at clock time 2018-05-21-15:28:40.313.
Taking step       92 at clock time 2018-05-21-15:28:40.329.
Taking step       93 at clock time 2018-05-21-15:28:40.329.
Taking step       94 at clock time 2018-05-21-15:28:40.329.
Taking step       95 at clock time 2018-05-21-15:28:40.329.
Taking step       96 at clock time 2018-05-21-15:28:40.329.
Taking step       97 at clock time 2018-05-21-15:28:40.329.
Taking step       98 at clock time 2018-05-21-15:28:40.329.
Taking step       99 at clock time 2018-05-21-15:28:40.329.
Taking step      100 at clock time 2018-05-21-15:28:40.329.

Domain: Dumping all at clock time 2018-05-21-15:28:40.329.
Globals dumped at 2018-05-21-15:28:40.329.
Dumping grid boundaries at 2018-05-21-15:28:40.329.
Dumped grid boundaries at 2018-05-21-15:28:40.329.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-05-21-15:28:40.329.
Domain: Histories dumped at 2018-05-21-15:28:40.329.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-21-15:28:40.329.
Dumping E at 2018-05-21-15:28:40.329.
Dumped E at 2018-05-21-15:28:40.329.
Dumping B at 2018-05-21-15:28:40.329.
Dumped B at 2018-05-21-15:28:40.329.
All multiFields dumped at 2018-05-21-15:28:40.329.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-21-15:28:40.329.

Main loop ended at clock time 2018-05-21-15:28:40.329
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-05-21-15:28:40.350.
NOTE: A VSimBase license was needed to run this simulation.
Lines from 'emPlaneWaveT.pre' processed.
Finished with 'emPlaneWaveT.pre'.
-------------------- END ENGINE OUTPUT ---------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 2.35: The Run Window at the end of execution.

The electric and magnetic field components can be found in the scalar data variables of the data overview tab.

- Make sure the *Data View* drop down is set to *Data Overview*.
- Here you can see *Variables*. Expand the *Scalar Data*.
- Expand E
- Select E_y

Initially, no field will be seen, as one is looking at Dump 0, the initial dump, when no fields are yet in the simulation. Move the slider at the bottom of the right pane to see the electric field at different times. The final time is shown in Fig. 2.36.

Further Experiments

To see more wavelengths, change the value of the WAVELENGTHS variable. What happens to the waves when there are very few cells in a wavelength?

See the wave reflect off the right boundary by running for more time steps.

To see a numerical instability, increase TIMESTEP_FACTOR to greater than one.

To see controlled dispersion, change to a 1D simulation and set TIMESTEP_FACTOR to unity.

Try changing NDIM to 3 to see the how the simulation is uniform across the z- dimension.

2.2.4 Electromagnetic Particle In Cell (emPtclInCellT.pre)

Keywords:

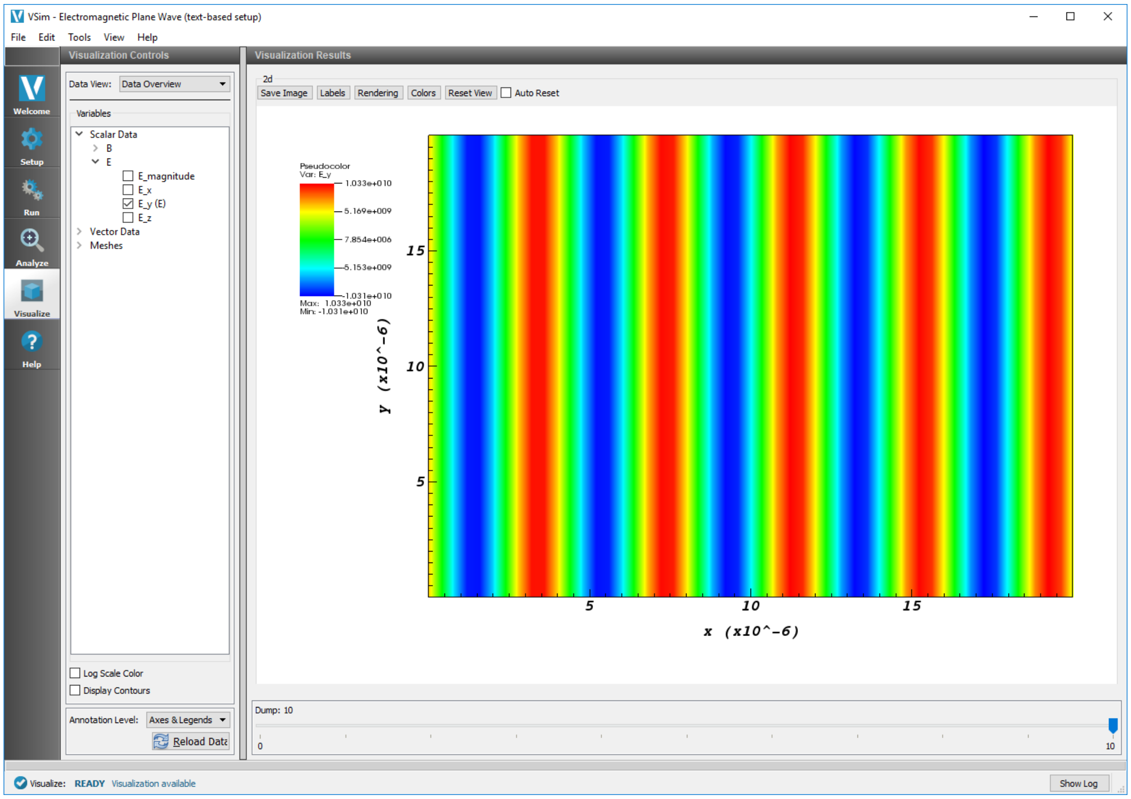


Fig. 2.36: Visualization of plane wave as a color contour plot.

`electromagnetics, particle in cell, sheath, box bounding`

Problem description

A point antenna launches a wave from a point that is midway in x and z (if two-dimensional) but 1/3 of the way up from the bottom in y. The electromagnetic field and plasma respond self consistently to an antenna current, if set, or else the plasma simply evolves due to its initial thermal distribution.

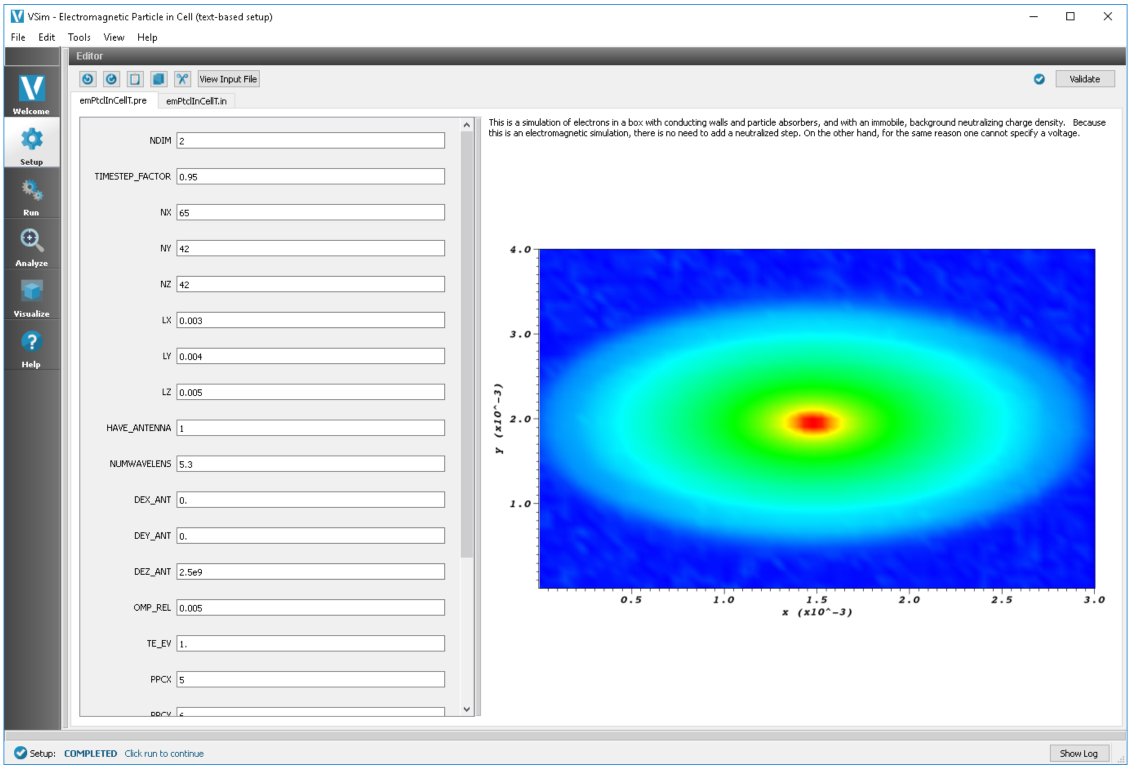
This simulation can be performed with a VSimBase license.

Opening the Simulation

The Electromagnetic Particle In Cell example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples (text-based setup)* option.
- Select “Electromagnetic Particle In Cell” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The key parameters of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 2.37.



This is a simulation of electrons in a box with conducting walls and particle absorbers, and with an immobile, background neutralizing charge density. Because this is an electromagnetic simulation, there is no need to add a neutralized step. On the other hand, for the same reason one cannot specify a voltage.

NDIM 2

TIMESTEP_FACTOR 0.95

NX 65

NY 42

NZ 42

LX 0.003

LY 0.004

LZ 0.005

HAVE_ANTENNA 1

NUMWAVELENS 5.3

DEX_ANT 0.

DEY_ANT 0.

DEZ_ANT 2.5e9

OMP_REL 0.005

TE_EV 1.

PPCX 5

PPCY 5

Setup: COMPLETED Click run to continue

Fig. 2.37: Setup Window for Electromagnetic Particle in Cell.

Input File Features

The input file sets the number of cells along each direction along with the lengths of each direction. The transverse (y, z) boundary conditions are periodic. There are slab conductors at the limits in x.

The number of vacuum wavelengths in the box can be chosen. The wave frequency ω, is then given from the vacuum dispersion relation $\omega = c/\lambda$.

Particles are loaded on a grid with a thermal velocity to generate a plasma with plasma frequency equal to OMP_REL times the wave frequency. For the wave amplitude, the antenna strength should satisfy DEZ_ANT > 1.e9*OMP_REL^2.

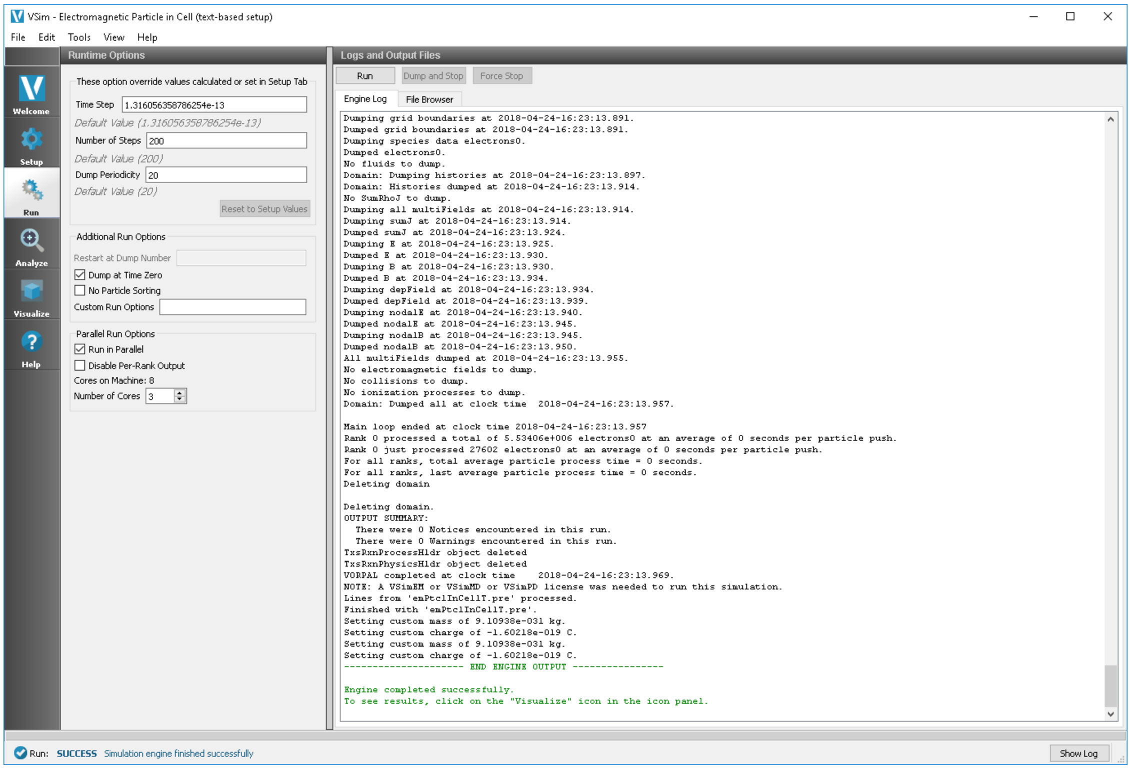
The time step is chosen to be TIMESTEP_FACTOR times the combined stability limit for plasma oscillations and that given by the CFL limit for FDTD electromagnetics.

One can turn the antenna off to see just the plasma noise and sheath generation.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner. of the window. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 2.38.



These option override values calculated or set in Setup Tab

Time Step: 1.316056358786254e-13
Default Value (1.316056358786254e-13)
Number of Steps: 200
Default Value (200)
Dump Periodicity: 20
Default Value (20)

Additional Run Options

Restart at Dump Number:
☑ Dump at Time Zero
□ No Particle Sorting
Custom Run Options:

Parallel Run Options

☑ Run in Parallel
□ Disable Per-Rank Output
Cores on Machine: 8
Number of Cores: 3

```
Dumping grid boundaries at 2018-04-24-16:23:13.891.
Dumped grid boundaries at 2018-04-24-16:23:13.891.
Dumping species data electrons0.
Dumped electrons0.
No fluids to dump.
Domain: Dumping histories at 2018-04-24-16:23:13.897.
Domain: Histories dumped at 2018-04-24-16:23:13.914.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-16:23:13.914.
Dumping sumJ at 2018-04-24-16:23:13.914.
Dumped sumJ at 2018-04-24-16:23:13.924.
Dumping E at 2018-04-24-16:23:13.925.
Dumped E at 2018-04-24-16:23:13.930.
Dumping B at 2018-04-24-16:23:13.930.
Dumped B at 2018-04-24-16:23:13.934.
Dumping depField at 2018-04-24-16:23:13.934.
Dumped depField at 2018-04-24-16:23:13.939.
Dumping nodalE at 2018-04-24-16:23:13.940.
Dumped nodalE at 2018-04-24-16:23:13.945.
Dumping nodalB at 2018-04-24-16:23:13.945.
Dumped nodalB at 2018-04-24-16:23:13.950.
All multiFields dumped at 2018-04-24-16:23:13.955.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-16:23:13.957.

Main loop ended at clock time 2018-04-24-16:23:13.957
Rank 0 processed a total of 5.53406e+006 electrons0 at an average of 0 seconds per particle push.
Rank 0 just processed 27602 electrons0 at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-16:23:13.969.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.
Lines from 'emPtclInCellT.pre' processed.
Finished with 'emPtclInCellT.pre'.
Setting custom mass of 9.10938e-031 kg.
Setting custom charge of -1.60218e-019 C.
Setting custom mass of 9.10938e-031 kg.
Setting custom charge of -1.60218e-019 C.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 2.38: The Run Window at the end of execution of Electromagnetic Particle in Cell.

Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the electric field, expand "Scalar Data", expand the E field box, and select E_magnitude. Move the slider at the bottom of the right pane to see the electric field at different times. At Dump 4, the rings of radiation surrounding the antenna can be seen, as shown in Fig. 2.2.4. box, as shown below.

Further Experiments

Vary the antenna amplitude, DEZ_ANT, to find out how low it can be before the signal is swamped by the plasma noise.

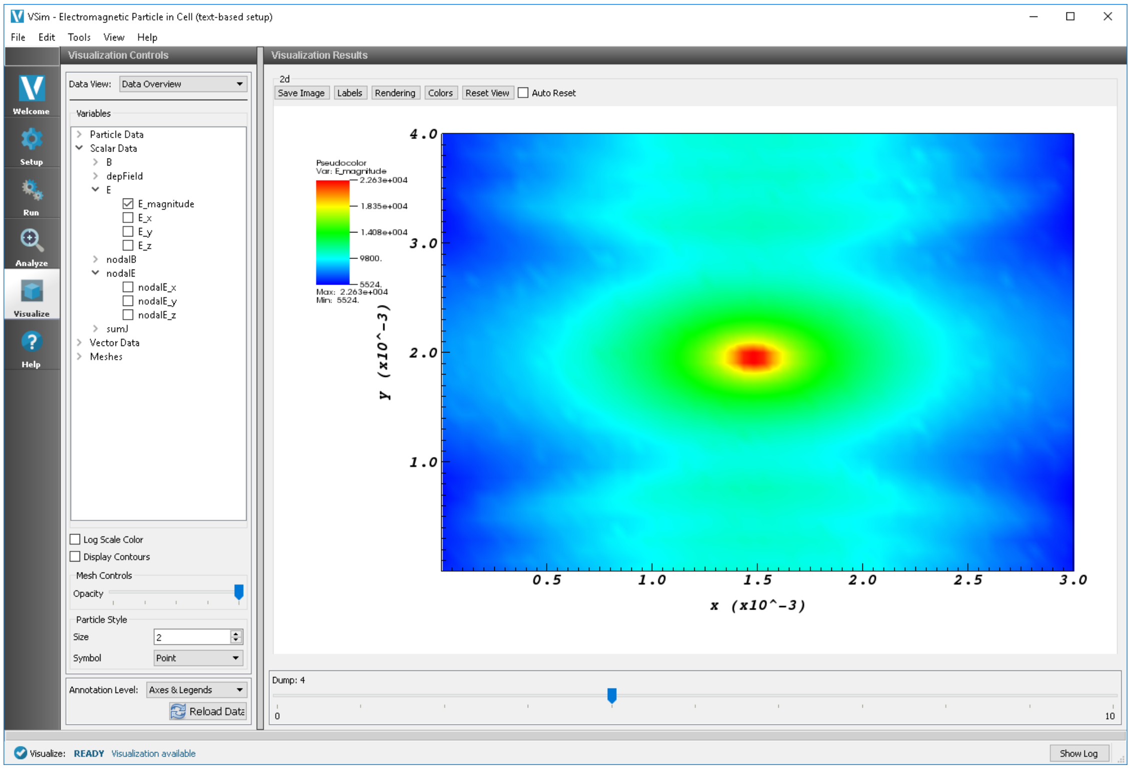
Increase OMP_REL to 1.5 to see that a wave is no longer launched, but what causes that front that travels far from the antenna.

Add in a magnetic field in the plane; remove the magnetic field out of the plane. Launch a mode with frequency above the upper-hybrid. Does the shape of the fronts change?

2.2.5 Vacuum Electromagnetic Pulse (emPulseInVacuumT.pre)

Keywords:

`electromagnetics, laser, plane wave pulse, field energy monitoring`



Problem description

A linearly-polarized (with electric field in the z-direction) electromagnetic pulse with a sinusoidal amplitude on a plane wave is launched from the left side (x=0). The transverse (y, z) boundary conditions are periodic, but the pulse has finite transverse extent.

This simulation can be performed with a VSimBase license.

Opening the Simulation

The Vacuum Electromagnetic Pulse example is accessed from within VSimComposer by the following actions:

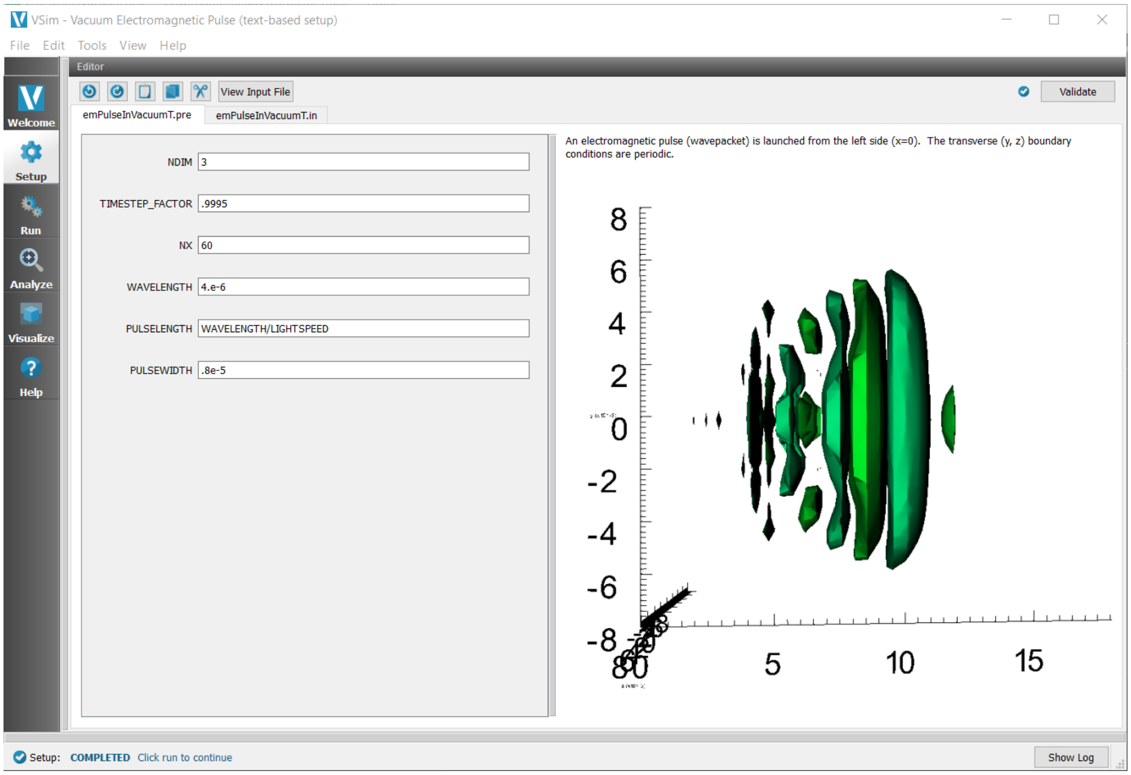
- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples (text-based setup)* option.
- Select *Vacuum Electromagnetic Pulse (text-based setup)* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window* as shown in Fig. 2.39.

Input File Features

The input file sets the wavelength to be one-eighth of the box length. It computes the CFL stability limit for the time step and sets the actual time step to TIMESTEP_FACTOR times the stability limit for the time step.

One can change the pulse width and length to see more or less diffraction or dispersion.



NDIM 3

TIMESTEP_FACTOR .9995

NX 60

WAVELENGTH 4.e-6

PULSELENGTH WAVELENGTH/LIGHTSPEED

PULSEWIDTH .8e-5

An electromagnetic pulse (wavepacket) is launched from the left side (x=0). The transverse (y, z) boundary conditions are periodic.

Setup: COMPLETED Click run to continue

Fig. 2.39: The Setup Window for the electromagnetic pulse.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully."

The Run Window, showing settable parameters with the engine output in the right pane, is shown in

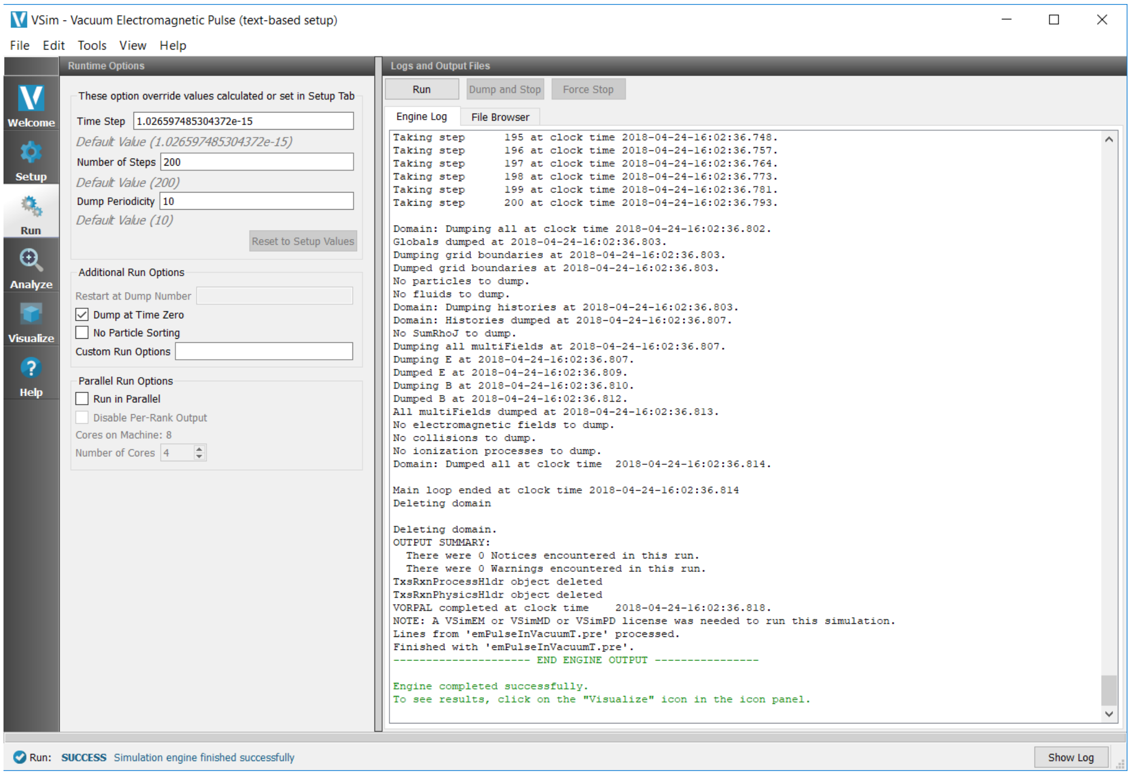
Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The electric and magnetic field components can be found in the scalar data variables of the data overview tab.

- Make sure the *Data View* drop down is set to *Data Overview*.
- Here you can see *Variables*. Expand the *Scalar Data*.
- Expand B
- Select B_x
- Check the box next to *Display Contours* and set the # of contours to 10
- Click and drag with your mouse to rotate the view



These option override values calculated or set in Setup Tab

Time Step 1.026597485304372e-15
Default Value (1.026597485304372e-15)
Number of Steps 200
Default Value (200)
Dump Periodicity 10
Default Value (10)

Additional Run Options

Restart at Dump Number
☑ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options

□ Run in Parallel
□ Disable Per-Rank Output
Cores on Machine: 8
Number of Cores 4

```
Taking step      195 at clock time 2018-04-24-16:02:36.748.
Taking step      196 at clock time 2018-04-24-16:02:36.757.
Taking step      197 at clock time 2018-04-24-16:02:36.764.
Taking step      198 at clock time 2018-04-24-16:02:36.773.
Taking step      199 at clock time 2018-04-24-16:02:36.781.
Taking step      200 at clock time 2018-04-24-16:02:36.793.

Domain: Dumping all at clock time 2018-04-24-16:02:36.802.
Globals dumped at 2018-04-24-16:02:36.803.
Dumping grid boundaries at 2018-04-24-16:02:36.803.
Dumped grid boundaries at 2018-04-24-16:02:36.803.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-04-24-16:02:36.803.
Domain: Histories dumped at 2018-04-24-16:02:36.807.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-16:02:36.807.
Dumping E at 2018-04-24-16:02:36.807.
Dumped E at 2018-04-24-16:02:36.809.
Dumping B at 2018-04-24-16:02:36.810.
Dumped B at 2018-04-24-16:02:36.812.
All multiFields dumped at 2018-04-24-16:02:36.813.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-16:02:36.814.

Main loop ended at clock time 2018-04-24-16:02:36.814
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-16:02:36.818.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.
Lines from 'emPulseInVacuumT.pre' processed.
Finished with 'emPulseInVacuumT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 2.40: The Run Window for the electromagnetic pulse.

Initially, no field will be seen, as one is looking at Dump 0, the initial dump, when no fields are yet in the simulation. Move the slider at the bottom of the right pane to see the magnetic field at different times.

Further Experiments

Increase CFL_FACTOR to greater than one. One should then see the simulation go numerically unstable, as one will have violated the CFL condition.

Increase NX to better resolve the wave and see whether it slips less with respect to the box.

Increase the pulse and box widths (you will also need to increase the number of cells in the transverse directions) to reduce diffraction.

Modify the boundary conditions to be a Matched Absorbing Layer (MAL) or Perfectly Matched Layer (PML) to see how the wave is absorbed when it reaches the end of the simulation.

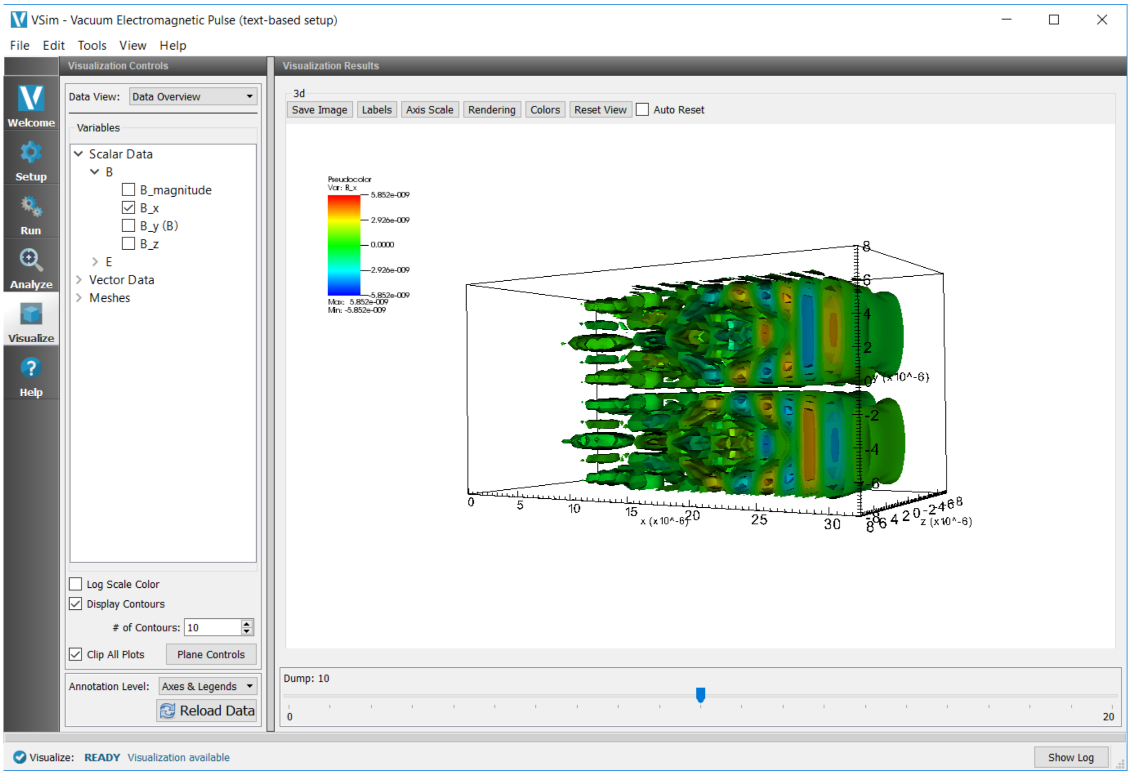
2.2.6 Electrostatic Particle In Cell (esPtclInCellT.pre)

Keywords:

`electrostatics, particle in cell, sheath, box bounding`

Problem description

This electrostatic particle in cell example computes the electrostatic potential and fields in a box with conducting walls and particle absorbers and with an immobile, background neutralizing charge density. The electrons move to the wall



Data View: Data Overview

□ B_magnitude
☑ B_x
□ B_y (B)
□ B_z

□ Log Scale Color
☑ Display Contours
# of Contours: 10
☑ Clip All Plots

Annotation Level: Axes & Legends

Dump: 10

Visualize: READY Visualization available

Fig. 2.41: Bx fields at Dump 10.

by the potential, creating a sheath.

This simulation can be performed with a VSimBase license.

Opening the Simulation

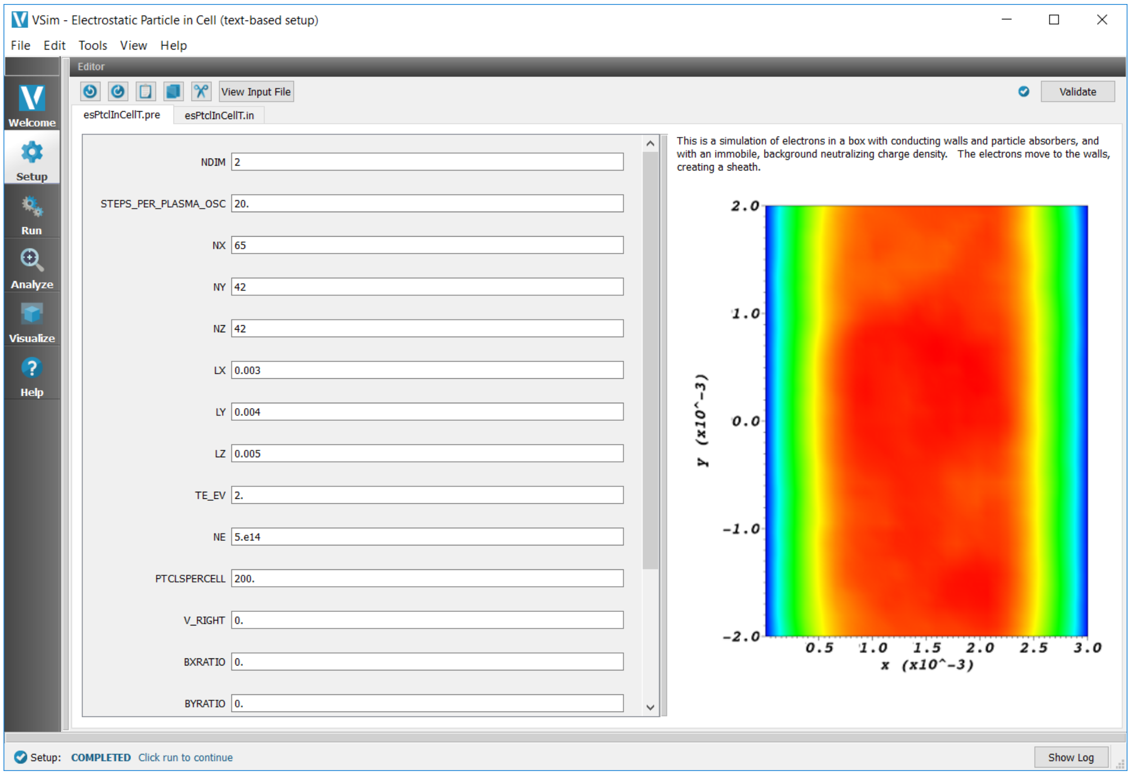
The electrostatic particle in cell example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples (Text-based setup)* option.
- Select “Electrostatic Particle in Cell (Text-based setup)” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 2.42.

Input File Features

The input file allows one to choose the simulation dimension, domain size, resolution (number of cells in each direction), electron temperature (eV), electron density, voltage, ratios of cyclotron frequencies to plasma frequency and electron marcroparticles per cell.



NDIM 2

STEPS_PER_PLASMA_OSC 20.

NX 65

NY 42

NZ 42

LX 0.003

LY 0.004

LZ 0.005

TE_EV 2.

NE 5.e14

PTCLSPERCELL 200.

V_RIGHT 0.

BXRATIO 0.

BYRATIO 0.

This is a simulation of electrons in a box with conducting walls and particle absorbers, and with an immobile, background neutralizing charge density. The electrons move to the walls, creating a sheath.

Setup: COMPLETED Click run to continue

Fig. 2.42: Setup Window for the electrostatic particle in cell example.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” A snapshot of the simulation run completion is shown in Fig. 2.43.

Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

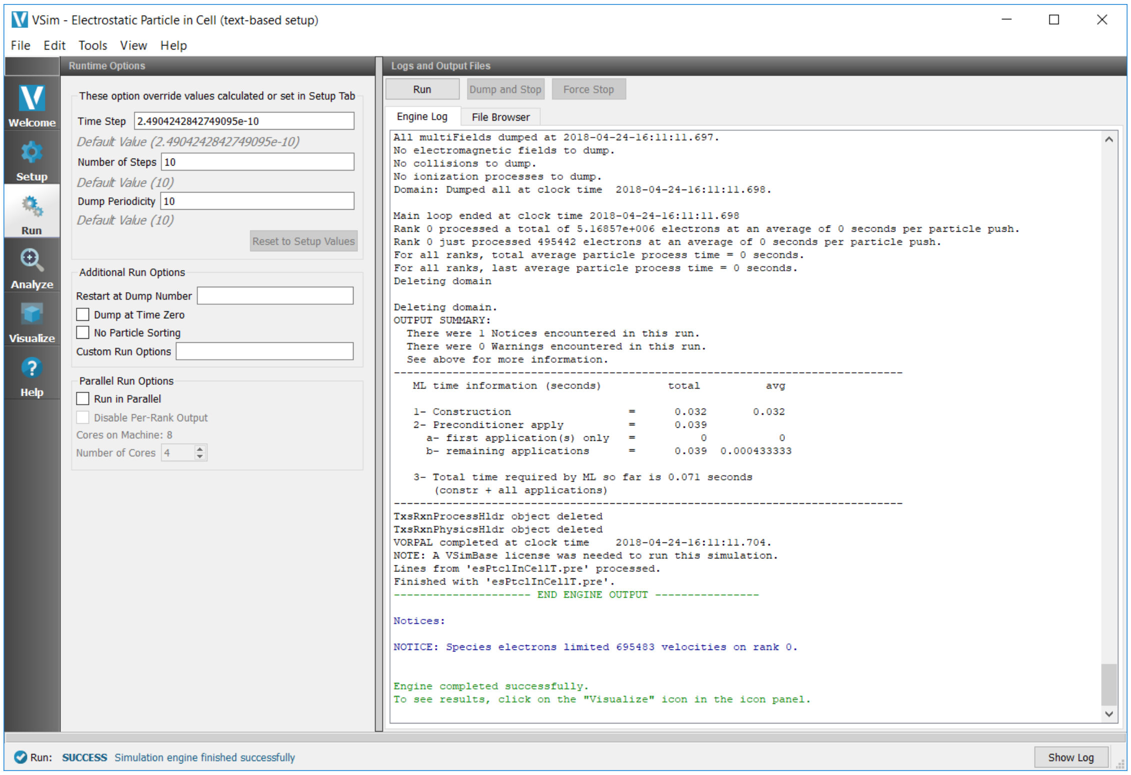
To view the electric potential

- Expand the *Scalar Data*
- Select *phi*
- Move the dump slider to the right to Dump 1

The potential in the visualization window resembles that shown in Fig. 2.44.

Further Experiments

Change the plasma density and see whether the frequency in the histories changes.



These option override values calculated or set in Setup Tab

Time Step 2.4904242842749095e-10

Default Value (2.4904242842749095e-10)

Number of Steps 10

Default Value (10)

Dump Periodicity 10

Default Value (10)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

□ Run in Parallel

□ Disable Per-Rank Output

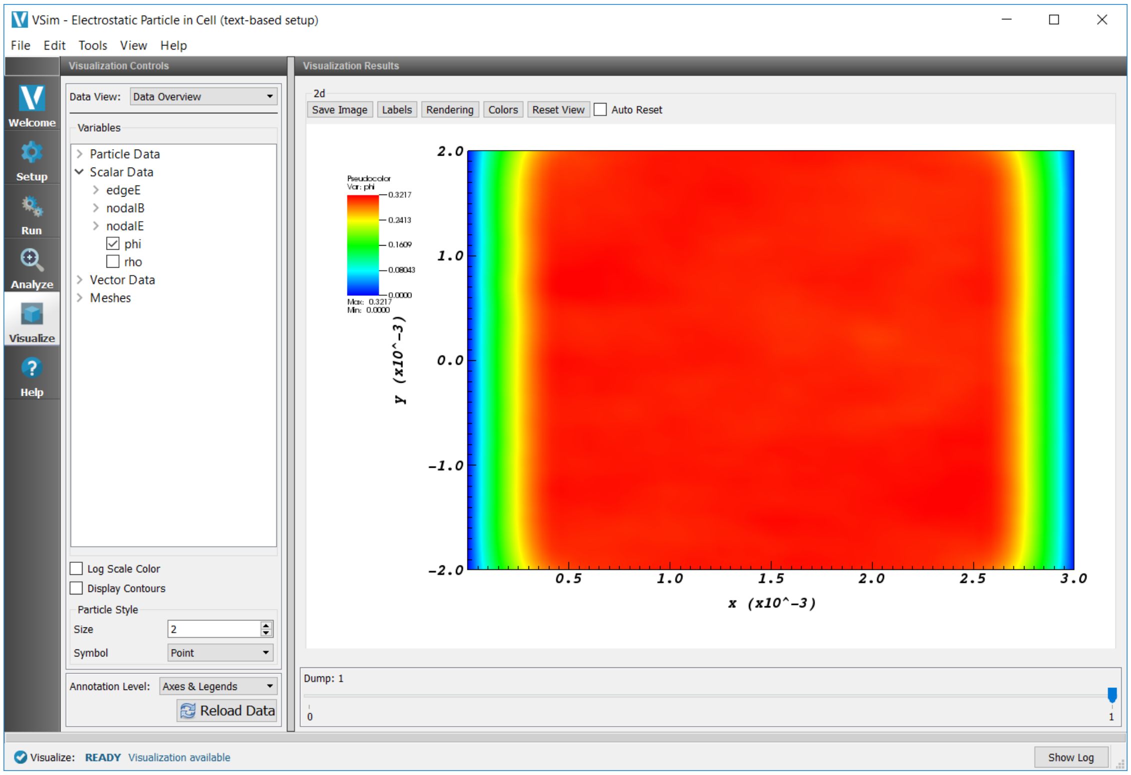
Cores on Machine: 8

Number of Cores 4

```
All multiFields dumped at 2018-04-24-16:11:11.697.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-16:11:11.698.

Main loop ended at clock time 2018-04-24-16:11:11.698
Rank 0 processed a total of 5.16857e+006 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 495442 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 1 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
  See above for more information.
------------------------------------------------------------------------------
   ML time information (seconds)          total          avg

   1- Construction                  =      0.032       0.032
   2- Preconditioner apply          =      0.039
     a- first application(s) only   =          0           0
     b- remaining applications      =      0.039  0.000433333

   3- Total time required by ML so far is 0.071 seconds
      (constr + all applications)
------------------------------------------------------------------------------
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-16:11:11.704.
NOTE: A VSimBase license was needed to run this simulation.
Lines from 'esPtclInCellT.pre' processed.
Finished with 'esPtclInCellT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: Species electrons limited 695483 velocities on rank 0.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 2.43: The Run Window at the end of execution.



Data View: Data Overview

Variables

- Particle Data
- Scalar Data
  - edgeE
  - nodalB
  - nodalE
  - ☑ phi
  - □ rho
- Vector Data
- Meshes

□ Log Scale Color

□ Display Contours

Particle Style

Size 2

Symbol Point

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 1

Visualize: READY Visualization available

Fig. 2.44: The Visualize Window showing the electric potential, phi, at dump 1.

Set BZRATIO to 1.0 to observe the upper-hybrid mode.

Set BXRATIO to 1.0 to observe both the plasma mode and the upper-hybrid mode.

Set a value to V_RIGHT to see how the potential distribution changes.

Change the parameter NDIM to 3 to see the potential in 3 dimensions. Note that this will take longer to simulate.

2.2.7 Parallel Plate Capacitor (parPlateCapacitorT.pre)

Keywords:

`electrostatics, parallel plate capacitor`

Problem description

This Parallel Plate Capacitor simulation computes the electrostatic potential and field for a parallel plate capacitor. It can be run in any number of dimensions. It is periodic in the y and z directions when they are present.

This simulation can be performed with a VSimBase license.

Opening the Simulation

The Parallel Plate Capacitor example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples (text-based setup)* option.
- Select *Parallel Plate Capacitor (text-based setup)* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 2.45.

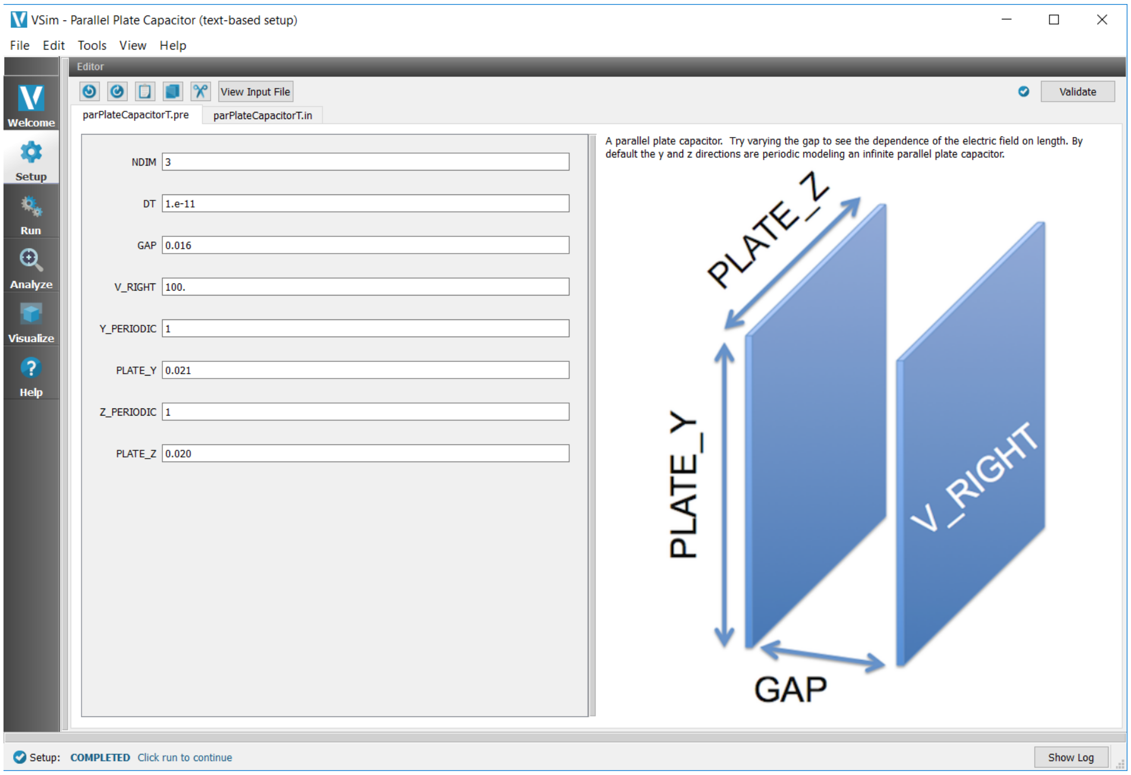
Input File Features

The input file allows one to choose the distance between the plates, width of the plates, voltage of the positive plate and the length of a time step (which is irrelevant as this is an electrostatic simulation)

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 2.46.



NDIM 3

DT 1.e-11

GAP 0.016

V_RIGHT 100.

Y_PERIODIC 1

PLATE_Y 0.021

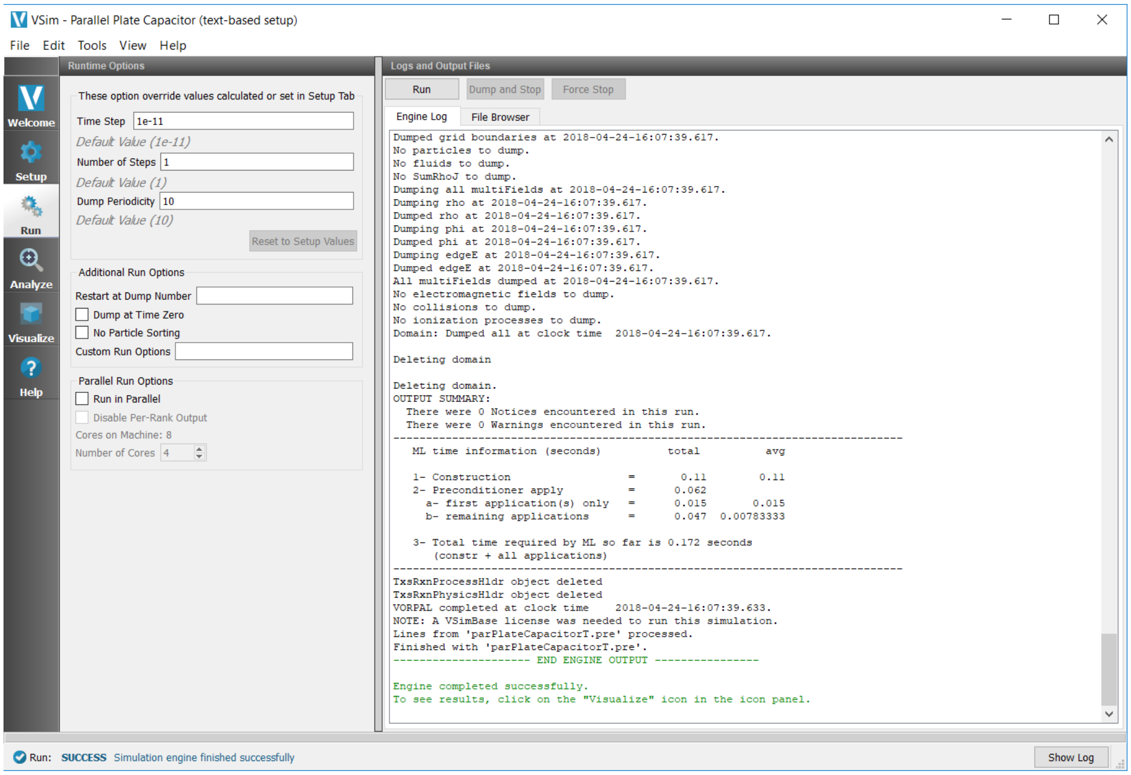
Z_PERIODIC 1

PLATE_Z 0.020

A parallel plate capacitor. Try varying the gap to see the dependence of the electric field on length. By default the y and z directions are periodic modeling an infinite parallel plate capacitor.

Setup: COMPLETED Click run to continue

Fig. 2.45: Setup Window for the Parallel Plate Capacitor example.



These option override values calculated or set in Setup Tab

Time Step 1e-11

Default Value (1e-11)

Number of Steps 1

Default Value (1)

Dump Periodicity 10

Default Value (10)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

□ Run in Parallel

□ Disable Per-Rank Output

Cores on Machine: 8

Number of Cores 4

```
Dumped grid boundaries at 2018-04-24-16:07:39.617.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-16:07:39.617.
Dumping rho at 2018-04-24-16:07:39.617.
Dumped rho at 2018-04-24-16:07:39.617.
Dumping phi at 2018-04-24-16:07:39.617.
Dumped phi at 2018-04-24-16:07:39.617.
Dumping edgeE at 2018-04-24-16:07:39.617.
Dumped edgeE at 2018-04-24-16:07:39.617.
All multiFields dumped at 2018-04-24-16:07:39.617.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-16:07:39.617.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
------------------------------------------------------------------------------
   ML time information (seconds)          total          avg

   1- Construction                  =       0.11        0.11
   2- Preconditioner apply          =      0.062
     a- first application(s) only   =      0.015       0.015
     b- remaining applications      =      0.047  0.00783333

   3- Total time required by ML so far is 0.172 seconds
      (constr + all applications)
------------------------------------------------------------------------------
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-16:07:39.633.
NOTE: A VSimBase license was needed to run this simulation.
Lines from 'parPlateCapacitorT.pre' processed.
Finished with 'parPlateCapacitorT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 2.46: The Run Window at the end of execution.

Visualizing the Results

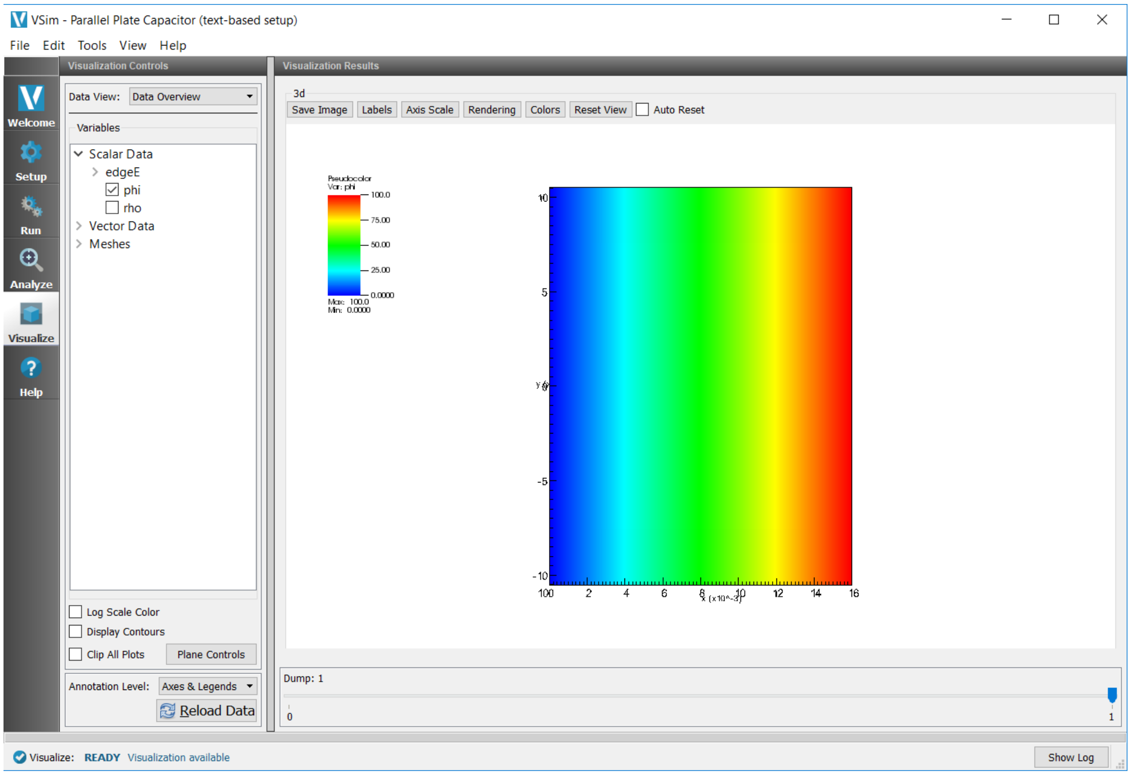
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To visualize the electrostatic potential

- Expand *Scalar Data*
- Select *phi*
- Move the time slider to the right to view Dump 1

Fig. 2.47 shows the visualization seen.



Data View: Data Overview

☑ phi

□ rho

□ Auto Reset

□ Log Scale Color

□ Display Contours

□ Clip All Plots

Annotation Level: Axes & Legends

Dump: 1

Visualize: READY Visualization available

Fig. 2.47: Visualization of plane wave as a color contour plot.

Further Experiments

Change the GAP between the plates and see how the electric field changes.

Change a width, e.g., LY and see whether it has an effect on the electric field.

Change the voltage on the right plate and see if it effects the electric field.

Set the boundary conditions to non-periodic (0) and see how the electric field changes.

2.2.8 Two-Stream Instability (twoStreamT.pre)

Keywords:

`electromagnetics, two-stream instability`

Problem Description

The two-stream instability is a rapidly growing collision-less plasma instability arising from small charge imbalances. A local imbalance leads to the acceleration or deceleration of particles in its vicinity, which in turn leads to an even stronger imbalance. One setup that allows to easily observe the instability is two counter-streaming beams of identical charge in a periodic system. The advantage of this configuration is that the generated plasma wave becomes a standing wave, thus allowing to easily observe the formation of the phase space vortices.

In this example, we use two electron streams. At $t = 0$ the streams have drift velocities of magnitude 7.78×10^6 m/s. In order to accelerate the onset of the instability, the two particle beams are given a small sinusoidal perturbation in velocity space.

This simulation can be performed with a VSimBase license.

Opening the Simulation

The Two-Stream Instability example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Basic Physics* option.
- Expand the *Basic Examples (text-based setup)* option.
- Select *Two-Stream Instability (text-based setup)* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 2.48.

Input File Features

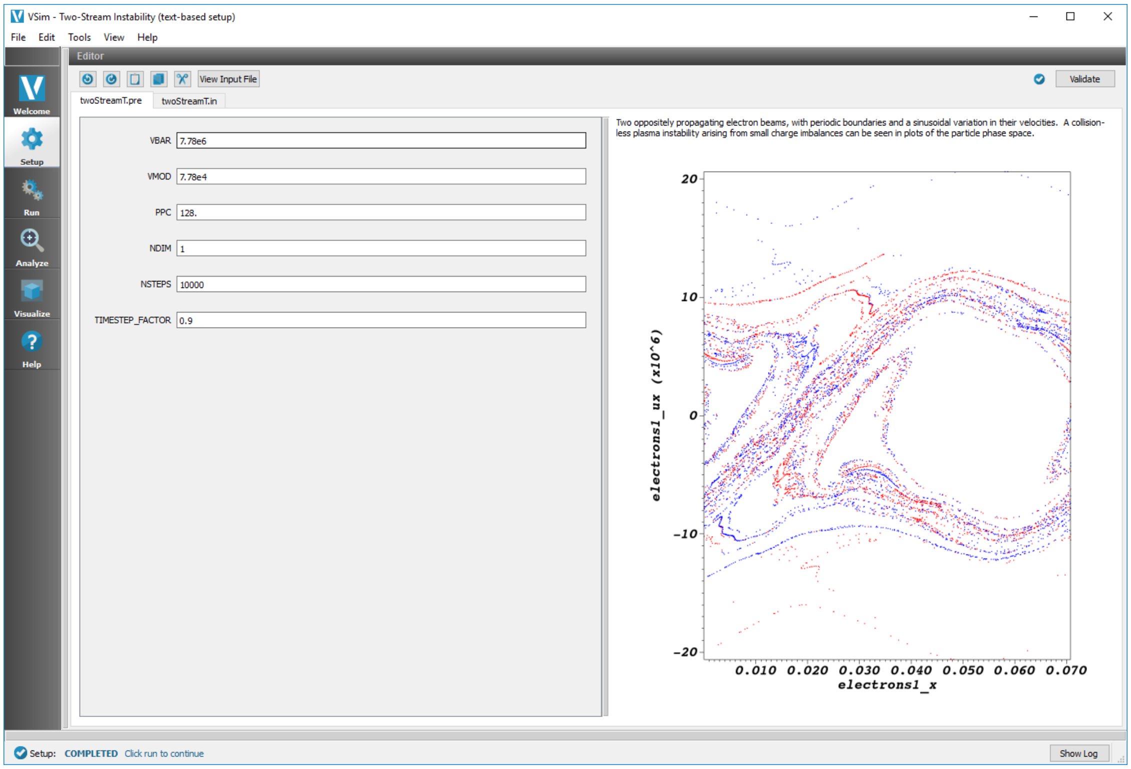
The simulation setup consists of an electromagnetic field and two particle species, one for each of the two counter propagating electron particle streams. Each electron stream is given a drift velocity and an additional velocity perturbation. For diagnostic purposes there is also output for species momentum, species energy and field energy histories.

The input file allows one to choose the average beam velocities and the amplitude of modulation. The number of particles per cell (PPC), dimensionality, length of the simulation and length of a time step can also be modified from the key parameters window.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper right corner of the window. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 2.49.



VBAR 7.78e6

VMOD 7.78e4

PPC 128.

NDIM 1

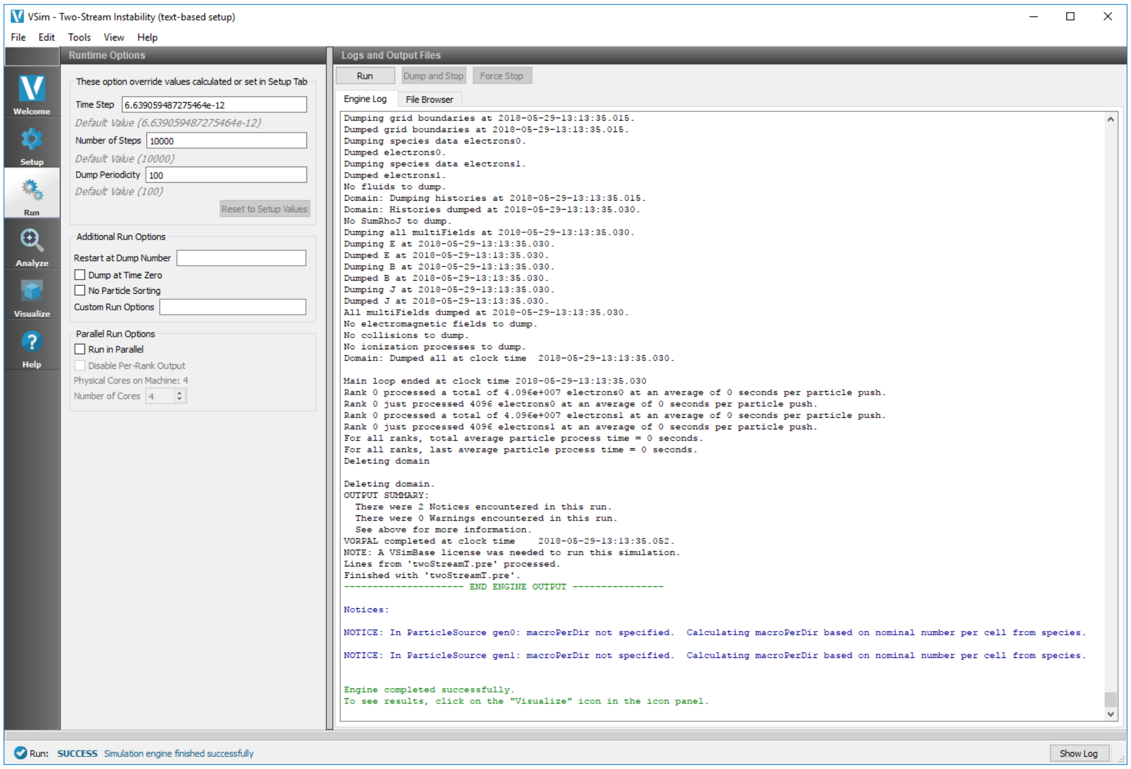
NSTEPS 10000

TIMESTEP_FACTOR 0.9

Two oppositely propagating electron beams, with periodic boundaries and a sinusoidal variation in their velocities. A collisionless plasma instability arising from small charge imbalances can be seen in plots of the particle phase space.

Setup: COMPLETED Click run to continue

Fig. 2.48: Setup Window for the Two-Stream Instability example.



These option override values calculated or set in Setup Tab

Time Step 6.639059487275464e-12

Default Value (6.639059487275464e-12)

Number of Steps 10000

Default Value (10000)

Dump Periodicity 100

Default Value (100)

Additional Run Options

Restart at Dump Number

☐ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☐ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Dumping grid boundaries at 2018-05-29-13:13:35.015.
Dumped grid boundaries at 2018-05-29-13:13:35.015.
Dumping species data electrons0.
Dumped electrons0.
Dumping species data electrons1.
Dumped electrons1.
No fluids to dump.
Domain: Dumping histories at 2018-05-29-13:13:35.015.
Domain: Histories dumped at 2018-05-29-13:13:35.030.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-29-13:13:35.030.
Dumping E at 2018-05-29-13:13:35.030.
Dumped E at 2018-05-29-13:13:35.030.
Dumping B at 2018-05-29-13:13:35.030.
Dumped B at 2018-05-29-13:13:35.030.
Dumping J at 2018-05-29-13:13:35.030.
Dumped J at 2018-05-29-13:13:35.030.
All multiFields dumped at 2018-05-29-13:13:35.030.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-29-13:13:35.030.

Main loop ended at clock time 2018-05-29-13:13:35.030
Rank 0 processed a total of 4.096e+007 electrons0 at an average of 0 seconds per particle push.
Rank 0 just processed 4096 electrons0 at an average of 0 seconds per particle push.
Rank 0 processed a total of 4.096e+007 electrons1 at an average of 0 seconds per particle push.
Rank 0 just processed 4096 electrons1 at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 2 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-05-29-13:13:35.052.
NOTE: A VSimBase license was needed to run this simulation.
Lines from 'twoStreamT.pre' processed.
Finished with 'twoStreamT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: In ParticleSource gen0: macroPerDir not specified.  Calculating macroPerDir based on nominal number per cell from species.

NOTICE: In ParticleSource gen1: macroPerDir not specified.  Calculating macroPerDir based on nominal number per cell from species.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 2.49: The Run Window at the end of execution.

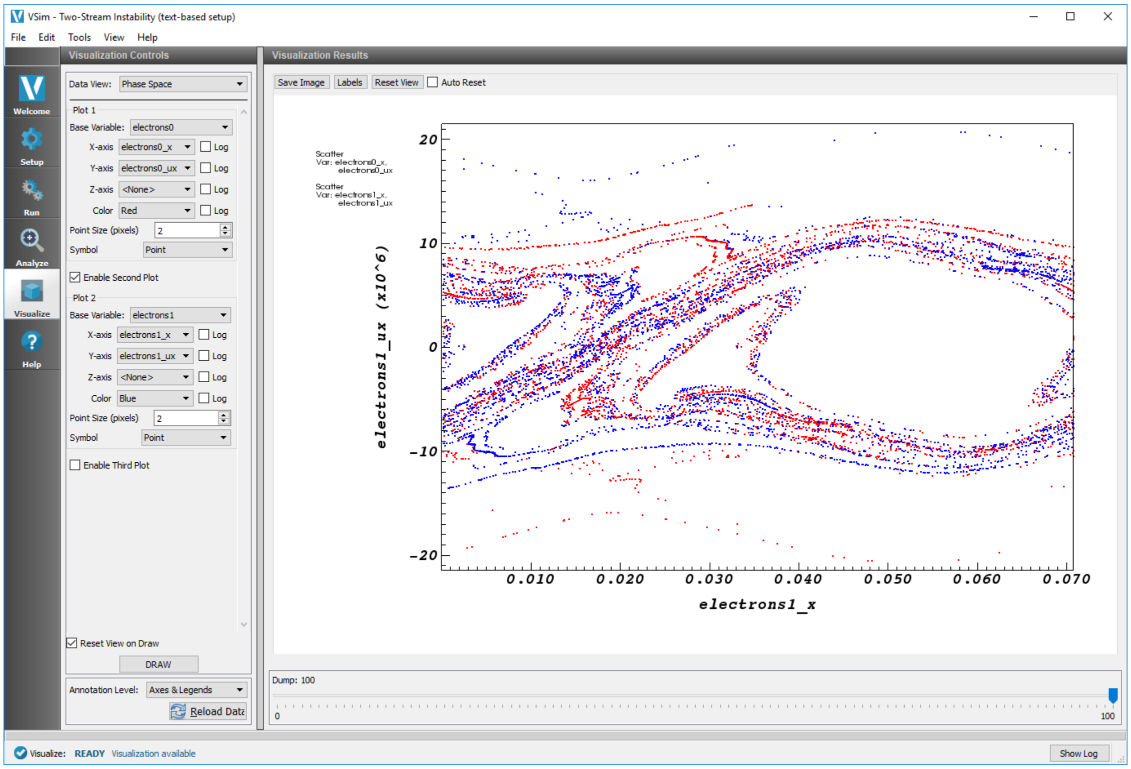
Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the instability in phase space:

- Select the *Phase Space* option from the *Data View* menu.
- In the Plot 1 box, change the *X-axis* to *electrons0_x*, the *Y-axis* to *electrons0_ux*, and *Color* to *Red*.
- Click the *Enable Second Plot* box.
- In the Plot 2 box, change the *X-axis* to *electrons1_x*, the *Y-axis* to *electrons1_ux* and *Color* to *Blue*.
- Click the *DRAW* button at the bottom
- Move the *Dump* slider over to dump number 100.



VSim - Two-Stream Instability (text-based setup)

Data View: Phase Space

Plot 1
Base Variable: electrons0
X-axis electrons0_x □ Log
Y-axis electrons0_ux □ Log
Z-axis <None> □ Log
Color Red □ Log
Point Size (pixels) 2
Symbol Point

☑ Enable Second Plot

Plot 2
Base Variable: electrons1
X-axis electrons1_x □ Log
Y-axis electrons1_ux □ Log
Z-axis <None> □ Log
Color Blue □ Log
Point Size (pixels) 2
Symbol Point

□ Enable Third Plot

☑ Reset View on Draw

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 100

Visualize: READY Visualization available

Fig. 2.50: Visualization of the two-stream instability developing in particle phase space.

Further Experiments

Look at the histories to see how the plasma waves exchange energy with fields and each other as the instability develops.

Change the average velocity and velocity modulation and see how the speed at which the instability sets in depends on the modulation.

To calculate the density of the particles in the simulation switch the key parameters to

1. *VBAR* = 7.78e3

2. *PPC* = 10. (note that decimal point is important here)
3. *NDIM* = 2

Then after the simulation has completed go to the *Analyze Window*, select ptclNumDensity from the drop down menu, input twoStreamT for the baseName and electrons0 for the speciesName and hit run. This process is detailed further in *VSim In Depth*

CHAPTER

THREE

VSIM FOR ELECTROMAGNETICS EXAMPLES

These examples illustrate how to solve complex problems in electromagnetics.

These examples can be run with a VSimEM license.

3.1 Antennas

3.1.1 Yagi-Uda (YagiUda2p4.sdf)

Keywords:

yagiUdaArrayWireModel, yagiT, far field, radiation

Problem description

A Yagi-Uda array is a directional antenna consisting of several parallel dipole elements. Only one of these dipole elements is driven, the other elements being parassitic . Directionality is acheived by requiring that there be one longer element adjacent to the source element, which is refereed to as the reflector. The rest of the elements beginning adjacent to the source but opposite to the reflector are shorter than the source element, these are called directors. Yagi antennas are ubiquitous and as such optimal parameters for dipole lengths and separations have been established, we go with values one would typically find in any text covering the matter. This example illustrates how to obtain the far field radiation pattern of a Yagi-Uda array.

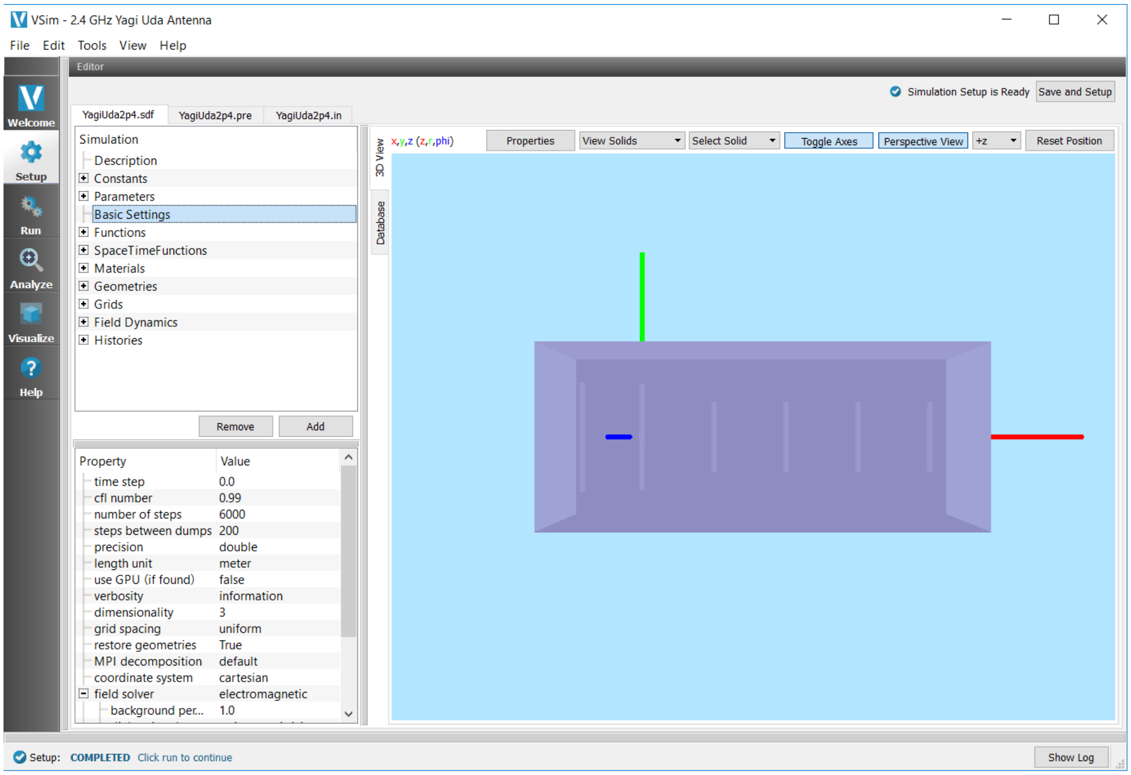
This simulation can be performed with a VSimEM license.

Opening the Simulation

The Yagi-Uda example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Antennas* option.
- Select *2.4 GHz Yagi Uda Antenna* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 3.1. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



Property	Value
time step	0.0
cfl number	0.99
number of steps	6000
steps between dumps	200
precision	double
length unit	meter
use GPU (if found)	false
verbosity	information
dimensionality	3
grid spacing	uniform
restore geometries	True
MPI decomposition	default
coordinate system	cartesian
field solver	electromagnetic
background per...	1.0

Fig. 3.1: Setup Window for the Yagi-Uda example.

Simulation Properties

This file allows the modification of the antenna operating frequency, antenna dimensions, and simulation domain size.

By adjusting the dimensions any sized Yagi-Uda array can be simulated.

Note: To obtain good far field resolution generally four or more antenna elements is desirable (One source, one reflector, two or more directors).

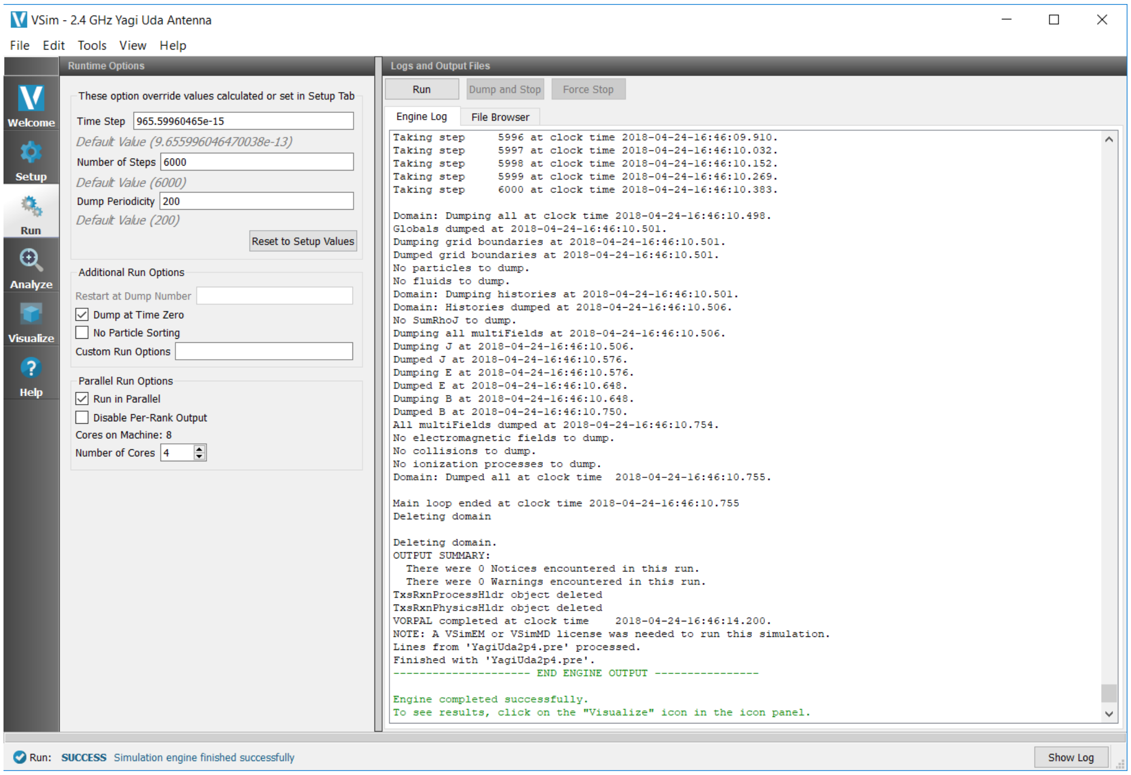
Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- Here you can set run parameters, including how many cores to run with (under the MPI tab).
- When you are finished setting run parameters, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 3.2.

Analyzing the Results

Click the *Show All Analyzers* button in the bottom left of the window, select “computeFarFieldFromKirchhoffBox.py” from the list, and click “Open.”



These option override values calculated or set in Setup Tab

Time Step 965.59960465e-15

Default Value (9.655996046470038e-13)

Number of Steps 6000

Default Value (6000)

Dump Periodicity 200

Default Value (200)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Cores on Machine: 8

Number of Cores 4

```
Taking step     5996 at clock time 2018-04-24-16:46:09.910.
Taking step     5997 at clock time 2018-04-24-16:46:10.032.
Taking step     5998 at clock time 2018-04-24-16:46:10.152.
Taking step     5999 at clock time 2018-04-24-16:46:10.269.
Taking step     6000 at clock time 2018-04-24-16:46:10.383.

Domain: Dumping all at clock time 2018-04-24-16:46:10.498.
Globals dumped at 2018-04-24-16:46:10.501.
Dumping grid boundaries at 2018-04-24-16:46:10.501.
Dumped grid boundaries at 2018-04-24-16:46:10.501.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-04-24-16:46:10.501.
Domain: Histories dumped at 2018-04-24-16:46:10.506.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-16:46:10.506.
Dumping J at 2018-04-24-16:46:10.506.
Dumped J at 2018-04-24-16:46:10.576.
Dumping E at 2018-04-24-16:46:10.576.
Dumped E at 2018-04-24-16:46:10.648.
Dumping B at 2018-04-24-16:46:10.648.
Dumped B at 2018-04-24-16:46:10.750.
All multiFields dumped at 2018-04-24-16:46:10.754.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-16:46:10.755.

Main loop ended at clock time 2018-04-24-16:46:10.755
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-16:46:14.200.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'YagiUda2p4.pre' processed.
Finished with 'YagiUda2p4.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 3.2: The Run Window at the end of execution.

Next, select the *computeFarFieldFromKirchhoffBox.py* analysis script from the *Active Analyzer* drop down.

The default parameters are sufficient for this problem. Input 10.0 for the *farFieldRadius* parameter and run the analyzer by clicking the "Analyze" button.

Visualizing the results

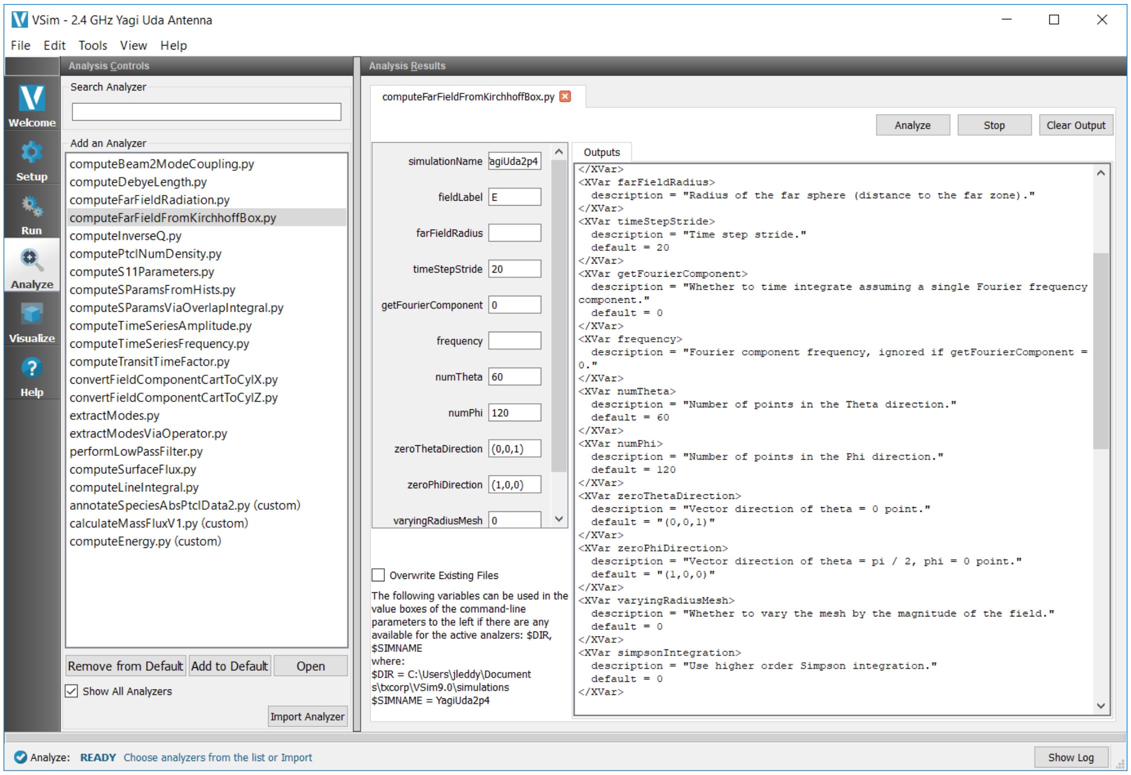
To view the near field pattern, do the following:

- Expand *Scalar Data*
- Expand *E*
- Select *E_x*
- Expand *Geometries*
- Select *poly (YagiUda2p4PecShapes)*
- Select *Clip All Plots*
- Move the dump slider forward in time

The far field radiation pattern can be found in the scalar data variables of the data overview tab underneath the farE field. Check the farE_magnitude box.

Further Experiments

Try adding more directors and changing their dimensions to see the affect on the far field pattern.



computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
computeSurfaceFlux.py
computeLineIntegral.py
annotateSpeciesAbsPtclData2.py (custom)
calculateMassFluxV1.py (custom)
computeEnergy.py (custom)

☑ Show All Analyzers

computeFarFieldFromKirchhoffBox.py

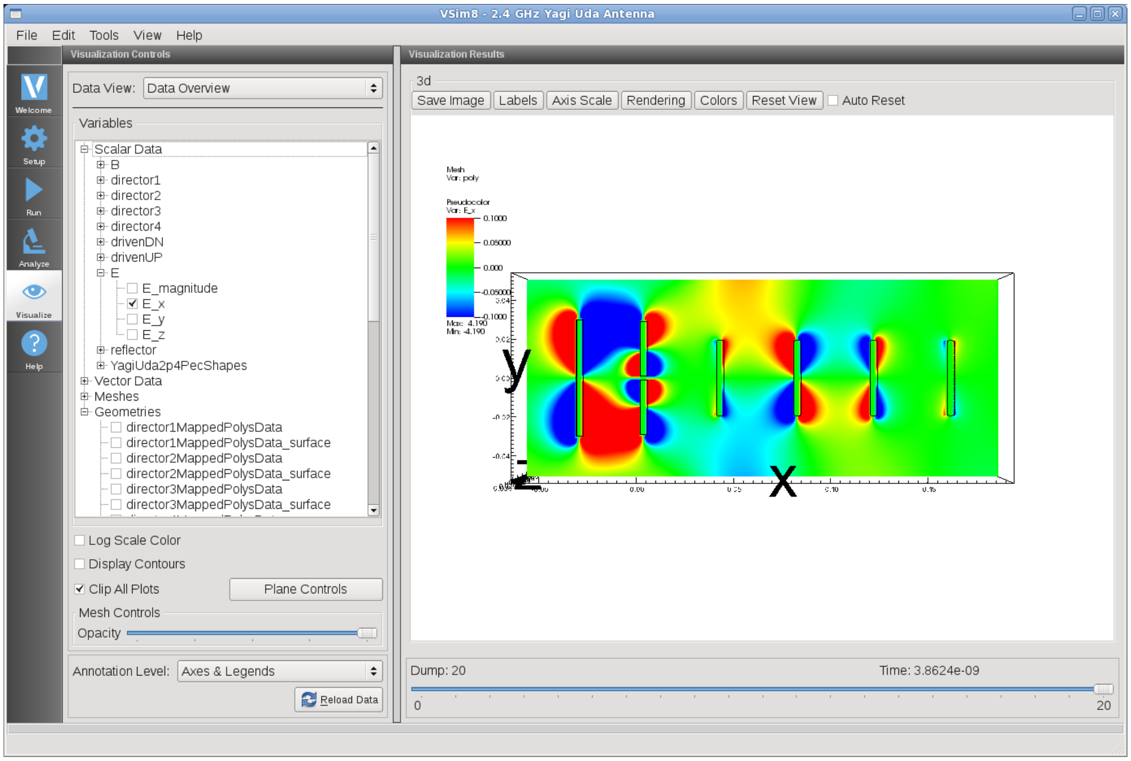
simulationName agiUda2p4
fieldLabel E
farFieldRadius
timeStepStride 20
getFourierComponent 0
frequency
numTheta 60
numPhi 120
zeroThetaDirection (0,0,1)
zeroPhiDirection (1,0,0)
varyingRadiusMesh 0

☐ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\jleddy\Documents\txcorp\VSim9.0\simulations
$SIMNAME = YagiUda2p4

```
</XVar>
<XVar farFieldRadius>
  description = "Radius of the far sphere (distance to the far zone)."
</XVar>
<XVar timeStepStride>
  description = "Time step stride."
  default = 20
</XVar>
<XVar getFourierComponent>
  description = "Whether to time integrate assuming a single Fourier frequency
component."
  default = 0
</XVar>
<XVar frequency>
  description = "Fourier component frequency, ignored if getFourierComponent =
0."
</XVar>
<XVar numTheta>
  description = "Number of points in the Theta direction."
  default = 60
</XVar>
<XVar numPhi>
  description = "Number of points in the Phi direction."
  default = 120
</XVar>
<XVar zeroThetaDirection>
  description = "Vector direction of theta = 0 point."
  default = "(0,0,1)"
</XVar>
<XVar zeroPhiDirection>
  description = "Vector direction of theta = pi / 2, phi = 0 point."
  default = "(1,0,0)"
</XVar>
<XVar varyingRadiusMesh>
  description = "Whether to vary the mesh by the magnitude of the field."
  default = 0
</XVar>
<XVar simpsonIntegration>
  description = "Use higher order Simpson integration."
  default = 0
</XVar>
```

Analyze: READY Choose analyzers from the list or Import



Data View: Data Overview

Variables
- Scalar Data
  - B
  - director1
  - director2
  - director3
  - director4
  - drivenDN
  - drivenUP
  - E
    - ☐ E_magnitude
    - ☑ E_x
    - ☐ E_y
    - ☐ E_z
  - reflector
  - YagiUda2p4PecShapes
- Vector Data
- Meshes
- Geometries
  - ☐ director1MappedPolysData
  - ☐ director1MappedPolysData_surface
  - ☐ director2MappedPolysData
  - ☐ director2MappedPolysData_surface
  - ☐ director3MappedPolysData
  - ☐ director3MappedPolysData_surface

☐ Auto Reset

☐ Log Scale Color

☐ Display Contours

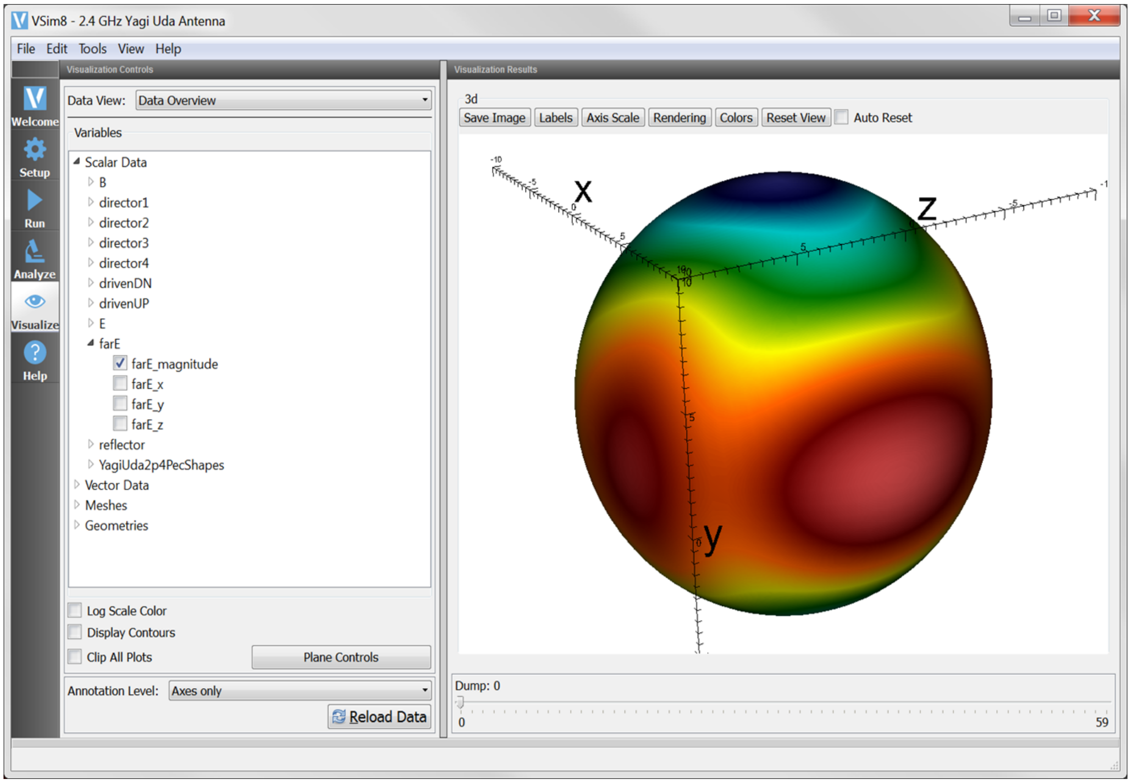
☑ Clip All Plots

Annotation Level: Axes & Legends

Dump: 20 Time: 3.8624e-09

0 20

Fig. 3.3: The electric field near-field pattern.



Data View: Data Overview

☑ farE_magnitude

□ farE_x

□ farE_y

□ farE_z

□ Log Scale Color

□ Display Contours

□ Clip All Plots

Annotation Level: Axes only

□ Auto Reset

Dump: 0

Fig. 3.4: The electric field manifestation of the far field pattern.

3.1.2 Antenna on Human Hand with Dielectric (antennaOnHand.sdf)

Keywords:

`antennaOnHand, far field, radiation`

Problem Description

This problem calculates the far-field radiation pattern of a small (cellular mobile) antenna mounted on a small curved dielectric (plastic/PVC). The fields interact with the human hand for which the bone structure was approximated by long thin cylinders. The antenna feed is a 850 MHz source.

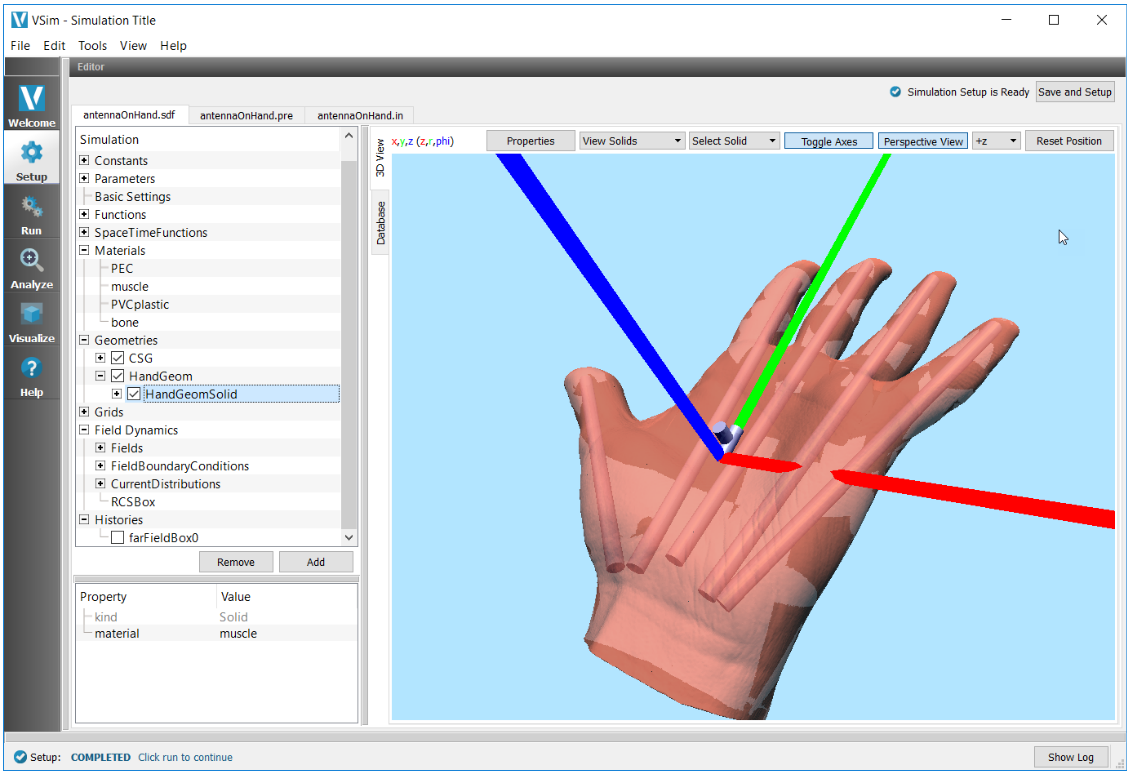
This simulation can be performed with a VSimEM license.

Opening the Simulation

The Antenna on Human Hand with Dielectric example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Antennas* option.
- Select “Antenna on Human Hand with Dielectric” and press the *Choose* button.
- In the resulting dialog, create a new folder if desired, and press the *Save* button to create a copy of this example.

The **Setup** window is now shown with all the implemented physics and geometries. See Fig. 3.5.



Simulation Setup is Ready

Property	Value
kind	Solid
material	muscle

Setup: COMPLETED Click run to continue

Fig. 3.5: Setup Window for the Antenna on Human Hand with Dielectric example.

Simulation Properties

This file allows the modification of antenna operating frequency, dimensions, orientation, simulation domain size.

Running the Simulation

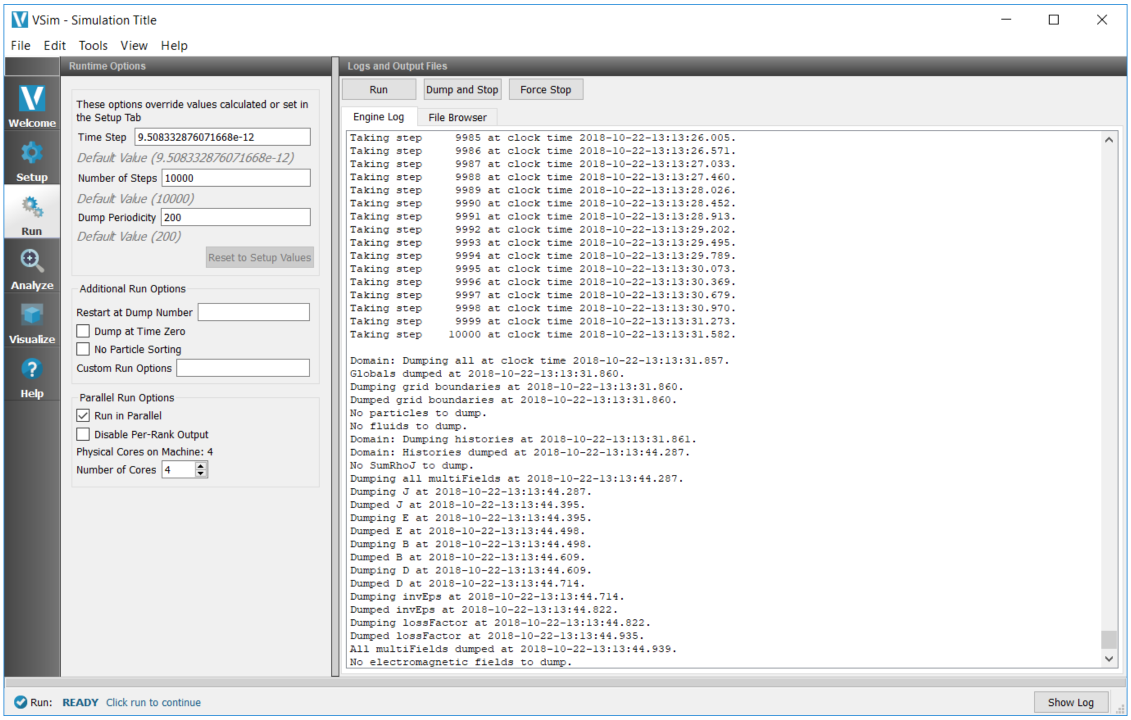
After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the **Run** button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane. You will see the output of the run in that pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in the window below.

Analyzing the Results

After performing the above actions, continue as follows:

- Proceed to the Analysis window by pressing the Analyze button in the left column of buttons.
- Tick the box “Show All Analyzers” in the bottom left corner of the window.
- Select computeFarFieldFromKirchhoffBox.py from the list and select “Open” (Fig. 3.7)
- Input values for the analyzer parameters. The analyzer may be run multiple times, allowing the user to experiment with different values.



These options override values calculated or set in the Setup Tab

Time Step 9.508332876071668e-12

Default Value (9.508332876071668e-12)

Number of Steps 10000

Default Value (10000)

Dump Periodicity 200

Default Value (200)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Taking step     9985 at clock time 2018-10-22-13:13:26.005.
Taking step     9986 at clock time 2018-10-22-13:13:26.571.
Taking step     9987 at clock time 2018-10-22-13:13:27.033.
Taking step     9988 at clock time 2018-10-22-13:13:27.460.
Taking step     9989 at clock time 2018-10-22-13:13:28.026.
Taking step     9990 at clock time 2018-10-22-13:13:28.452.
Taking step     9991 at clock time 2018-10-22-13:13:28.913.
Taking step     9992 at clock time 2018-10-22-13:13:29.202.
Taking step     9993 at clock time 2018-10-22-13:13:29.495.
Taking step     9994 at clock time 2018-10-22-13:13:29.789.
Taking step     9995 at clock time 2018-10-22-13:13:30.073.
Taking step     9996 at clock time 2018-10-22-13:13:30.369.
Taking step     9997 at clock time 2018-10-22-13:13:30.679.
Taking step     9998 at clock time 2018-10-22-13:13:30.970.
Taking step     9999 at clock time 2018-10-22-13:13:31.273.
Taking step    10000 at clock time 2018-10-22-13:13:31.582.

Domain: Dumping all at clock time 2018-10-22-13:13:31.857.
Globals dumped at 2018-10-22-13:13:31.860.
Dumping grid boundaries at 2018-10-22-13:13:31.860.
Dumped grid boundaries at 2018-10-22-13:13:31.860.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-10-22-13:13:31.861.
Domain: Histories dumped at 2018-10-22-13:13:44.287.
No SumRhoJ to dump.
Dumping all multiFields at 2018-10-22-13:13:44.287.
Dumping J at 2018-10-22-13:13:44.287.
Dumped J at 2018-10-22-13:13:44.395.
Dumping E at 2018-10-22-13:13:44.395.
Dumped E at 2018-10-22-13:13:44.498.
Dumping B at 2018-10-22-13:13:44.498.
Dumped B at 2018-10-22-13:13:44.609.
Dumping D at 2018-10-22-13:13:44.609.
Dumped D at 2018-10-22-13:13:44.714.
Dumping invEps at 2018-10-22-13:13:44.714.
Dumped invEps at 2018-10-22-13:13:44.822.
Dumping lossFactor at 2018-10-22-13:13:44.822.
Dumped lossFactor at 2018-10-22-13:13:44.935.
All multiFields dumped at 2018-10-22-13:13:44.939.
No electromagnetic fields to dump.
```

Run: **READY** Click run to continue

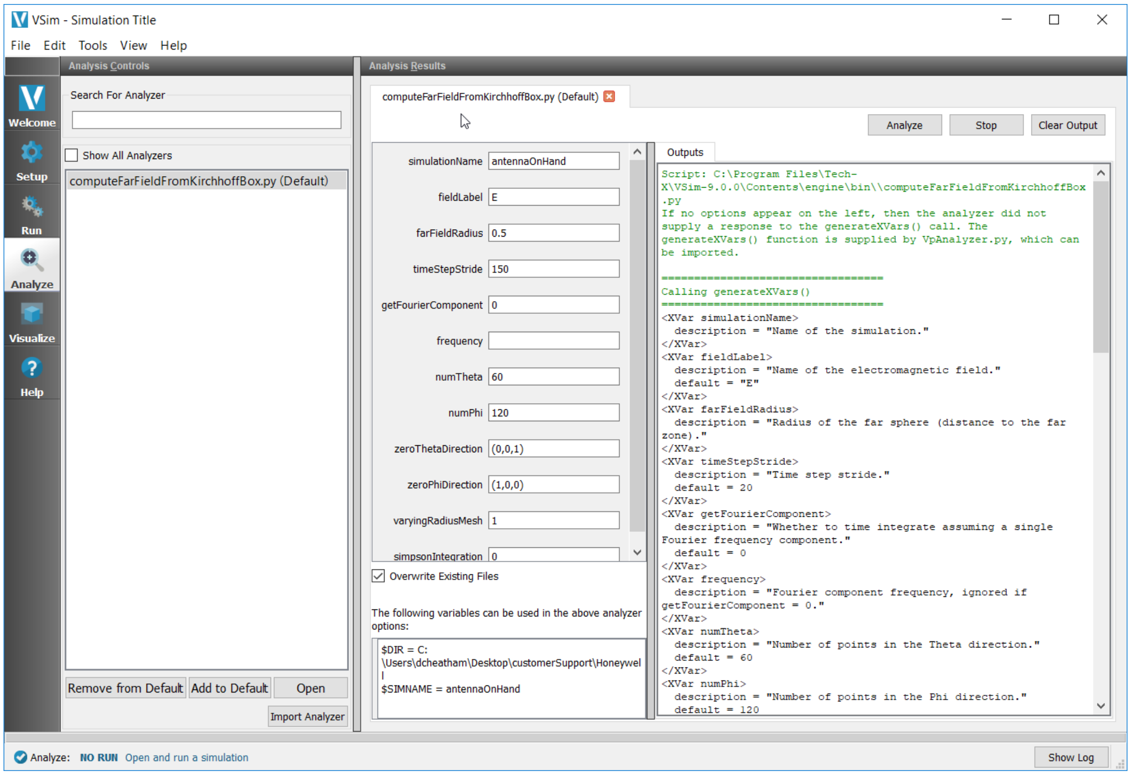
Fig. 3.6: The Run Window at the end of execution.

- simulationName - antennaOnHand (name of the input file)
- fieldLabel - E (name of the electric field)
- farFieldRadius - 0.5 (distance to far field in m, 10.0 is a good value)
- timeStepStride - 150 (number of timesteps between far field calculations; determines how many far fields are output)
- getFourierComponent - 0, do not integrate assuming single fourier frequency
- frequency - not used because getFourierComponent is false
- numTheta - 60 (number of theta points in the far field, 30 for a quick calculation, 60 for finer resolution)
- numPhi - 120 (number of phi points in the far field, 60 for a quick calculation, 120 for finer resolution)
- zeroThetaDirection - (0,0,1) (determines orientation of far field coordinate system)
- zeroPhiDirection - (1,0,0) (determines orientation of far field coordinate system
- varyingRadiusMesh - 1 (Set to 1 in order to make far field mesh adapt to magnitude of far field solution: the classic lobe view - Note: using a varrying mesh option will make the analyzer run very slow.)

- Click “Analyze”
- Depending on the values of numTheta, numPhi, and timeStepStride, the script may need to run for several minutes or longer.

Visualizing the Results

The far field radiation pattern can be found in the *Scalar Data* variables of the *Data Overview* tab. Check the *farE* box. The far field mesh can also be plotted; it can be found under *Geometries*.



simulationName: antennaOnHand
fieldLabel: E
farFieldRadius: 0.5
timeStepStride: 150
getFourierComponent: 0
frequency:
numTheta: 60
numPhi: 120
zeroThetaDirection: (0,0,1)
zeroPhiDirection: (1,0,0)
varyingRadiusMesh: 1
simpsonIntegration: 0

☑ Overwrite Existing Files

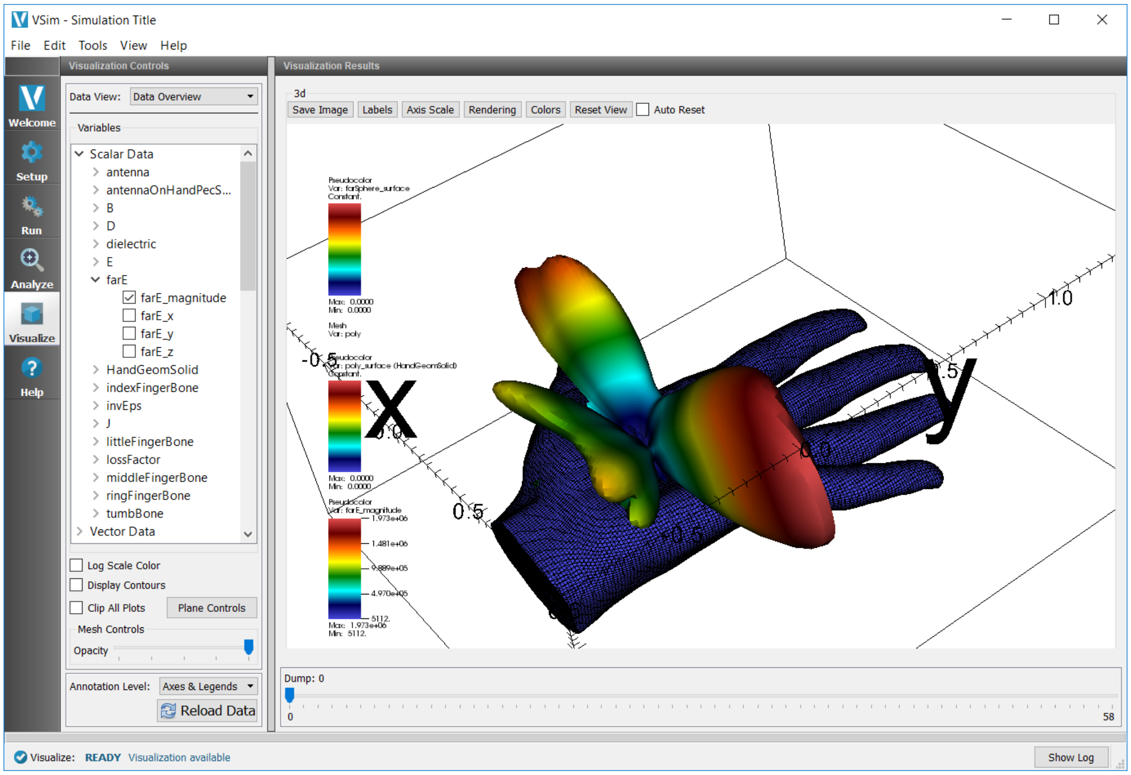
The following variables can be used in the above analyzer options:

```
$DIR = C:
\Users\dcheatham\Desktop\customerSupport\Honeywel
l
$SIMNAME = antennaOnHand
```

```
Script: C:\Program Files\Tech-
X\VSim-9.0.0\Contents\engine\bin\\computeFarFieldFromKirchhoffBox
.py
If no options appear on the left, then the analyzer did not
supply a response to the generateXVars() call. The
generateXVars() function is supplied by VpAnalyzer.py, which can
be imported.

==================================
Calling generateXVars()
==================================
<XVar simulationName>
  description = "Name of the simulation."
</XVar>
<XVar fieldLabel>
  description = "Name of the electromagnetic field."
  default = "E"
</XVar>
<XVar farFieldRadius>
  description = "Radius of the far sphere (distance to the far
zone)."
</XVar>
<XVar timeStepStride>
  description = "Time step stride."
  default = 20
</XVar>
<XVar getFourierComponent>
  description = "Whether to time integrate assuming a single
Fourier frequency component."
  default = 0
</XVar>
<XVar frequency>
  description = "Fourier component frequency, ignored if
getFourierComponent = 0."
</XVar>
<XVar numTheta>
  description = "Number of points in the Theta direction."
  default = 60
</XVar>
<XVar numPhi>
  description = "Number of points in the Phi direction."
  default = 120
```

Analyze: NO RUN Open and run a simulation

Fig. 3.7: The **Analyze** panel after running computeFarFieldFromKirchhoffBox.py.



Data View: Data Overview
☑ farE_magnitude
□ farE_x
□ farE_y
□ farE_z
□ Log Scale Color
□ Display Contours
□ Clip All Plots
Annotation Level: Axes & Legends
□ Auto Reset
Dump: 0

Visualize: READY Visualization available

Fig. 3.8: The Far Field Radiation Pattern.

Further Experiments

The skin can be included as an additional geometry by simply importing the hand geometry a second time within the same set-up, but with a very slightly higher scaling factor and setting the *Skin* material for the hand geometry with the higher scaling factor. Some "by eye" adjustments of the x-, y-, and z- traslation velues may be needed.

3.1.3 Coaxial Loop Antenna (coaxialLoopAntenna.sdf)

Keywords:

`coaxial, coaxial waveguide, coaxial cable`

Problem description

This example illustrates how to use the coaxial cable Field Boundary Condition and Constructive Solid Geometry to create a coaxial loop antenna.

This simulation can be run with a VSimEM, VSimMD, VSimPD, or VSimPA license.

Opening the Simulation

The Coaxial Loop Antenna example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Antennas* option.
- Select "A Loop Antenna created from a coaxial cable" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is shown Fig. 3.9.

Simulation Properties

This simulation makes use of the new coaxial waveguide Field Boundary Condition in VSim 8.1.

A coaxial waveguide is first constructed by creating a physical coaxial cable that enters the simulation domain. It is very important that this cable exist from atleast 1 cell outside of to 1 cell inside the simulation boundary. This is done by first creating a box primitive and setting it along the desired simulation boundary.

A cylinder corresponding to the outer diameter of the coaxial cable is then created, and subtracted from the plate.

A cylinder corresponding to the inner diameter of the coaxial cable is then created and extended into the simulation space.

It is then made into a loop antenna by adding a second, intersecting cylinder.

The wave itself is specified by a Field Boundary Condition.

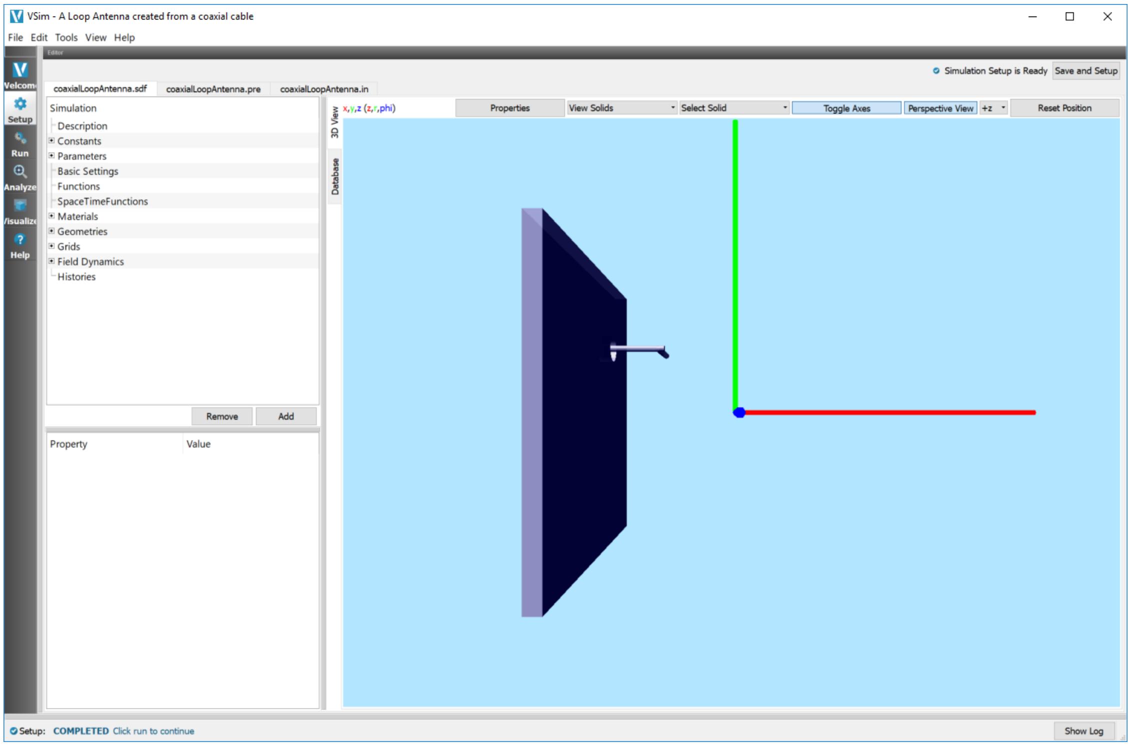


Fig. 3.9: Setup Window for the Coaxial Loop Antenna example.

Running the Simulation

Once finished with the problem setup, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- Choose parallel computing options on the MPI tab
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in this pane. The run has completed when you see the output, “Engine completed successfully.”

Visualizing the Results

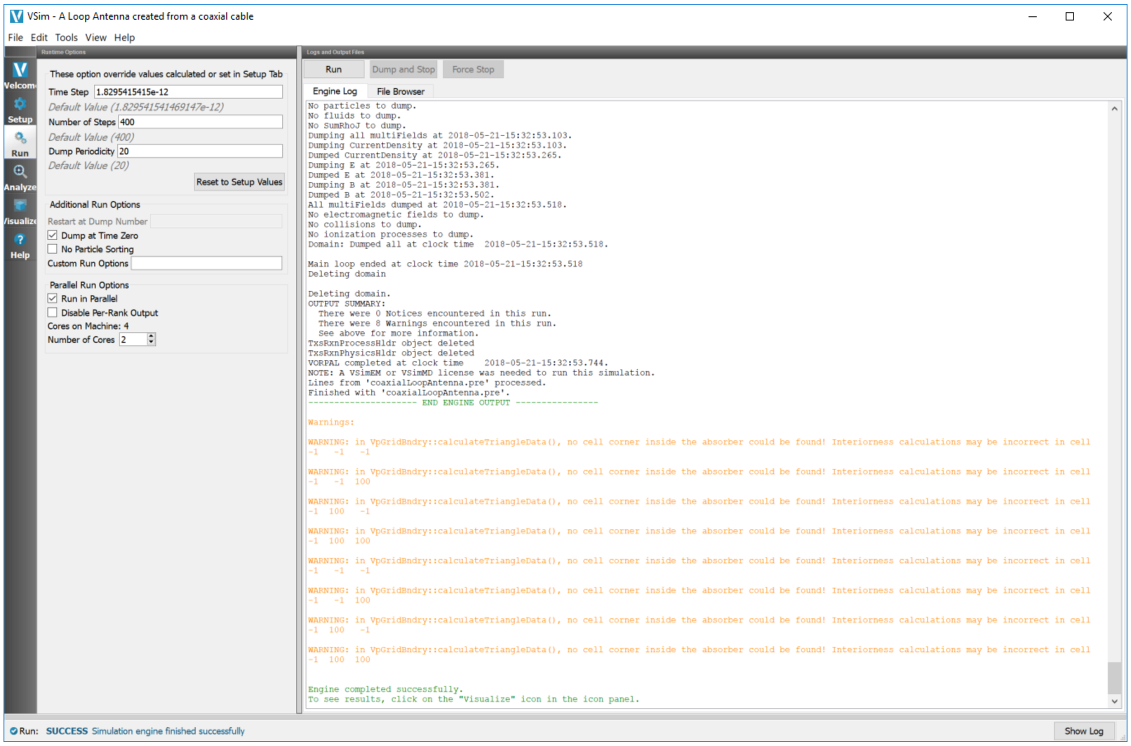
Under *Scalar Data* select *E_z*. To slice inside the field, select *Clip All Plots* in the lower left hand corner. Click on *Plane Controls* and change the cut-plane normal to lie along Y instead of Z, and adjust the origin of the normal vector to .05 . Under the colors button set the min to -10 and the max to 10 to see the field.

Further Experiments

3.1.4 Dipole Above Conducting Plane (dipoleOnConductingPlane.sdf)

Keywords:

`dipoleOnConductingPlane, far field, radiation`



These option override values calculated or set in Setup Tab

Time Step 1.8295415415e-12
Default Value (1.829541541469147e-12)
Number of Steps 400
Default Value (400)
Dump Periodicity 20
Default Value (20)

Additional Run Options

Restart at Dump Number
☑ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
□ Disable Per-Rank Output
Cores on Machine: 4
Number of Cores 2

```
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-21-15:32:53.103.
Dumping CurrentDensity at 2018-05-21-15:32:53.103.
Dumped CurrentDensity at 2018-05-21-15:32:53.265.
Dumping E at 2018-05-21-15:32:53.265.
Dumped E at 2018-05-21-15:32:53.381.
Dumping B at 2018-05-21-15:32:53.381.
Dumped B at 2018-05-21-15:32:53.502.
All multiFields dumped at 2018-05-21-15:32:53.518.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-21-15:32:53.518.

Main loop ended at clock time 2018-05-21-15:32:53.518
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 8 Warnings encountered in this run.
  See above for more information.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-05-21-15:32:53.744.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'coaxialLoopAntenna.pre' processed.
Finished with 'coaxialLoopAntenna.pre'.
--------------------- END ENGINE OUTPUT ----------------

Warnings:

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may be incorrect in cell
-1   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may be incorrect in cell
-1   -1  100

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may be incorrect in cell
-1  100   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may be incorrect in cell
-1  100  100

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may be incorrect in cell
-1   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may be incorrect in cell
-1   -1  100

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may be incorrect in cell
-1  100   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may be incorrect in cell
-1  100  100


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 3.10: The Run Window at the end of execution.

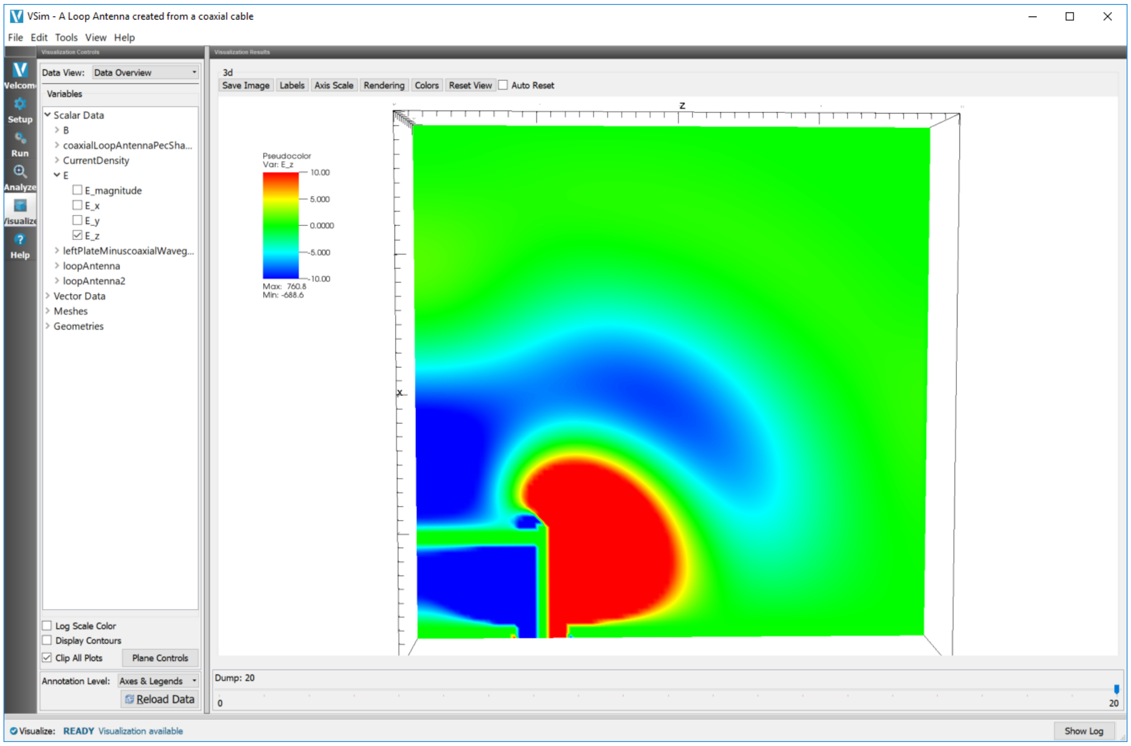


Fig. 3.11: The E_z field propagating off of the loop antenna.

Problem Description

This problem illustrates how to obtain far fields within VSim by simulating an infinitesimally short dipole mounted a variable height above a conducting plane. The conducting plane is simulated by using the method of images and utilizes an equal magnitude dipole with direction rotated azimuthally by PI, on the opposite side of the plane. This example is similar to the Oscillating Dipole Above Conducting Plane of VSimBase, but modified with functionality available as part of the VSimEM package to obtain the far field radiation pattern. The number of lobes in the far field will vary as a function of height above the conducting plane. There will be 2*HEIGHT/WAVELENGTH + 1 lobes.

This simulation can be performed with a VSimEM license.

Opening the Simulation

The Dipole Above Conducting Plane example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Antennas* option.
- Select “Dipole Above Conducting Plane” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 3.12. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.

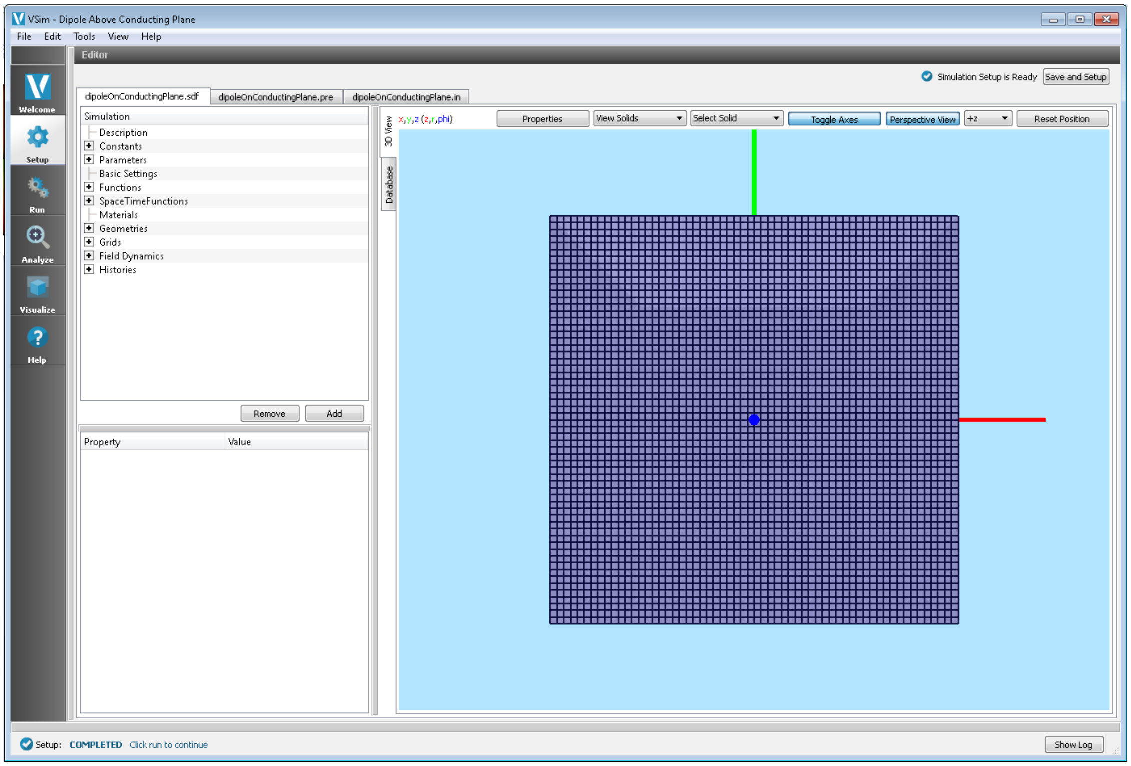


Fig. 3.12: Setup Window for the Dipole Above Conducting Plane example.

Simulation Properties

This setup includes several *Constants* and *Parameters* to help define the dipole signals, including the frequency and height of the antenna.

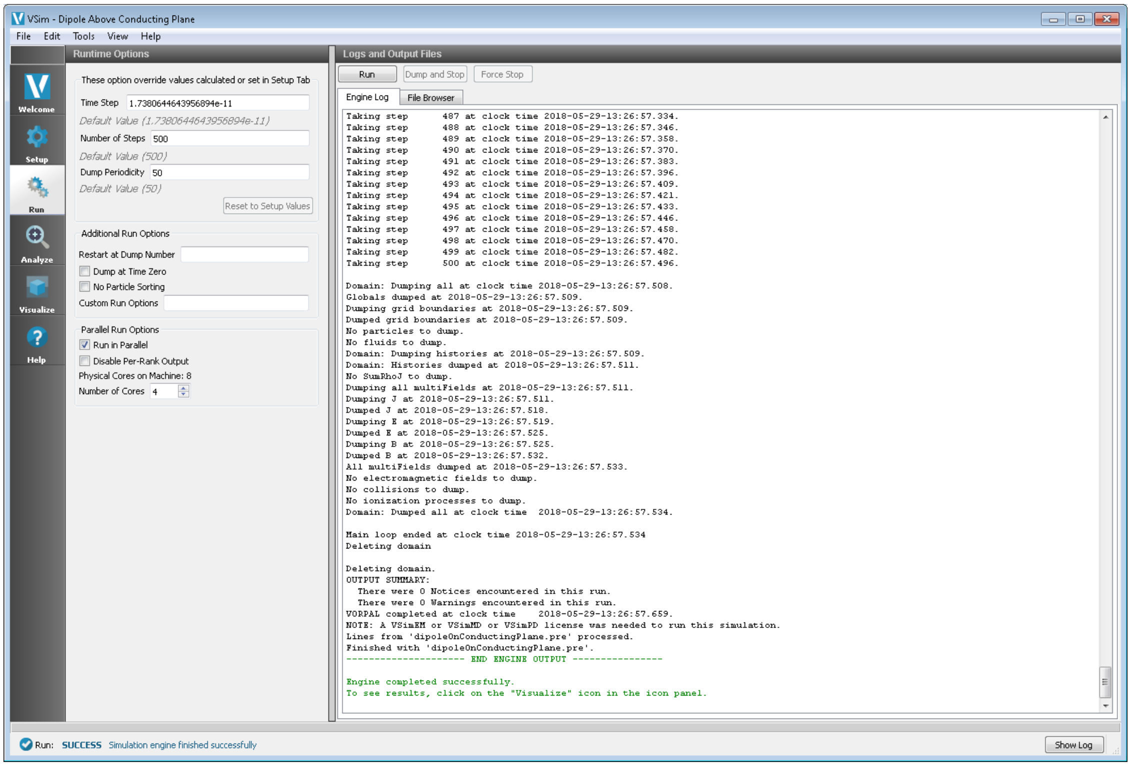
There are open boundary conditions on each side of the simulation domain.

The conducting plane is simulated by using the method of images and utilizes an equal magnitude dipole with direction rotated azimuthally by PI, on the opposite side of the plane.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner. of the window. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 3.13.



These option override values calculated or set in Setup Tab

Time Step 1.7380644643956894e-11
Default Value (1.7380644643956894e-11)
Number of Steps 500
Default Value (500)
Dump Periodicity 50
Default Value (50)

Additional Run Options

Restart at Dump Number
☐ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
☐ Disable Per-Rank Output
Physical Cores on Machine: 8
Number of Cores 4

```
Taking step      487 at clock time 2018-05-29-13:26:57.334.
Taking step      488 at clock time 2018-05-29-13:26:57.346.
Taking step      489 at clock time 2018-05-29-13:26:57.358.
Taking step      490 at clock time 2018-05-29-13:26:57.370.
Taking step      491 at clock time 2018-05-29-13:26:57.383.
Taking step      492 at clock time 2018-05-29-13:26:57.396.
Taking step      493 at clock time 2018-05-29-13:26:57.409.
Taking step      494 at clock time 2018-05-29-13:26:57.421.
Taking step      495 at clock time 2018-05-29-13:26:57.433.
Taking step      496 at clock time 2018-05-29-13:26:57.446.
Taking step      497 at clock time 2018-05-29-13:26:57.458.
Taking step      498 at clock time 2018-05-29-13:26:57.470.
Taking step      499 at clock time 2018-05-29-13:26:57.482.
Taking step      500 at clock time 2018-05-29-13:26:57.496.

Domain: Dumping all at clock time 2018-05-29-13:26:57.508.
Globals dumped at 2018-05-29-13:26:57.509.
Dumping grid boundaries at 2018-05-29-13:26:57.509.
Dumped grid boundaries at 2018-05-29-13:26:57.509.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-05-29-13:26:57.509.
Domain: Histories dumped at 2018-05-29-13:26:57.511.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-29-13:26:57.511.
Dumping J at 2018-05-29-13:26:57.511.
Dumped J at 2018-05-29-13:26:57.518.
Dumping E at 2018-05-29-13:26:57.519.
Dumped E at 2018-05-29-13:26:57.525.
Dumping B at 2018-05-29-13:26:57.525.
Dumped B at 2018-05-29-13:26:57.532.
All multiFields dumped at 2018-05-29-13:26:57.533.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-29-13:26:57.534.

Main loop ended at clock time 2018-05-29-13:26:57.534
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-05-29-13:26:57.659.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.
Lines from 'dipoleOnConductingPlane.pre' processed.
Finished with 'dipoleOnConductingPlane.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

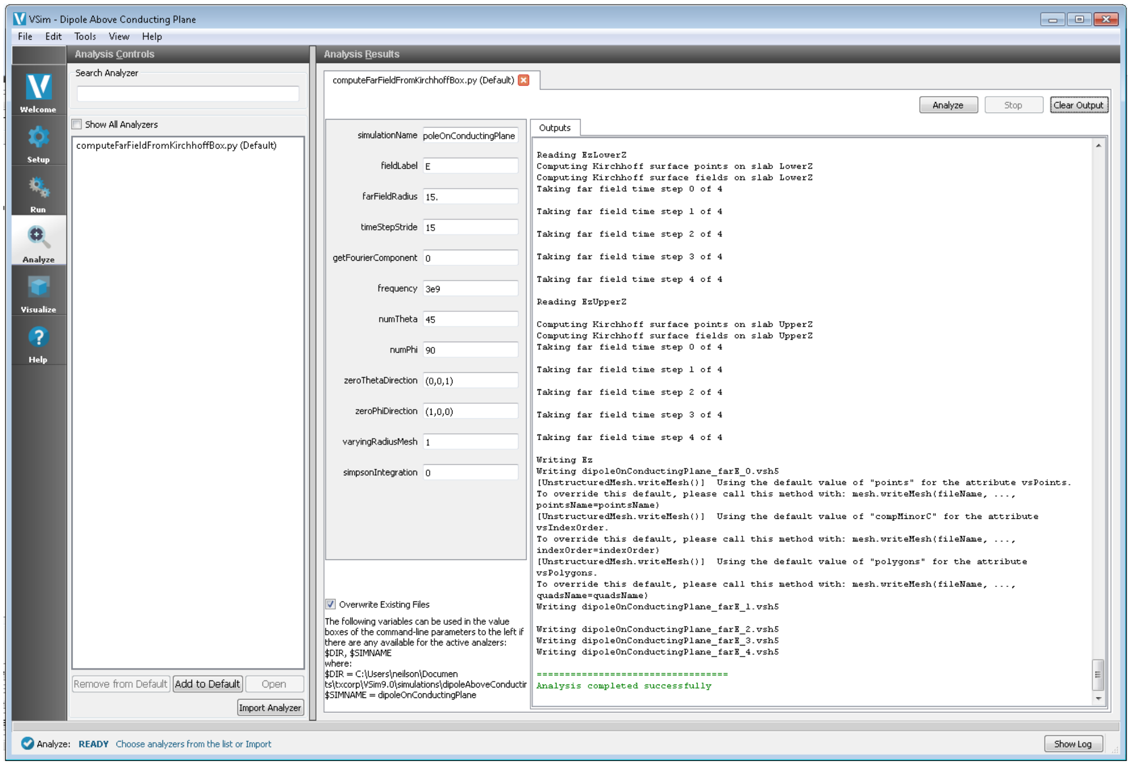
Run: SUCCESS Simulation engine finished successfully

Fig. 3.13: The Run Window at the end of execution.

Analyzing the Results

- Proceed to the Analysis window by pressing the Analyze button in the left column of buttons.
- Select the Default *computeFarFieldFromKirchhoffBox.py* Analyzer
- Input values for the variables given on the left hand side of the screen. Check that these have the following values.

   - simulationName - dipoleOnConductingPlane (name of the input file)
   - fieldLabel - E
   - farFieldRadius - 15.
   - timeStepStride - 15
   - getFourierComponent - 0
   - frequency - 3e9
   - numTheta - 45 (number of points in the theta direction)
   - numPhi - 90 (number of points in the phi direction)
   - zeroThetaDirection - (0,0,1)
   - zeroPhiDirection - (1,0,0)
   - varyingRadiusMesh - 1 (Varys the radius of the field based on field magnitude)
   - simpsonIntegration - 0
- Click Run



☐ Show All Analyzers

computeFarFieldFromKirchhoffBox.py (Default)

simulationName	poleOnConductingPlane
fieldLabel	E
farFieldRadius	15.
timeStepStride	15
getFourierComponent	0
frequency	3e9
numTheta	45
numPhi	90
zeroThetaDirection	(0,0,1)
zeroPhiDirection	(1,0,0)
varyingRadiusMesh	1
simpsonIntegration	0

☑ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analzers:
$DIR, $SIMNAME
where:
$DIR = C:\Users\neilson\Documents\txcorp\VSim9.0\simulations\dipoleAboveConductir
$SIMNAME = dipoleOnConductingPlane

```
Reading EzLowerZ
Computing Kirchhoff surface points on slab LowerZ
Computing Kirchhoff surface fields on slab LowerZ
Taking far field time step 0 of 4

Taking far field time step 1 of 4

Taking far field time step 2 of 4

Taking far field time step 3 of 4

Taking far field time step 4 of 4

Reading EzUpperZ

Computing Kirchhoff surface points on slab UpperZ
Computing Kirchhoff surface fields on slab UpperZ
Taking far field time step 0 of 4

Taking far field time step 1 of 4

Taking far field time step 2 of 4

Taking far field time step 3 of 4

Taking far field time step 4 of 4

Writing Ez
Writing dipoleOnConductingPlane_farE_0.vsh5
[UnstructuredMesh.writeMesh()]  Using the default value of "points" for the attribute vsPoints.
To override this default, please call this method with: mesh.writeMesh(fileName, ...,
pointsName=pointsName)
[UnstructuredMesh.writeMesh()]  Using the default value of "compMinorC" for the attribute
vsIndexOrder.
To override this default, please call this method with: mesh.writeMesh(fileName, ...,
indexOrder=indexOrder)
[UnstructuredMesh.writeMesh()]  Using the default value of "polygons" for the attribute
vsPolygons.
To override this default, please call this method with: mesh.writeMesh(fileName, ...,
quadsName=quadsName)
Writing dipoleOnConductingPlane_farE_1.vsh5

Writing dipoleOnConductingPlane_farE_2.vsh5
Writing dipoleOnConductingPlane_farE_3.vsh5
Writing dipoleOnConductingPlane_farE_4.vsh5

==================================
Analysis completed successfully
```

Analyze: READY Choose analyzers from the list or Import

Fig. 3.14: The Analyze Window at the end of execution.

Visualizing the Results

The far field radiation pattern can be found in the scalar data variables of the data overview tab. Check the farE_magnitude box.

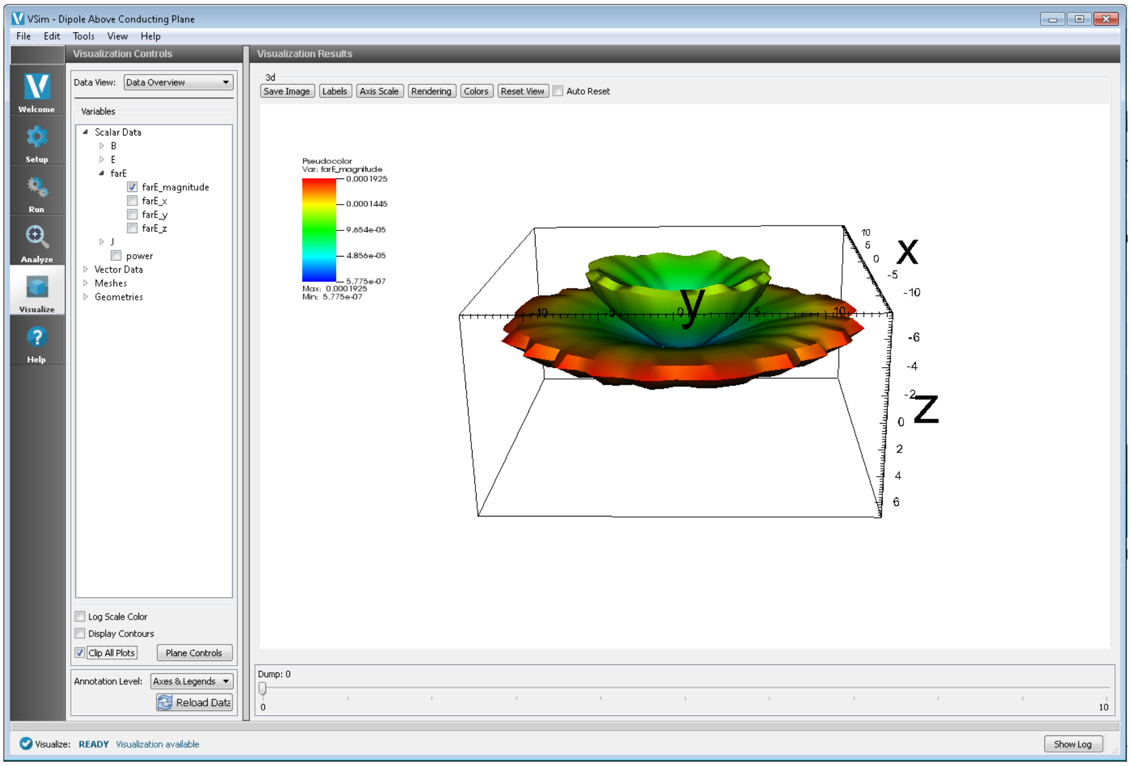


Fig. 3.15: The far field radiation pattern

Further Experiments

The number of lobes in the far field is dependent on Antenna Orientation and height. If vertically oriented there will be 2*Height/Wavelength +1 lobes. A horizontally oriented dipole will produce 2*Height/Wavelength lobes.

The resolution of the far field pattern can be changed by editing the number of theta and phi points in the analyis.

If the Simulation domain is made too small, the results will be distorted as the entire near field must be within the simulation domain in order to acheive a proper transformation to the far field.

Note that an infinite perfect electric conducting plane is simulated in the computational engine via image theory. An equal infintesimal dipole is placed the same distance from the conducting "plane" in order to acheive the result of having an infinite electric conductor.

3.1.5 Dish Antenna (dishAntenna.sdf)

Keywords:

electromagnetics, antennas

Problem Description

The Dish Antenna simulation illustrates how to get the radiation pattern from a source in the presence of a complex shape.

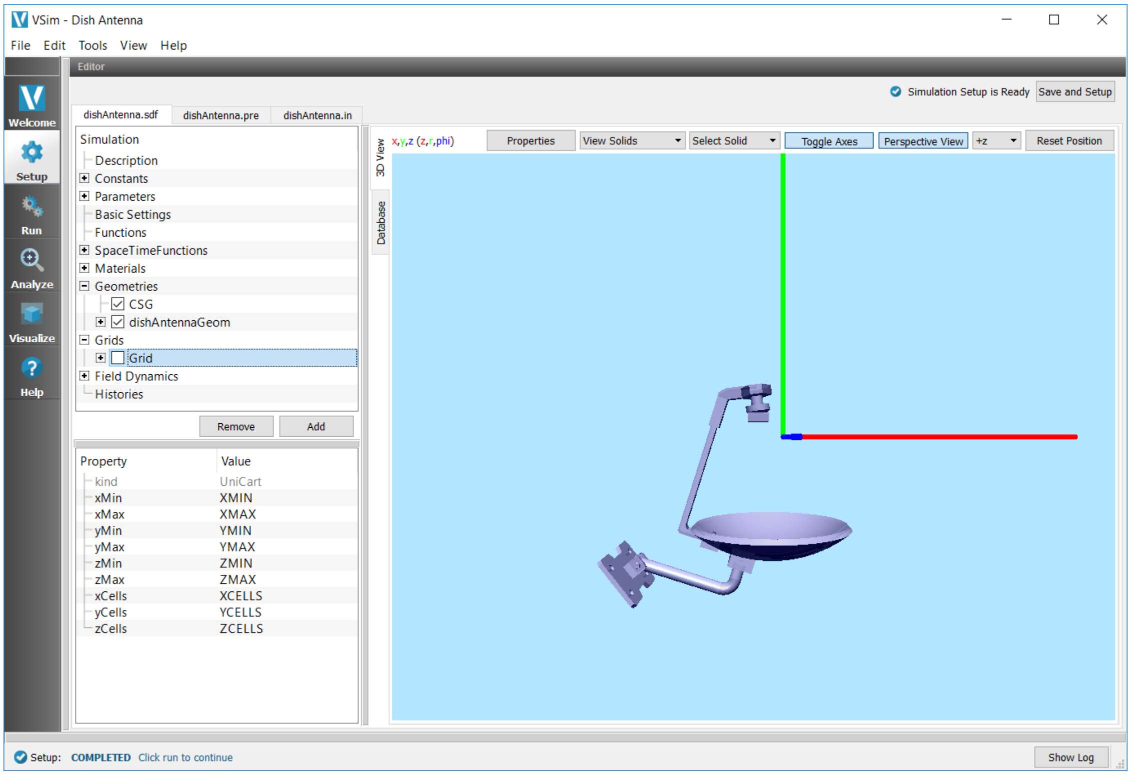
This simulation can be performed with a VSimEM, VSimMD or VSimPD license.

Opening the Simulation

The Dish Antenna example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Antennas* option.
- Select "Dish Antenna" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 3.16. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



Simulation
- Description
- ⊞ Constants
- ⊞ Parameters
- Basic Settings
- Functions
- ⊞ SpaceTimeFunctions
- ⊞ Materials
- ⊟ Geometries
  - ☑ CSG
  - ⊞ ☑ dishAntennaGeom
- ⊟ Grids
  - ⊞ □ Grid
- ⊞ Field Dynamics
- Histories

Property	Value
kind	UniCart
xMin	XMIN
xMax	XMAX
yMin	YMIN
yMax	YMAX
zMin	ZMIN
zMax	ZMAX
xCells	XCELLS
yCells	YCELLS
zCells	ZCELLS

Setup: COMPLETED Click run to continue

Fig. 3.16: Setup Window for the Dish Antenna example.

Simulation Properties

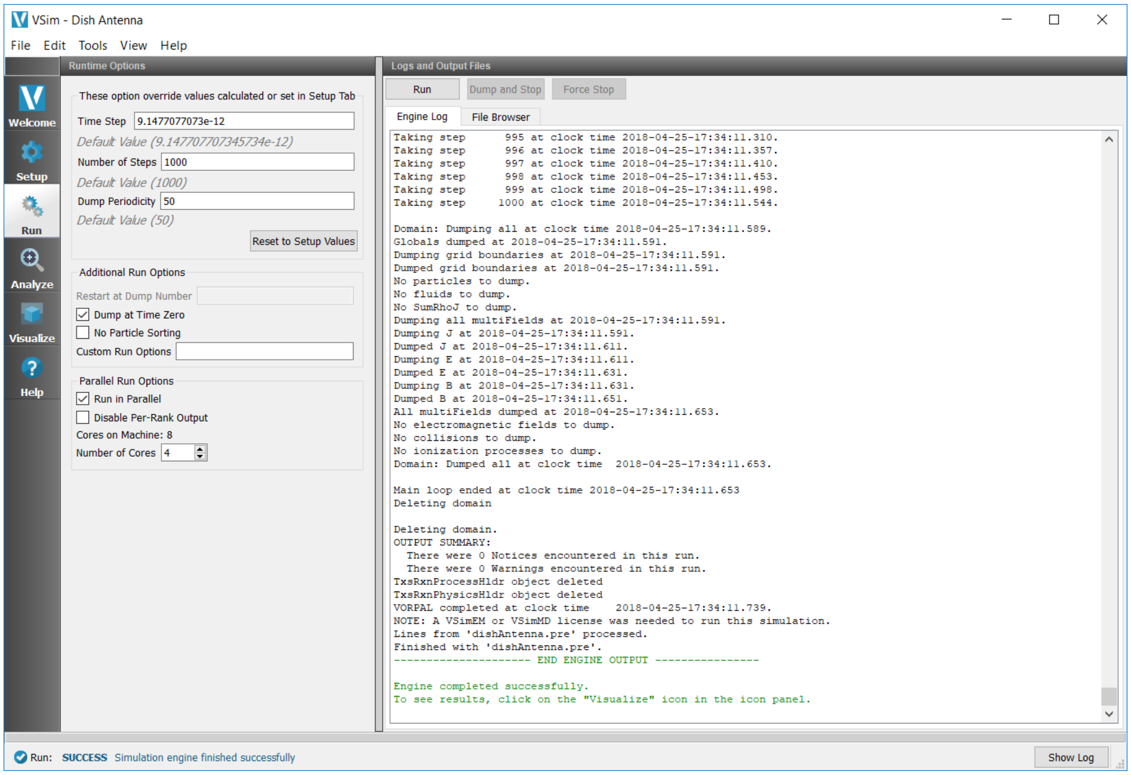
One can set the parameters of the grid and the source through the setup tree. The parameters are put under the Constants section.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.

- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in the *Logs and Output Files* pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 3.17.



These option override values calculated or set in Setup Tab

Time Step 9.1477077073e-12

Default Value (9.147707707345734e-12)

Number of Steps 1000

Default Value (1000)

Dump Periodicity 50

Default Value (50)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Cores on Machine: 8

Number of Cores 4

```
Taking step     995 at clock time 2018-04-25-17:34:11.310.
Taking step     996 at clock time 2018-04-25-17:34:11.357.
Taking step     997 at clock time 2018-04-25-17:34:11.410.
Taking step     998 at clock time 2018-04-25-17:34:11.453.
Taking step     999 at clock time 2018-04-25-17:34:11.498.
Taking step    1000 at clock time 2018-04-25-17:34:11.544.

Domain: Dumping all at clock time 2018-04-25-17:34:11.589.
Globals dumped at 2018-04-25-17:34:11.591.
Dumping grid boundaries at 2018-04-25-17:34:11.591.
Dumped grid boundaries at 2018-04-25-17:34:11.591.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-25-17:34:11.591.
Dumping J at 2018-04-25-17:34:11.591.
Dumped J at 2018-04-25-17:34:11.611.
Dumping E at 2018-04-25-17:34:11.611.
Dumped E at 2018-04-25-17:34:11.631.
Dumping B at 2018-04-25-17:34:11.631.
Dumped B at 2018-04-25-17:34:11.651.
All multiFields dumped at 2018-04-25-17:34:11.653.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-25-17:34:11.653.

Main loop ended at clock time 2018-04-25-17:34:11.653
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-25-17:34:11.739.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'dishAntenna.pre' processed.
Finished with 'dishAntenna.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 3.17: The Run Window at the end of execution.

Visualizing the Results

After performing the above actions, continue as follows:

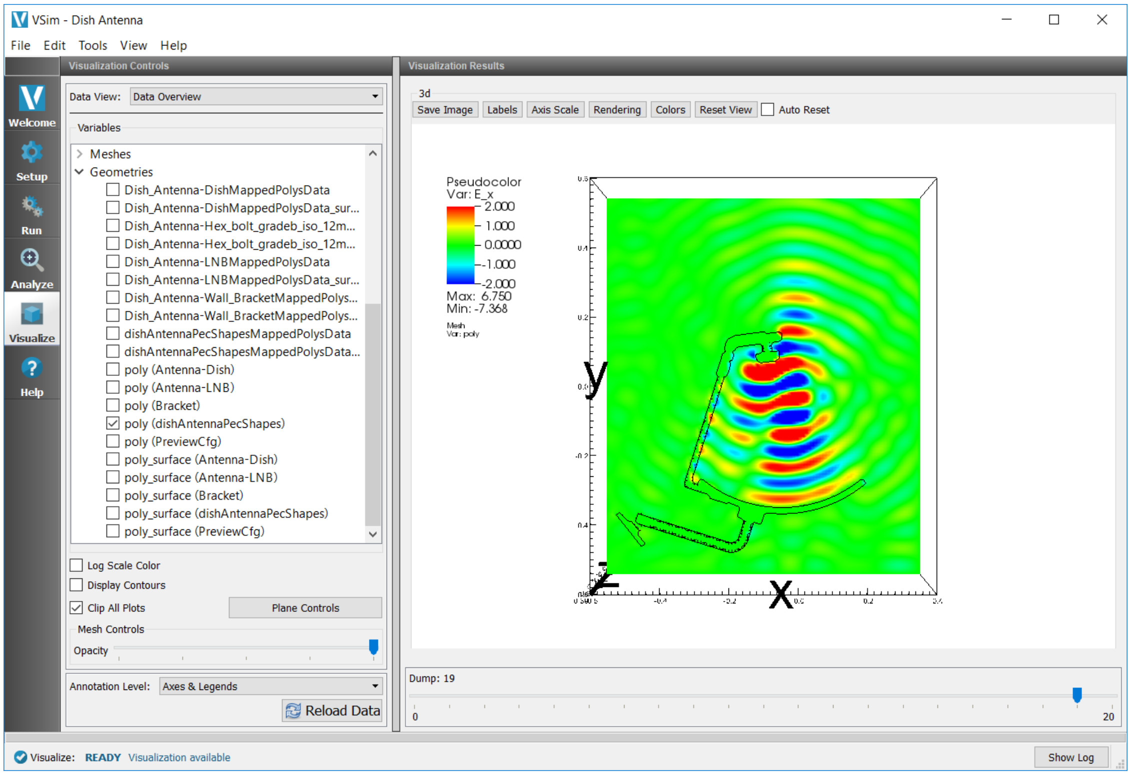
- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the electric field reflected from the dish antenna as shown in Fig. 3.18, do the following:

- Expand *Scalar Data*
- Expand *E*
- Select *E_x*
- Expand *Geometries*
- Select *poly (dishAntennaPecShapes)*
- Select *Clip All Plots*

It is easier to see the fields if you change the color scale minimum and maximum. To do so, click on the *Colors* button, and set a fixed minimum of -2 and a fixed maximum of 2.

Move the slider at the bottom of the right pane to see the electric field at different times.


VSim - Dish Antenna
File Edit Tools View Help
Welcome
Setup
Run
Analyze
Visualize
Help
Visualization Controls
Variables
Meshes
Geometries
Dish_Antenna-DishMappedPolysData
Dish_Antenna-DishMappedPolysData_sur...
Dish_Antenna-Hex_bolt_gradeb_iso_12m...
Dish_Antenna-Hex_bolt_gradeb_iso_12m...
Dish_Antenna-LNBMappedPolysData
Dish_Antenna-LNBMappedPolysData_sur...
Dish_Antenna-Wall_BracketMappedPolys...
Dish_Antenna-Wall_BracketMappedPolys...
dishAntennaPecShapesMappedPolysData
dishAntennaPecShapesMappedPolysData...
poly (Antenna-Dish)
poly (Antenna-LNB)
poly (Bracket)
poly (dishAntennaPecShapes)
poly (PreviewCfg)
poly_surface (Antenna-Dish)
poly_surface (Antenna-LNB)
poly_surface (Bracket)
poly_surface (dishAntennaPecShapes)
poly_surface (PreviewCfg)
Log Scale Color
Display Contours
Clip All Plots
Plane Controls
Mesh Controls
Opacity
Reload Data
Visualization Results
3d
Save Image
Labels
Axis Scale
Rendering
Colors
Reset View
Auto Reset
Pseudocolor
Var: E_x
2.000
1.000
0.0000
-1.000
-2.000
Max: 6.750
Min: -7.368
Mesh
Var: poly
y
x
z
0
20
Visualize: READY Visualization available
Show Log

Data View: Data Overview

☑ poly (dishAntennaPecShapes)

☐ Log Scale Color

☐ Display Contours

☑ Clip All Plots

☐ Auto Reset

Annotation Level: Axes & Legends

Dump: 19

Fig. 3.18: Visualization of a slice of the electric field as a color contour plot at dump 19.

Further Experiments

Additional experiments worth investigating are:

- Change the resolution to see whether more resolution gives a different answer.
- Change the frequency of the source. Be careful, because at high frequencies with the chosen resolution, one will require a large amount of memory.

3.1.6 Half-Wave Dipole in Free Space (halfWaveDipoleAntenna.sdf)

Keywords:

`halfWaveDipoleAntenna, far field, radiation`

Problem Description

This problem illustrates how to obtain far field radiation patterns from VSim simulation data. The simulation itself consists of a half-wavelength long current source in free space.

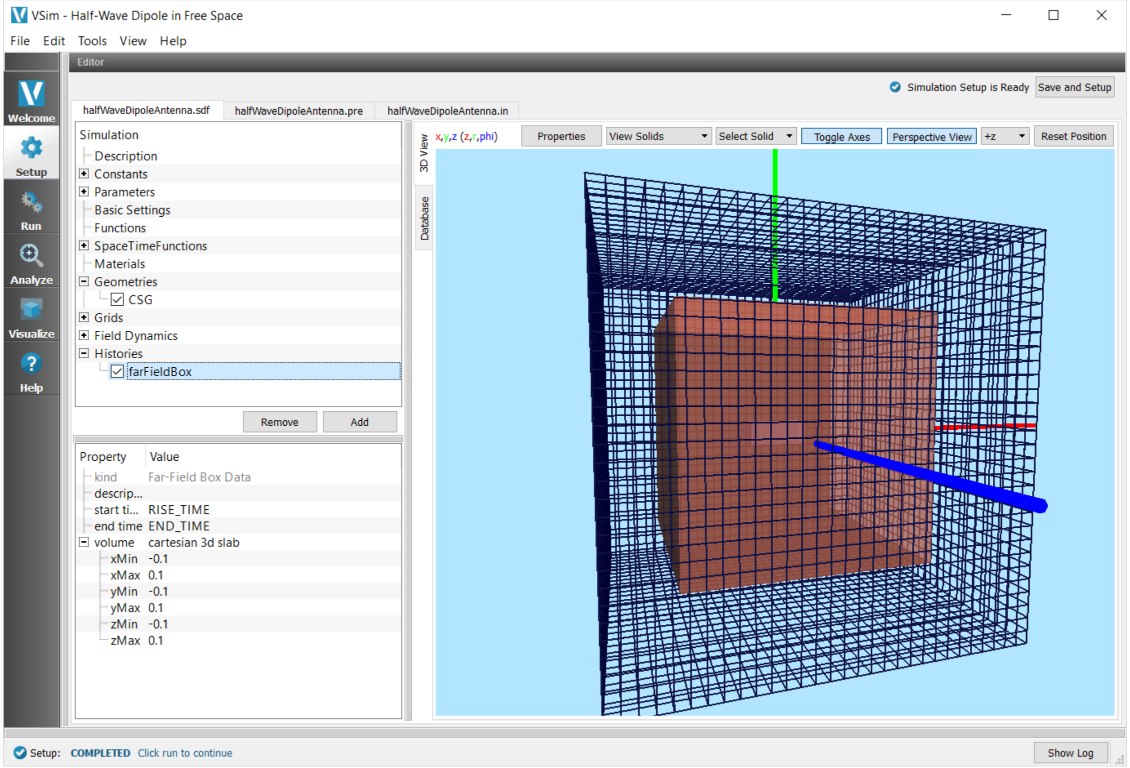
This simulation can be performed with a VSimEM license.

Opening the Simulation

The Half Wave Dipole Antenna example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Antennas* option.
- Select "Half-Wave Dipole in Free Space" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 3.19. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



Simulation
- Description
- ⊞ Constants
- ⊞ Parameters
- Basic Settings
- Functions
- ⊞ SpaceTimeFunctions
- Materials
- ⊟ Geometries
  - ☑ CSG
- ⊞ Grids
- ⊞ Field Dynamics
- ⊟ Histories
  - ☑ farFieldBox

Property	Value
kind	Far-Field Box Data
descrip…	
start ti…	RISE_TIME
end time	END_TIME
⊟ volume	cartesian 3d slab
xMin	-0.1
xMax	0.1
yMin	-0.1
yMax	0.1
zMin	-0.1
zMax	0.1

Setup: COMPLETED Click run to continue

Fig. 3.19: Setup Window for the Half Wave Dipole Antenna example.

Simulation Properties

This example includes *Constants* for easy adjustment of simulation properties, Including:

- WAVELENGTH: The wavelength of the antenna
- FREQUENCY: The frequency of the antenna

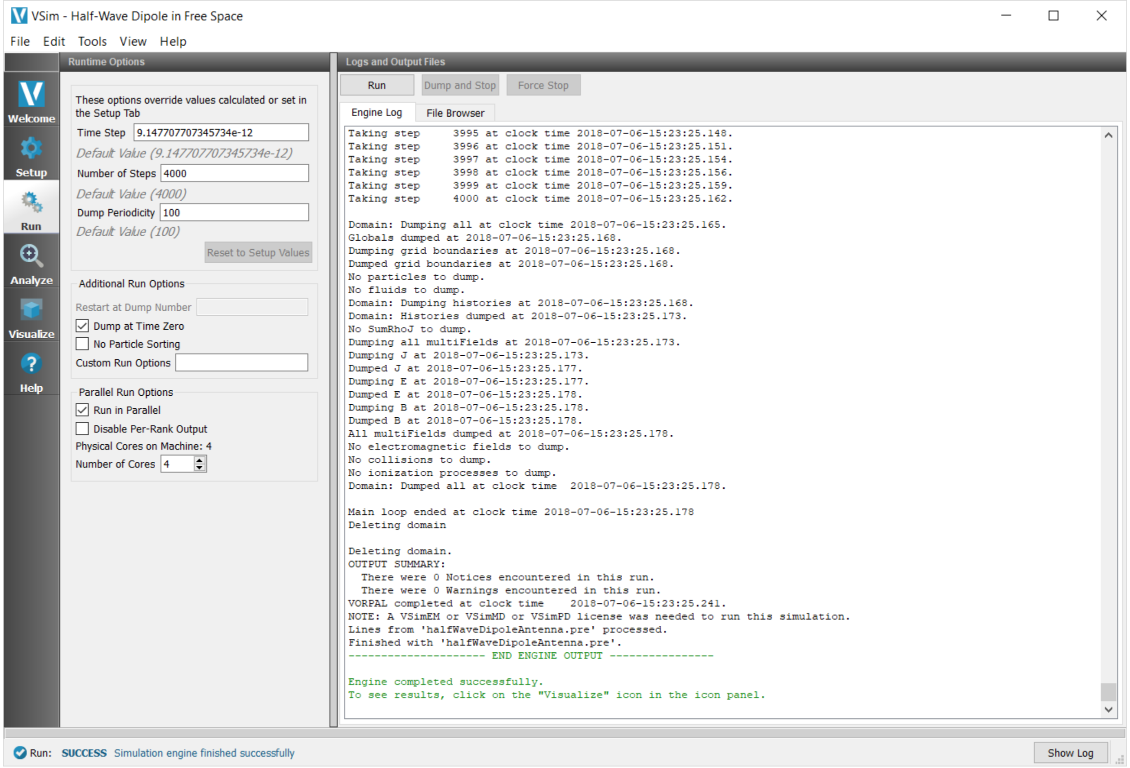
There are also *SpaceTimeFunctions* to define the current driver of the half wavelength source.

Other properties of the simulation include open boundaries on all sides. A Distributed Current source is used to set the current of the half wavelength antenna.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 3.20.



These options override values calculated or set in the Setup Tab

Time Step 9.147707707345734e-12
Default Value (9.147707707345734e-12)
Number of Steps 4000
Default Value (4000)
Dump Periodicity 100
Default Value (100)

Additional Run Options
Restart at Dump Number
☑ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options
☑ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
Taking step     3995 at clock time 2018-07-06-15:23:25.148.
Taking step     3996 at clock time 2018-07-06-15:23:25.151.
Taking step     3997 at clock time 2018-07-06-15:23:25.154.
Taking step     3998 at clock time 2018-07-06-15:23:25.156.
Taking step     3999 at clock time 2018-07-06-15:23:25.159.
Taking step     4000 at clock time 2018-07-06-15:23:25.162.

Domain: Dumping all at clock time 2018-07-06-15:23:25.165.
Globals dumped at 2018-07-06-15:23:25.168.
Dumping grid boundaries at 2018-07-06-15:23:25.168.
Dumped grid boundaries at 2018-07-06-15:23:25.168.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-07-06-15:23:25.168.
Domain: Histories dumped at 2018-07-06-15:23:25.173.
No SumRhoJ to dump.
Dumping all multiFields at 2018-07-06-15:23:25.173.
Dumping J at 2018-07-06-15:23:25.173.
Dumped J at 2018-07-06-15:23:25.177.
Dumping E at 2018-07-06-15:23:25.177.
Dumped E at 2018-07-06-15:23:25.178.
Dumping B at 2018-07-06-15:23:25.178.
Dumped B at 2018-07-06-15:23:25.178.
All multiFields dumped at 2018-07-06-15:23:25.178.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-06-15:23:25.178.

Main loop ended at clock time 2018-07-06-15:23:25.178
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-07-06-15:23:25.241.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.
Lines from 'halfWaveDipoleAntenna.pre' processed.
Finished with 'halfWaveDipoleAntenna.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

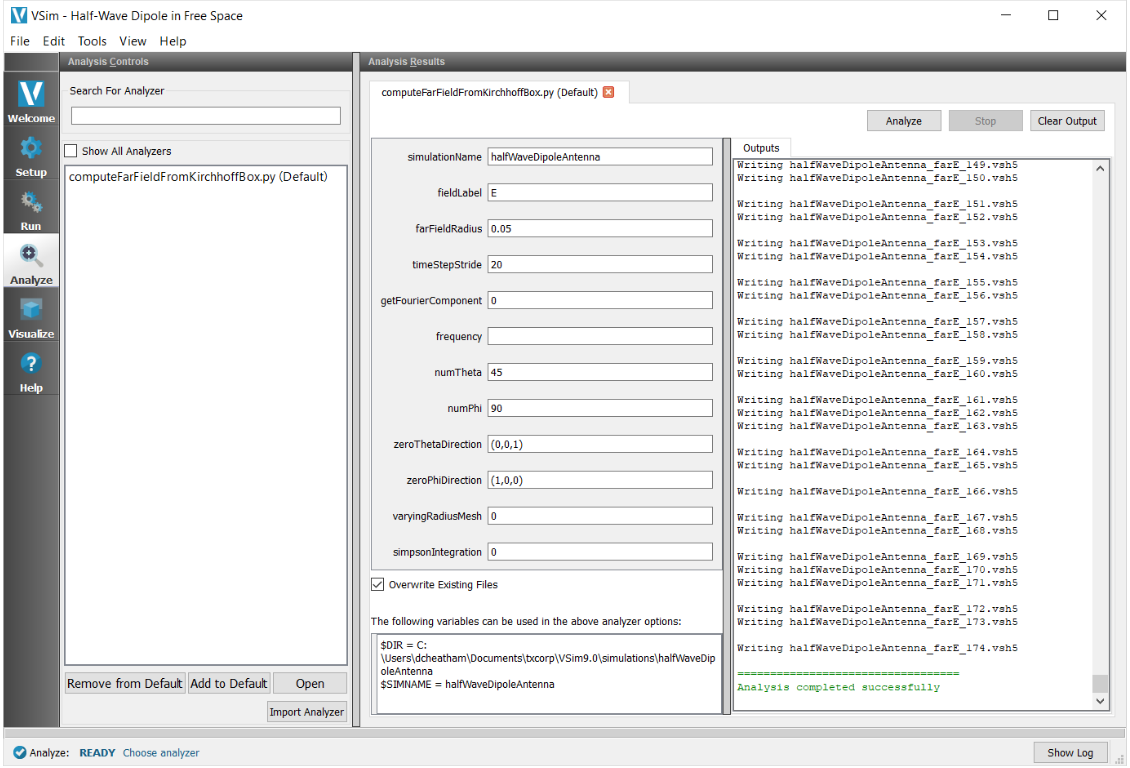
Run: SUCCESS Simulation engine finished successfully

Fig. 3.20: The Run Window at the end of execution.

Analyzing the Results

After performing the above actions, continue as follows:

- Proceed to the Analysis window by pressing the Analyze button in the left column of buttons.
- Select *computeFarFieldFromKirchhoffBox.py (default)*. Then click *Open*.
- For this example, edit the following input parameters (the rest should remain at their default setting):
  - simulationName - halfWaveDipoleAntenna (name of the input file)
  - fieldLabel - E (name of the electromagnetic field)
  - farFieldRadius - 0.05 (radius of the far sphere, i.e., distance to the far zone)
  - numTheta - 45 (number of points in the theta direction)
  - numPhi - 90 (number of points in the phi direction)
- Click the *Analyze* button in the upper right corner.



simulationName: halfWaveDipoleAntenna
fieldLabel: E
farFieldRadius: 0.05
timeStepStride: 20
getFourierComponent: 0
frequency:
numTheta: 45
numPhi: 90
zeroThetaDirection: (0,0,1)
zeroPhiDirection: (1,0,0)
varyingRadiusMesh: 0
simpsonIntegration: 0

☑ Overwrite Existing Files

The following variables can be used in the above analyzer options:

$DIR = C:\Users\dcheatham\Documents\txcorp\VSim9.0\simulations\halfWaveDipoleAntenna
$SIMNAME = halfWaveDipoleAntenna

```
Writing halfWaveDipoleAntenna_farE_149.vsh5
Writing halfWaveDipoleAntenna_farE_150.vsh5

Writing halfWaveDipoleAntenna_farE_151.vsh5
Writing halfWaveDipoleAntenna_farE_152.vsh5

Writing halfWaveDipoleAntenna_farE_153.vsh5
Writing halfWaveDipoleAntenna_farE_154.vsh5

Writing halfWaveDipoleAntenna_farE_155.vsh5
Writing halfWaveDipoleAntenna_farE_156.vsh5

Writing halfWaveDipoleAntenna_farE_157.vsh5
Writing halfWaveDipoleAntenna_farE_158.vsh5

Writing halfWaveDipoleAntenna_farE_159.vsh5
Writing halfWaveDipoleAntenna_farE_160.vsh5

Writing halfWaveDipoleAntenna_farE_161.vsh5
Writing halfWaveDipoleAntenna_farE_162.vsh5
Writing halfWaveDipoleAntenna_farE_163.vsh5

Writing halfWaveDipoleAntenna_farE_164.vsh5
Writing halfWaveDipoleAntenna_farE_165.vsh5

Writing halfWaveDipoleAntenna_farE_166.vsh5

Writing halfWaveDipoleAntenna_farE_167.vsh5
Writing halfWaveDipoleAntenna_farE_168.vsh5

Writing halfWaveDipoleAntenna_farE_169.vsh5
Writing halfWaveDipoleAntenna_farE_170.vsh5
Writing halfWaveDipoleAntenna_farE_171.vsh5

Writing halfWaveDipoleAntenna_farE_172.vsh5
Writing halfWaveDipoleAntenna_farE_173.vsh5

Writing halfWaveDipoleAntenna_farE_174.vsh5

==================================
Analysis completed successfully
```

Fig. 3.21: The Analysis window at the end of execution.

Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The far field radiation pattern can be found in the scalar data variables of the data overview tab:

- Expand *Scalar Data*
- Expand *farE*
- Select *farE_magnitude*
- Move the dump slider forward in time to see the evolution
- Click and drag to rotate the image

Further Experiments

The resolution of the far field pattern can be changed by editing the number of theta, phi, and sphere points in the far field history.

Try implementing a conducting plane to see how it affects the far field.

If the Simulation domain is made to small, the results will be distorted as the entire near field must be within the simulation domain in order to acheive a proper transformation to the far field.

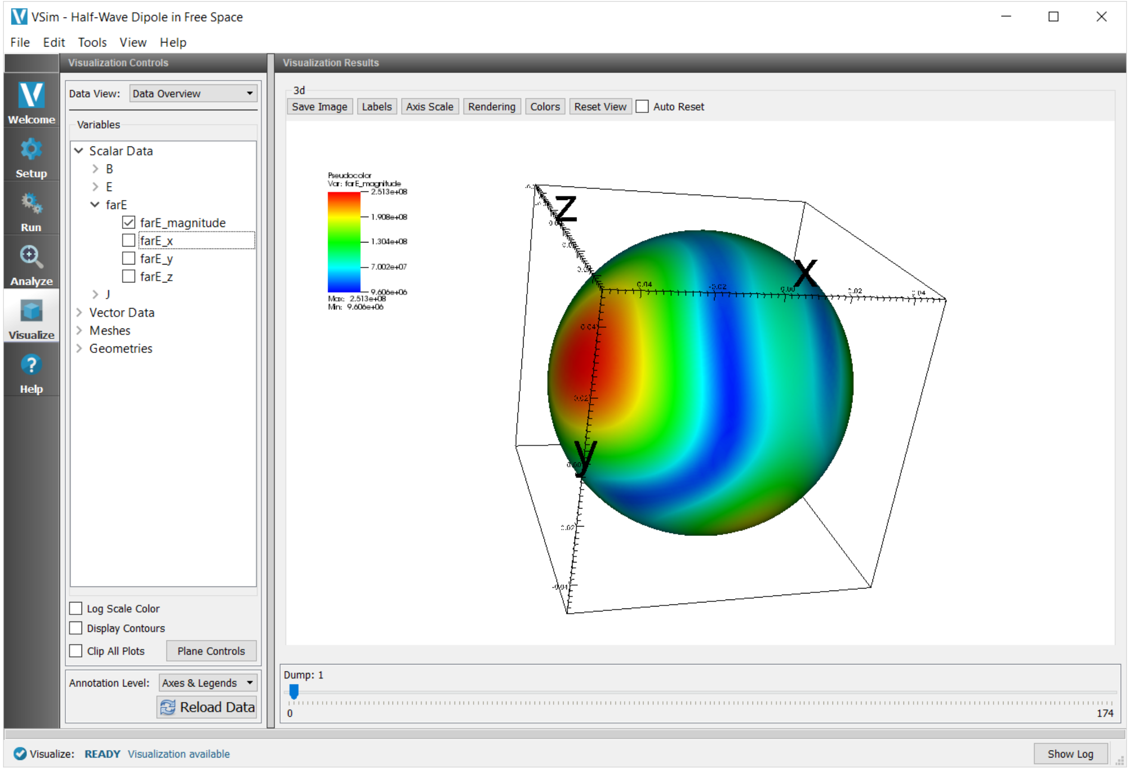


Fig. 3.22: The far field radiation pattern

3.1.7 Horn Antenna (hornAntenna.sdf)

Keywords:

`sectoral, horn antenna, far field, radiation`

Problem description

This example illustrates how to obtain the far field radiation pattern of a sectoral horn antenna. A horn antenna consists of a flaring metal waveguide shaped like a horn that directs radio waves into a beam. Horns are widely used as antennas at UHF and microwave frequencies. A sectoral horn is only flared along one axis, the other horn axis has constant width and is equivalent to the width of the waveguide. Sectoral horns produce a fan shaped beam, wider in the plane of the narrow sides.

This simulation can be run with a VSimEM license.

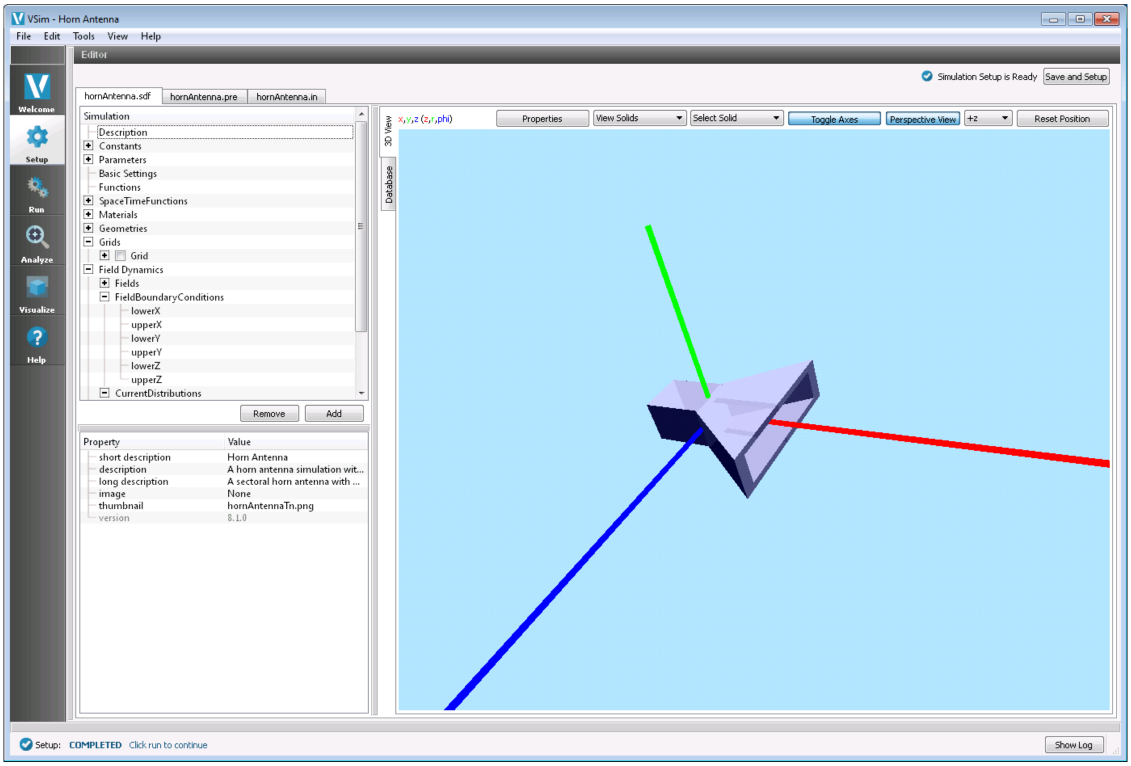
Opening the Simulation

The Horn Antenna example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Antennas* option.
- Select "Horn Antenna" and press the *Choose* button.

- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is shown Fig. 3.23. To obtain the same view as in the figure, in the tree click on Histories to open that part of the tree. then click on the box next to farFieldBox0 to make the far field box invisible. Right-clicking and dragging rotates the view.



Property	Value
short description	Horn Antenna
description	A horn antenna simulation wit...
long description	A sectoral horn antenna with ...
image	None
thumbnail	hornAntennaTn.png
version	8.1.0

Setup: COMPLETED Click run to continue

Fig. 3.23: Setup Window for the Horn Antenna example.

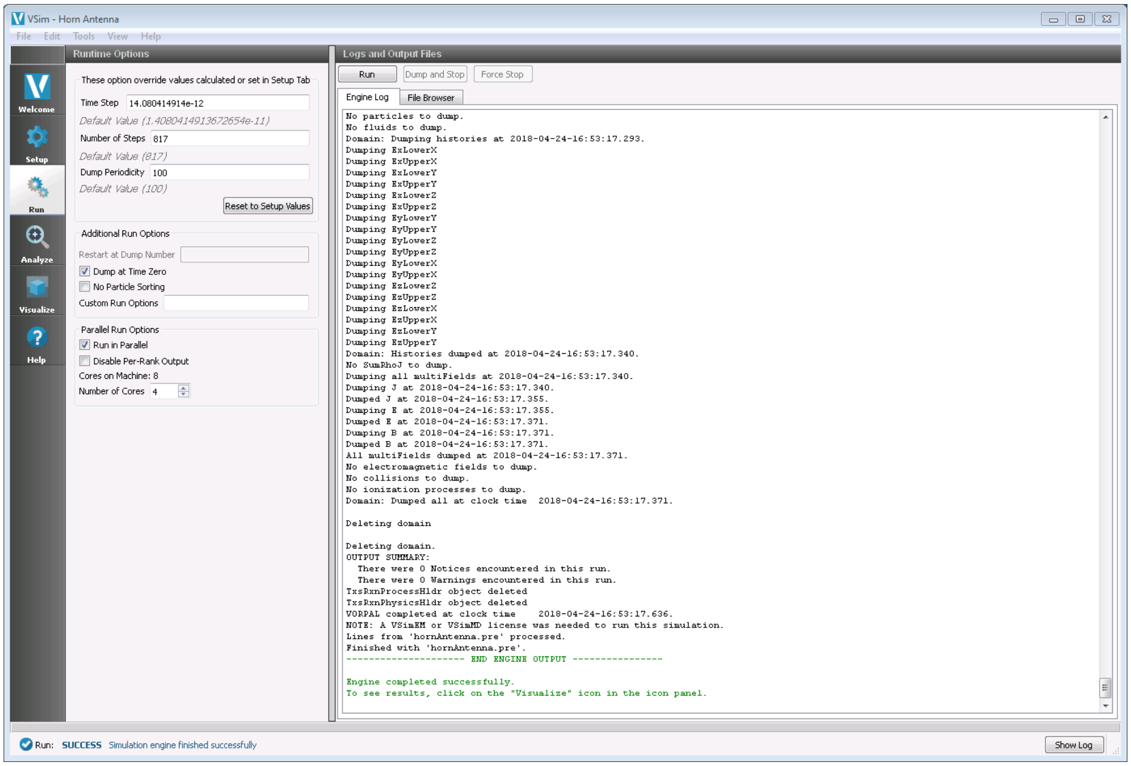
Simulation Properties

The antenna geometry in this example has been setup using CSG in the graphical setup interface. The dimensions of the antenna can be adjusted by tuning the sizes of the various wedges and cubes used in the antenna's construction. Under *Constants*, the wavelength may be modified, as well as the grid size and resolution. The polarization of the antenna may be altered by going into *CurrentDistributions* and changing the components of the driving current source.

Running the Simulation

Once finished with the problem setup, continue as follows:

- Proceed to the Run Window by pressing the **Run** button in the left column of buttons.
- Choose parallel computing options on the MPI tab.
- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane. You will see the output of the run in that pane. The run has completed when you see the output, "Engine completed successfully." This is shown in the window below.



These option override values calculated or set in Setup Tab

Time Step 14.080414914e-12
Default Value (1.4080414913672654e-11)
Number of Steps 817
Default Value (817)
Dump Periodicity 100
Default Value (100)

Additional Run Options

Restart at Dump Number
☑ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
□ Disable Per-Rank Output
Cores on Machine: 8
Number of Cores 4

```
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-04-24-16:53:17.293.
Dumping ExLowerX
Dumping ExUpperX
Dumping ExLowerY
Dumping ExUpperY
Dumping ExLowerZ
Dumping ExUpperZ
Dumping EyLowerY
Dumping EyUpperY
Dumping EyLowerZ
Dumping EyUpperZ
Dumping EyLowerX
Dumping EyUpperX
Dumping EzLowerZ
Dumping EzUpperZ
Dumping EzLowerX
Dumping EzUpperX
Dumping EzLowerY
Dumping EzUpperY
Domain: Histories dumped at 2018-04-24-16:53:17.340.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-24-16:53:17.340.
Dumping J at 2018-04-24-16:53:17.340.
Dumped J at 2018-04-24-16:53:17.355.
Dumping E at 2018-04-24-16:53:17.355.
Dumped E at 2018-04-24-16:53:17.371.
Dumping B at 2018-04-24-16:53:17.371.
Dumped B at 2018-04-24-16:53:17.371.
All multiFields dumped at 2018-04-24-16:53:17.371.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-24-16:53:17.371.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-24-16:53:17.636.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'hornAntenna.pre' processed.
Finished with 'hornAntenna.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

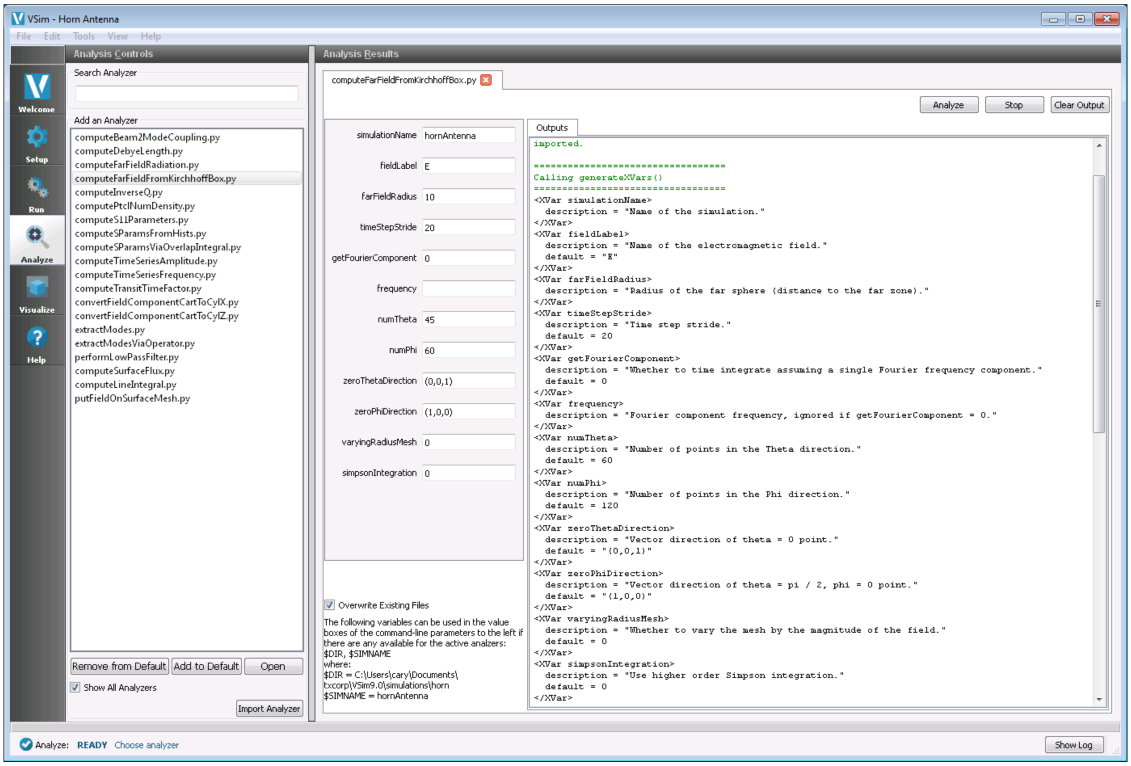
Fig. 3.24: The Run Window at the end of execution.

Analyzing the Results

After performing the above actions, continue as follows:

- Proceed to the Analysis window by pressing the Analyze button in the left column of buttons.
- Click ‘Show All Analyzers’
- In the resulting dialog, select computeFarFieldFromKirhhoffBox.py (Fig. 3.25) and press Open.
- Input values for the analyzer parameters. The analyzer may be run multiple times, allowing the user to experiment with different values.
  - simulationName - hornAntenna (name of the input file)
  - fieldLabel - E (name of the electric field)
  - farFieldRadius - 10.0 (distance to far field in m, 10.0 is a good value)
  - timeStepStride - 20 (number of timesteps between far field calculations; determines how many far fields are output; 20 steps should yield 4 far fields in this case)
  - getFourierComponent - 0 (whether to fourier analyze for a particular frequency)
  - frequency - the frequency to use in the fourier analysis.
  - numTheta - 18 (number of theta points in the far field, 18 for a quick calculation, 45 for finer resolution)
  - numPhi - 36 (number of phi points in the far field, 36 for a quick calculation, 90 for finer resolution)
  - zeroThetaDirection - (0,0,1) (determines orientation of far field coordinate system)
  - zeroPhiDirection - (1,0,0) (determines orientation of far field coordinate system

- varyingRadiusMesh - 1 (Set to 1 in order to make far field mesh adapt to magnitude of far field solution: the classic lobe view)
- simpsonIntegration - 0 (Set to 1 for more accurate integration)



computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
computeSurfaceFlux.py
computeLineIntegral.py
putFieldOnSurfaceMesh.py

☑ Show All Analyzers

computeFarFieldFromKirchhoffBox.py

simulationName: hornAntenna
fieldLabel: E
farFieldRadius: 10
timeStepStride: 20
getFourierComponent: 0
frequency:
numTheta: 45
numPhi: 60
zeroThetaDirection: (0,0,1)
zeroPhiDirection: (1,0,0)
varyingRadiusMesh: 0
simpsonIntegration: 0

☑ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analzers:
$DIR, $SIMNAME
where:
$DIR = C:\Users\cary\Documents\txcorp\VSim9.0\simulations\horn
$SIMNAME = hornAntenna

```
imported.

==================================
Calling generateXVars()
==================================
<XVar simulationName>
  description = "Name of the simulation."
</XVar>
<XVar fieldLabel>
  description = "Name of the electromagnetic field."
  default = "E"
</XVar>
<XVar farFieldRadius>
  description = "Radius of the far sphere (distance to the far zone)."
</XVar>
<XVar timeStepStride>
  description = "Time step stride."
  default = 20
</XVar>
<XVar getFourierComponent>
  description = "Whether to time integrate assuming a single Fourier frequency component."
  default = 0
</XVar>
<XVar frequency>
  description = "Fourier component frequency, ignored if getFourierComponent = 0."
</XVar>
<XVar numTheta>
  description = "Number of points in the Theta direction."
  default = 60
</XVar>
<XVar numPhi>
  description = "Number of points in the Phi direction."
  default = 120
</XVar>
<XVar zeroThetaDirection>
  description = "Vector direction of theta = 0 point."
  default = "(0,0,1)"
</XVar>
<XVar zeroPhiDirection>
  description = "Vector direction of theta = pi / 2, phi = 0 point."
  default = "(1,0,0)"
</XVar>
<XVar varyingRadiusMesh>
  description = "Whether to vary the mesh by the magnitude of the field."
  default = 0
</XVar>
<XVar simpsonIntegration>
  description = "Use higher order Simpson integration."
  default = 0
</XVar>
```

Analyze: READY Choose analyzer

Fig. 3.25: Add the computeFarFieldFromKirhhoffBox.py script to your simulation.

- Click Analyze
- The analysis is completed when you see “Analysis completed successfully” in the Outputs. Depending on the values of numTheta, numPhi, and timeStepStride, the script may need to run for several minutes or longer.

Visualizing the Results

Under *Scalar Data* plot E_z. To slice inside the field, select *Clip All Plots* in the lower left hand corner. Click on *Plane Controls* and change the cut-plane normal to lie along Y instead of Z. You can plot the horn antenna geometry by selecting poly (hornAntenna) under *Geometries*. Move the dump slider to view the electric field emanating from the horn. You can get a better look by adjusting the color scale. Select *Log Scale Color* in the lower left hand corner and then click the *Colors* button above the 3D view. Try adjusting the min and max until the signal is well resolved.

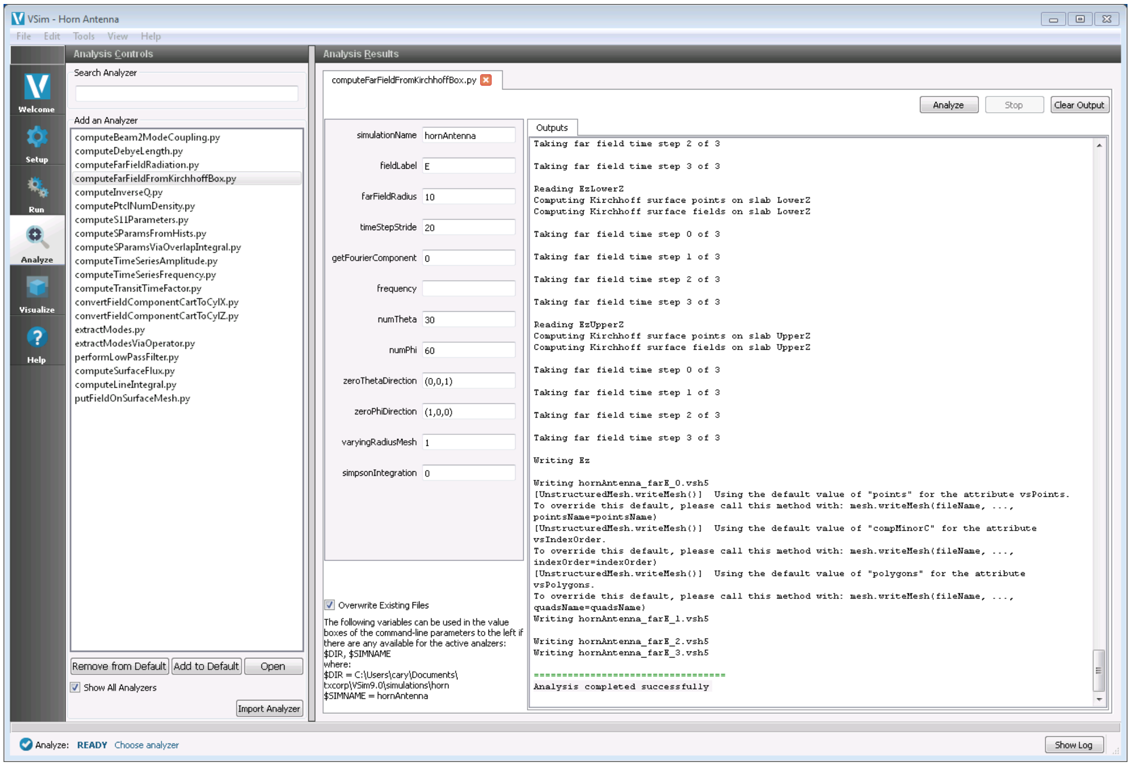
The far field radiation pattern can be found in the *Scalar Data* variables of the *Data Overview* tab. Open the farE tree element and check the *farE_magnitude* box. The far field mesh can also be plotted; it can be found under *Geometries*.

Further Experiments

The physical dimensions of the pyramidal horn can be modified in the GUI.

To turn the antenna into an E-plane sectoral horn, try changing the polarization to lie along the flared direction (z).

Try experimenting with different far field reolutions by changing the values of numTheta and numPhi during the *Analyze* step. You can also experiment with different far field distances by changing the value of farFieldRadius.



computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
computeSurfaceFlux.py
computeLineIntegral.py
putFieldOnSurfaceMesh.py

☑ Show All Analyzers

computeFarFieldFromKirchhoffBox.py

simulationName	hornAntenna
fieldLabel	E
farFieldRadius	10
timeStepStride	20
getFourierComponent	0
frequency	
numTheta	30
numPhi	60
zeroThetaDirection	(0,0,1)
zeroPhiDirection	(1,0,0)
varyingRadiusMesh	1
simpsonIntegration	0

☑ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analzers:
$DIR, $SIMNAME
where:
$DIR = C:\Users\cary\Documents\txcorp\VSim9.0\simulations\horn
$SIMNAME = hornAntenna

```
Taking far field time step 2 of 3

Taking far field time step 3 of 3

Reading EzLowerZ
Computing Kirchhoff surface points on slab LowerZ
Computing Kirchhoff surface fields on slab LowerZ

Taking far field time step 0 of 3

Taking far field time step 1 of 3

Taking far field time step 2 of 3

Taking far field time step 3 of 3

Reading EzUpperZ
Computing Kirchhoff surface points on slab UpperZ
Computing Kirchhoff surface fields on slab UpperZ

Taking far field time step 0 of 3

Taking far field time step 1 of 3

Taking far field time step 2 of 3

Taking far field time step 3 of 3

Writing Ez

Writing hornAntenna_farE_0.vsh5
[UnstructuredMesh.writeMesh()]  Using the default value of "points" for the attribute vsPoints.
To override this default, please call this method with: mesh.writeMesh(fileName, ...,
pointsName=pointsName)
[UnstructuredMesh.writeMesh()]  Using the default value of "compMinorC" for the attribute
vsIndexOrder.
To override this default, please call this method with: mesh.writeMesh(fileName, ...,
indexOrder=indexOrder)
[UnstructuredMesh.writeMesh()]  Using the default value of "polygons" for the attribute
vsPolygons.
To override this default, please call this method with: mesh.writeMesh(fileName, ...,
quadsName=quadsName)
Writing hornAntenna_farE_1.vsh5

Writing hornAntenna_farE_2.vsh5
Writing hornAntenna_farE_3.vsh5

===================================
Analysis completed successfully
```

Fig. 3.26: The **Analyze** panel after running computeFarFieldFromKirhhoffBox.py.

Try making the domain and the size of the Kirchhoff box larger or smaller (size of the Kirchhoff box is tied to the domain size by default). If the simulation domain is made too small, the results may appear distorted because the entire near field must be resolved within the simulation domain in order to achieve a proper transformation to the far field.

3.1.8 Patch Antenna with Far Fields (patchAntennaFarField.sdf)

Keywords:

`patchAntenna, far field, radiation`

Problem Description

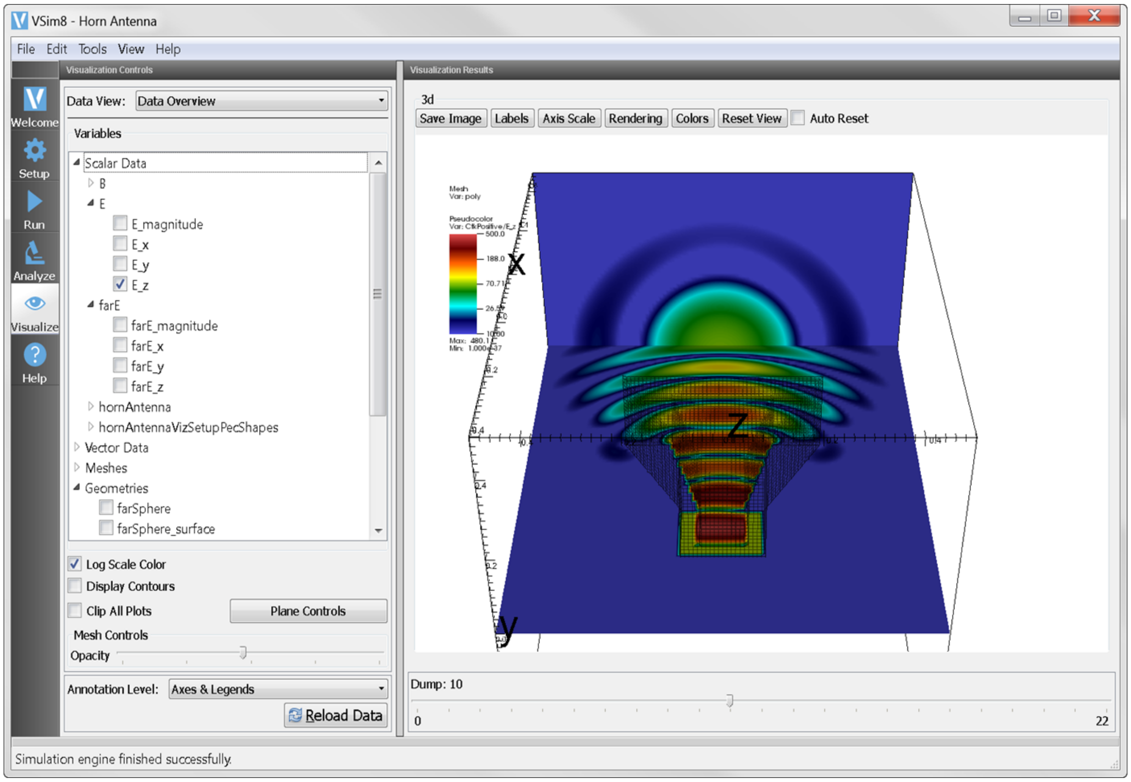
This problem takes the same patch antenna from the *Patch Antenna* example (currently text-based only, visual setup coming soon) and modifies it to calculate the far-field radiation pattern. It is fed with a 5.5GHz source on a microstrip feed line. The patch itself is mounted on a dielectric substrate made of alumina.

This simulation can be performed with a VSimEM license.

Opening the Simulation

The Patch Antenna example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example. . .* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.


VSim8 - Horn Antenna

Data View: Data Overview

Variables

- Scalar Data
  - B
  - E
    - ☐ E_magnitude
    - ☐ E_x
    - ☐ E_y
    - ☑ E_z
  - farE
    - ☐ farE_magnitude
    - ☐ farE_x
    - ☐ farE_y
    - ☐ farE_z
  - hornAntenna
  - hornAntennaVizSetupPecShapes
- Vector Data
- Meshes
- Geometries
  - ☐ farSphere
  - ☐ farSphere_surface

☑ Log Scale Color

☐ Display Contours

☐ Clip All Plots

Annotation Level: Axes & Legends

☐ Auto Reset

Dump: 10

Simulation engine finished successfully.

Fig. 3.27: The E_z field propagating out of the horn. The color scale has been log scaled and the min and max have been fixed to 10.0 and 500.0, respectively. The geometry's wireframe has been plotted and the opacity of the mesh reduced, making it slightly transparent. For convenience, a three-slice operator has been added through the VisIt GUI (hidden by default) in order to show as much information through one plot as possible. Various clips can be performed to show the fields on different coordinate planes, one at a time.

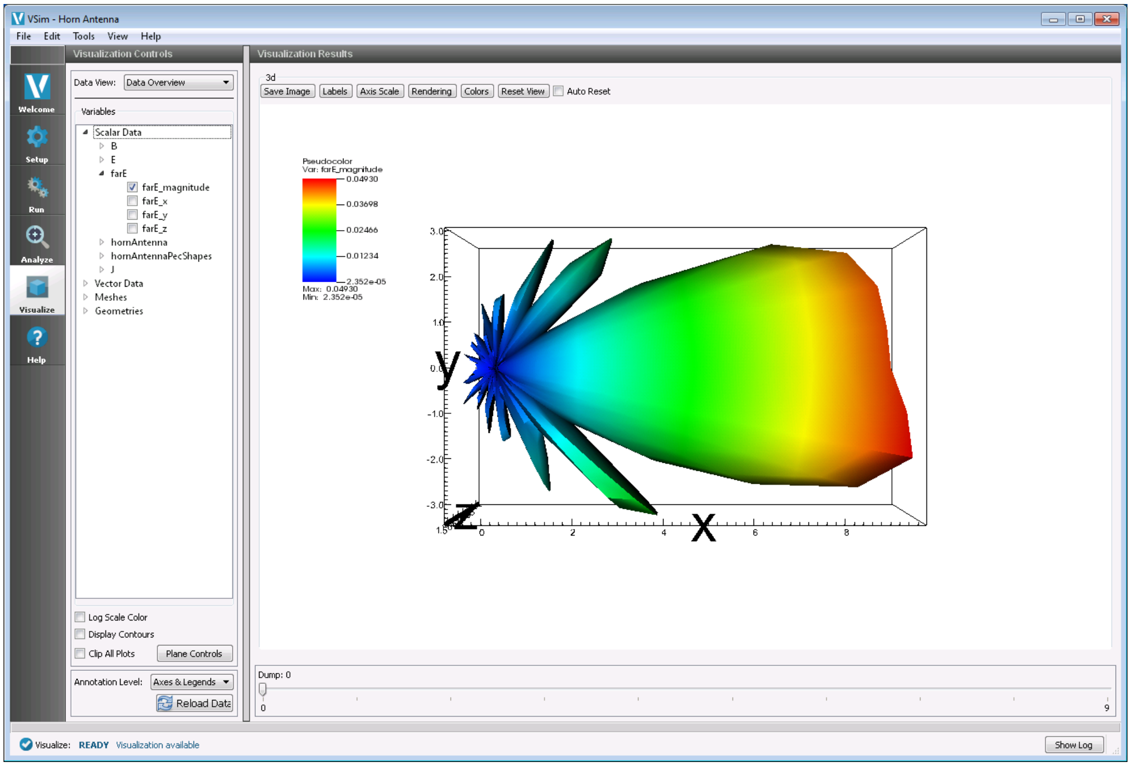


Fig. 3.28: The far field radiation pattern.

- Expand the *Antennas* option.
- Select “Patch Antenna with Far Fields” and press the *Choose* button.
- In the resulting dialog, create a new folder if desired, and press the *Save* button to create a copy of this example.

The **Setup** window is now shown with all the implemented physics and geometries. See Fig. 3.29.

Simulation Properties

This file allows the modification of antenna operating frequency, dimensions, orientation, simulation domain size.

Running the Simulation

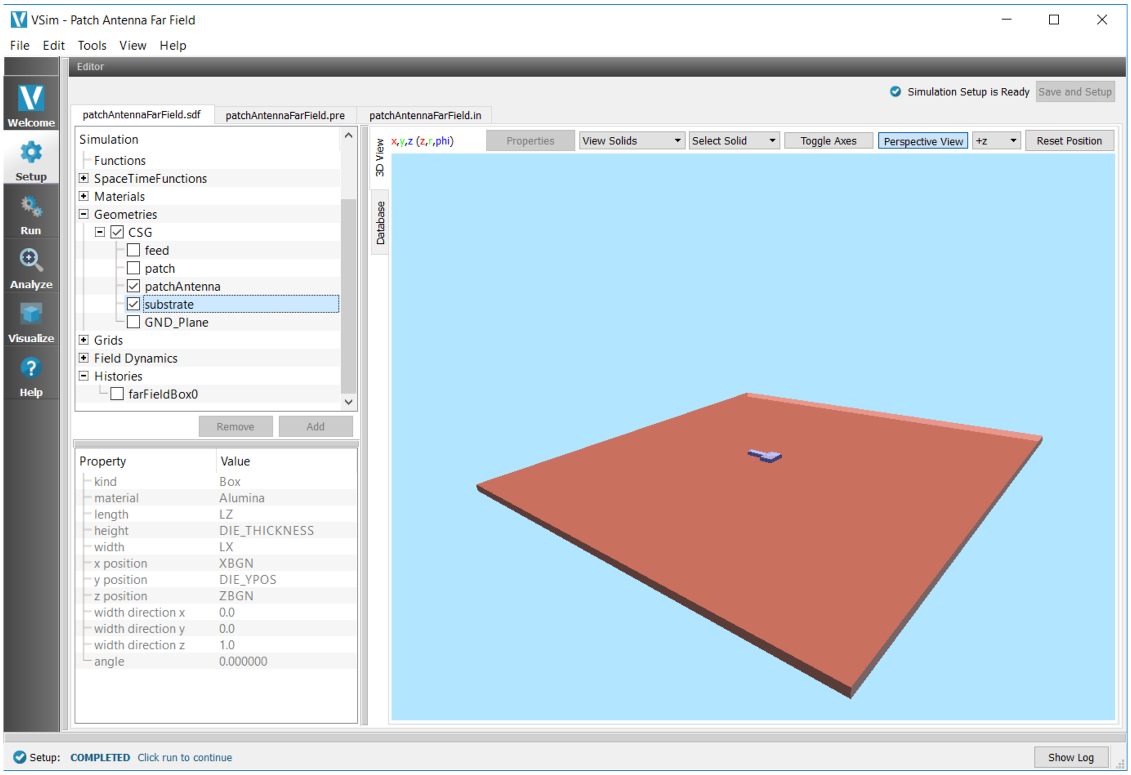
After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the **Run** button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane. You will see the output of the run in that pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in the window below.

Analyzing the Results

After performing the above actions, continue as follows:

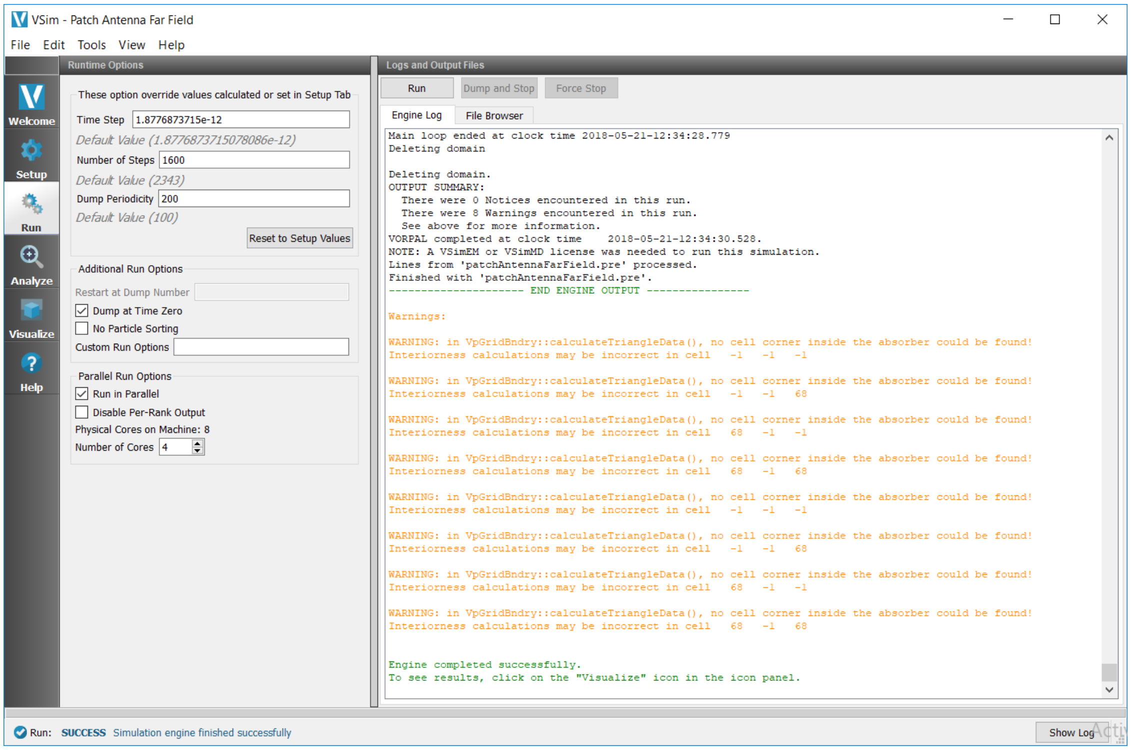
- Proceed to the Analysis window by pressing the Analyze button in the left column of buttons.



Property	Value
kind	Box
material	Alumina
length	LZ
height	DIE_THICKNESS
width	LX
x position	XBGN
y position	DIE_YPOS
z position	ZBGN
width direction x	0.0
width direction y	0.0
width direction z	1.0
angle	0.000000

Setup: COMPLETED Click run to continue

Fig. 3.29: Setup Window for the Patch Antenna example.



These option override values calculated or set in Setup Tab

Time Step 1.8776873715e-12

Default Value (1.8776873715078086e-12)

Number of Steps 1600

Default Value (2343)

Dump Periodicity 200

Default Value (100)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 8

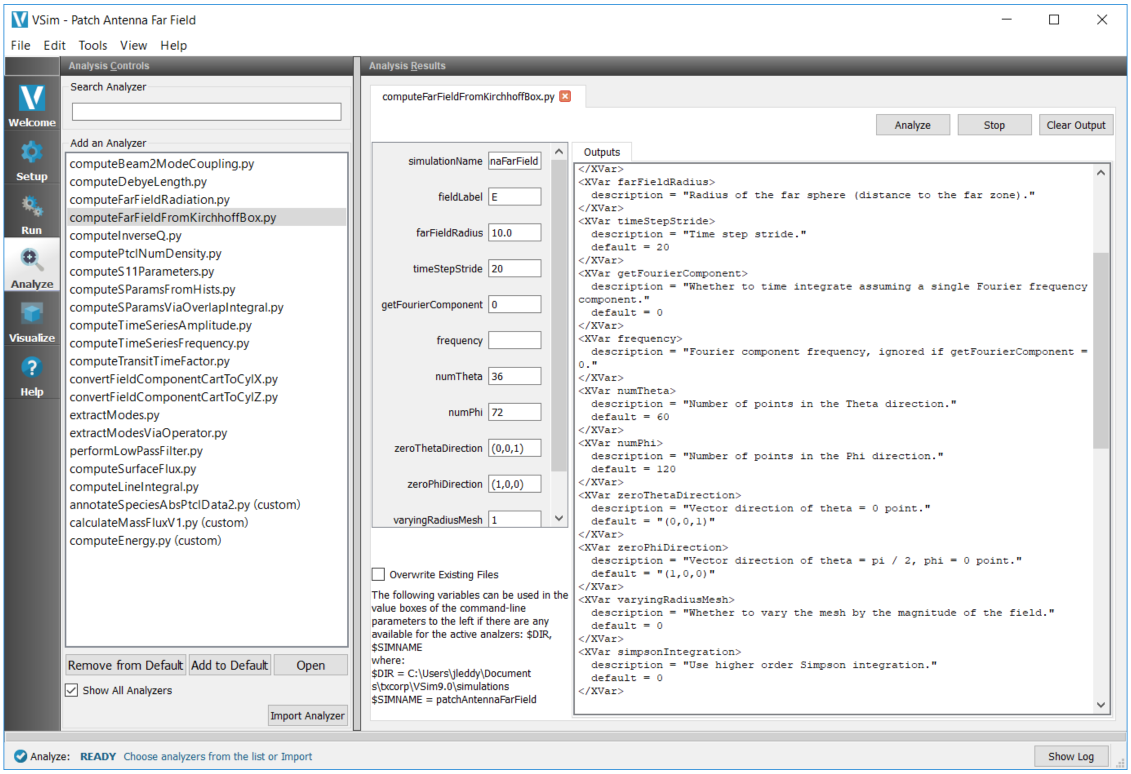
Number of Cores 4

```
Main loop ended at clock time 2018-05-21-12:34:28.779
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 8 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-05-21-12:34:30.528.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'patchAntennaFarField.pre' processed.
Finished with 'patchAntennaFarField.pre'.
--------------------- END ENGINE OUTPUT ----------------

Warnings:

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found!
Interiorness calculations may be incorrect in cell   -1   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found!
Interiorness calculations may be incorrect in cell   -1   -1   68

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found!
Interiorness calculations may be incorrect in cell   68   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found!
Interiorness calculations may be incorrect in cell   68   -1   68

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found!
Interiorness calculations may be incorrect in cell   -1   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found!
Interiorness calculations may be incorrect in cell   -1   -1   68

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found!
Interiorness calculations may be incorrect in cell   68   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found!
Interiorness calculations may be incorrect in cell   68   -1   68


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 3.30: The Run Window at the end of execution.

- Tick the box “Show All Analyzers” in the bottom left corner of the window.
- Select computeFarFieldFromKorhhoffBox.py from the list and select “Open” (Fig. 3.31)
- Input values for the analyzer parameters. The analyzer may be run multiple times, allowing the user to experiment with different values.
  - simulationName - patchAntennaFarField (name of the input file)
  - fieldLabel - E (name of the electric field)
  - farFieldRadius - 10.0 (distance to far field in m, 10.0 is a good value)
  - timeStepStride - 20 (number of timesteps between far field calculations; determines how many far fields are output; 20 steps should yield 4 far fields in this case)
  - getFourierComponent - 0, do not integrate assuming single fourier frequency
  - frequency - not used because getFourierComponent is false
  - numTheta - 36 (number of theta points in the far field, 36 for a quick calculation, 45 for finer resolution)
  - numPhi - 72 (number of phi points in the far field, 72 for a quick calculation, 90 for finer resolution)
  - zeroThetaDirection - (0,0,1) (determines orientation of far field coordinate system)
  - zeroPhiDirection - (1,0,0) (determines orientation of far field coordinate system
  - varyingRadiusMesh - 0 (Set to 1 in order to make far field mesh adapt to magnitude of far field solution: the classic lobe view)


VSim - Patch Antenna Far Field
File
Edit
Tools
View
Help
Analysis Controls
Search Analyzer
Add an Analyzer
computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
computeSurfaceFlux.py
computeLineIntegral.py
annotateSpeciesAbsPtclData2.py (custom)
calculateMassFluxV1.py (custom)
computeEnergy.py (custom)
Remove from Default
Add to Default
Open
Import Analyzer
Welcome
Setup
Run
Analyze
Visualize
Help
Analysis Results
computeFarFieldFromKirchhoffBox.py
Analyze
Stop
Clear Output
Outputs
Show Log

☑ Show All Analyzers

simulationName naFarField

fieldLabel E

farFieldRadius 10.0

timeStepStride 20

getFourierComponent 0

frequency

numTheta 36

numPhi 72

zeroThetaDirection (0,0,1)

zeroPhiDirection (1,0,0)

varyingRadiusMesh 1

□ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\jleddy\Documents\txcorp\VSim9.0\simulations
$SIMNAME = patchAntennaFarField

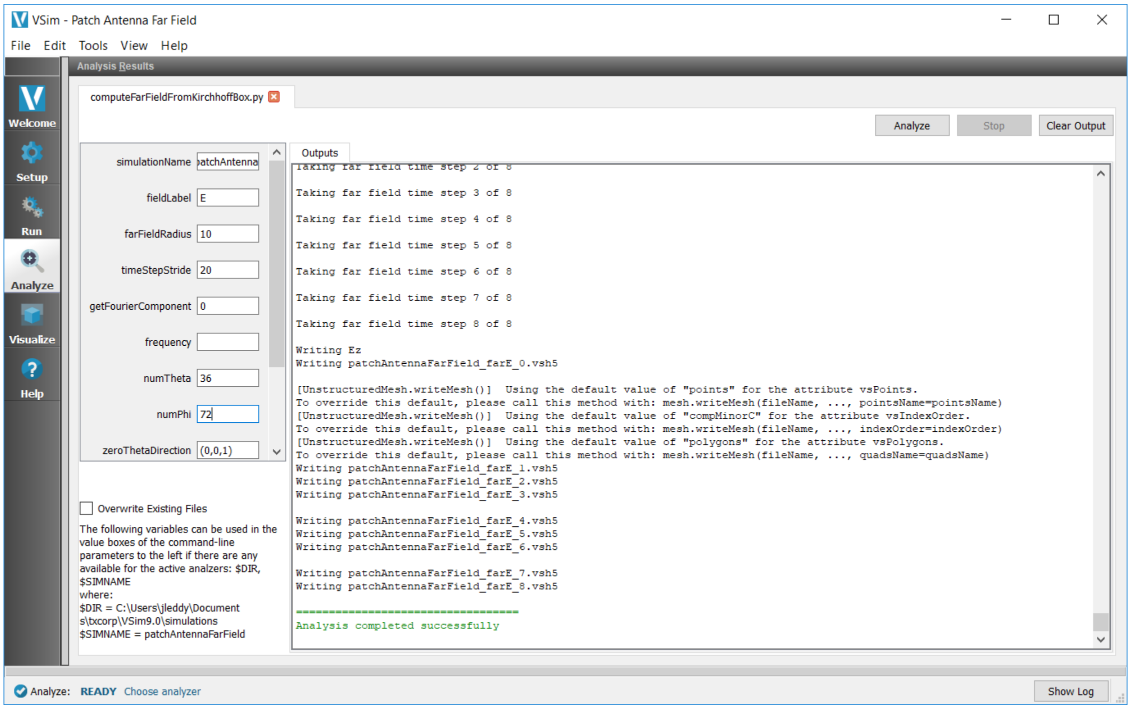
```
</XVar>
<XVar farFieldRadius>
  description = "Radius of the far sphere (distance to the far zone)."
</XVar>
<XVar timeStepStride>
  description = "Time step stride."
  default = 20
</XVar>
<XVar getFourierComponent>
  description = "Whether to time integrate assuming a single Fourier frequency
component."
  default = 0
</XVar>
<XVar frequency>
  description = "Fourier component frequency, ignored if getFourierComponent =
0."
</XVar>
<XVar numTheta>
  description = "Number of points in the Theta direction."
  default = 60
</XVar>
<XVar numPhi>
  description = "Number of points in the Phi direction."
  default = 120
</XVar>
<XVar zeroThetaDirection>
  description = "Vector direction of theta = 0 point."
  default = "(0,0,1)"
</XVar>
<XVar zeroPhiDirection>
  description = "Vector direction of theta = pi / 2, phi = 0 point."
  default = "(1,0,0)"
</XVar>
<XVar varyingRadiusMesh>
  description = "Whether to vary the mesh by the magnitude of the field."
  default = 0
</XVar>
<XVar simpsonIntegration>
  description = "Use higher order Simpson integration."
  default = 0
</XVar>
```

Analyze: READY Choose analyzers from the list or Import

Fig. 3.31: Add the computeFarFieldFromKirhhoffBox.py script to your simulation.

- Click “Analyze”

- Depending on the values of numTheta, numPhi, and timeStepStride, the script may need to run for several minutes or longer.



simulationName: patchAntenna
fieldLabel: E
farFieldRadius: 10
timeStepStride: 20
getFourierComponent: 0
frequency:
numTheta: 36
numPhi: 72
zeroThetaDirection: (0,0,1)

□ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\jleddy\Documents\txcorp\VSim9.0\simulations
$SIMNAME = patchAntennaFarField

```
Taking far field time step 2 of 8

Taking far field time step 3 of 8

Taking far field time step 4 of 8

Taking far field time step 5 of 8

Taking far field time step 6 of 8

Taking far field time step 7 of 8

Taking far field time step 8 of 8

Writing Ez
Writing patchAntennaFarField_farE_0.vsh5

[UnstructuredMesh.writeMesh()]  Using the default value of "points" for the attribute vsPoints.
To override this default, please call this method with: mesh.writeMesh(fileName, ..., pointsName=pointsName)
[UnstructuredMesh.writeMesh()]  Using the default value of "compMinorC" for the attribute vsIndexOrder.
To override this default, please call this method with: mesh.writeMesh(fileName, ..., indexOrder=indexOrder)
[UnstructuredMesh.writeMesh()]  Using the default value of "polygons" for the attribute vsPolygons.
To override this default, please call this method with: mesh.writeMesh(fileName, ..., quadsName=quadsName)
Writing patchAntennaFarField_farE_1.vsh5
Writing patchAntennaFarField_farE_2.vsh5
Writing patchAntennaFarField_farE_3.vsh5

Writing patchAntennaFarField_farE_4.vsh5
Writing patchAntennaFarField_farE_5.vsh5
Writing patchAntennaFarField_farE_6.vsh5

Writing patchAntennaFarField_farE_7.vsh5
Writing patchAntennaFarField_farE_8.vsh5

==================================
Analysis completed successfully
```

Analyze: READY Choose analyzer

Fig. 3.32: The **Analyze** panel after running computeFarFieldFromKirhhoffBox.py.

Visualizing the Results

The far field radiation pattern can be found in the *Scalar Data* variables of the *Data Overview* tab. Check the *farE* box. The far field mesh can also be plotted; it can be found under *Geometries*.

Further Experiments

The physical dimensions of the patch can be modified to turn it into any rectangular patch. This model can in fact be used to simulate any form of patch antenna, simply modify the geometry in the Setup Window by expaning the *Parameters* tree node and adjusting the values of PATCH_WIDTH, PATCH_LENGTH, PATCH_THICKNESS, FEED_WIDTH, FEED_LENGTH, and FEED_OFFSET. The thickness of the alumina die may also be adjusted by modifying DIE_THICKNESS.

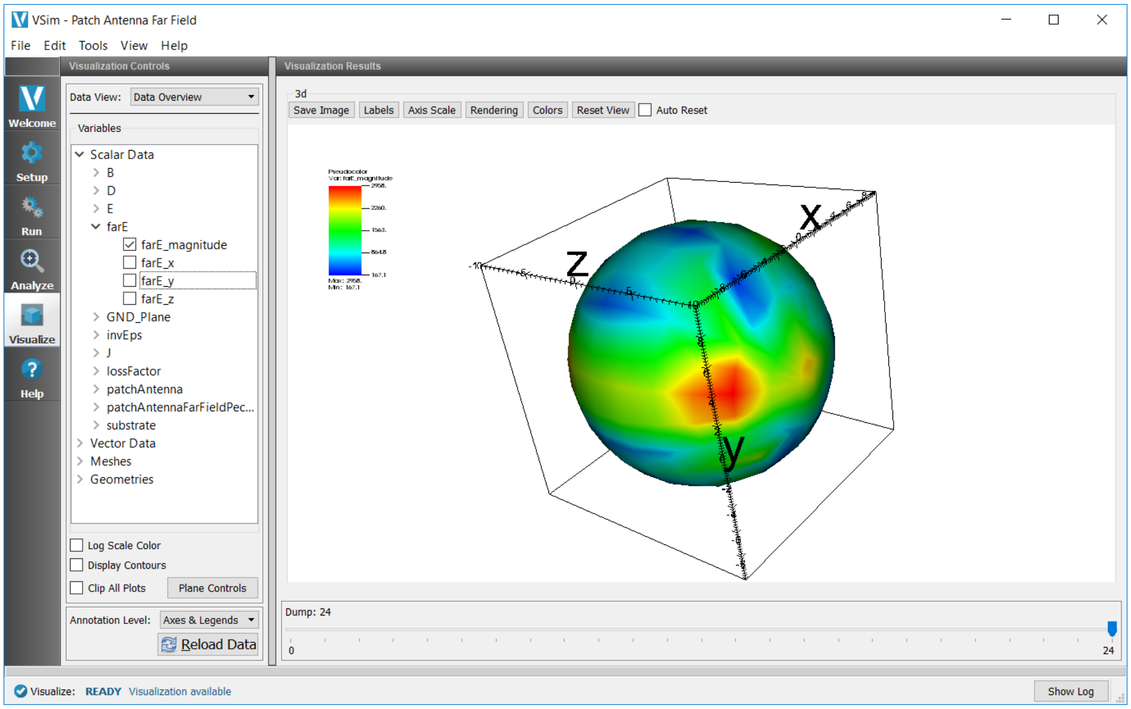
3.1.9 Antenna on Predator Drone (predatorDrone.sdf)

Keywords:

`predatorDrone, far field, radiation`

Problem Description

This problem illustrates how to obtain the far field radiation pattern of a current source antenna mounted on a Predator Drone.



Data View: Data Overview

☑ farE_magnitude

Annotation Level: Axes & Legends

Dump: 24

Visualize: READY Visualization available

Fig. 3.33: The Far Field Radiation Pattern

This simulation can be performed with a VSimEM license.

Opening the Simulation

The Predator Drone example is accessed from within VSimComposer by the following actions:

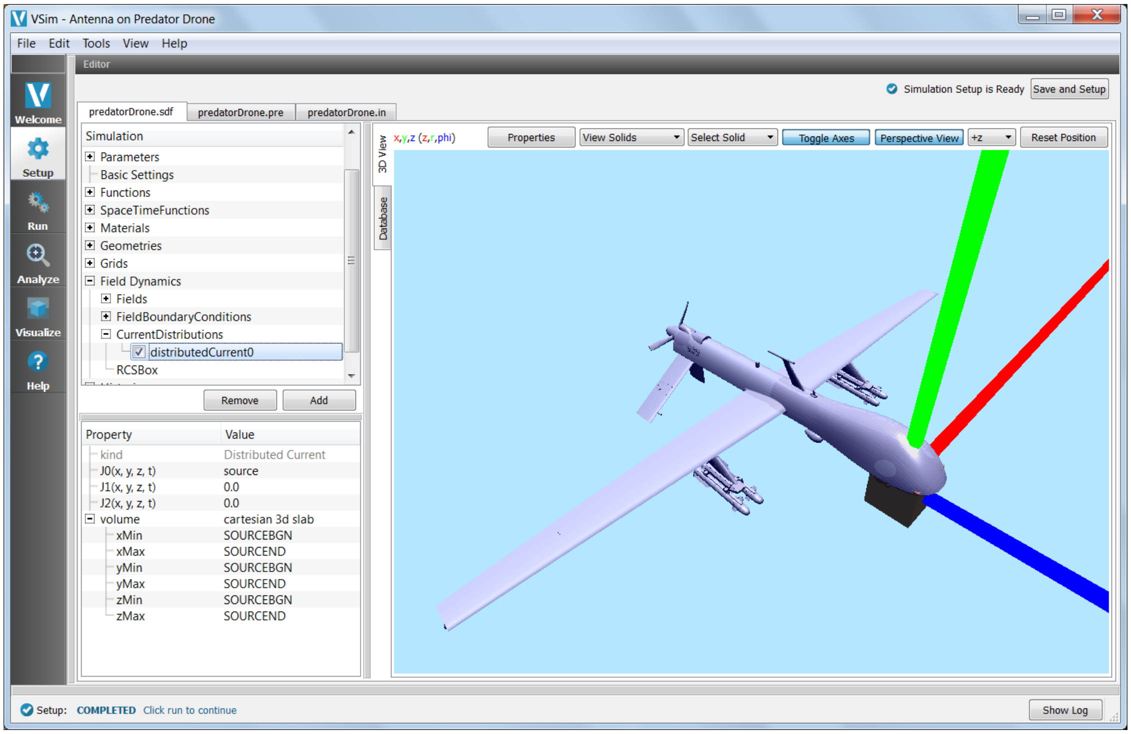
- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Antennas* option.
- Select “Antenna on Predator Drone” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is now shown with the CAD imported geometry and antenna current distribution accessible to the user. See Fig. 3.34.

One can click and unclick the grid, the farFieldBox0 in the histories, the current distribution, and so forth to see where those objects are. One can change locations through changing the values under Constants or, in some cases, the numbers directly in the objects.

Simulation Properties

This file allows the modification of antenna operating frequency, source amplitude, dimensions of the source and the Kirchhoff box by changing the associated variable values under the Constants.



Simulation
- Parameters
- Basic Settings
- Functions
- SpaceTimeFunctions
- Materials
- Geometries
- Grids
- Field Dynamics
  - Fields
  - FieldBoundaryConditions
  - CurrentDistributions
    - ☑ distributedCurrent0
  - RCSBox

Property	Value
kind	Distributed Current
J0(x, y, z, t)	source
J1(x, y, z, t)	0.0
J2(x, y, z, t)	0.0
volume	cartesian 3d slab
xMin	SOURCEBGN
xMax	SOURCEND
yMin	SOURCEBGN
yMax	SOURCEND
zMin	SOURCEBGN
zMax	SOURCEND

Setup: COMPLETED Click run to continue

Fig. 3.34: Setup Window for the Predator Drone example.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the *Run* button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 3.35.

Analyzing the Results

After the run, one must analyze the Kirchhoff box data to get the far fields. This is done as follows:

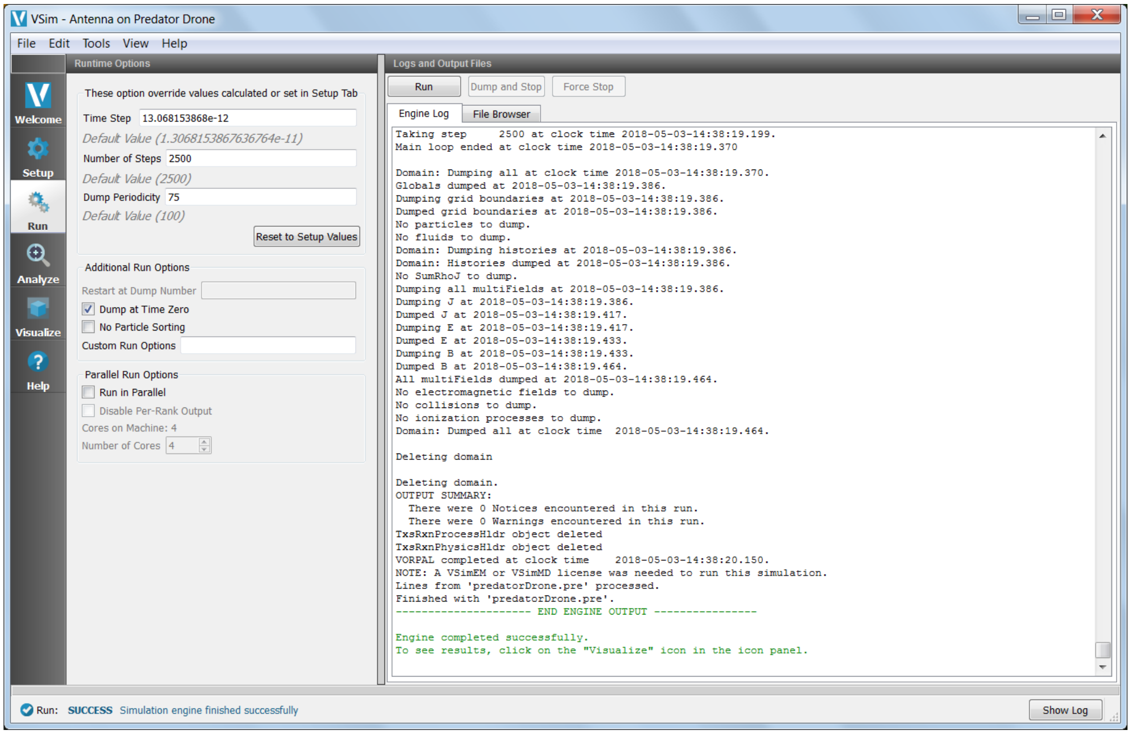
- Proceed to the Analysis window by pressing the *Analyze* button in the left column of buttons.
- Click on the *Show All Analyzers* button, then click *computeFarFieldFromKirchhoffBox.py*, then click on the *Open* button. as shown in Fig. 3.36.

If you want, you can grab the divding bar between the list of Analyzers in the *Analysis Controls* window and the *Analysis Results* window, and slide it left to cover up the *Analysis Controls* window, making more room for the *Analysis Results* window.

The default values should all be good, but one must set *farFieldRadius*. Since the box size is one, it is sufficient to choose 10.

The analysis script will compute the instantaneous far fields at the timeStepStride. You may also optionally compute the Fourier amplitude at a single frequency. As this is a single frequency excitation, at 1 GHz, the Fourier amplitude is indeed of interest, and so you should also set getFourierComponent to 1, and set frequency to 1.0e9.

If you want the script to run faster, lower numTheta to 8 and numPhi to 16.



These option override values calculated or set in Setup Tab

Time Step 13.068153868e-12

Default Value (1.3068153867636764e-11)

Number of Steps 2500

Default Value (2500)

Dump Periodicity 75

Default Value (100)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

□ Run in Parallel

□ Disable Per-Rank Output

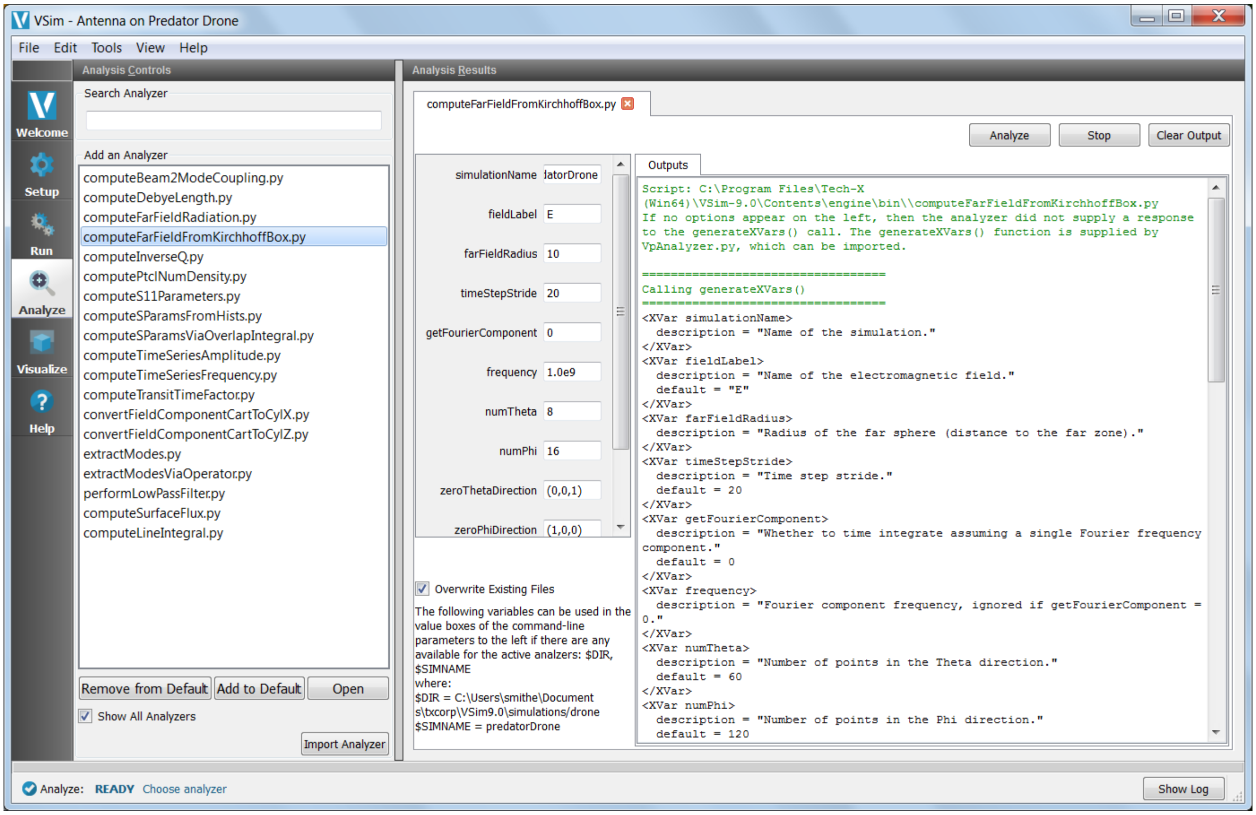
Cores on Machine: 4

Number of Cores 4

```
Taking step     2500 at clock time 2018-05-03-14:38:19.199.
Main loop ended at clock time 2018-05-03-14:38:19.370

Domain: Dumping all at clock time 2018-05-03-14:38:19.370.
Globals dumped at 2018-05-03-14:38:19.386.
Dumping grid boundaries at 2018-05-03-14:38:19.386.
Dumped grid boundaries at 2018-05-03-14:38:19.386.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-05-03-14:38:19.386.
Domain: Histories dumped at 2018-05-03-14:38:19.386.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-03-14:38:19.386.
Dumping J at 2018-05-03-14:38:19.386.
Dumped J at 2018-05-03-14:38:19.417.
Dumping E at 2018-05-03-14:38:19.417.
Dumped E at 2018-05-03-14:38:19.433.
Dumping B at 2018-05-03-14:38:19.433.
Dumped B at 2018-05-03-14:38:19.464.
All multiFields dumped at 2018-05-03-14:38:19.464.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-03-14:38:19.464.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-05-03-14:38:20.150.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'predatorDrone.pre' processed.
Finished with 'predatorDrone.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 3.35: The Run Window at the end of execution.



computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
computeSurfaceFlux.py
computeLineIntegral.py

☑ Show All Analyzers

simulationName	latorDrone
fieldLabel	E
farFieldRadius	10
timeStepStride	20
getFourierComponent	0
frequency	1.0e9
numTheta	8
numPhi	16
zeroThetaDirection	(0,0,1)
zeroPhiDirection	(1,0,0)

☑ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\smithe\Documents\txcorp\VSim9.0\simulations/drone
$SIMNAME = predatorDrone

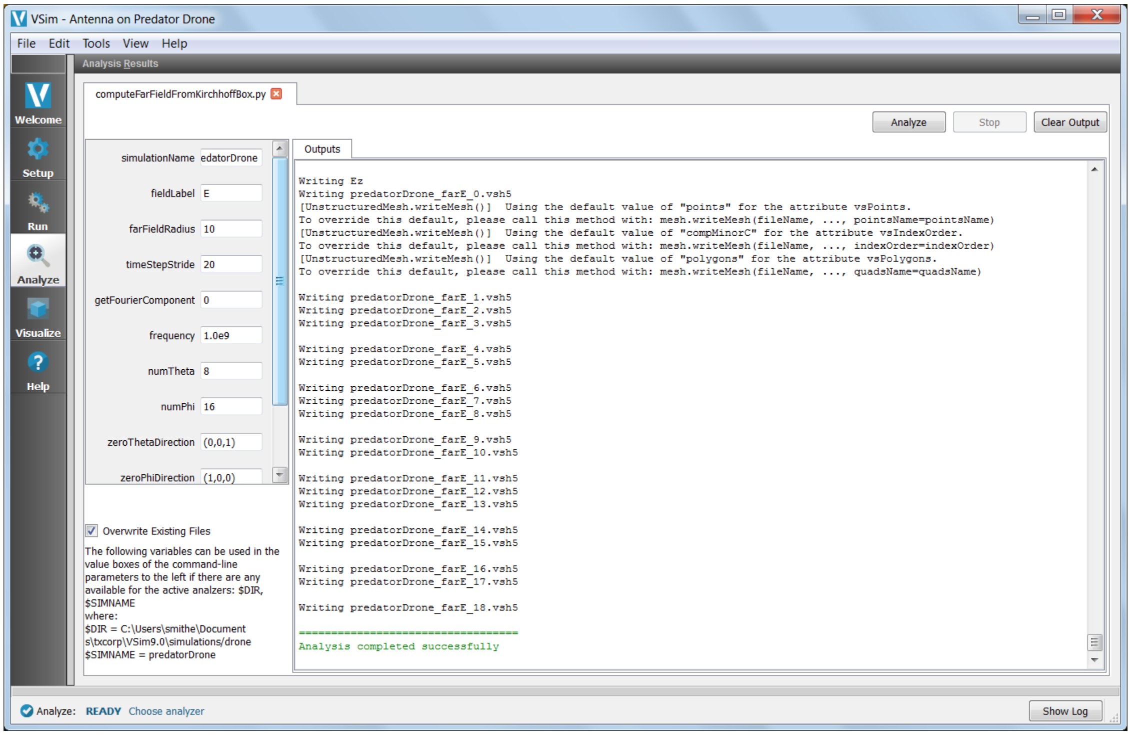
```
Script: C:\Program Files\Tech-X
(Win64)\VSim-9.0\Contents\engine\bin\\computeFarFieldFromKirchhoffBox.py
If no options appear on the left, then the analyzer did not supply a response
to the generateXVars() call. The generateXVars() function is supplied by
VpAnalyzer.py, which can be imported.

==================================
Calling generateXVars()
==================================
<XVar simulationName>
  description = "Name of the simulation."
</XVar>
<XVar fieldLabel>
  description = "Name of the electromagnetic field."
  default = "E"
</XVar>
<XVar farFieldRadius>
  description = "Radius of the far sphere (distance to the far zone)."
</XVar>
<XVar timeStepStride>
  description = "Time step stride."
  default = 20
</XVar>
<XVar getFourierComponent>
  description = "Whether to time integrate assuming a single Fourier frequency
component."
  default = 0
</XVar>
<XVar frequency>
  description = "Fourier component frequency, ignored if getFourierComponent =
0."
</XVar>
<XVar numTheta>
  description = "Number of points in the Theta direction."
  default = 60
</XVar>
<XVar numPhi>
  description = "Number of points in the Phi direction."
  default = 120
```

Analyze: **READY** Choose analyzer

Fig. 3.36: Choosing the Kirchhoff Box Analyzer.

- You will also usually want to choose *Overwrite Existing Files*.
- Hit the *Analyze* button in the top left of the window.

At completion, you will see Fig. 3.37. The far field data is written to vsh5 files in the simulation directory.


VSim - Antenna on Predator Drone
File Edit Tools View Help
Analysis Results
Welcome
Setup
Run
Analyze
Visualize
Help
computeFarFieldFromKirchhoffBox.py
Analyze
Stop
Clear Output
Outputs
Show Log

simulationName edatorDrone
fieldLabel E
farFieldRadius 10
timeStepStride 20
getFourierComponent 0
frequency 1.0e9
numTheta 8
numPhi 16
zeroThetaDirection (0,0,1)
zeroPhiDirection (1,0,0)

☑ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\smithe\Documents\txcorp\VSim9.0\simulations/drone
$SIMNAME = predatorDrone

```
Writing Ez
Writing predatorDrone_farE_0.vsh5
[UnstructuredMesh.writeMesh()]  Using the default value of "points" for the attribute vsPoints.
To override this default, please call this method with: mesh.writeMesh(fileName, ..., pointsName=pointsName)
[UnstructuredMesh.writeMesh()]  Using the default value of "compMinorC" for the attribute vsIndexOrder.
To override this default, please call this method with: mesh.writeMesh(fileName, ..., indexOrder=indexOrder)
[UnstructuredMesh.writeMesh()]  Using the default value of "polygons" for the attribute vsPolygons.
To override this default, please call this method with: mesh.writeMesh(fileName, ..., quadsName=quadsName)

Writing predatorDrone_farE_1.vsh5
Writing predatorDrone_farE_2.vsh5
Writing predatorDrone_farE_3.vsh5

Writing predatorDrone_farE_4.vsh5
Writing predatorDrone_farE_5.vsh5

Writing predatorDrone_farE_6.vsh5
Writing predatorDrone_farE_7.vsh5
Writing predatorDrone_farE_8.vsh5

Writing predatorDrone_farE_9.vsh5
Writing predatorDrone_farE_10.vsh5

Writing predatorDrone_farE_11.vsh5
Writing predatorDrone_farE_12.vsh5
Writing predatorDrone_farE_13.vsh5

Writing predatorDrone_farE_14.vsh5
Writing predatorDrone_farE_15.vsh5

Writing predatorDrone_farE_16.vsh5
Writing predatorDrone_farE_17.vsh5

Writing predatorDrone_farE_18.vsh5

==================================
Analysis completed successfully
```

Analyze: READY Choose analyzer

Fig. 3.37: The Analysis window at the end of execution.

Visualizing the Results

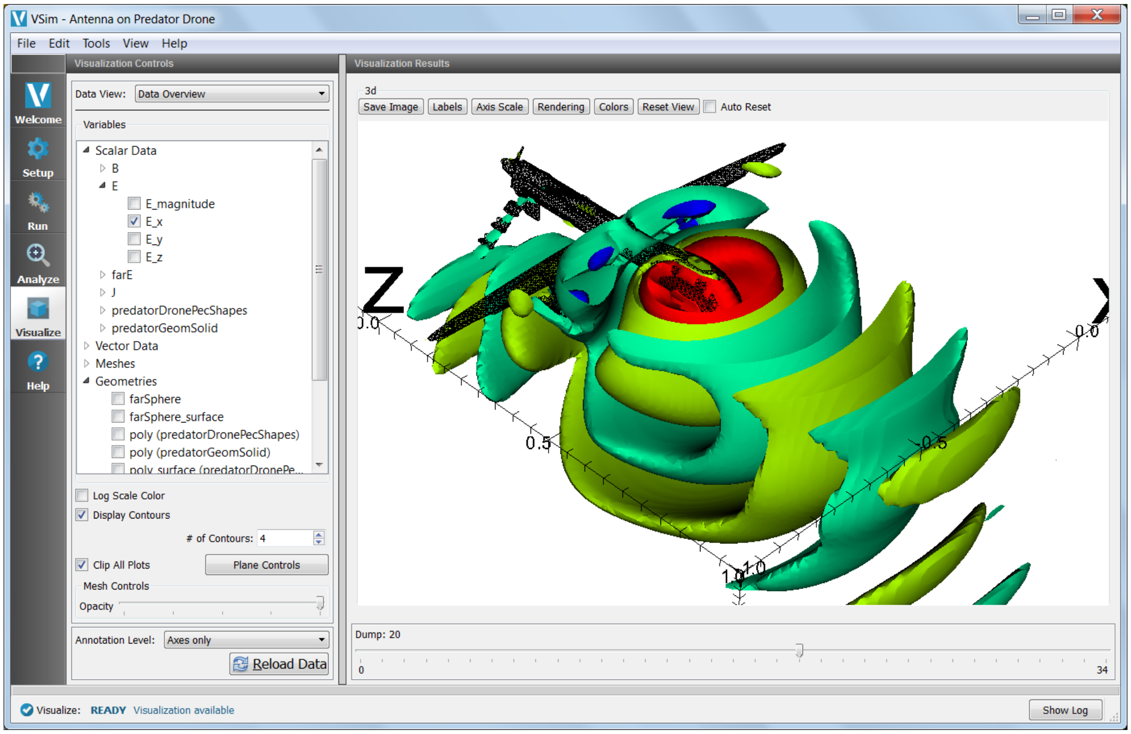
- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The radiation pattern in real space can be visualized by doing the following:

- Expand *Scalar Data*
- Expand *E*
- Select one of the scalar fields, such as E_x
- Move the Dump slider to see the far field pattern at different times.

It is useful to reset the color minimum and maximum using the Colors button and the number of Contours, to give a pleasing pattern, as shown in Fig. 3.38. An odd number of contours will result in a contour at zero field, which often leads to a less attractive plot with the zero contour filling up the space. Thus, in this case, an even number of contours is suggested.

The far field radiation pattern, which was computed in the section on Analyzing the data can also be displayed. Remove the previous image. Then check the *farE_magnitude* box under Scalar Data, and move the dump slider until the image stops changing. You will see a spherical surface, representing the Far Field Sphere, and colors indicating the strength of radiation at each angle that was processed. A notable peak in the radiation pattern is evident in the forward direction, as seen in Fig. 3.39.



Data View: Data Overview

□ E_magnitude
☑ E_x
□ E_y
□ E_z

□ farSphere
□ farSphere_surface
□ poly (predatorDronePecShapes)
□ poly (predatorGeomSolid)
□ poly_surface (predatorDronePe...

□ Log Scale Color
☑ Display Contours
# of Contours: 4
☑ Clip All Plots

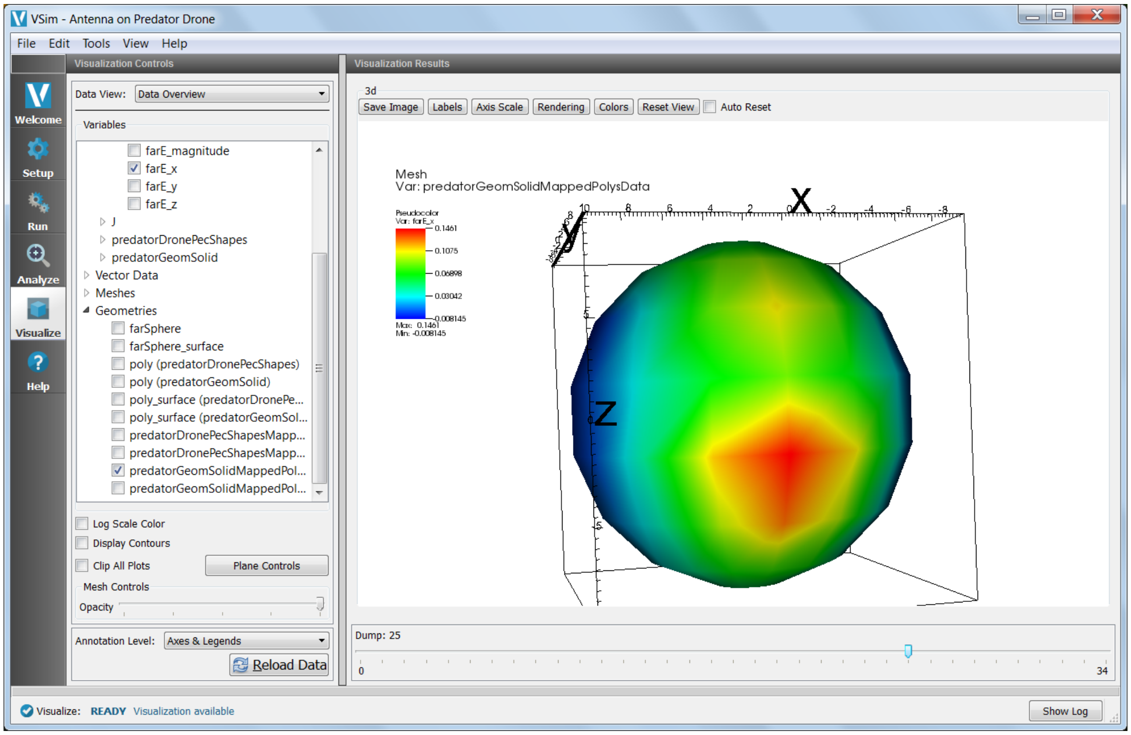
Annotation Level: Axes only

□ Auto Reset

Dump: 20

Visualize: READY Visualization available

Fig. 3.38: The radiation pattern in real space



Data View: Data Overview

□ farE_magnitude
☑ farE_x
□ farE_y
□ farE_z

□ farSphere
□ farSphere_surface
□ poly (predatorDronePecShapes)
□ poly (predatorGeomSolid)
□ poly_surface (predatorDronePe...
□ poly_surface (predatorGeomSol...
□ predatorDronePecShapesMapp...
□ predatorDronePecShapesMapp...
☑ predatorGeomSolidMappedPol...
□ predatorGeomSolidMappedPol...

□ Log Scale Color
□ Display Contours
□ Clip All Plots

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 25

Visualize: READY Visualization available

Fig. 3.39: The far field radiation pattern

Further Experiments

Upon close inspection you will note that the mesh size is slightly too large to fully resolve the thin wing structures of the tail section. You can experiment with smaller cell size to resolve these structures. Beware that more cells will increase the run time.

This example can be extended to meet any antenna placement problem with by addition of parameters to define the current distribution center. The vertical extent of the simulation box could be shrunk to reduce the simulation time, which would then allow greater resolution of the wavelength.

The main driver of simulation accuracy is the number of points per wavelength. Because of this lower frequencies will simulate in less time as they require fewer cells to achieve the same resolution in the wave.

3.2 Antennas (text-based setup)

3.2.1 Half-Wave Dipole in Free Space (halfWaveDipoleAntennaT.pre)

Keywords:

`halfWaveDipoleAntennaT, far field, radiation`

Problem Description

This problem illustrates how to obtain far field radiation patterns from VSim simulation data. The simulation itself consists of a half-wavelength long current source in free space.

This simulation can be performed with a VSimEM license.

Opening the Simulation

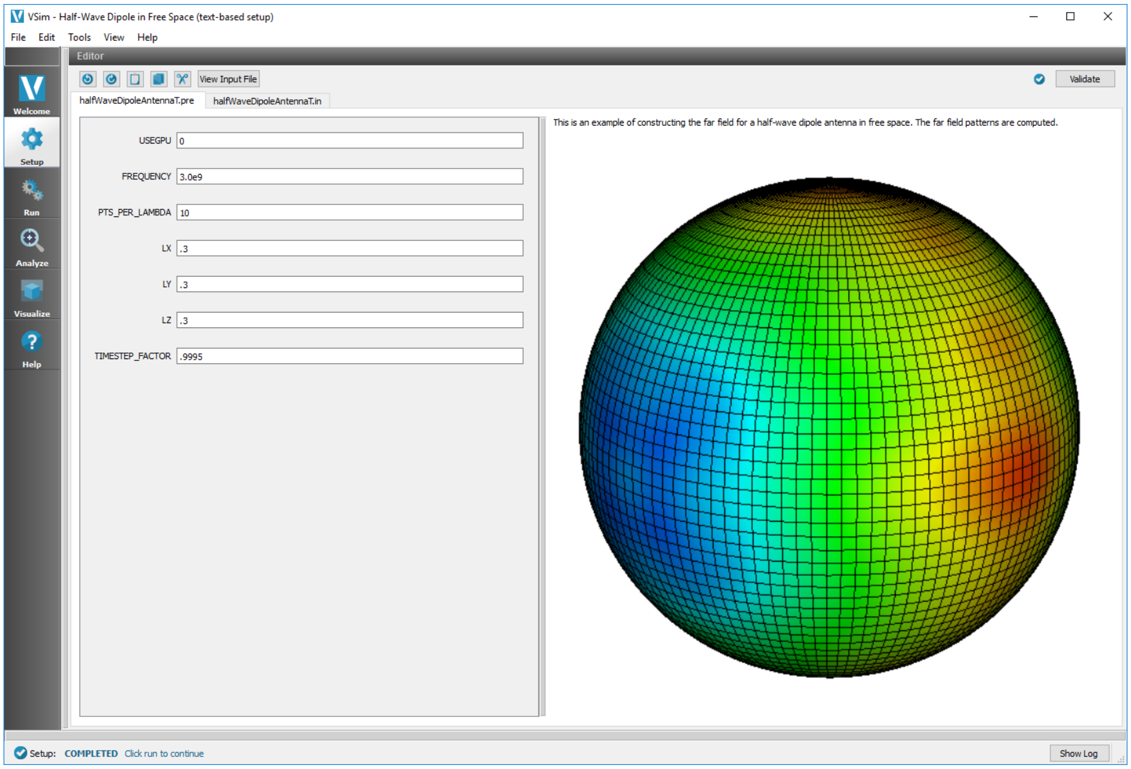
The Half Wave Dipole Antenna example is accessed from within VSimComposer by the following actions:

- Select the *New From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Antennas (text-based setup)* option.
- Select “Half Wave Dipole in Free Space (text-based setup)” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 3.40.

Input File Features

This file allows the modification of antenna operating frequency, as well as simulation domain size and far field resolution. It is also possible to perform this computation with a GPU.


VSim - Half-Wave Dipole in Free Space (text-based setup)
File
Edit
Tools
View
Help
Editor
View Input File
Validate
halfWaveDipoleAntennaT.pre
halfWaveDipoleAntennaT.in
Welcome
Setup
Run
Analyze
Visualize
Help
Show Log

USEGPU 0

FREQUENCY 3.0e9

PTS_PER_LAMBDA 10

LX .3

LY .3

LZ .3

TIMESTEP_FACTOR .9995

This is an example of constructing the far field for a half-wave dipole antenna in free space. The far field patterns are computed.

Setup: COMPLETED Click run to continue

Fig. 3.40: Setup Window for the Half Wave Dipole Antenna example.

Running the Simulation

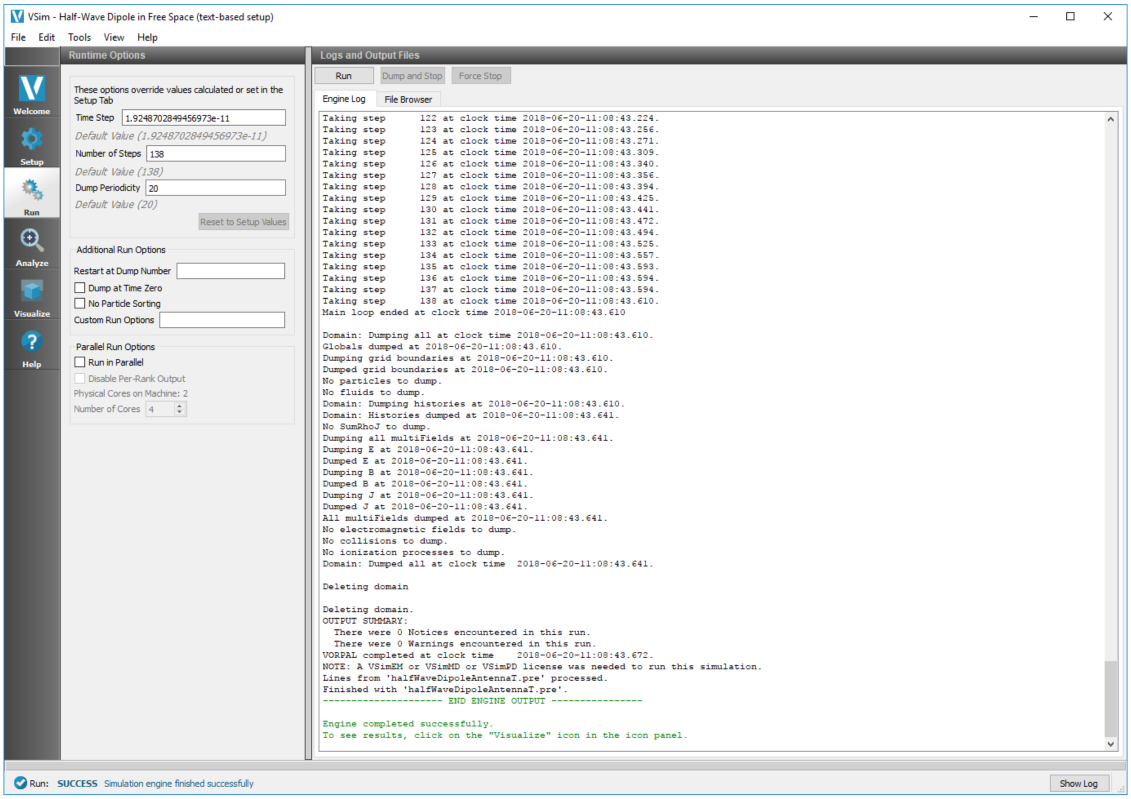
After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner. of the window. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in the window below.

Analyzing the Results

After performing the above actions, continue as follows:

- Proceed to the Analysis window by pressing the Analyze button in the left column of buttons.
- Click ‘Show All Analyzers’
- In the resulting dialog, select computeFarFieldFromKirhhoffBox.py and press Open.
- Input values for the analyzer parameters. The analyzer may be run multiple times, allowing the user to experiment with different values.
  - simulationName - halfWaveDipoleAntennaT (name of the input file)
  - fieldLabel - E (name of the electric field)
  - farFieldRadius - 10.0 (distance to far field in m, 10.0 is a good value)
  - timeStepStride - 3 (number of timesteps between far field calculations; determines how many far fields are output; 3 steps should yield 6 far fields in this case)



These options override values calculated or set in the Setup Tab

Time Step 1.9248702849456973e-11

Default Value (1.9248702849456973e-11)

Number of Steps 138

Default Value (138)

Dump Periodicity 20

Default Value (20)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

□ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 2

Number of Cores 4

```
Taking step      122 at clock time 2018-06-20-11:08:43.224.
Taking step      123 at clock time 2018-06-20-11:08:43.256.
Taking step      124 at clock time 2018-06-20-11:08:43.271.
Taking step      125 at clock time 2018-06-20-11:08:43.309.
Taking step      126 at clock time 2018-06-20-11:08:43.340.
Taking step      127 at clock time 2018-06-20-11:08:43.356.
Taking step      128 at clock time 2018-06-20-11:08:43.394.
Taking step      129 at clock time 2018-06-20-11:08:43.425.
Taking step      130 at clock time 2018-06-20-11:08:43.441.
Taking step      131 at clock time 2018-06-20-11:08:43.472.
Taking step      132 at clock time 2018-06-20-11:08:43.494.
Taking step      133 at clock time 2018-06-20-11:08:43.525.
Taking step      134 at clock time 2018-06-20-11:08:43.557.
Taking step      135 at clock time 2018-06-20-11:08:43.593.
Taking step      136 at clock time 2018-06-20-11:08:43.594.
Taking step      137 at clock time 2018-06-20-11:08:43.594.
Taking step      138 at clock time 2018-06-20-11:08:43.610.
Main loop ended at clock time 2018-06-20-11:08:43.610

Domain: Dumping all at clock time 2018-06-20-11:08:43.610.
Globals dumped at 2018-06-20-11:08:43.610.
Dumping grid boundaries at 2018-06-20-11:08:43.610.
Dumped grid boundaries at 2018-06-20-11:08:43.610.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-06-20-11:08:43.610.
Domain: Histories dumped at 2018-06-20-11:08:43.641.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-20-11:08:43.641.
Dumping E at 2018-06-20-11:08:43.641.
Dumped E at 2018-06-20-11:08:43.641.
Dumping B at 2018-06-20-11:08:43.641.
Dumped B at 2018-06-20-11:08:43.641.
Dumping J at 2018-06-20-11:08:43.641.
Dumped J at 2018-06-20-11:08:43.641.
All multiFields dumped at 2018-06-20-11:08:43.641.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-20-11:08:43.641.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-20-11:08:43.672.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.
Lines from 'halfWaveDipoleAntennaT.pre' processed.
Finished with 'halfWaveDipoleAntennaT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 3.41: The Run Window at the end of execution.

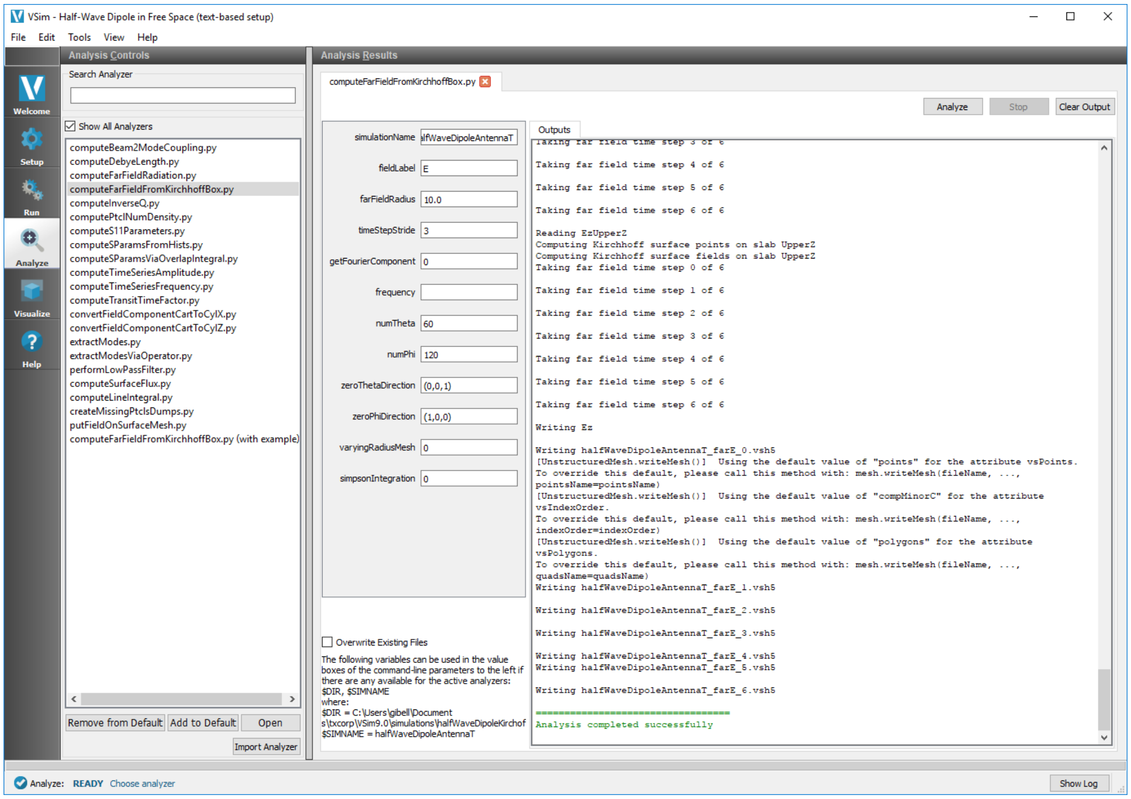
- getFourierComponent - 0 (whether to fourier analyze for a particular frequency)
- frequency - the frequency to use in the fourier analysis (not needed here).
- numTheta - 60 (number of theta points in the far field, 18 for a quick calculation, 45 for finer resolution)
- numPhi - 120 (number of phi points in the far field, 36 for a quick calculation, 90 for finer resolution)
- zeroThetaDirection - (0,0,1) (determines orientation of far field coordinate system)
- zeroPhiDirection - (1,0,0) (determines orientation of far field coordinate system
- varyingRadiusMesh - 0 (Set to 1 in order to make far field mesh adapt to magnitude of far field solution: the classic lobe view)
- simpsonIntegration - 0 (Set to 1 for more accurate integration)

Visualizing the Results

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The far field radiation pattern can be viewed as follows:

- Expand *Scalar Data*
- Expand *farE*
- Select *farE_magnitude*
- Expand *Geometries*


VSim - Half-Wave Dipole in Free Space (text-based setup)
File
Edit
Tools
View
Help
Analysis Controls
Welcome
Setup
Run
Analyze
Visualize
Help
Search Analyzer
☑ Show All Analyzers
computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
computeSurfaceFlux.py
computeLineIntegral.py
createMissingPtclsDumps.py
putFieldOnSurfaceMesh.py
computeFarFieldFromKirchhoffBox.py (with example)
Remove from Default
Add to Default
Open
Import Analyzer
Analysis Results
computeFarFieldFromKirchhoffBox.py
Analyze
Stop
Clear Output
Outputs
Analyze: READY Choose analyzer
Show Log

simulationName: alfWaveDipoleAntennaT
fieldLabel: E
farFieldRadius: 10.0
timeStepStride: 3
getFourierComponent: 0
frequency:
numTheta: 60
numPhi: 120
zeroThetaDirection: (0,0,1)
zeroPhiDirection: (1,0,0)
varyingRadiusMesh: 0
simpsonIntegration: 0

□ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analyzers:
$DIR, $SIMNAME
where:
$DIR = C:\Users\gibell\Documents\txcorp\VSim9.0\simulations\halfWaveDipoleKirchof
$SIMNAME = halfWaveDipoleAntennaT

```
Taking far field time step 3 of 6

Taking far field time step 4 of 6

Taking far field time step 5 of 6

Taking far field time step 6 of 6

Reading EzUpperZ
Computing Kirchhoff surface points on slab UpperZ
Computing Kirchhoff surface fields on slab UpperZ
Taking far field time step 0 of 6

Taking far field time step 1 of 6

Taking far field time step 2 of 6

Taking far field time step 3 of 6

Taking far field time step 4 of 6

Taking far field time step 5 of 6

Taking far field time step 6 of 6

Writing Ez

Writing halfWaveDipoleAntennaT_farE_0.vsh5
[UnstructuredMesh.writeMesh()]  Using the default value of "points" for the attribute vsPoints.
To override this default, please call this method with: mesh.writeMesh(fileName, ...,
pointsName=pointsName)
[UnstructuredMesh.writeMesh()]  Using the default value of "compMinorC" for the attribute
vsIndexOrder.
To override this default, please call this method with: mesh.writeMesh(fileName, ...,
indexOrder=indexOrder)
[UnstructuredMesh.writeMesh()]  Using the default value of "polygons" for the attribute
vsPolygons.
To override this default, please call this method with: mesh.writeMesh(fileName, ...,
quadsName=quadsName)
Writing halfWaveDipoleAntennaT_farE_1.vsh5

Writing halfWaveDipoleAntennaT_farE_2.vsh5

Writing halfWaveDipoleAntennaT_farE_3.vsh5

Writing halfWaveDipoleAntennaT_farE_4.vsh5
Writing halfWaveDipoleAntennaT_farE_5.vsh5

Writing halfWaveDipoleAntennaT_farE_6.vsh5

==================================
Analysis completed successfully
```

Fig. 3.42: The Analysis window at the end of execution.

- Select *farSphere*
- Click and drag with your mouse to rotate the image

Further Experiments

The resolution of the far field pattern can be changed by editing the numTheta and numPhi values in the Analyzer Window.

To improve computational speed the size of the simulation domain can be optimized by adjusting LX/LY/LZ and PTS_PER_LAMBDA. Note that for far field calculations the simulation domain must be square.

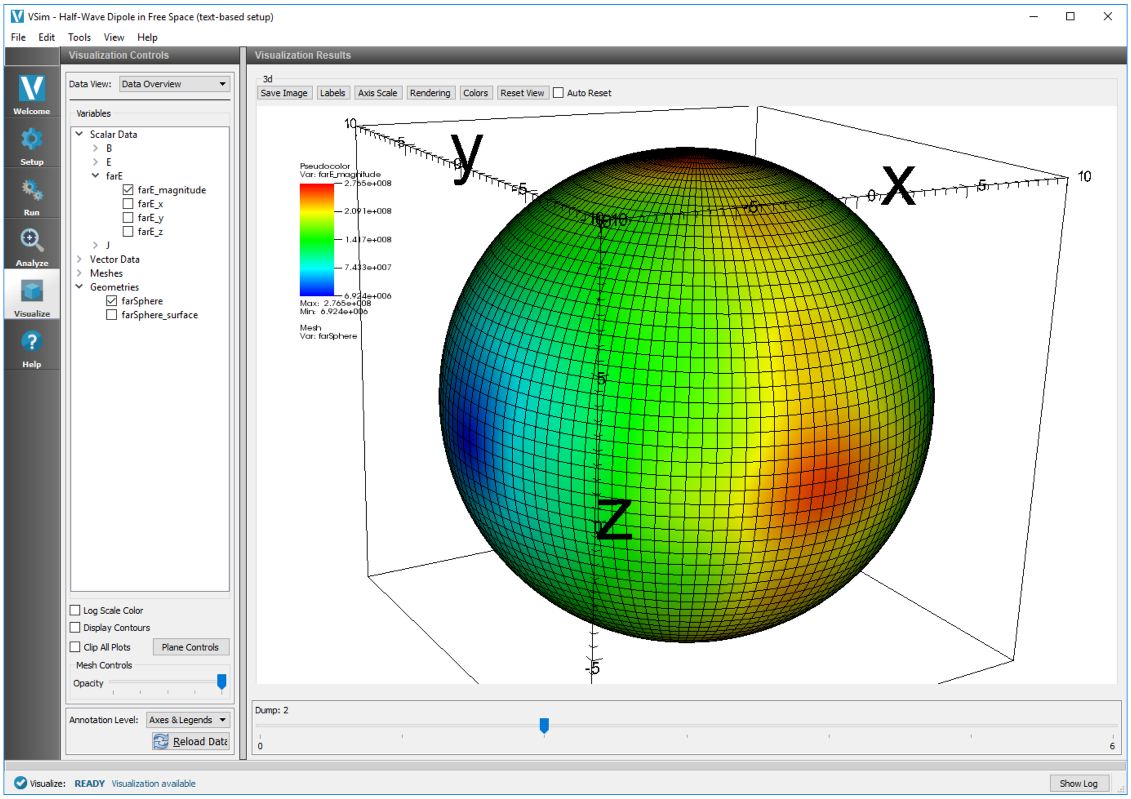
If the Simulation domain is made to small, the results will be distorted as the entire near field must be within the simulation domain in order to acheive a proper transformation to the far field.

3.3 Electrostatics

3.3.1 Like-Charge Dipole (esChargedSpheres.sdf)

Keywords:

`electrostatics, like-charge dipole`



Data View: Data Overview

Variables
- Scalar Data
  - B
  - E
  - farE
    - ☑ farE_magnitude
    - □ farE_x
    - □ farE_y
    - □ farE_z
  - J
- Vector Data
- Meshes
- Geometries
  - ☑ farSphere
  - □ farSphere_surface

□ Log Scale Color
□ Display Contours
□ Clip All Plots
Mesh Controls
Opacity
Annotation Level: Axes & Legends

Pseudocolor
Var: farE_magnitude
2.765e+008
2.091e+008
1.417e+008
7.433e+007
6.924e+006
Max: 2.765e+008
Min: 6.924e+006
Mesh
Var: farSphere

Visualize: READY Visualization available

Fig. 3.43: The far field radiation pattern

Problem description

This Like-charge Dipole simulation computes the electrostatic potential and field for a dipole of two spheres with given radius at the same potential.

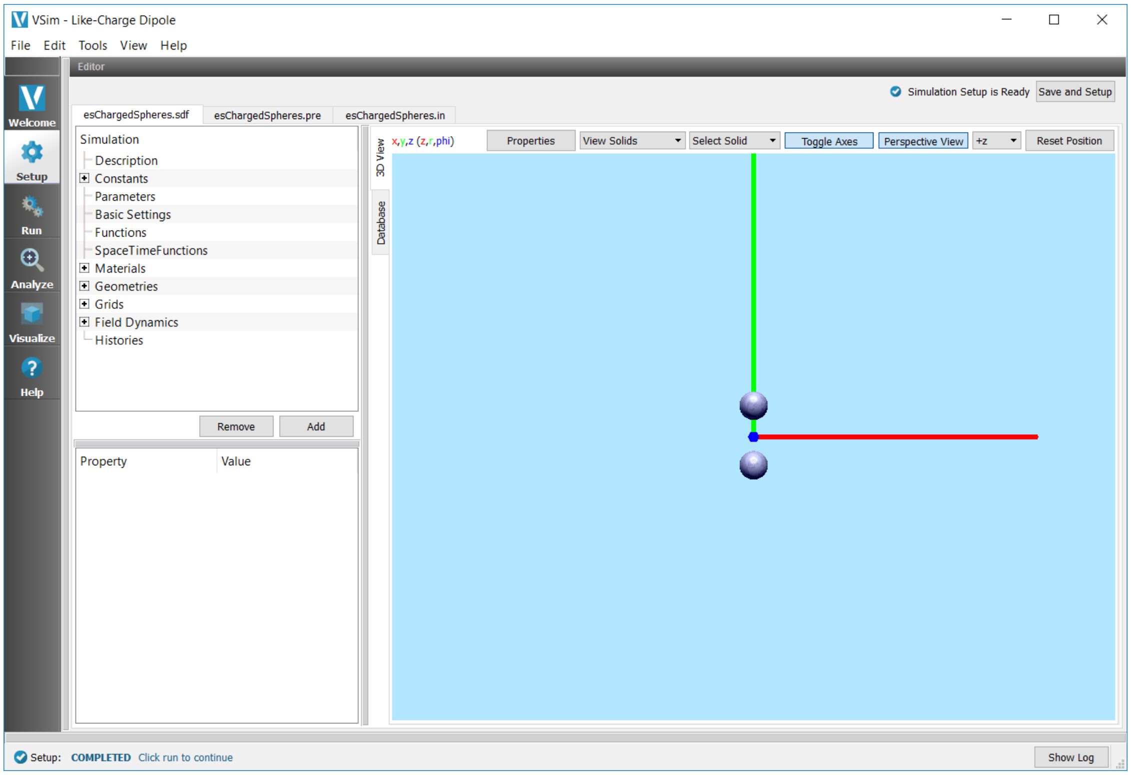
This simulation can be performed with a VSimEM or VSimPD license, with Composer licensed for Visual Setup.

Opening the Simulation

The like-charge dipole example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Electrostatics* option.
- Select *Like-charge Dipole* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 3.44. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



Simulation Setup is Ready

Setup: COMPLETED Click run to continue

Fig. 3.44: Setup Window for the Like-charge Dipole example.

Simulation Properties

This simulation uses visual setup to create a simple dipole. To do this we employ a few simple techniques such as Constructive Solid Geometries (CSG), and field Boundary Conditions. The dipole is constructed as two spheres of identical size. A Dirichlet boundary condition with the desired voltage is applied on both spheres.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in this pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 3.45.

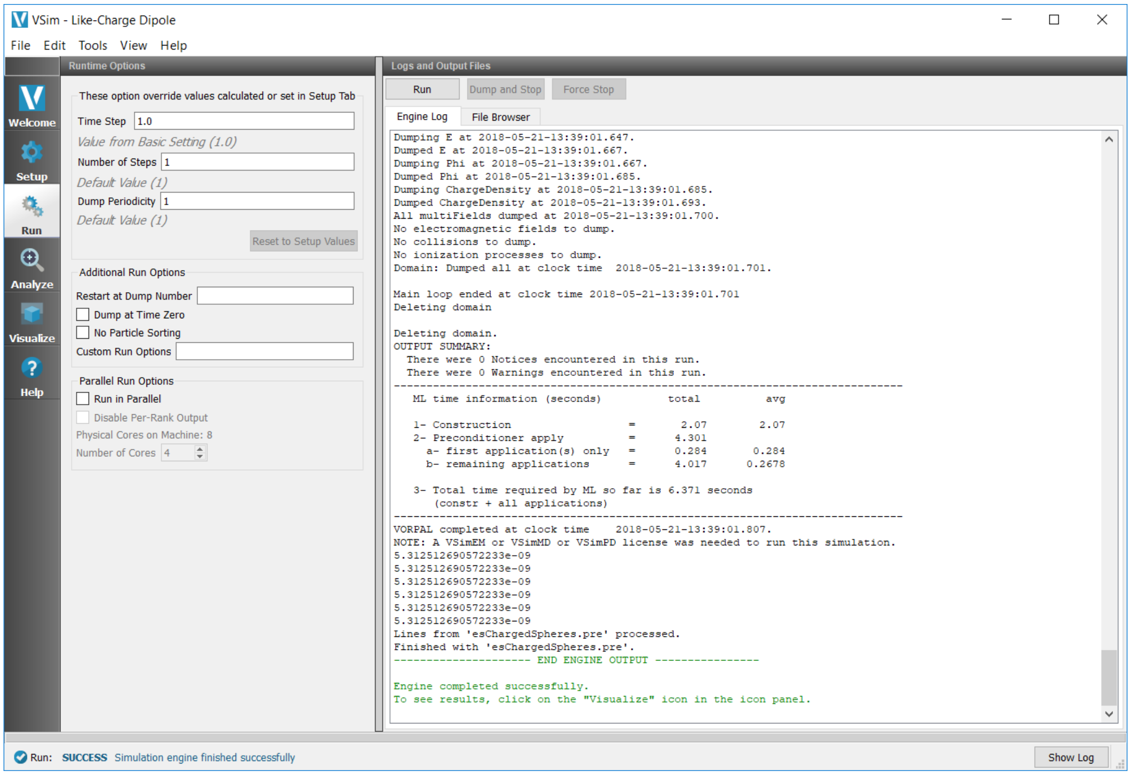
Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the electrostatic potential of the dipole as seen in Fig. 3.46 do the following:

- Expand *Scalar Data*
- Select *Phi*



These option override values calculated or set in Setup Tab

Time Step 1.0

Value from Basic Setting (1.0)

Number of Steps 1

Default Value (1)

Dump Periodicity 1

Default Value (1)

Additional Run Options

Restart at Dump Number

☐ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☐ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 8

Number of Cores 4

```
Dumping E at 2018-05-21-13:39:01.647.
Dumped E at 2018-05-21-13:39:01.667.
Dumping Phi at 2018-05-21-13:39:01.667.
Dumped Phi at 2018-05-21-13:39:01.685.
Dumping ChargeDensity at 2018-05-21-13:39:01.685.
Dumped ChargeDensity at 2018-05-21-13:39:01.693.
All multiFields dumped at 2018-05-21-13:39:01.700.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-21-13:39:01.701.

Main loop ended at clock time 2018-05-21-13:39:01.701
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
------------------------------------------------------------------------------
   ML time information (seconds)          total          avg

   1- Construction                 =       2.07        2.07
   2- Preconditioner apply         =      4.301
     a- first application(s) only  =      0.284       0.284
     b- remaining applications     =      4.017      0.2678

   3- Total time required by ML so far is 6.371 seconds
      (constr + all applications)
------------------------------------------------------------------------------
VORPAL completed at clock time    2018-05-21-13:39:01.807.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.
5.312512690572233e-09
5.312512690572233e-09
5.312512690572233e-09
5.312512690572233e-09
5.312512690572233e-09
5.312512690572233e-09
Lines from 'esChargedSpheres.pre' processed.
Finished with 'esChargedSpheres.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 3.45: The Run Window at the end of execution.

- Select *Display Contours*
- Select *Clip All Plots*

Further Experiments

Change the distance between spheres and see how the electric field changes.

Change the sphere radius and see how the electric field changes.

Change the sphere's potential to observe a change in the electric field.

Determine how the electric field changes with varying resolution.

3.4 Electrostatics (text-based setup)

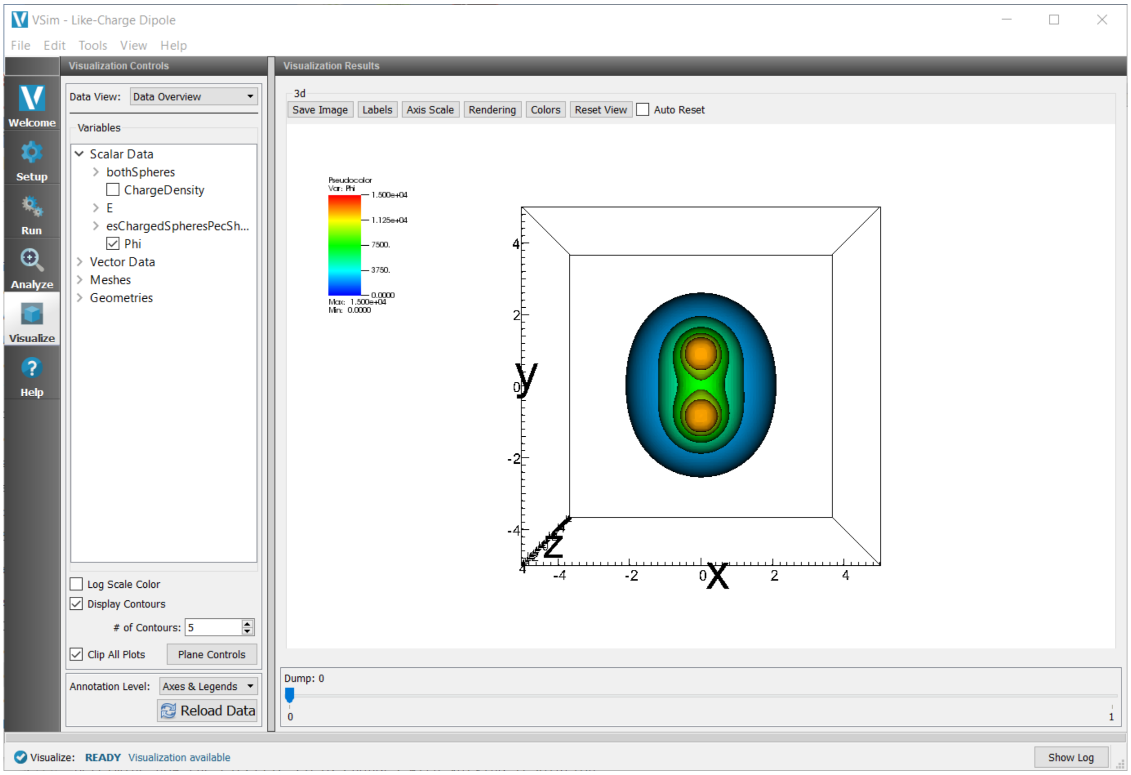
3.4.1 Like-Charge Dipole (esChargedSpheresT.pre)

Keywords:

`electrostatics, like-charge dipole`

Problem description

This ike-charge Dipole simulation computes the electrostatic potential and field for a dipole of two spheres with given radius, at the same potential.



Data View: Data Overview

☑ Phi

□ Log Scale Color

☑ Display Contours

# of Contours: 5

☑ Clip All Plots

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 0

Visualize: READY Visualization available

Fig. 3.46: Visualization of like-charge dipole as a pseudocolor plot.

This simulation can be performed with a VSimEM or VSimPD license.

Opening the Simulation

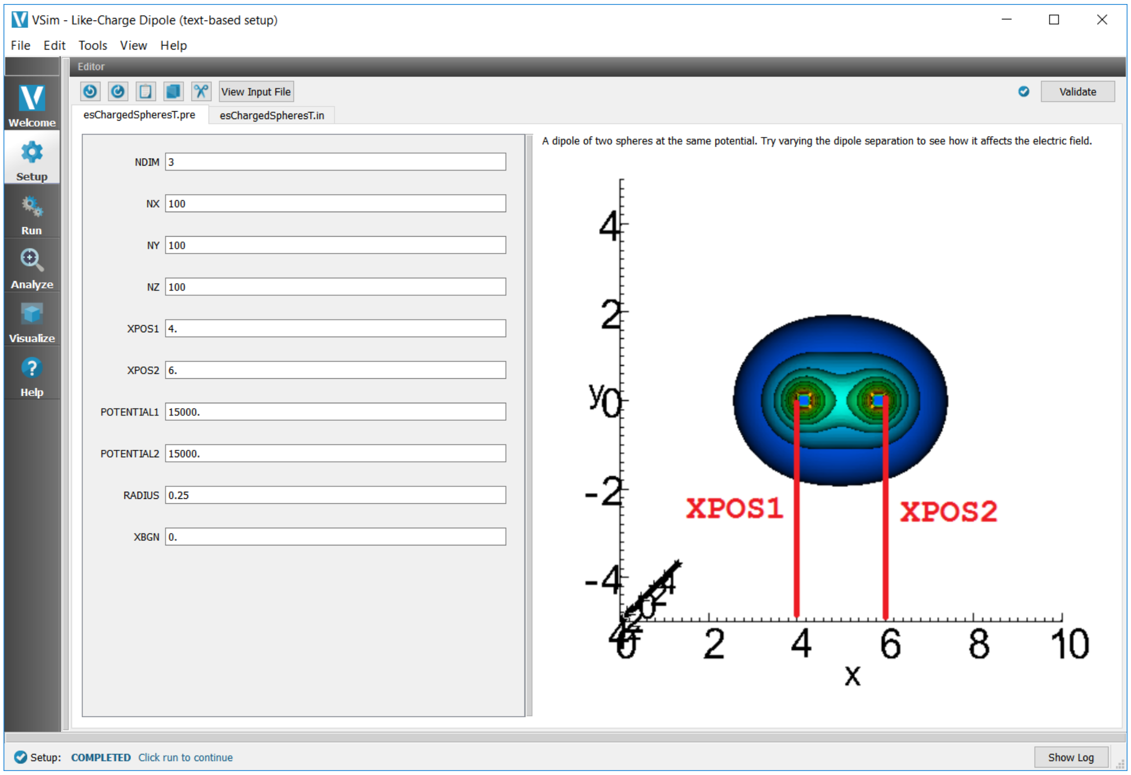
The like-charge dipole example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Electrostatics (text-based setup)* option.
- Select *Like-charge Dipole (text-based setup)* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 3.47.

Input File Features

The input file allows one to choose the location,radius and electrostatic potential of the spheres. It is also possible to adjust the size of the simulation domain, start point (m) of the simulation in the X direction, resolution (number of cells) in the simulation domain and number of dimensions to run the simulation in. The length of one timestep, DT is also available for modification. As the simulation is electrostatic it does not matter what value is selected for DT.



NDIM	3
NX	100
NY	100
NZ	100
XPOS1	4.
XPOS2	6.
POTENTIAL1	15000.
POTENTIAL2	15000.
RADIUS	0.25
XBGN	0.

A dipole of two spheres at the same potential. Try varying the dipole separation to see how it affects the electric field.

Setup: COMPLETED Click run to continue

Fig. 3.47: Setup Window for the Like-charge Dipole example.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in this pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 3.48.

Visualizing the results

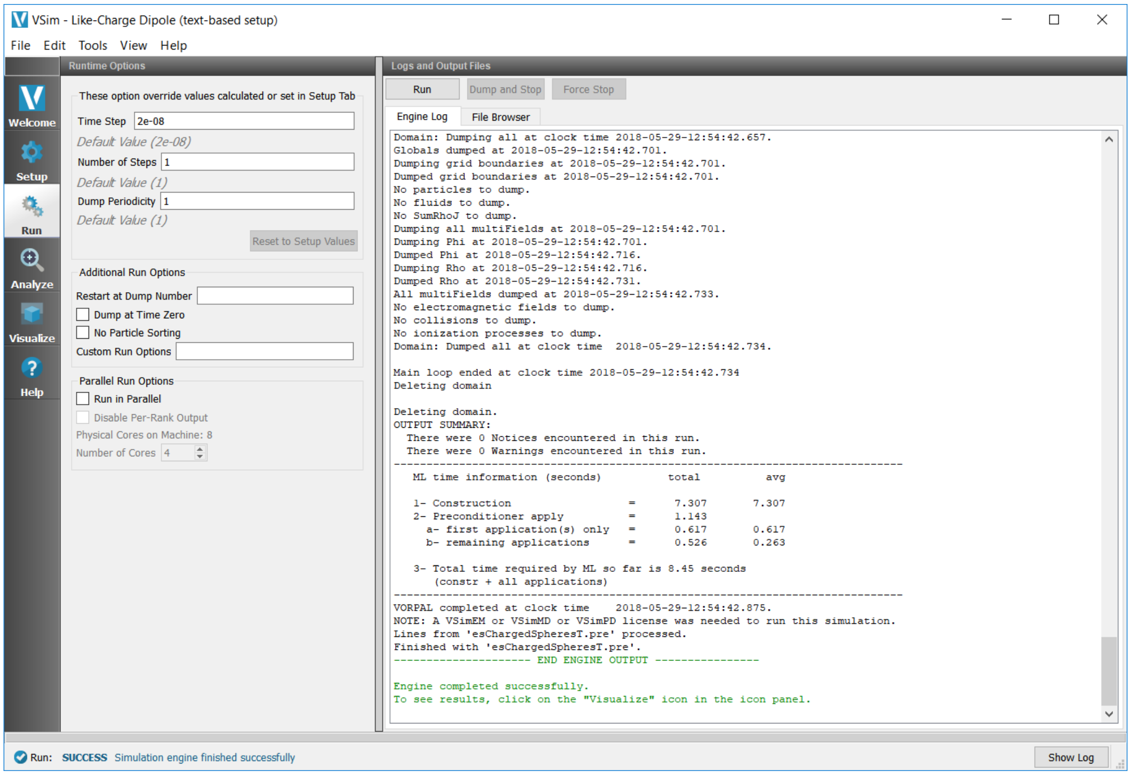
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The electrostatic potential can be viewed as follows:

- Expand *Scalar Data*
- Select *Phi*
- Select *Display Contours*
- Select the *Clip All Plots* check box
- Move the time slider forward to Dump 1

Fig. 3.49 shows the visualization seen.



These option override values calculated or set in Setup Tab

Time Step 2e-08
Default Value (2e-08)
Number of Steps 1
Default Value (1)
Dump Periodicity 1
Default Value (1)

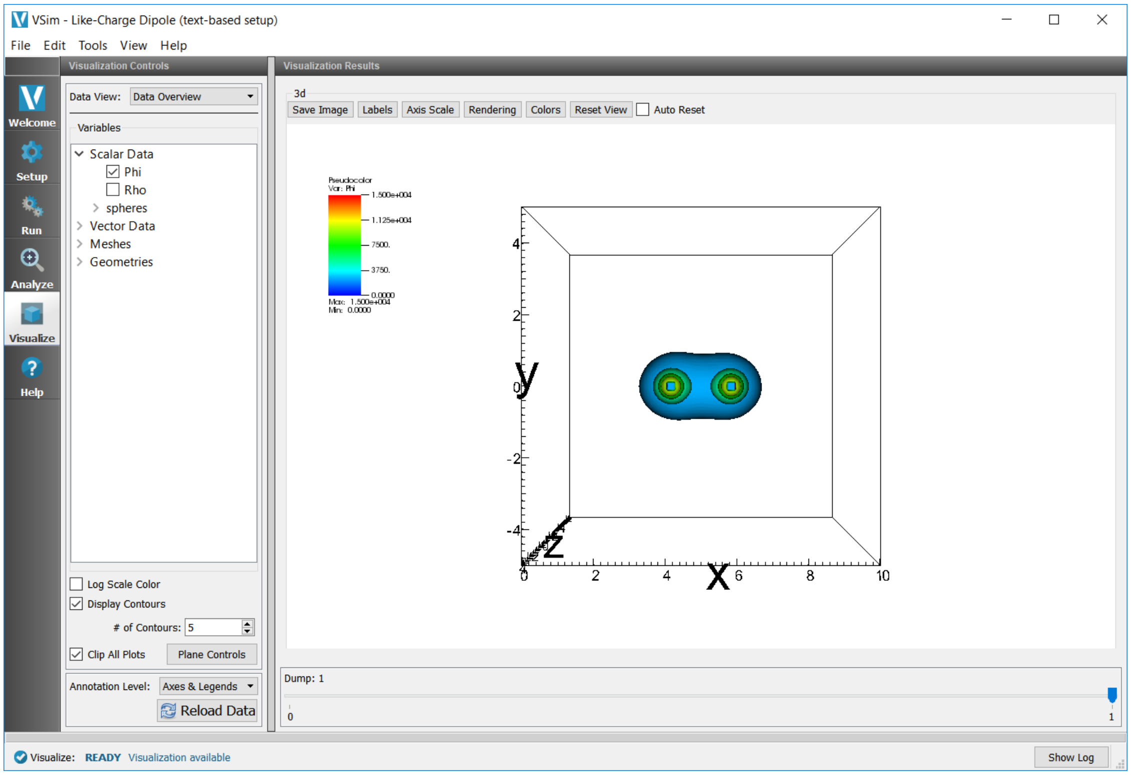
Additional Run Options
Restart at Dump Number
□ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options
□ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 8
Number of Cores 4

```
Domain: Dumping all at clock time 2018-05-29-12:54:42.657.
Globals dumped at 2018-05-29-12:54:42.701.
Dumping grid boundaries at 2018-05-29-12:54:42.701.
Dumped grid boundaries at 2018-05-29-12:54:42.701.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-29-12:54:42.701.
Dumping Phi at 2018-05-29-12:54:42.701.
Dumped Phi at 2018-05-29-12:54:42.716.
Dumping Rho at 2018-05-29-12:54:42.716.
Dumped Rho at 2018-05-29-12:54:42.731.
All multiFields dumped at 2018-05-29-12:54:42.733.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-29-12:54:42.734.

Main loop ended at clock time 2018-05-29-12:54:42.734
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
-----------------------------------------------------------------------------
   ML time information (seconds)          total          avg

   1- Construction                  =      7.307       7.307
   2- Preconditioner apply          =      1.143
     a- first application(s) only   =      0.617       0.617
     b- remaining applications      =      0.526       0.263

   3- Total time required by ML so far is 8.45 seconds
      (constr + all applications)
-----------------------------------------------------------------------------
VORPAL completed at clock time    2018-05-29-12:54:42.875.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.
Lines from 'esChargedSpheresT.pre' processed.
Finished with 'esChargedSpheresT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 3.48: The Run Window at the end of execution.



Data View: Data Overview

Variables
- Scalar Data
  - ☑ Phi
  - □ Rho
  - spheres
- Vector Data
- Meshes
- Geometries

□ Log Scale Color
☑ Display Contours
# of Contours: 5
☑ Clip All Plots
Annotation Level: Axes & Legends
□ Auto Reset
Dump: 1
0
1

Visualize: **READY** Visualization available

Fig. 3.49: Visualization of like-charge dipole as a pseudocolor plot.

Further Experiments

Change the distance between spheres by changing XPOS1 and XPOS2, and see how the electric field changes.

Change the sphere radius RADIUS and see how the electric field changes.

Change the sphere's potential to observe a change in the electric field.

Determine how the electric field changes with varying resolution.

To reduce computational time NDIM can be switched to 2 to make it a 2D simulation.

3.5 Photonics

3.5.1 Cylindrical Dielectric Fiber (cylFiber.sdf)

Keywords:

`Photonics, dielectric fiber`

Problem Description

This example illustrates how to compute the modes of a cylindrical fiber for a given propagation constant, β, which, because the primary direction of propagation in VSim is along the x-axis, is also denoted as k_x. The calculation is performed using excitation of a system with only two cells in the x-dimension. (The simulation can be performed with one cell in x, but this cannot be easily visualized so two are used instead.) This document will show how to extract the modes and their frequencies, as well as how to get a text-based setup file for further exploration, including solving for the propagation constant as a function of the frequency.

This simulation can be performed with a VSimEM license.

Opening the Simulation

The Cylindrical Dielectric Fiber example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Photonics* option.
- Select *Cylindrical Dielectric Fiber* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 3.50. Expanding the Constants section of the Simulation Tree shows the user-defined constants of this simulation:

`LENGTH_UNIT` (real): The length-scale of the simulation. All lengths, such as that of the fiber radius and box size, are divided by this number to make the geometry lengths of order unity as needed by the geometry engine. In this case, `LENGTH_UNIT` is unity, so it has no effect.

`WAVELENGTH_VAC` (real): Wavelength of the signal in vacuum. Determines the excitation frequency.

`N_EFF` (real): Estimate of the value of n_{eff} of the expected modes. This determines the wavenumber, $k_x = 2\pi n_{eff}/\lambda_{vac}$, of the modes to be found.

`RESOLUTION` (real): The resolution of the simulation grid. The cell size is set to be this number multiplied by the smallest simulation feature, i.e., the fiber radius.

XCELLS (integer): The number of cells to simulate in the x direction.

CFL_NUMBER (real): This times the stable time step gives the time step chosen for the simulation.

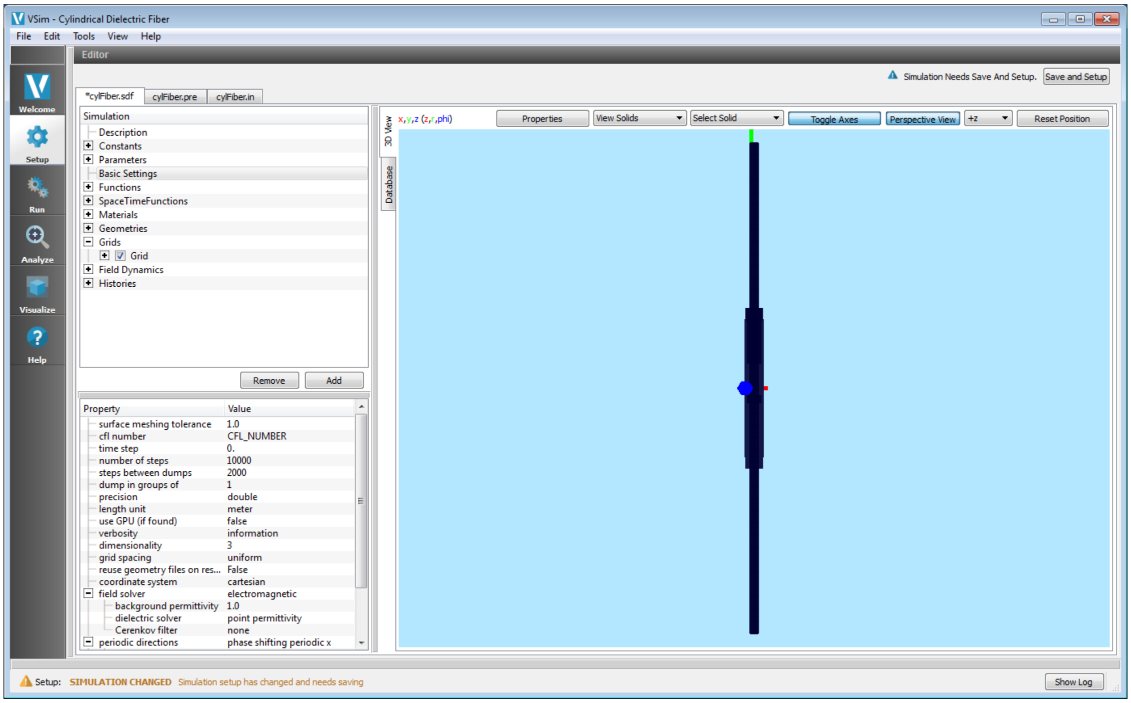
FREQ_GAP_REL (real): This times the stable time step gives the time step chosen for the simulation.

Expanding the Parameters section of the tree shows how the other simulation parameters are computed from the constants. For example, the grid size DX along the x-axis is set to the resolution multiplied by the vacuum wavelength, divided by n_{eff} and scaled by LENGTH_UNIT. The excitation central frequency, frequency, is computed from the vacuum wavelength scaled by the LENGTH_UNIT. The longitudinal wavenumber, KAY, is computed from the desired n_{eff}, and from that the phase shift across cells along the x-axis is calculated.

The range of frequencies to be excited is [FREQ_LOW, FREQ_HIGH]. Outside of this range by FREQ_GAP, the excitation drops off to the suppression value. This requires a successively longer excitation time, TIME_EXCITE and so a successivelong larger NSTEPS_EXCITE, the number of steps for the excitation.

Absorbing layers have been placed at the y and z limits to damp out modes that would be outgoing for a fiber in infinite space.

In 3D View tab of the right pane of the Setup Window, the fiber and the grid are visible. Right-click and drag to rotate the view. The simulation has been constructed so that the fiber extends beyond the grid in both the positive and negative x-directions, with a fiber diameter one-half the perpendicular domain length.



Property	Value
surface meshing tolerance	1.0
cfl number	CFL_NUMBER
time step	0.
number of steps	10000
steps between dumps	2000
dump in groups of	1
precision	double
length unit	meter
use GPU (if found)	false
verbosity	information
dimensionality	3
grid spacing	uniform
reuse geometry files on res...	False
coordinate system	cartesian
field solver	electromagnetic
background permittivity	1.0
dielectric solver	point permittivity
Cerenkov filter	none
periodic directions	phase shifting periodic x

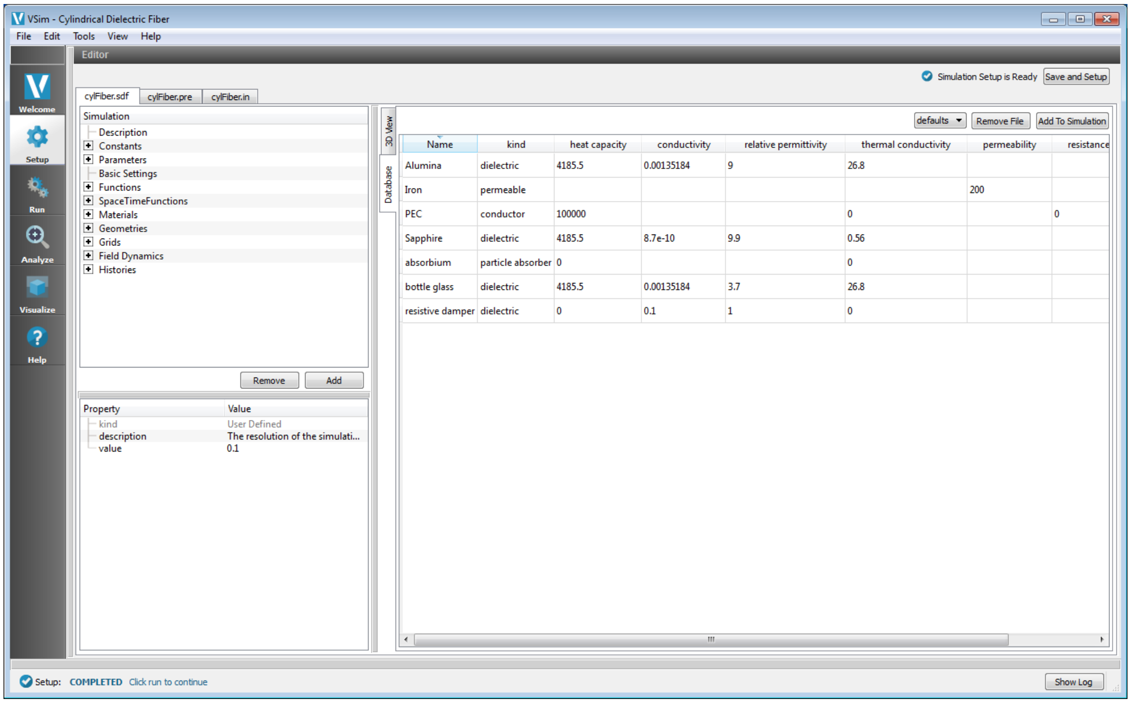
Setup: SIMULATION CHANGED Simulation setup has changed and needs saving

Fig. 3.50: Setup Window for the Cylindrical Dielectric Fiber example.

Expanding the Materials section of the Simulation tree shows that the simulation includes FiberMaterial. This was created by importing a material from the Database tab of the right pane of the Setup Window, then changing the name of the material and changing its properties. A material can be changed arbitrarily once it is in the simulation, as shown in Fig. 3.51.

Expanding the Geometries of the Elements Tree shows that the simulation includes one geometry, the fiber, and its material is FiberMaterial. This is seen in Fig. 3.52.

This simulation is excited with the freqBand function. This is a function that has a fairly uniform excitation over a band of frequencies, falling off steeply outside of the band. The band has been chosen to be centered near where we

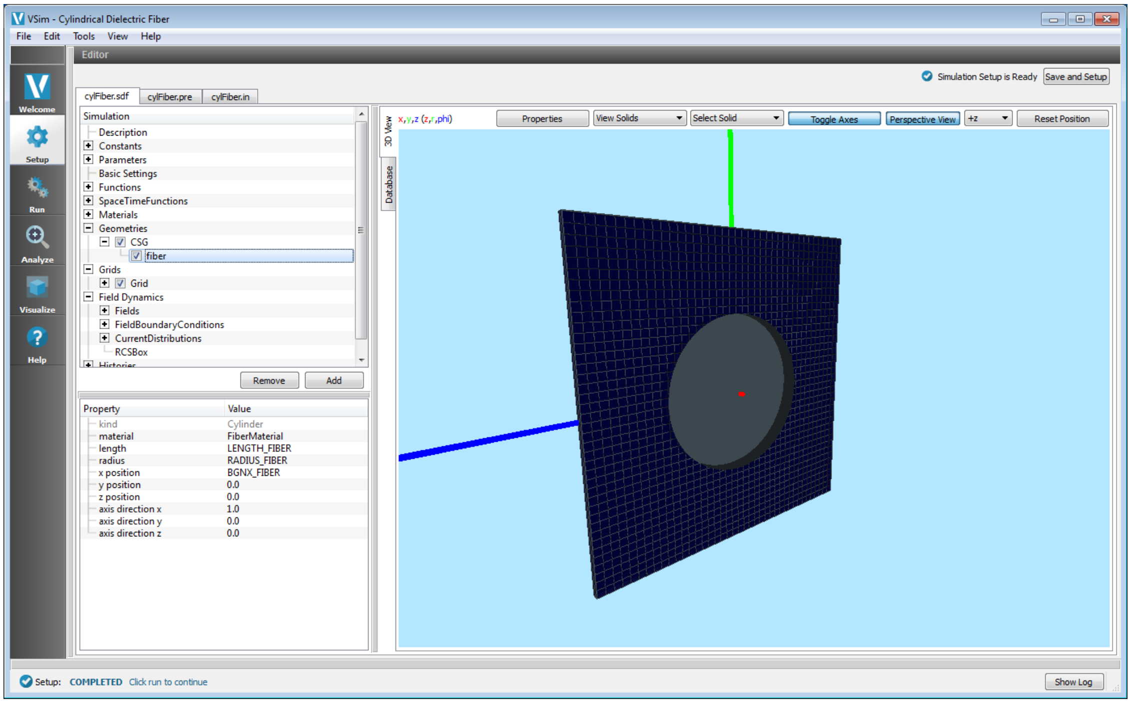


Property	Value
kind	User Defined
description	The resolution of the simulati...
value	0.1

Name	kind	heat capacity	conductivity	relative permittivity	thermal conductivity	permeability	resistance
Alumina	dielectric	4185.5	0.00135184	9	26.8		
Iron	permeable					200	
PEC	conductor	100000			0		0
Sapphire	dielectric	4185.5	8.7e-10	9.9	0.56		
absorbium	particle absorber	0			0		
bottle glass	dielectric	4185.5	0.00135184	3.7	26.8		
resistive damper	dielectric	0	0.1	1	0		

Setup: COMPLETED Click run to continue

Fig. 3.51: Setup Window for the Cylindrical Dielectric Fiber materials.



Property	Value
kind	Cylinder
material	FiberMaterial
length	LENGTH_FIBER
radius	RADIUS_FIBER
x position	BGNX_FIBER
y position	0.0
z position	0.0
axis direction x	1.0
axis direction y	0.0
axis direction z	0.0

Setup: COMPLETED Click run to continue

Fig. 3.52: Setup Window for the Cylindrical Dielectric Fiber geometries.

expect to find the modes.

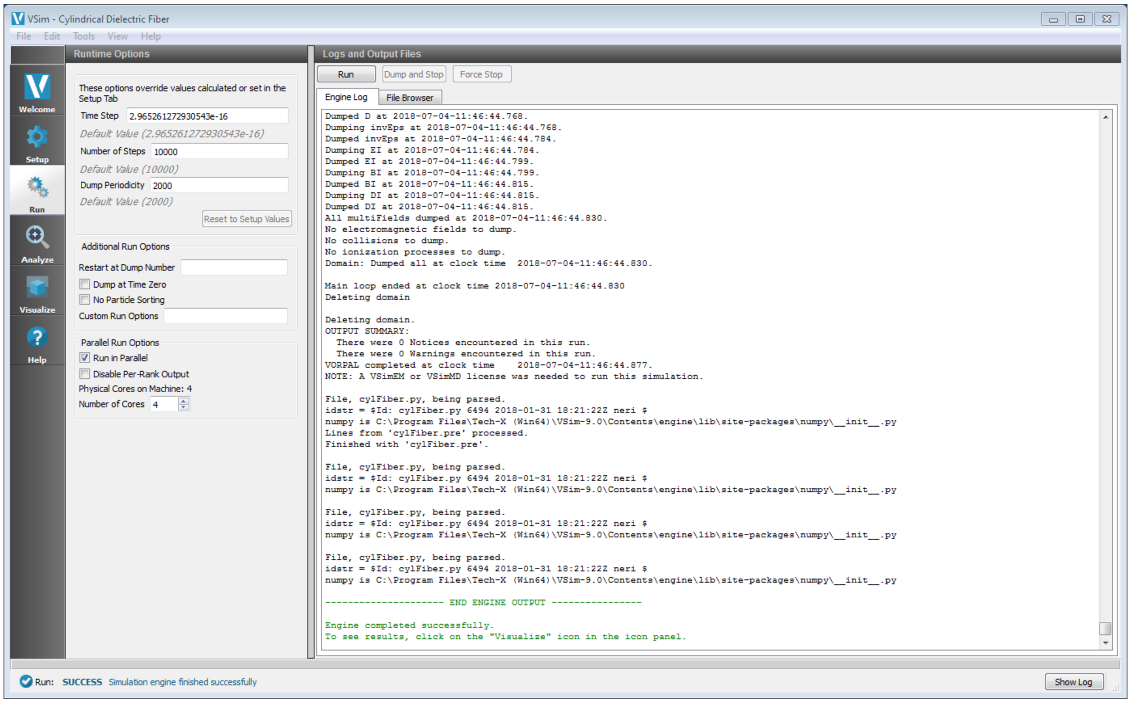
A field history has also been implimented in this simulation, so that the Fourier transform of what has been excited can be seen.

As noted above, under the Parameters section of the Tree Elements is defined `NSTEPS_EXCITE` which specifies the number of steps to excite the desired frequency content. Because `FREQ_GAP` also distinguishes the peaks, this excitation time will distinguish the peaks.

Running the Simulation

Once finished with the problem setup, continue as follows:

- Proceed to the Run Window by pressing the **Run** button in the left column of buttons.
- For this run we choose 10000 steps, much greater than that (4000) required for the excitation. This further reduces any effect of the excitation on the signal in free oscillation.
- Choose parallel computing options on the MPI tab.
- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane. You will see the output of the run in that pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 3.53.



These options override values calculated or set in the Setup Tab

Time Step 2.965261272930543e-16

Default Value (2.965261272930543e-16)

Number of Steps 10000

Default Value (10000)

Dump Periodicity 2000

Default Value (2000)

Additional Run Options

Restart at Dump Number

☐ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Dumped D at 2018-07-04-11:46:44.768.
Dumping invEps at 2018-07-04-11:46:44.768.
Dumped invEps at 2018-07-04-11:46:44.784.
Dumping EI at 2018-07-04-11:46:44.784.
Dumped EI at 2018-07-04-11:46:44.799.
Dumping BI at 2018-07-04-11:46:44.799.
Dumped BI at 2018-07-04-11:46:44.815.
Dumping DI at 2018-07-04-11:46:44.815.
Dumped DI at 2018-07-04-11:46:44.815.
All multiFields dumped at 2018-07-04-11:46:44.830.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-04-11:46:44.830.

Main loop ended at clock time 2018-07-04-11:46:44.830
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-07-04-11:46:44.877.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.

File, cylFiber.py, being parsed.
idstr = $Id: cylFiber.py 6494 2018-01-31 18:21:22Z neri $
numpy is C:\Program Files\Tech-X (Win64)\VSim-9.0\Contents\engine\lib\site-packages\numpy\__init__.py
Lines from 'cylFiber.pre' processed.
Finished with 'cylFiber.pre'.

File, cylFiber.py, being parsed.
idstr = $Id: cylFiber.py 6494 2018-01-31 18:21:22Z neri $
numpy is C:\Program Files\Tech-X (Win64)\VSim-9.0\Contents\engine\lib\site-packages\numpy\__init__.py

File, cylFiber.py, being parsed.
idstr = $Id: cylFiber.py 6494 2018-01-31 18:21:22Z neri $
numpy is C:\Program Files\Tech-X (Win64)\VSim-9.0\Contents\engine\lib\site-packages\numpy\__init__.py

File, cylFiber.py, being parsed.
idstr = $Id: cylFiber.py 6494 2018-01-31 18:21:22Z neri $
numpy is C:\Program Files\Tech-X (Win64)\VSim-9.0\Contents\engine\lib\site-packages\numpy\__init__.py

--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

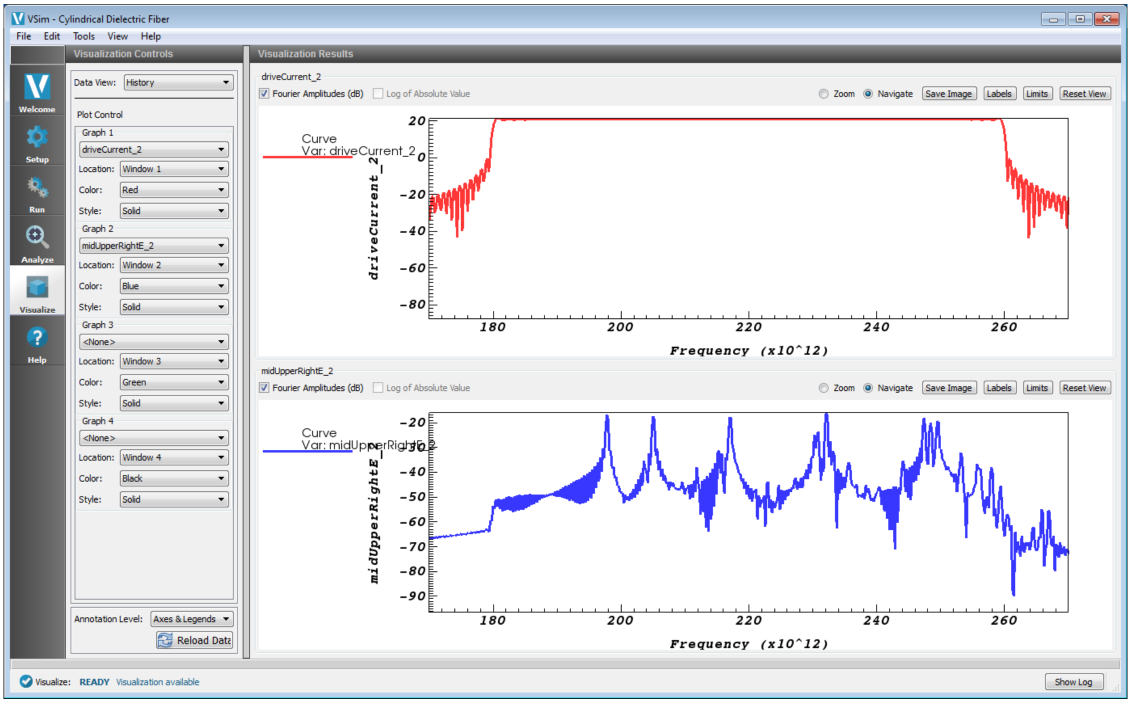
Fig. 3.53: The Run Window at the end of execution.

This simulation takes approximately 10 seconds on 4 cores of a modern processor.

Visualizing the spectrum

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.
- In the Visualization Controls pane, in the dropdown menu for *Data View* select *History*.

- For graph 1, set the quantity to be plotted to `driveCurrent_2`.
- For graph 2, set the quantity to be plotted to `midUpperRightE_2`.
- For graphs 3 and 4 set the quantity to be plotted to `<None>`.
- In the Visualization Results pane, for each plot, click the *Fourier Amplitudes (dB)* check box.
- Graph 1 shows the square window in frequency space.
- Graph 2 shows several peaks between 200 and 250 THz.
- To see this region in more detail, for each graph press the limits button and set the minimum to 1.7e14 and the maximum to 2.7e14. Peaks in the spectrum are seen at the frequencies, 198 THz, 205THz, 217THz, 233, and several around 250 THz in Fig. 3.54.


VSim - Cylindrical Dielectric Fiber

Visualization Controls

Data View: History

Plot Control

Graph 1
driveCurrent_2
Location: Window 1
Color: Red
Style: Solid

Graph 2
midUpperRightE_2
Location: Window 2
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 3
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualization Results

driveCurrent_2: ☑ Fourier Amplitudes (dB) ☐ Log of Absolute Value ○ Zoom ● Navigate

midUpperRightE_2: ☑ Fourier Amplitudes (dB) ☐ Log of Absolute Value ○ Zoom ● Navigate

Visualize: READY Visualization available

Fig. 3.54: The observed spectrum.

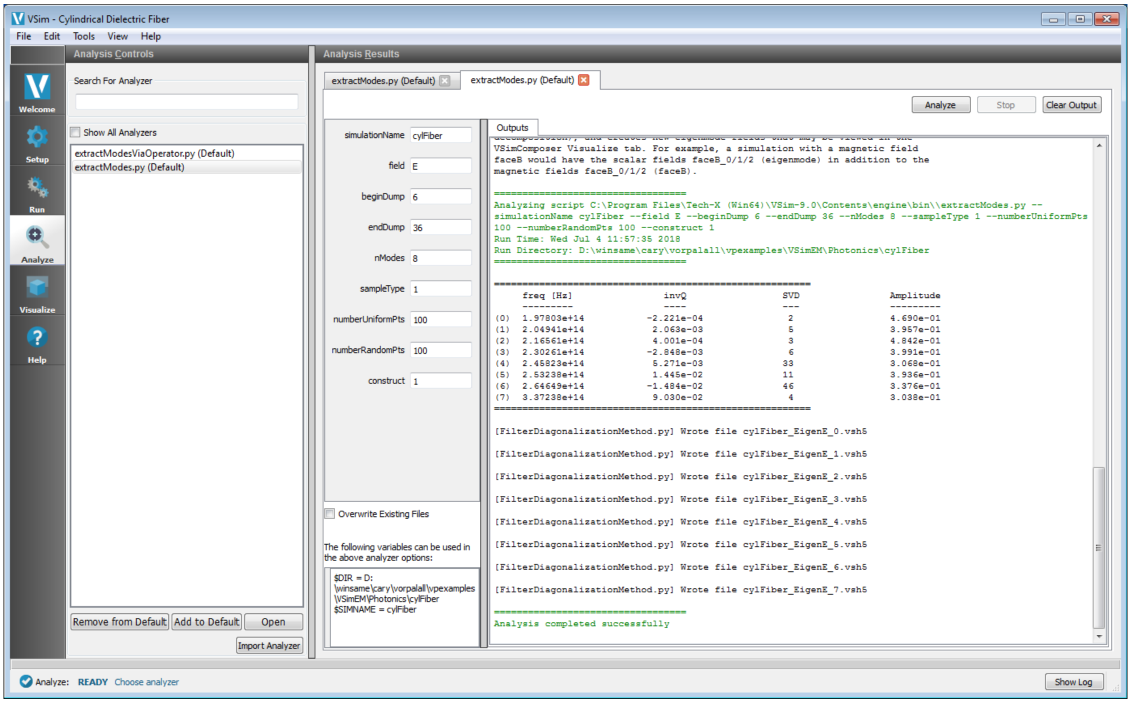
Analyzing the data

Since 8 modes were seen, we look for 8. This will requires 3 dumps per mode, or about 27 additional dumps. The dumps should be spread out over a few oscillations (35 steps each) or so, and it is good to do a few extra. So upon returning to the Run pane, select 200 time steps, dump periodicity of 5 time steps, and restart at dump 5.

Now go to the Analyze pane, select the *extractModes.py* analyzer, and press the *Open* button. Set the field to E, choose beginDump to be 6, endDump to be 36, nModes to be 8, sampleType of 1, construct to be 1, and 100 of each kind of points. Upon hitting the Analyze button of the Analyze pane, we sees the list of detected frequencies, of which the first (mode 0) is the mode of interest, it having frequency of 1.98e14. This is seen in Fig. 3.55.

Visualizing the eigenmodes

Return to the Visualize Window, reload the data, open Scalar Data -> E, and click on E_z (EigenE). Keep the slider on position 0. With the mouse, turn the image sideways to see the cross section, as shown in Fig. 3.56.



□ Show All Analyzers

extractModesViaOperator.py (Default)
extractModes.py (Default)

simulationName: cylFiber
field: E
beginDump: 6
endDump: 36
nModes: 8
sampleType: 1
numberUniformPts: 100
numberRandomPts: 100
construct: 1

□ Overwrite Existing Files

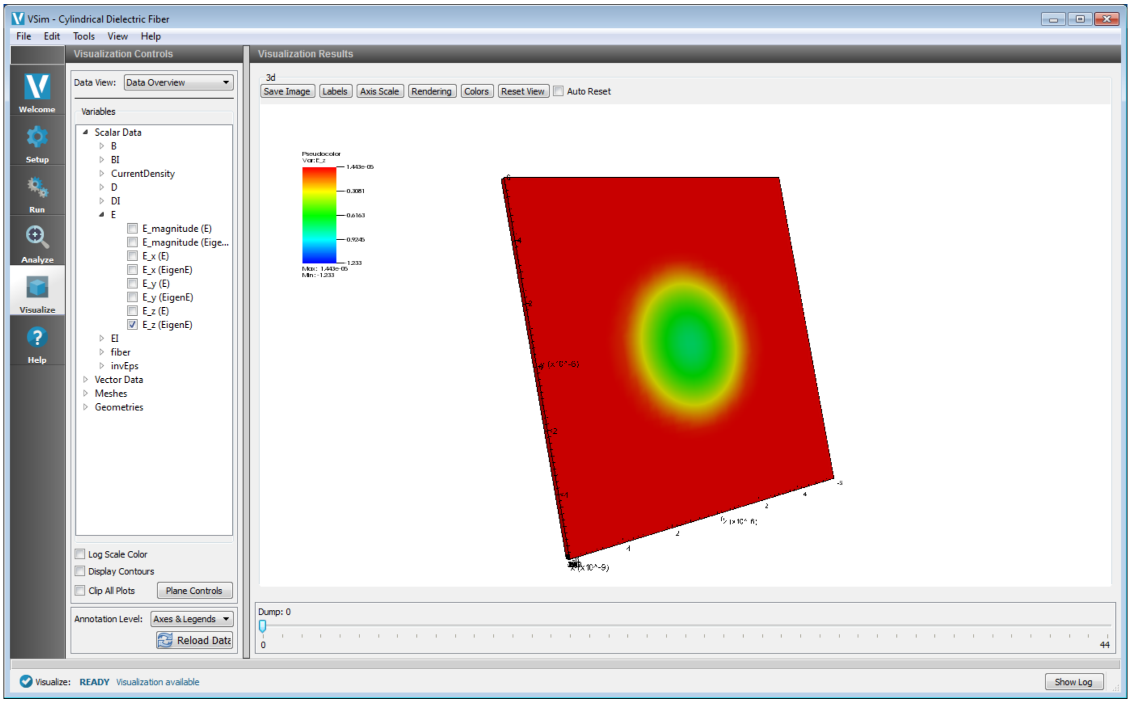
The following variables can be used in the above analyzer options:

$DIR = D:\winsame\cary\vorpalall\vpexamples\VSimEM\Photonics\cylFiber
$SIMNAME = cylFiber

```
VSimComposer Visualize tab. For example, a simulation with a magnetic field
faceB would have the scalar fields faceB_0/1/2 (eigenmode) in addition to the
magnetic fields faceB_0/1/2 (faceB).

==================================
Analyzing script C:\Program Files\Tech-X (Win64)\VSim-9.0\Contents\engine\bin\\extractModes.py --
simulationName cylFiber --field E --beginDump 6 --endDump 36 --nModes 8 --sampleType 1 --numberUniformPts
100 --numberRandomPts 100 --construct 1
Run Time: Wed Jul 4 11:57:35 2018
Run Directory: D:\winsame\cary\vorpalall\vpexamples\VSimEM\Photonics\cylFiber
==================================

========================================================
     freq [Hz]              invQ               SVD               Amplitude
     ---------              ----               ---               ---------
(0)  1.97803e+14         -2.221e-04             2                4.690e-01
(1)  2.04941e+14          2.063e-03             5                3.957e-01
(2)  2.16561e+14          4.001e-04             3                4.842e-01
(3)  2.30261e+14         -2.848e-03             6                3.991e-01
(4)  2.45823e+14          5.271e-03            33                3.068e-01
(5)  2.53238e+14          1.445e-02            11                3.936e-01
(6)  2.64649e+14         -1.484e-02            46                3.376e-01
(7)  3.37238e+14          9.030e-02             4                3.038e-01
========================================================

[FilterDiagonalizationMethod.py] Wrote file cylFiber_EigenE_0.vsh5

[FilterDiagonalizationMethod.py] Wrote file cylFiber_EigenE_1.vsh5

[FilterDiagonalizationMethod.py] Wrote file cylFiber_EigenE_2.vsh5

[FilterDiagonalizationMethod.py] Wrote file cylFiber_EigenE_3.vsh5

[FilterDiagonalizationMethod.py] Wrote file cylFiber_EigenE_4.vsh5

[FilterDiagonalizationMethod.py] Wrote file cylFiber_EigenE_5.vsh5

[FilterDiagonalizationMethod.py] Wrote file cylFiber_EigenE_6.vsh5

[FilterDiagonalizationMethod.py] Wrote file cylFiber_EigenE_7.vsh5

==================================
Analysis completed successfully
```

Analyze: READY Choose analyzer

Fig. 3.55: Extraction of the mode frequencies.



Data View: Data Overview

Variables
- Scalar Data
  - B
  - BI
  - CurrentDensity
  - D
  - DI
  - E
    - □ E_magnitude (E)
    - □ E_magnitude (Eige…
    - □ E_x (E)
    - □ E_x (EigenE)
    - □ E_y (E)
    - □ E_y (EigenE)
    - □ E_z (E)
    - ☑ E_z (EigenE)
  - EI
  - fiber
  - invEps
- Vector Data
- Meshes
- Geometries

□ Log Scale Color
□ Display Contours
□ Clip All Plots

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 0

Visualize: READY Visualization available

Fig. 3.56: The extracted eigenmode.

Convergence

This simulation can be repeated for different values of RESOLUTION to see how the frequency varies with the meshing. We carried out this experiment with RESOLUTION varying over 0.1, 0.05, 0.02, 0.01 and plotted the frequency versus the inverse grid length in Fig. 3.57.

For each value of the resolution do the excitation run followed by the extraction run:

- Excitation run: Press *Reset to Setup Values* to get the correct value for the time step. Then set the number of step in the run panel to what is given by NSTEPS_EXCITE, and also modify the number of steps in the second run proportionately. E.g., for RESOLUTION = 0.05, NSTEPS_EXCITE = 7916, so in the Run panel choose *Number of Steps* = 8000 and *Dump Periodicity* = 2000. Clear the *Restart at Dump Number* box. Press the *Run* button.
- Extraction run: E.g., for RESOLUTION = 0.05, set the *Number of Steps* to 400, the *Dump Periodicity* to 10, and *Restart at Dump Number* to be 4.
- Analysis: Same as originally, as all numbers have been scaled.

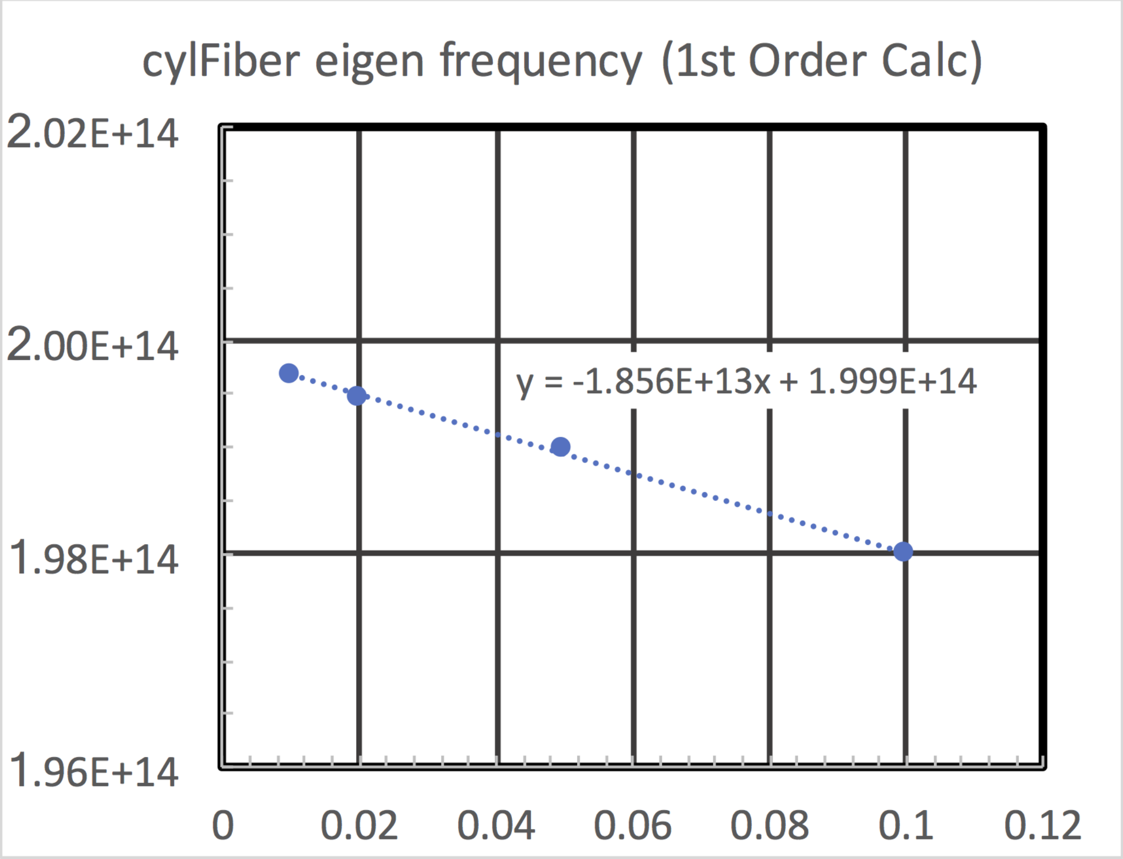


Fig. 3.57: Convergence of the first mode.

The linear approach to the axis indicates that this is a first order accurate calculation. In other examples we will show higher-order accuracy. Even so, one can see that the frequency is obtained on the sub-percent level with the finest grid used.

Further Experiments

This same process can be used to get the frequency of modes of different wavelengths or of waveguides of different cross sections or made of different dielectrics.

3.5.2 Dielectric Waveguide with Gaussian Launcher (dielectricWaveguideGaussian.sdf)

Keywords:

`Photonic Waveguide, Unidirectional Mode Launcher, MAL,`
`Guided Mode, Semiconductor`

Problem description

The dielectric waveguide consists of a single, straight silicon waveguide that is parallel to the x-axis and centered at the origin. The waveguide is surrounded by silica. Matched Absorbing Layers (MALs) are used to dampen the E and B fields near the boundary of the simulation to supress reflected fields.

A gaussian approximation of the fundamental guided mode is launched in the silicon waveguide in the +x direction. The fundamental mode was extracted in the Dielectric Waveguide Mode Calculation Example.

This simulation can be performed with a VSimEM license.

Opening the Simulation

The dielectric waveguide example can be accessed from within VSimComposer through the following steps:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window, expand the *VSim for Electromagnetics* option.
- Expand the *Photonics* option.
- Select *Dielectric Waveguide with Gaussian Launcher* and press the *Choose* button.
- In the resulting dialog, create a *New Folder* if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation will now be available in the Setup window as shown in Fig. 3.58. You can expand the tree elements and navigate through the various properties. The right pane shows a 3D view of the geometry, as well as the grid. To show or hide the grid, expand the *Grid* element and select or deselect the box next to *Grid*.

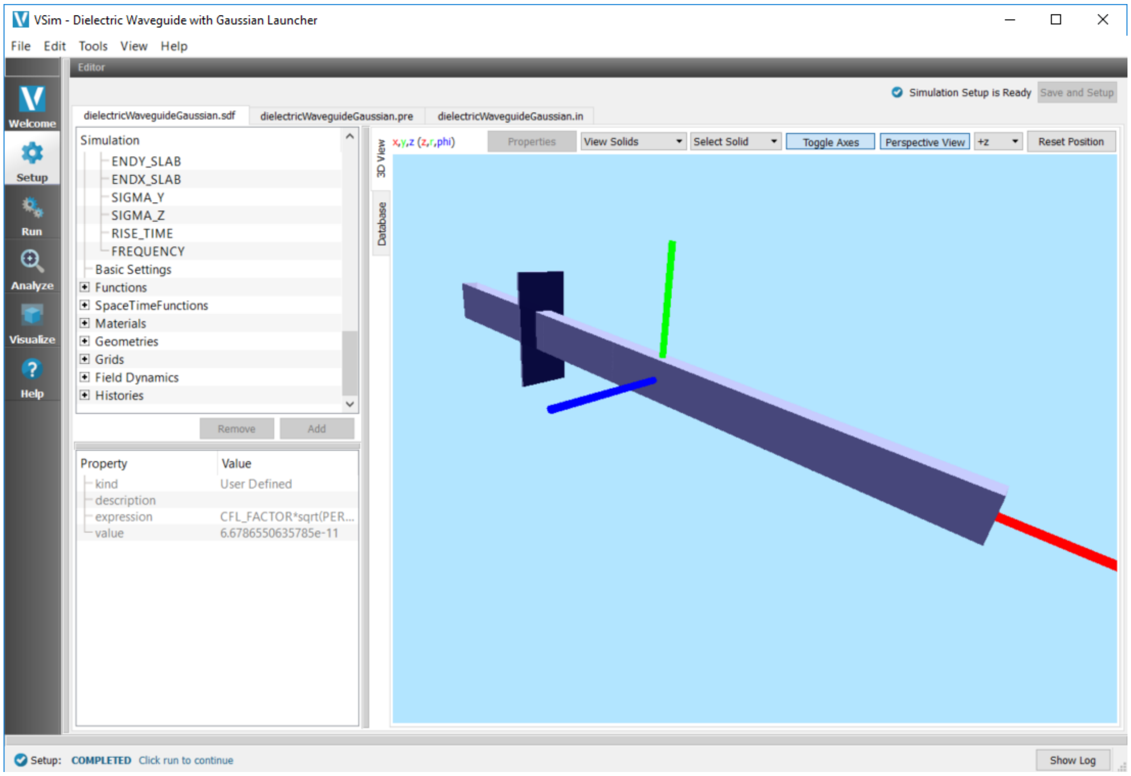
Simulation Properties

This example contains a number of constants defined to make the simulation easily modifiable. Some relevant constants are listed below.

PERMITTVITY_WAVEGUIDE and PERMITTVITY_BACKGROUND: Relative permittivities of silicon and silica. These constants are used in multiple parameters and in the accompanying Python file for solving the waveguide modes.

LENGTH_UNIT: The constant factor by which VSim will scale all simulation lengths.

WAVELENGTH_VAC: Wavelength of the input signal. This wavelength is also used for the calculation of the fundamental guided mode of the device.


VSim - Dielectric Waveguide with Gaussian Launcher
File Edit Tools View Help
Editor
Welcome
Setup
Run
Analyze
Visualize
Help
Simulation Setup is Ready
Save and Setup
dielectricWaveguideGaussian.sdf
dielectricWaveguideGaussian.pre
dielectricWaveguideGaussian.in
3D View
Database
x,y,z (z,r,phi)
Properties
View Solids
Select Solid
Toggle Axes
Perspective View
+z
Reset Position
Remove
Add
Show Log

Simulation
- ENDY_SLAB
- ENDX_SLAB
- SIGMA_Y
- SIGMA_Z
- RISE_TIME
- FREQUENCY
- Basic Settings
- Functions
- SpaceTimeFunctions
- Materials
- Geometries
- Grids
- Field Dynamics
- Histories

Property	Value
kind	User Defined
description	
expression	CFL_FACTOR*sqrt(PER...
value	6.6786550635785e-11

Setup: COMPLETED Click run to continue

Fig. 3.58: The Setup window for the dielectric waveguide example showing some relevant constants.

NWAVELENGTH_MAL: Approximate number of wavelengths that can fit in a MAL region. The thickness of the MAL regions in this example are measured in wavelengths.

The *Materials* section contains just silicon and silica.

The *Geometries* includes the CSG waveguide and its defining parameters.

In *Field Dynamics*, there are *FieldBoundaryConditions* and *CurrentDistributions* to be aware of. In photonics simulations, Matched Absorbing Layers (MALs) are the most stable boundary conditions for preventing reflections. The gaussian approximation is defined under *SpaceTimeFunctions* and is set to drive the y-component of the *currentSource*.

Running the simulation

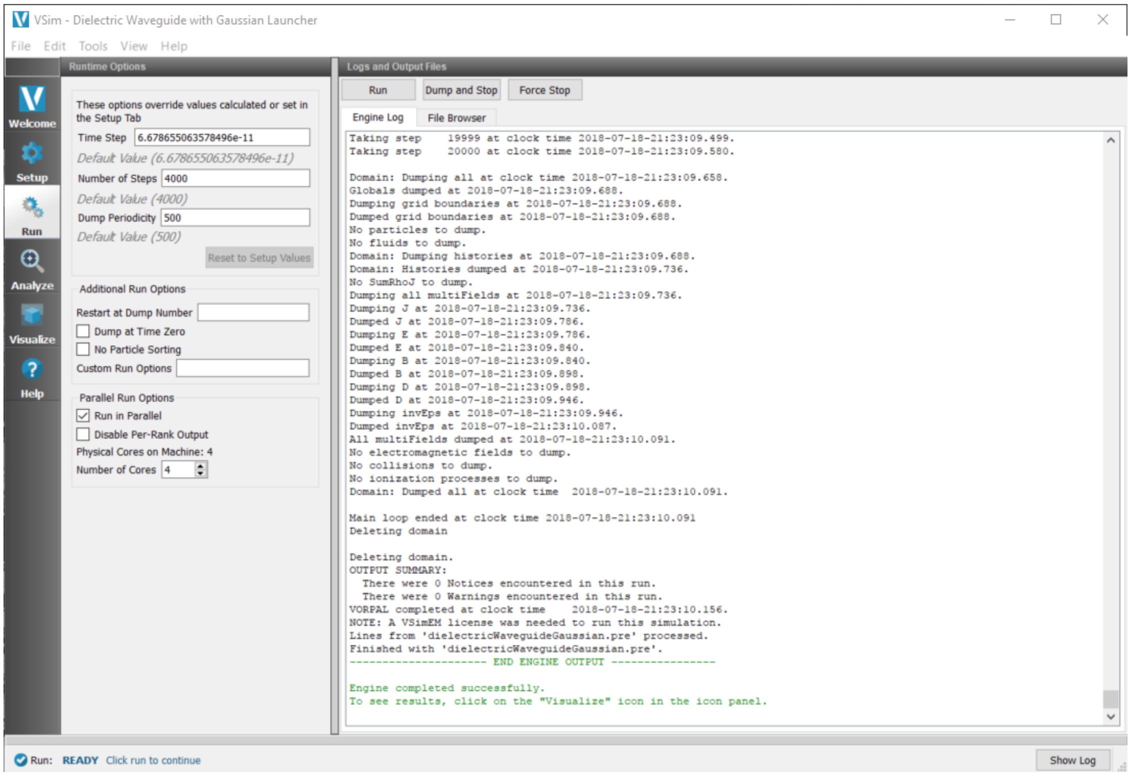
After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the *Run* button in the left column. You will be asked to Save. Click *Save*.
- To run the file, click on the *Run* button in the upper left corner of the right pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” The result is shown in Fig. 3.59.

Visualizing the Results

Then proceed to the Visualize window by pressing the *Visualize* button in the left column.

You can verify that the geometry is correct by visualizing the inverse permittivity as follows:



These options override values calculated or set in the Setup Tab

Time Step 6.678655063578496e-11

Default Value (6.678655063578496e-11)

Number of Steps 4000

Default Value (4000)

Dump Periodicity 500

Default Value (500)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Taking step    19999 at clock time 2018-07-18-21:23:09.499.
Taking step    20000 at clock time 2018-07-18-21:23:09.580.

Domain: Dumping all at clock time 2018-07-18-21:23:09.658.
Globals dumped at 2018-07-18-21:23:09.688.
Dumping grid boundaries at 2018-07-18-21:23:09.688.
Dumped grid boundaries at 2018-07-18-21:23:09.688.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-07-18-21:23:09.688.
Domain: Histories dumped at 2018-07-18-21:23:09.736.
No SumRhoJ to dump.
Dumping all multiFields at 2018-07-18-21:23:09.736.
Dumping J at 2018-07-18-21:23:09.736.
Dumped J at 2018-07-18-21:23:09.786.
Dumping E at 2018-07-18-21:23:09.786.
Dumped E at 2018-07-18-21:23:09.840.
Dumping B at 2018-07-18-21:23:09.840.
Dumped B at 2018-07-18-21:23:09.898.
Dumping D at 2018-07-18-21:23:09.898.
Dumped D at 2018-07-18-21:23:09.946.
Dumping invEps at 2018-07-18-21:23:09.946.
Dumped invEps at 2018-07-18-21:23:10.087.
All multiFields dumped at 2018-07-18-21:23:10.091.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-18-21:23:10.091.

Main loop ended at clock time 2018-07-18-21:23:10.091
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-07-18-21:23:10.156.
NOTE: A VSimEM license was needed to run this simulation.
Lines from 'dielectricWaveguideGaussian.pre' processed.
Finished with 'dielectricWaveguideGaussian.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: READY Click run to continue

Fig. 3.59: The output after a successful run.

- Near the top left corner of the window, make sure *Data View* is set to *Data Overview*.
- Expand *Scalar Data*, expand *invEps*, and select *invEps_z*
- In the controls below the variables frame, select *Clip All Plots*.

By default, the clipping plane is at z = 0, which is in the middle of dielectric waveguide structure height-wise. As such, this will reveal the dielectric waveguide's 2D layout geometry as seen in Fig. 3.60.

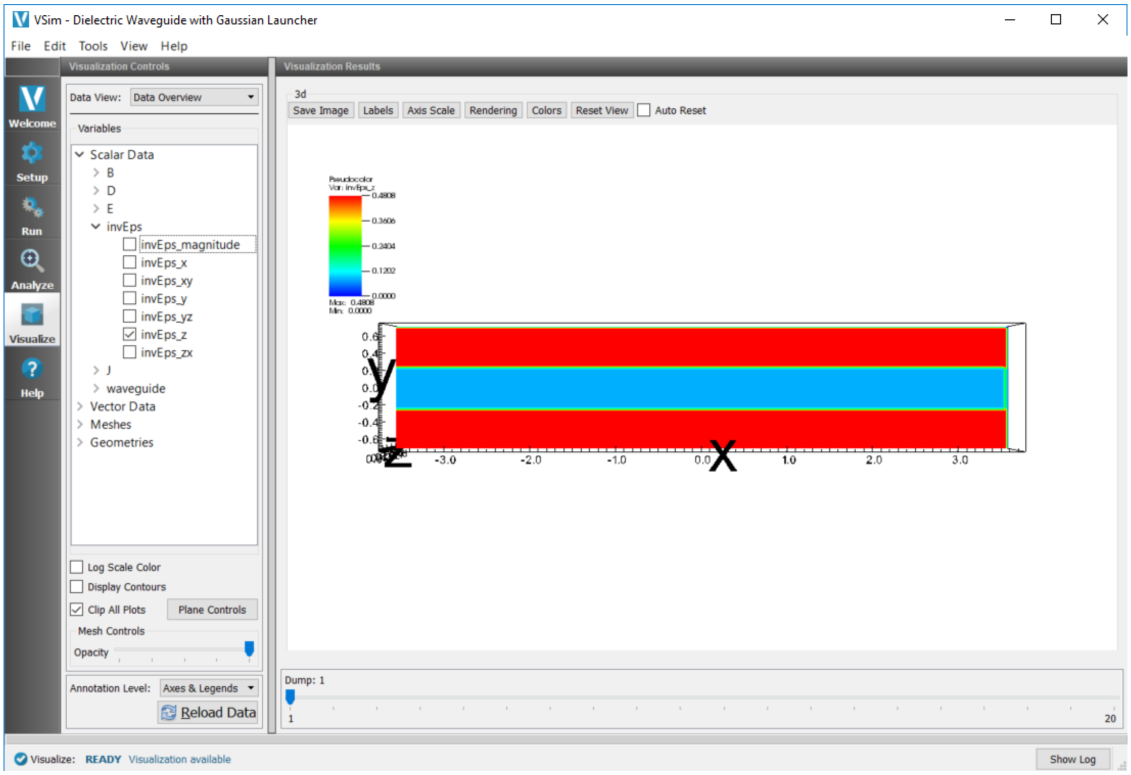
A useful visualization of the dielectric waveguide would be to view the Z component of the B field to qualitatively see the mode propagate down the waveguide.

- Near the top left corner of the window, make sure *Data View* is set to *Data Overview*.
- Expand *Scalar Data*, expand *B*, and select *B_z*
- In the controls below the variables frame, select *Clip All Plots*.
- In the top of the screen, press the button that's titled *Colors*, check the Fix Minimum and Fix Maximum buttons, and input {-5e-08, 5e-08} for the min and max, respectively.

Select the highest dump number on the lower right of the screen using the slide bar. Fig. 3.61 shows an example of what one should expect if one has run the simulation for enough cycles.

Further Experiments

One can experiment by changing constants or introducing a different signal to drive the waveguide.



Data View: Data Overview

☑ invEps_z

☐ Log Scale Color

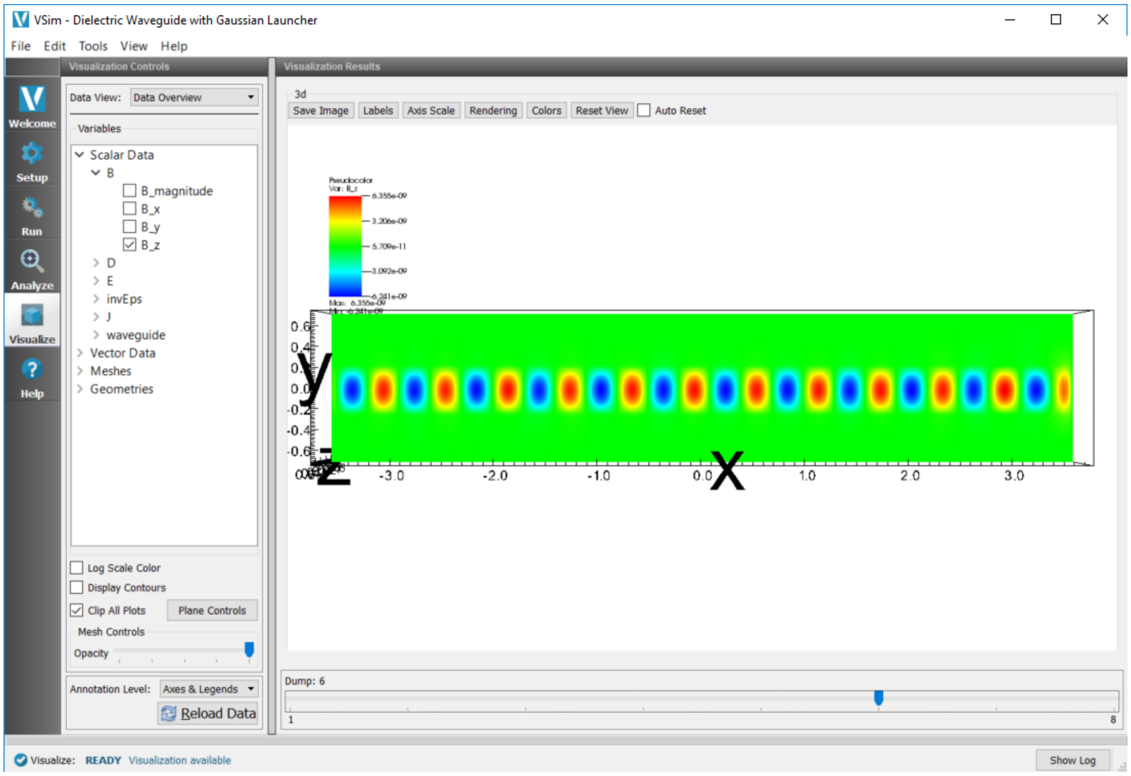
☐ Display Contours

☑ Clip All Plots

Annotation Level: Axes & Legends

Dump: 1

Fig. 3.60: Visualization of inverse epsilon field's Z component



Data View: Data Overview

☑ B_z

☐ Log Scale Color

☐ Display Contours

☑ Clip All Plots

Annotation Level: Axes & Legends

Dump: 6

Fig. 3.61: Visualization of the B field's Z component

3.5.3 Dielectric Waveguide Mode Calculation (dielectricWaveguideModeCalc.sdf)

Keywords:

`Mode Extraction, Photonic Waveguide, Guided Mode, Semiconductor`

Problem Description

This example demonstrates running a quasi-2D simulation of a rectangular dielectric waveguide and extracting the frequency and fields of a guided mode using the filter diagonalization method. The waveguide axis runs parallel to the x-axis, and is surrounded by Silica in the y and z directions. The y and z boundaries are set to absorb radiation. The x simulation boundaries are phase-shifting; i.e., fields on the +x boundary are set to the values of the fields on the -x boundary multiplied by $e^{i\phi}$.

Eigenmodes in such a simulation have the form:

$$\mathbf{E}(\mathbf{x}, t) = \mathbf{E}(y, z)e^{i(kx-\omega t)}$$

where the wavenumbers k that may be simulated are given by $kL_x = \phi + 2\pi n$, where n is any integer and L_x is the length of the simulation in the x direction (in this case, two grid cells; or $L_x = 2\Delta x$). Since we will be searching for the lowest-frequency guided modes, we can safely assume $n = 0$ and set $\phi = kL_x$ for a desired k.

The "modal index" or "effective index of refraction" of a waveguide mode is given by $\bar{n} = k/k_0$ where $k_0 = \omega/c$. If the waveguide has index of refraction n_w and the cladding $n_c < n_w$, then a *guided* mode will have a modal index in the range, $n_c < \bar{n} < n_w$.

This simulation can be performed with a VSimEM license.

Opening the Simulation

The Dielectric Waveguide example is accessed from within VSimComposer by the following actions:

- Select the *New → From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Photonics* option.
- Select *Dielectric Waveguide Mode Calculation* and press the *Choose* button.
- In the resulting dialog, create a *New Folder* if desired, and press the *Save* button to create a copy of this example.

Simulation Properties

Constants

This example contains a number of constants defined to make the simulation easily modifiable. The user should avoid altering constants not mentioned below.

LENGTH_UNIT: This is chosen to make the size of the system of order unity, as sometimes helps with the CSG construction.

WAVELENGTH_VAC: The estimated vacuum wavelength (frequency divided by the speed of light) for the eigenmode of interest.

N_EFF_EST: The estimated modal index of the eigenmode of interest; it should have a value between n_c and n_w.

RESOLUTION: The number of cells per estimated wavelength in the propagation (x) direction.

HALF_BANDWIDTH: The relative amount above and below the estimated frequency of the mode for excitation. Found modes should be in this range to be accurate.

SUPPRESSION_FACTOR: a fall of by this factor is considered zero.

CFL_FACTOR: The time step will be chosen to be this times the limit for numerical stability.

PERMITTIVITY_BACKGROUND: This value of electrical permittivity is assigned to the space outside the waveguide. It is set to Silica.

Parameters

Derived from the above are other parameters in the simulation. Some examples:

KAY: the estimated wavenumber, given by N_EFF_EST*(TWOPI/(WAVELENGTH_VAC/LENGTH_UNIT)). Here one sees a conversion using LENGTH_UNIT.

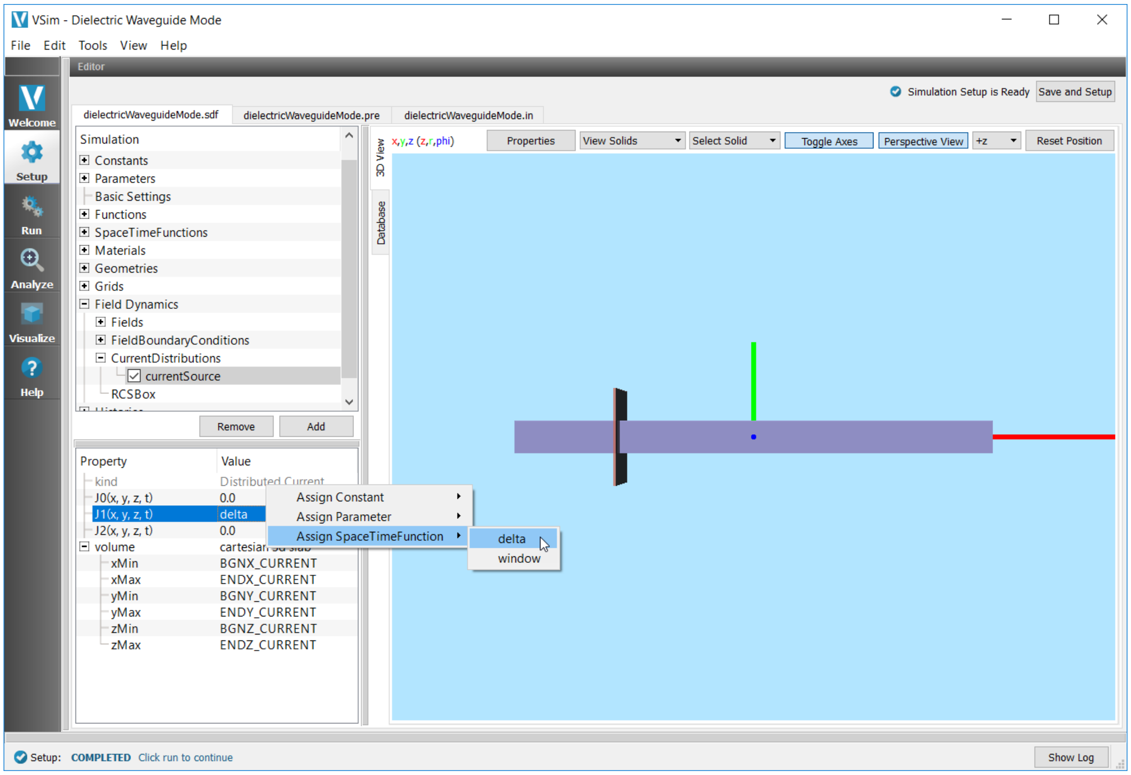
PHASE: is calculated as described at the beginning.

HEIGHT_WAVEGUIDE: .22e-6/LENGTH_UNIT: is the actual height scaled by *LENGTH_UNIT*.

NSTEPS_EXCITE: is the computed number of steps to excite the simulation with the desired frequency content.

Setting up the Broad-band Excitation

As delivered, the system is set up for exciting a broad-band with a delta-function in time. A history (eHist) has been added so that we can see the frequency content of the excitation. A *delta* current excitation has been selected. This is shown in Fig. 3.62.



Simulation Setup is Ready

Property	Value
kind	Distributed Current
J0(x, y, z, t)	0.0
J1(x, y, z, t)	delta
J2(x, y, z, t)	0.0
volume	cartesian 3d slab
xMin	BGNX_CURRENT
xMax	ENDX_CURRENT
yMin	BGNY_CURRENT
yMax	ENDY_CURRENT
zMin	BGNZ_CURRENT
zMax	ENDZ_CURRENT

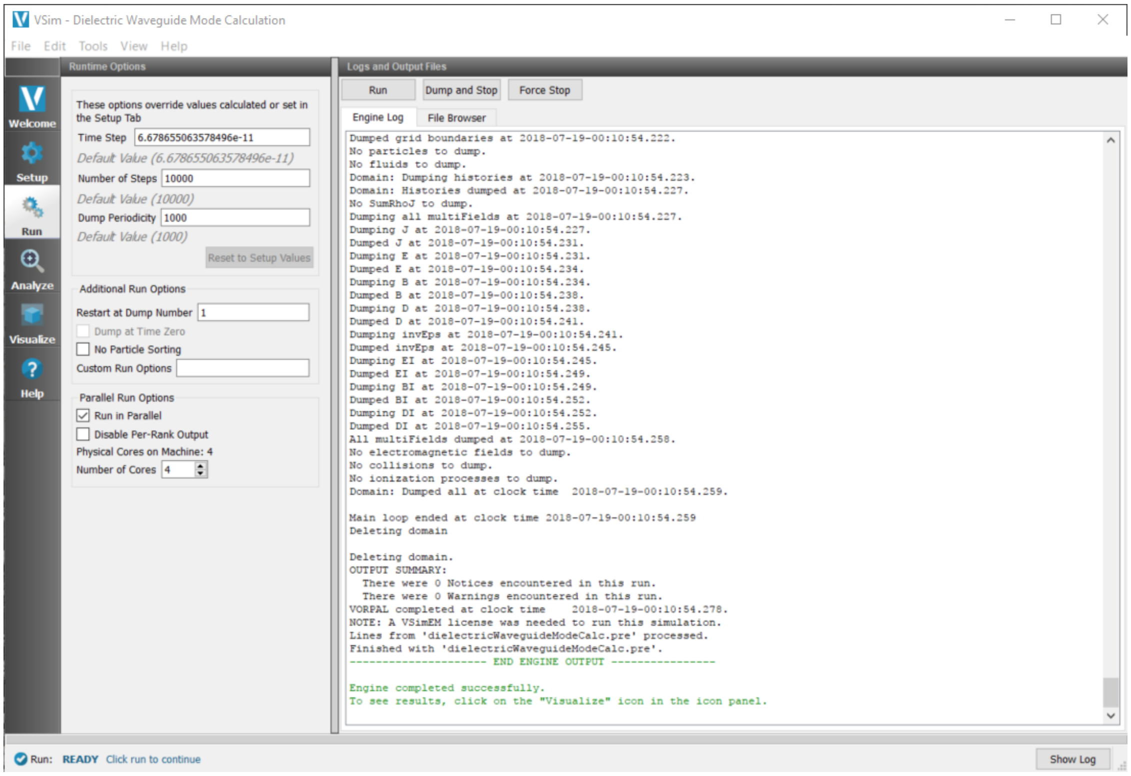
Setup: COMPLETED Click run to continue

Fig. 3.62: Choosing the delta-function current source

Running the Broad-band Excitation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the *Run* button in the left column of buttons. You will be asked to Save. Click *Save* upon the request to save.
- To run the file, click on the *Run* button in the upper left corner of the right pane. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." The result is shown in Fig. 3.63.



These options override values calculated or set in the Setup Tab

Time Step 6.678655063578496e-11

Default Value (6.678655063578496e-11)

Number of Steps 10000

Default Value (10000)

Dump Periodicity 1000

Default Value (1000)

Additional Run Options

Restart at Dump Number 1

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Dumped grid boundaries at 2018-07-19-00:10:54.222.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-07-19-00:10:54.223.
Domain: Histories dumped at 2018-07-19-00:10:54.227.
No SumRhoJ to dump.
Dumping all multiFields at 2018-07-19-00:10:54.227.
Dumping J at 2018-07-19-00:10:54.227.
Dumped J at 2018-07-19-00:10:54.231.
Dumping E at 2018-07-19-00:10:54.231.
Dumped E at 2018-07-19-00:10:54.234.
Dumping B at 2018-07-19-00:10:54.234.
Dumped B at 2018-07-19-00:10:54.238.
Dumping D at 2018-07-19-00:10:54.238.
Dumped D at 2018-07-19-00:10:54.241.
Dumping invEps at 2018-07-19-00:10:54.241.
Dumped invEps at 2018-07-19-00:10:54.245.
Dumping EI at 2018-07-19-00:10:54.245.
Dumped EI at 2018-07-19-00:10:54.249.
Dumping BI at 2018-07-19-00:10:54.249.
Dumped BI at 2018-07-19-00:10:54.252.
Dumping DI at 2018-07-19-00:10:54.252.
Dumped DI at 2018-07-19-00:10:54.255.
All multiFields dumped at 2018-07-19-00:10:54.258.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-19-00:10:54.259.

Main loop ended at clock time 2018-07-19-00:10:54.259
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-07-19-00:10:54.278.
NOTE: A VSimEM license was needed to run this simulation.
Lines from 'dielectricWaveguideModeCalc.pre' processed.
Finished with 'dielectricWaveguideModeCalc.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

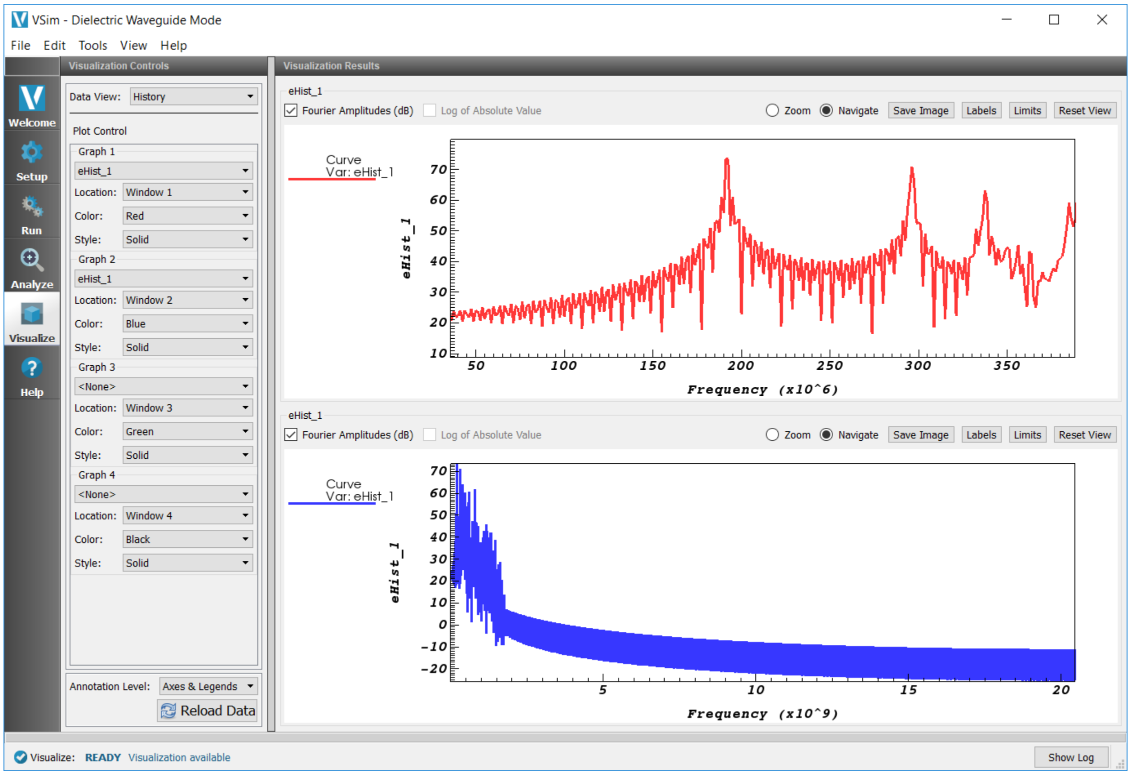
Run: **READY** Click run to continue

Fig. 3.63: Exciting the mode with a broad-band current.

Visualizing the broad-band excitation simulation

- Proceed to the Visualize Window by pressing the *Visualize* button in the left column of buttons.
- From the *Data View* dropdown menu select *History*
- Set Graphs 1 and 2 to eHist_1 (the y component).
- Set Graph 3 to <None>.
- In the both graphs, click FFT.
- In the upper graph, select the *Zoom* radio button, and select a box around the lower peak. The result is shown in Fig. 3.64.

Both spectra show that there is a mode at around *190e6 Hz*, with many more modes above. We will excite the waveguide with a window of frequencies centered about this mode.



Data View: History

Plot Control

Graph 1
eHist_1
Location: Window 1
Color: Red
Style: Solid

Graph 2
eHist_1
Location: Window 2
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 3
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

☑ Fourier Amplitudes (dB) □ Log of Absolute Value ○ Zoom ● Navigate

Visualize: READY Visualization available

Fig. 3.64: Spectrum seen from the broad-band excitation.

The Narrow-band Excitation

Now return to *Setup*, for the narrow-band excitation. HALF_BANDWIDTH is set to 0.1, so that the excitation will start to fall off at FREQ_MID +- 10%. To change to the narrow-band simulation, choose window for *J1* under *currentSource*. Under *Basic Settings*, change *steps between dumps* to 0. This will make 1 dump at the end of the run. Under *Parameters*, check the value of NSTEPS_EXCITE. Now on the *Run* window, set the *Number of Steps* to somewhere around the value of NSTEPS_EXCITE. The *Dump Periodicity* should be set to 0. Run the simulation and proceed to the visualization window, setting it up as before. The result is shown in Fig. 3.65.

As seen in Fig. 3.65, our excitation has isolated a single frequency peak, which we can now find precisely using mode extraction.

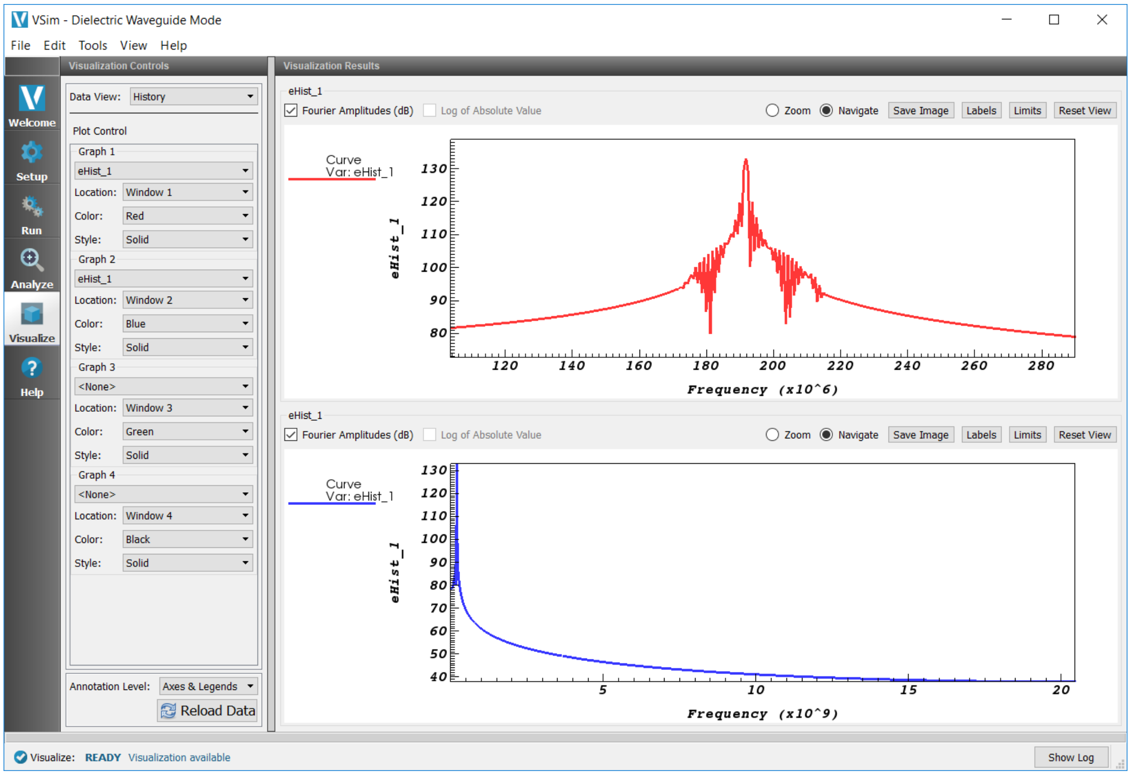
Mode extraction by operator

Mode extraction by operator requires that one have sequences of 3 dumps during the time of free oscillation. To get these, we return to the Setup window, and under *Basic Settings*, we set *dump in groups of* to 3 and set *steps between dumps* to 20.

We next move to the Run window, make sure *Dump at Time Zero* is unclicked and put in 1 for *Restart at Dump Number*. We set the *Number of Steps* to 202 (dumps will be at multiples of 20 + 0, 1, or 2). We then empty out (blank) the *Dump Periodicity* text box and immediately hit *Run*.

Once the run finishes and the fields are dumped, proceed to the analysis screen by clicking the *Analyze* navigation button. Select *extractModesViaOperator.py* and click *Open*. Change the following options:

- *electricField*: D,DI
- *magneticField*: B,BI



Data View: History

Graph 1: eHist_1, Location: Window 1, Color: Red, Style: Solid

Graph 2: eHist_1, Location: Window 2, Color: Blue, Style: Solid

Graph 3: <None>, Location: Window 3, Color: Green, Style: Solid

Graph 4: <None>, Location: Window 4, Color: Black, Style: Solid

Annotation Level: Axes & Legends

eHist_1: ☑ Fourier Amplitudes (dB) □ Log of Absolute Value ○ Zoom ● Navigate

eHist_1: ☑ Fourier Amplitudes (dB) □ Log of Absolute Value ○ Zoom ● Navigate

Visualize: READY Visualization available

Fig. 3.65: Spectrum seen from the narrow-band excitation.

- *operator*: d2dt2
- *dumpRange*: 1:
- *cellSamples*: 0:2,25:36:5,25:36:5

Also check *Overwrite Existing Files*. Run the analyzer by clicking *Analyze* button in the upper right corner. The analyzer output should resemble the following image.

The output in Fig. 3.66 shows that Mode 0 (on the first line) has the smallest relative error.

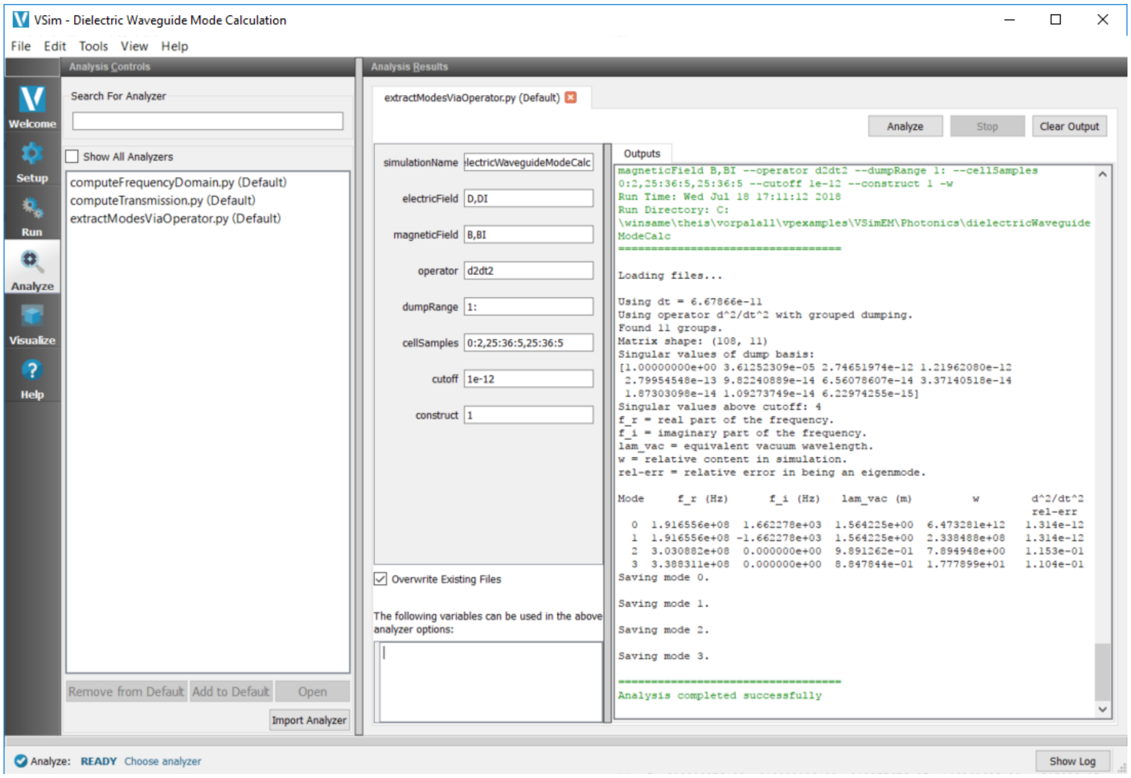
Visualizing the results

After performing the above actions proceed to the Visualize window by pressing the *Visualize* button in the left column of buttons. You may need to *Reload Data* (bottom left). Visualize an eigenmode by following these steps:

- From the *Data View* dropdown select *Data Overview*.
- Expand *Scalar Data*, expand *EigenD*, and select *EigenD_1*.
- Below the visualization, select the dump number (0) that corresponds to a mode number with the lowest operator error (Mode 0).

The resulting visualization pane should resemble Fig. 3.67.

One can select other components of the electric field to see how they vary for this eigenmode. The profile of the eigenmode is now saved in a vsh5 file in the folder where the simulation was run.



computeFrequencyDomain.py (Default)
computeTransmission.py (Default)
extractModesViaOperator.py (Default)

extractModesViaOperator.py (Default)

simulationName	lectricWaveguideModeCalc
electricField	D,DI
magneticField	B,BI
operator	d2dt2
dumpRange	1:
cellSamples	0:2,25:36:5,25:36:5
cutoff	1e-12
construct	1

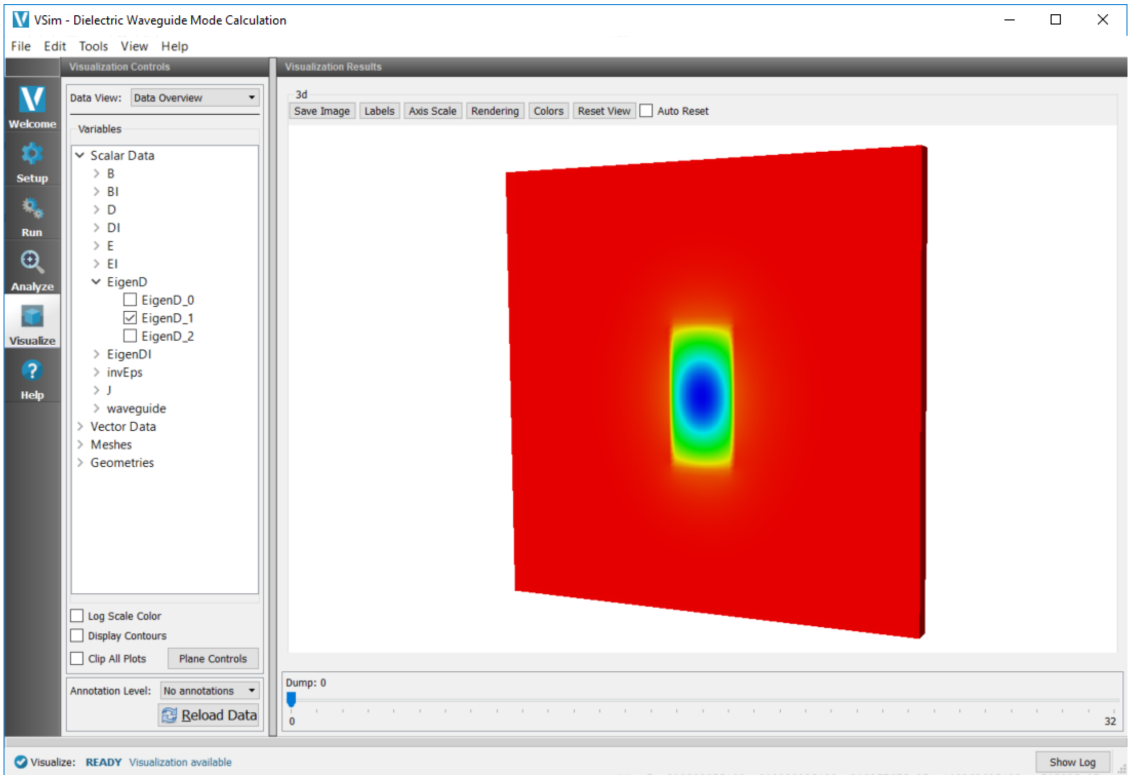
☑ Overwrite Existing Files

The following variables can be used in the above analyzer options:

```
magneticField B,BI --operator d2dt2 --dumpRange 1: --cellSamples
0:2,25:36:5,25:36:5 --cutoff 1e-12 --construct 1 -w
Run Time: Wed Jul 18 17:11:12 2018
Run Directory: C:
\winsame\theis\vorpalall\vpexamples\VSimEM\Photonics\dielectricWaveguide
ModeCalc
==================================

Loading files...

Using dt = 6.67866e-11
Using operator d^2/dt^2 with grouped dumping.
Found 11 groups.
Matrix shape: (108, 11)
Singular values of dump basis:
[1.00000000e+00 3.61252309e-05 2.74651974e-12 1.21962080e-12
 2.79954548e-13 9.82240889e-14 6.56078607e-14 3.37140518e-14
 1.87303098e-14 1.09273749e-14 6.22974255e-15]
Singular values above cutoff: 4
f_r = real part of the frequency.
f_i = imaginary part of the frequency.
lam_vac = equivalent vacuum wavelength.
w = relative content in simulation.
rel-err = relative error in being an eigenmode.

Mode     f_r (Hz)      f_i (Hz)    lam_vac (m)         w        d^2/dt^2
                                                                 rel-err
  0  1.916556e+08  1.662278e+03  1.564225e+00  6.473281e+12  1.314e-12
  1  1.916556e+08 -1.662278e+03  1.564225e+00  2.338488e+08  1.314e-12
  2  3.030882e+08  0.000000e+00  9.891262e-01  7.894948e+00  1.153e-01
  3  3.388311e+08  0.000000e+00  8.847844e-01  1.777899e+01  1.104e-01
Saving mode 0.

Saving mode 1.

Saving mode 2.

Saving mode 3.

==================================
Analysis completed successfully
```

Analyze: READY Choose analyzer

Fig. 3.66: Output from the mode extraction script.



Data View: Data Overview

□ EigenD_0
☑ EigenD_1
□ EigenD_2

Annotation Level: No annotations

Dump: 0

Visualize: READY Visualization available

Fig. 3.67: Color map of the Dy component of the guided mode.

Further Experiments

One can run a full convergence study of eigenmode frequencies by altering the RESOLUTION parameter in the Setup Window and re-running the simulation and mode extraction script.

3.5.4 Dielectric Waveguide Mode Calculation using Point Permittivity (dielectricWaveguideModeCalcPP.sdf)

Keywords:

`Mode Extraction, Photonic Waveguide, Guided Mode, Semiconductor`

Problem Description

This example demonstrates running a quasi-2D simulation of a rectangular dielectric waveguide and extracting the frequency and fields of a guided mode using the filter diagonalization method. The waveguide axis runs parallel to the x-axis, and is surrounded by Silica in the y and z directions. The y and z boundaries are set to absorb radiation. This simulation uses point permittivity as opposed to permittivity averaging, which is done in Dielectric Waveguide Mode Calculation. The x simulation boundaries are phase-shifting; i.e., fields on the +x boundary are set to the values of the fields on the -x boundary multiplied by $e^{i\phi}$.

Eigenmodes in such a simulation have the form:

$$\mathbf{E}(\mathbf{x},t) = \mathbf{E}(y,z)e^{i(kx-\omega t)}$$

where the wavenumbers k that may be simulated are given by $kL_x = \phi + 2\pi n$, where n is any integer and L_x is the length of the simulation in the x direction (in this case, two grid cells; or $L_x = 2\Delta x$). Since we will be searching for the lowest-frequency guided modes, we can safely assume $n = 0$ and set $\phi = kL_x$ for a desired k.

The "modal index" or "effective index of refraction" of a waveguide mode is given by $\bar{n} = k/k_0$ where $k_0 = \omega/c$. If the waveguide has index of refraction n_w and the cladding $n_c < n_w$, then a *guided* mode will have a modal index in the range, $n_c < \bar{n} < n_w$.

This simulation can be performed with a VSimEM license.

Opening the Simulation

The Dielectric Waveguide example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Photonics* option.
- Select *Dielectric Waveguide Mode Calculation using Point Permittivity* and press the *Choose* button.
- In the resulting dialog, create a *New Folder* if desired, and press the *Save* button to create a copy of this example.

Simulation Properties

Constants

This example contains a number of constants defined to make the simulation easily modifiable. The user should avoid altering constants not mentioned below.

LENGTH_UNIT: This is chosen to make the size of the system of order unity, as sometimes helps with the CSG construction.

WAVELENGTH_VAC: The estimated vacuum wavelength (frequency divided by the speed of light) for the eigenmode of interest.

N_EFF_EST: The estimated modal index of the eigenmode of interest; it should have a value between n_c and n_w.

RESOLUTION: The number of cells per estimated wavelength in the propagation (x) direction.

HALF_BANDWIDTH: The relative amount above and below the estimated frequency of the mode for excitation. Found modes should be in this range to be accurate.

SUPPRESSION_FACTOR: a fall of by this factor is considered zero.

CFL_FACTOR: The time step will be chosen to be this times the limit for numerical stability.

PERMITTIVITY_BACKGROUND: This value of electrical permittivity is assigned to the space outside the waveguide. It is set to Silica.

Parameters

Derived from the above are other parameters in the simulation. Some examples:

KAY: the estimated wavenumber, given by N_EFF_EST*(TWOPI/(WAVELENGTH_VAC/LENGTH_UNIT)). Here one sees a conversion using LENGTH_UNIT.

PHASE: is calculated as described at the beginning.

HEIGHT_WAVEGUIDE: .22e-6/LENGTH_UNIT: is the actual height scaled by LENGTH_UNIT.

NSTEPS_EXCITE: is the computed number of steps to excite the simulation with the desired frequency content.

Setting up the Broad-band Excitation

As delivered, the system is set up for exciting a broad-band with a delta-function in time. A history (eHist) has been added so that we can see the frequency content of the excitation. A *delta* current excitation has been selected. This is shown in Fig. 3.68.

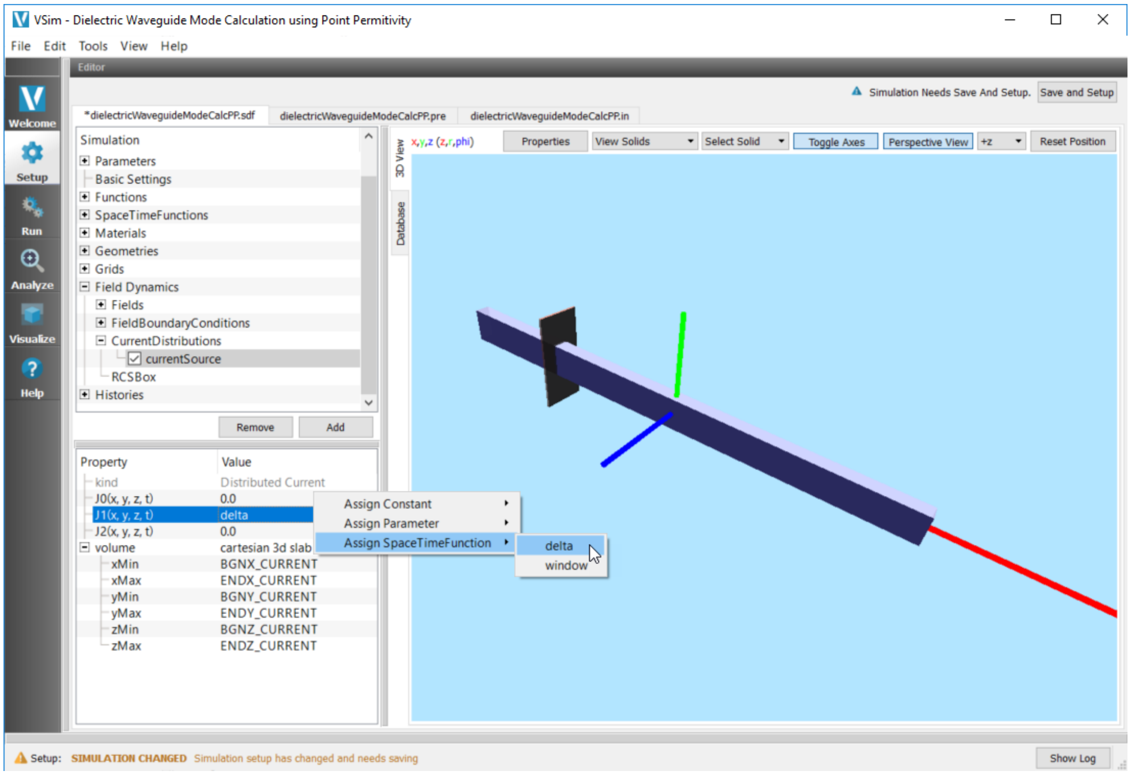
Running the Broad-band Excitation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the *Run* button in the left column of buttons. You will be asked to Save. Click *Save* upon the request to save.
- To run the file, click on the *Run* button in the upper left corner of the right pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” The result is shown in Fig. 3.69.

Visualizing the Broad-band Excitation

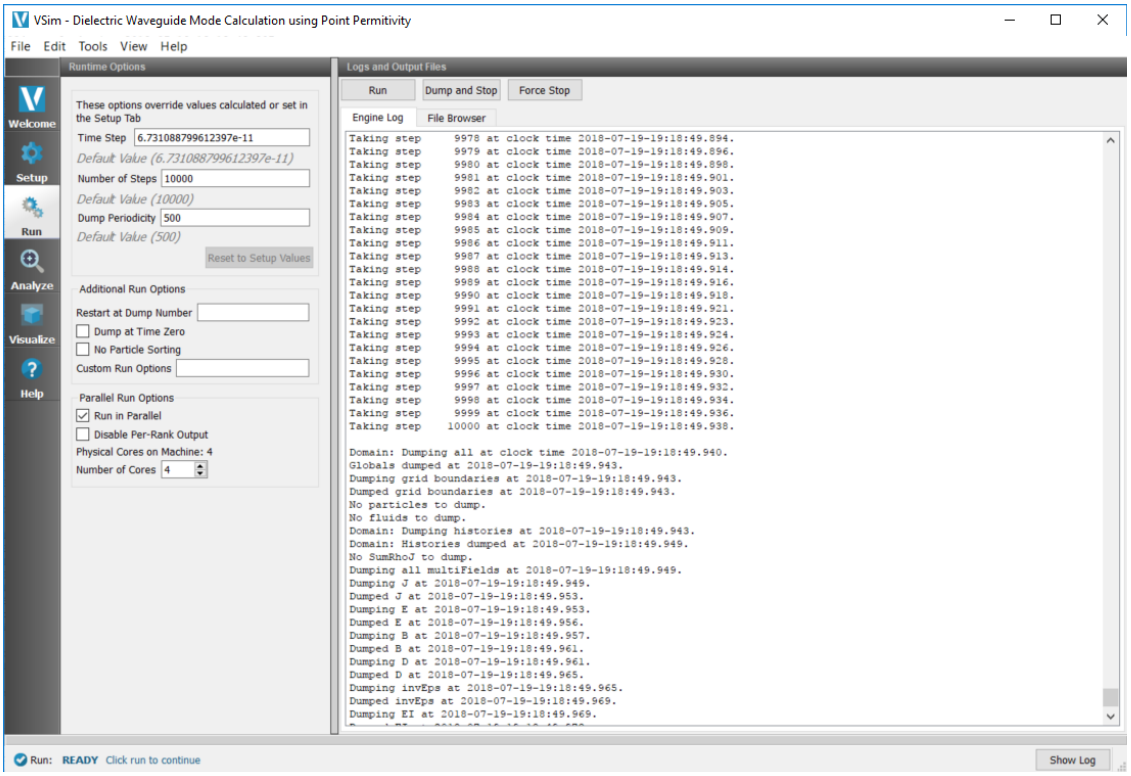
- Proceed to the Visualize Window by pressing the *Visualize* button in the left column of buttons.
- From the *Data View* dropdown menu select *History*
- Set Graphs 1 and 2 to eHist_1 (the y component).
- Set Graph 3 to <None>.
- In the both graphs, click FFT.



Property	Value
kind	Distributed Current
J0(x, y, z, t)	0.0
J1(x, y, z, t)	delta
J2(x, y, z, t)	0.0
volume	cartesian 3d slab
xMin	BGNX_CURRENT
xMax	ENDX_CURRENT
yMin	BGNY_CURRENT
yMax	ENDY_CURRENT
zMin	BGNZ_CURRENT
zMax	ENDZ_CURRENT

Setup: SIMULATION CHANGED Simulation setup has changed and needs saving

Fig. 3.68: Choosing the delta-function current source



These options override values calculated or set in the Setup Tab

Time Step 6.731088799612397e-11
Default Value (6.731088799612397e-11)
Number of Steps 10000
Default Value (10000)
Dump Periodicity 500
Default Value (500)

Additional Run Options
Restart at Dump Number
□ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

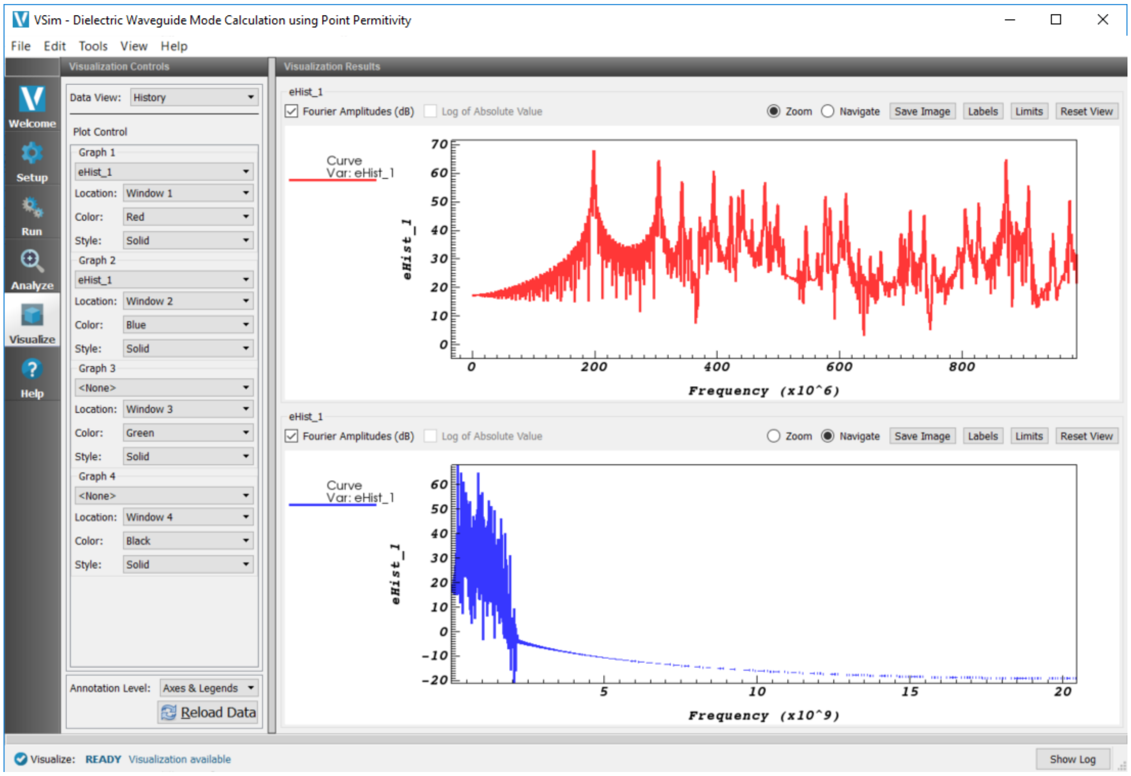
Parallel Run Options
☑ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
Taking step    9978 at clock time 2018-07-19-19:18:49.894.
Taking step    9979 at clock time 2018-07-19-19:18:49.896.
Taking step    9980 at clock time 2018-07-19-19:18:49.898.
Taking step    9981 at clock time 2018-07-19-19:18:49.901.
Taking step    9982 at clock time 2018-07-19-19:18:49.903.
Taking step    9983 at clock time 2018-07-19-19:18:49.905.
Taking step    9984 at clock time 2018-07-19-19:18:49.907.
Taking step    9985 at clock time 2018-07-19-19:18:49.909.
Taking step    9986 at clock time 2018-07-19-19:18:49.911.
Taking step    9987 at clock time 2018-07-19-19:18:49.913.
Taking step    9988 at clock time 2018-07-19-19:18:49.914.
Taking step    9989 at clock time 2018-07-19-19:18:49.916.
Taking step    9990 at clock time 2018-07-19-19:18:49.918.
Taking step    9991 at clock time 2018-07-19-19:18:49.921.
Taking step    9992 at clock time 2018-07-19-19:18:49.923.
Taking step    9993 at clock time 2018-07-19-19:18:49.924.
Taking step    9994 at clock time 2018-07-19-19:18:49.926.
Taking step    9995 at clock time 2018-07-19-19:18:49.928.
Taking step    9996 at clock time 2018-07-19-19:18:49.930.
Taking step    9997 at clock time 2018-07-19-19:18:49.932.
Taking step    9998 at clock time 2018-07-19-19:18:49.934.
Taking step    9999 at clock time 2018-07-19-19:18:49.936.
Taking step   10000 at clock time 2018-07-19-19:18:49.938.

Domain: Dumping all at clock time 2018-07-19-19:18:49.940.
Globals dumped at 2018-07-19-19:18:49.943.
Dumping grid boundaries at 2018-07-19-19:18:49.943.
Dumped grid boundaries at 2018-07-19-19:18:49.943.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-07-19-19:18:49.943.
Domain: Histories dumped at 2018-07-19-19:18:49.949.
No SumRhoJ to dump.
Dumping all multiFields at 2018-07-19-19:18:49.949.
Dumping J at 2018-07-19-19:18:49.949.
Dumped J at 2018-07-19-19:18:49.953.
Dumping E at 2018-07-19-19:18:49.953.
Dumped E at 2018-07-19-19:18:49.956.
Dumping B at 2018-07-19-19:18:49.957.
Dumped B at 2018-07-19-19:18:49.961.
Dumping D at 2018-07-19-19:18:49.961.
Dumped D at 2018-07-19-19:18:49.965.
Dumping invEps at 2018-07-19-19:18:49.965.
Dumped invEps at 2018-07-19-19:18:49.969.
Dumping EI at 2018-07-19-19:18:49.969.
```

Run: READY Click run to continue

Fig. 3.69: Run window at the end of a successful run.

- In the upper graph, select the *Zoom* radio button, and select a box around the lower peak. The result is shown in Fig. 3.70.



VSim - Dielectric Waveguide Mode Calculation using Point Permitivity

Data View: History

Plot Control

Graph 1: eHist_1, Location: Window 1, Color: Red, Style: Solid

Graph 2: eHist_1, Location: Window 2, Color: Blue, Style: Solid

Graph 3: <None>, Location: Window 3, Color: Green, Style: Solid

Graph 4: <None>, Location: Window 4, Color: Black, Style: Solid

Annotation Level: Axes & Legends

eHist_1: ☑ Fourier Amplitudes (dB) □ Log of Absolute Value ● Zoom ○ Navigate

eHist_1: ☑ Fourier Amplitudes (dB) □ Log of Absolute Value ○ Zoom ● Navigate

Visualize: READY Visualization available

Fig. 3.70: Spectrum seen from the broad-band excitation.

Both spectra show that there is a mode at around *190e6 Hz*, with many more modes above. We will excite the waveguide with a window of frequencies centered about the lowest mode.

The Narrow-band Excitation

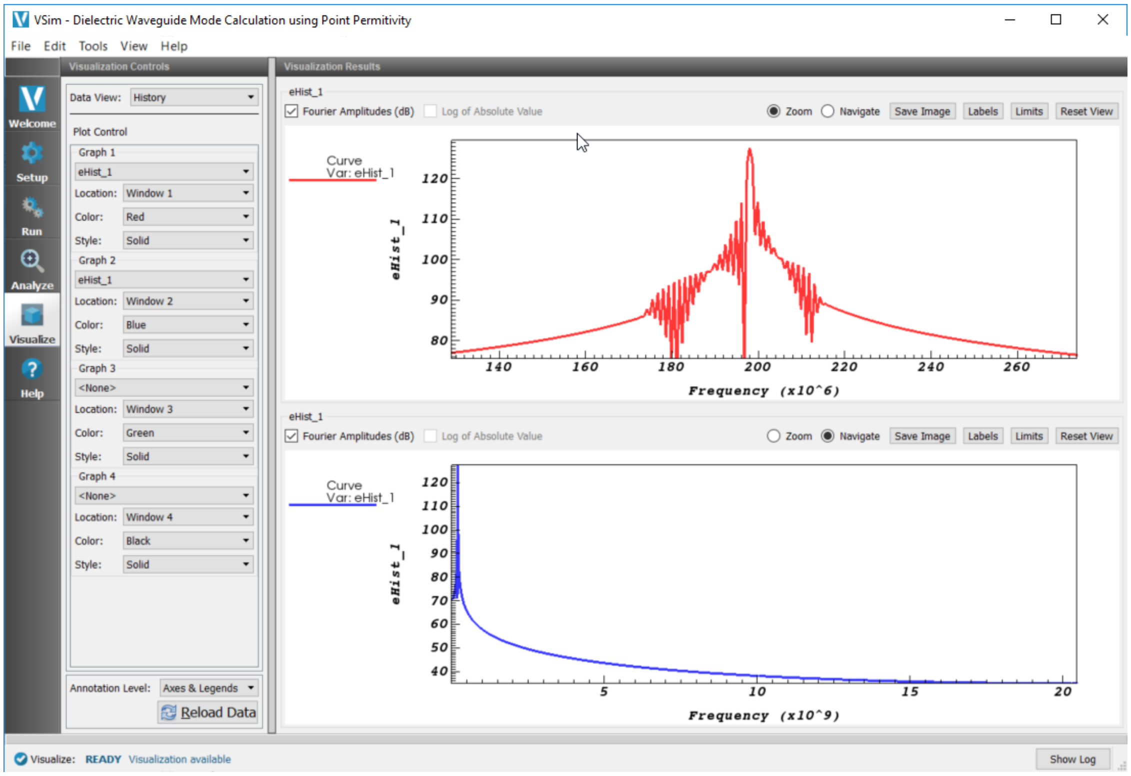
Now return to *Setup*, for the narrow-band excitation. HALF_BANDWIDTH is set to 0.1, so that the excitation will start to fall off at FREQ_MID +- 10%. To change to the narrow-band simulation, choose window for *J1* under *currentSource*. Under *Basic Settings*, change *steps between dumps* to 0. This will make 1 dump at the end of the run. Under *Parameters*, check the value of NSTEPS_EXCITE. Now on the Run window, set the *Number of Steps* to somewhere around the value of NSTEPS_EXCITE. The *Dump Periodicity* should be set to 0. Run the simulation and proceed to the visualization window, setting it up as before. The result is shown in Fig. 3.71.

As seen in Fig. 3.71, our excitation has isolated a single frequency peak, which we can now find precisely using mode extraction.

Mode Extraction by Operator

Mode extraction by operator requires that one have sequences of 3 dumps during the time of free oscillation. To get these, we return to the Setup window, and under *Basic Settings*, we set *dump in groups of* to 3 and set *steps between dumps* to 20.

We next move to the Run window, make sure *Dump at Time Zero* is unclicked and put in 1 for *Restart at Dump Number*. We set the *Number of Steps* to 202 (dumps will be at multiples of 20 + 0, 1, or 2). We then empty out (blank) the *Dump Periodicity* text box and hit *Run*.



Data View: History

Graph 1: eHist_1, Location: Window 1, Color: Red, Style: Solid

Graph 2: eHist_1, Location: Window 2, Color: Blue, Style: Solid

Graph 3: <None>, Location: Window 3, Color: Green, Style: Solid

Graph 4: <None>, Location: Window 4, Color: Black, Style: Solid

Annotation Level: Axes & Legends

eHist_1: ☑ Fourier Amplitudes (dB) □ Log of Absolute Value ● Zoom ○ Navigate

eHist_1: ☑ Fourier Amplitudes (dB) □ Log of Absolute Value ○ Zoom ● Navigate

Visualize: READY Visualization available

Fig. 3.71: Spectrum seen from the narrow-band excitation.

Once the run finishes and the fields are dumped, proceed to the analysis screen by clicking the *Analyze* navigation button. Select *extractModesViaOperator.py* and click *Open*. Change the following options:

- *electricField*: D,DI
- *magneticField*: B,BI
- *operator*: d2dt2
- *dumpRange*: 1:
- *cellSamples*: 0:2,25:36:5,25:36:5

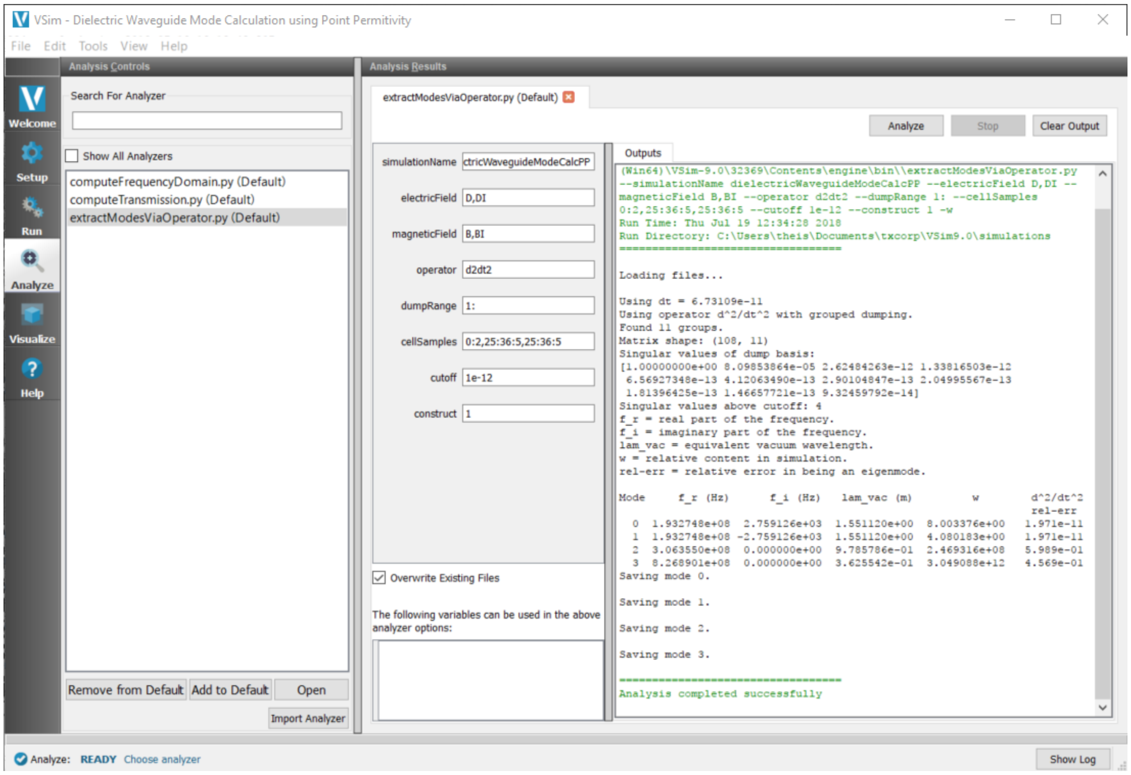
Also check *Overwrite Existing Files*. Run the analyzer by clicking *Analyze* button in the upper right corner. The analyzer output should resemble Fig. 3.72.

The output in Fig. 3.72 shows that Mode 0 (on the first line) has the smallest relative error.

Visualizing the Fundamental Mode

After performing the above actions proceed to the Visualize window by pressing the *Visualize* button in the left column of buttons. You may need to *Reload Data* (bottom left). Visualize an eigenmode by following these steps:

- From the *Data View* dropdown select *Data Overview*.
- Expand *Scalar Data*, expand *EigenD*, and select *EigenD_1*.
- Below the visualization, select the dump number (0) that corresponds to a mode number with the lowest operator error (Mode 0).



computeFrequencyDomain.py (Default)
computeTransmission.py (Default)
extractModesViaOperator.py (Default)

extractModesViaOperator.py (Default)

simulationName: ctricWaveguideModeCalcPP
electricField: D,DI
magneticField: B,BI
operator: d2dt2
dumpRange: 1:
cellSamples: 0:2,25:36:5,25:36:5
cutoff: 1e-12
construct: 1

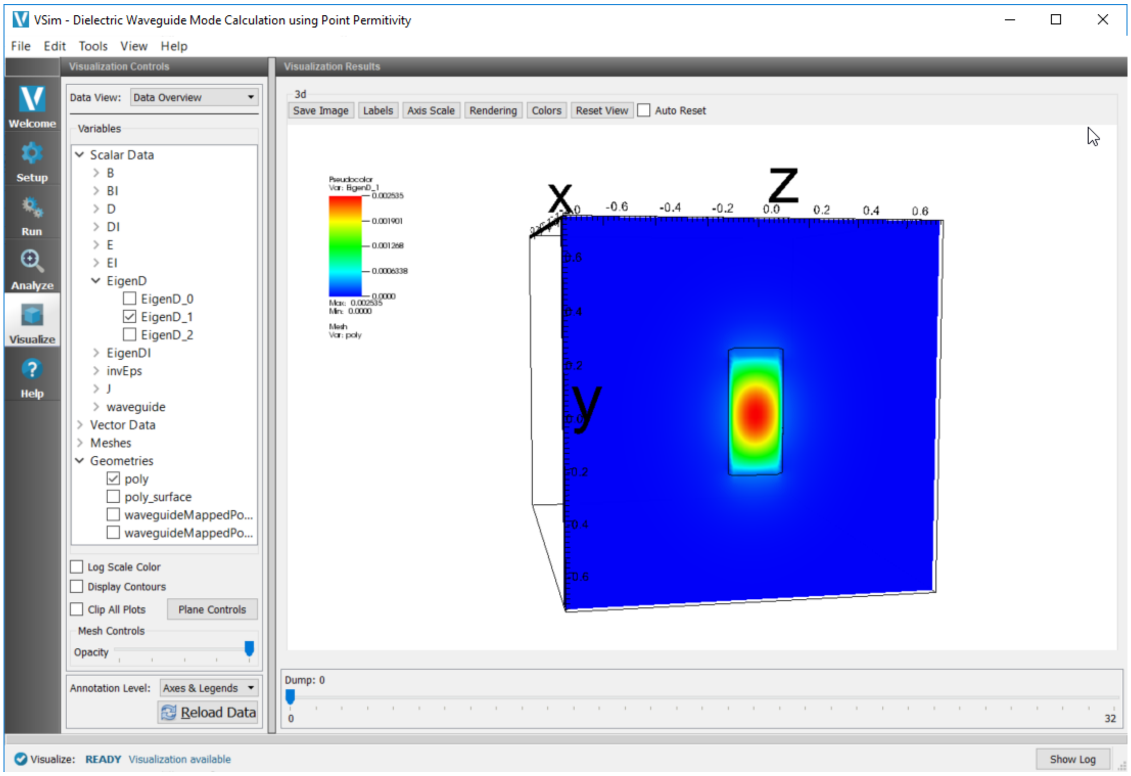
☑ Overwrite Existing Files

The following variables can be used in the above analyzer options:

```
(Win64)\VSim-9.0\32369\Contents\engine\bin\\extractModesViaOperator.py
--simulationName dielectricWaveguideModeCalcPP --electricField D,DI --
magneticField B,BI --operator d2dt2 --dumpRange 1: --cellSamples
0:2,25:36:5,25:36:5 --cutoff 1e-12 --construct 1 -w
Run Time: Thu Jul 19 12:34:28 2018
Run Directory: C:\Users\theis\Documents\txcorp\VSim9.0\simulations
==================================

Loading files...

Using dt = 6.73109e-11
Using operator d^2/dt^2 with grouped dumping.
Found 11 groups.
Matrix shape: (108, 11)
Singular values of dump basis:
[1.00000000e+00 8.09853864e-05 2.62484263e-12 1.33816503e-12
 6.56927348e-13 4.12063490e-13 2.90104847e-13 2.04995567e-13
 1.81396425e-13 1.46657721e-13 9.32459792e-14]
Singular values above cutoff: 4
f_r = real part of the frequency.
f_i = imaginary part of the frequency.
lam_vac = equivalent vacuum wavelength.
w = relative content in simulation.
rel-err = relative error in being an eigenmode.

Mode     f_r (Hz)      f_i (Hz)   lam_vac (m)         w        d^2/dt^2
                                                                rel-err
  0  1.932748e+08  2.759126e+03  1.551120e+00  8.003376e+00   1.971e-11
  1  1.932748e+08 -2.759126e+03  1.551120e+00  4.080183e+00   1.971e-11
  2  3.063550e+08  0.000000e+00  9.785786e-01  2.469316e+08   5.989e-01
  3  8.268901e+08  0.000000e+00  3.625542e-01  3.049088e+12   4.569e-01
Saving mode 0.

Saving mode 1.

Saving mode 2.

Saving mode 3.

==================================
Analysis completed successfully
```

Analyze: READY Choose analyzer

Fig. 3.72: Output from the mode extraction script.



Data View: Data Overview

Variables
- Scalar Data
  - B
  - BI
  - D
  - DI
  - E
  - EI
  - EigenD
    - ☐ EigenD_0
    - ☑ EigenD_1
    - ☐ EigenD_2
  - EigenDI
  - invEps
  - J
  - waveguide
- Vector Data
- Meshes
- Geometries
  - ☑ poly
  - ☐ poly_surface
  - ☐ waveguideMappedPo...
  - ☐ waveguideMappedPo...

☐ Log Scale Color
☐ Display Contours
☐ Clip All Plots
☐ Auto Reset

Annotation Level: Axes & Legends

Dump: 0 (0–32)

Visualize: READY Visualization available

Fig. 3.73: Color map of the Dy component of the guided mode.

The resulting visualization pane should resemble Fig. 3.73.

One can select other components of the electric field to see how they vary for this eigenmode. The profile of the eigenmode is now saved in a vsh5 file in the folder where the simulation was run. This mode is launched into the waveguide in the Dielectric Waveguide Mode Launch example.

Further Experiments

One can run a full convergence study of eigenmode frequencies by altering the RESOLUTION parameter in the Setup Window and re-running the simulation and mode extraction script.

3.5.5 Microring Resonator with Gaussian Launcher (microringResonatorGaussian.sdf)

Keywords:

`Microring Resonator, Unidirectional Mode Launcher, MAL, Guided Mode, Photonic Device, Semic`

Problem Description

The Microring Resonator consist of two straight Silicon waveguides and a Silicon waveguide ring that sits between the straight Waveguides. All three waveguides rest on top of a Silicon Dioxide slab. The rest of the simulation domain is set to vacuum. Matched Absorbing Layers (MALs) are used to dampen the E, B and D fields near the boundary of the simulation, this is a way to dampen reflected fields from the simulation boundaries.

An approximation of the fundamental guided mode profile is launched as a wide band pulse in the input waveguide (the waveguide in -y). This input signal is launching in the +x direction.

This simulation can be performed with a VSimEM license.

Opening the Simulation

The Microring Resonator example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Photonics* option.
- Select *Microring Resonator with Gaussian Launcher* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

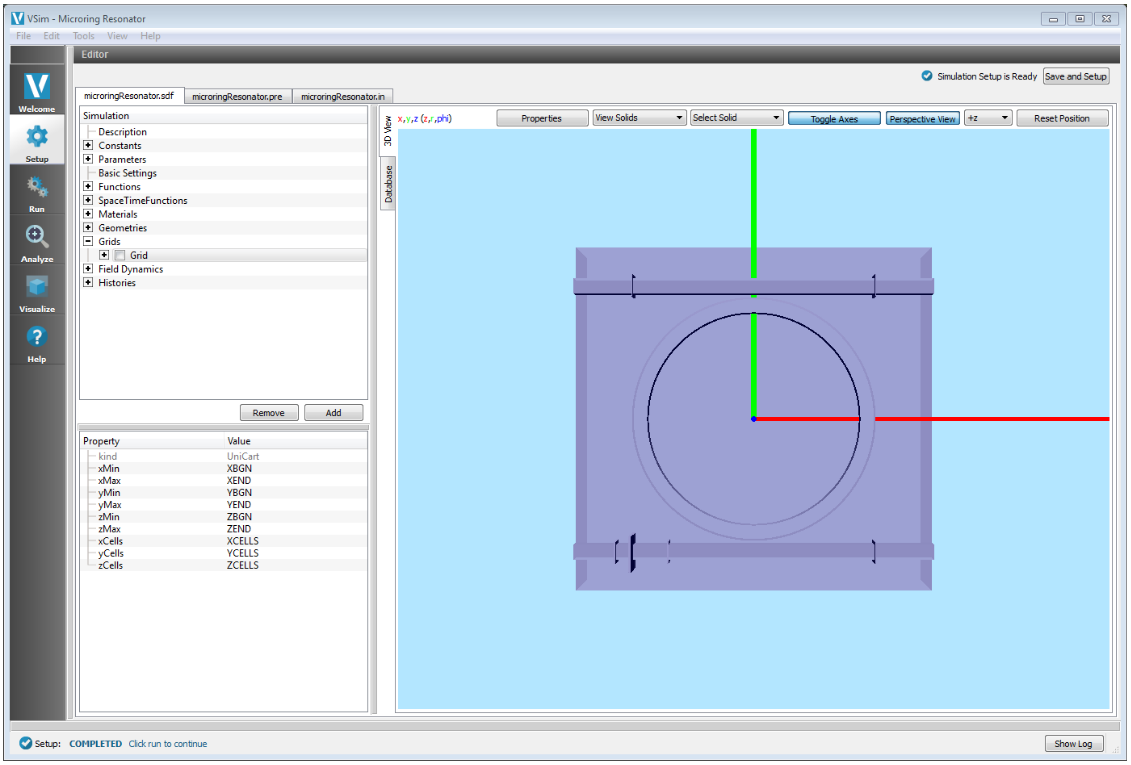
All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 3.74. You can expand the tree elements and navigate through the various properties. The right pane shows a 3D view of the geometry, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.

Simulation Properties

This example contains a number of *Constants* defined to make the simulation easily modifiable.

General Simulation Parameters (Constants):

- *WAVEL_MAX* = largest wavelength in the wide band signal



Simulation Setup is Ready

Property	Value
kind	UniCart
xMin	XBGN
xMax	XEND
yMin	YBGN
yMax	YEND
zMin	ZBGN
zMax	ZEND
xCells	XCELLS
yCells	YCELLS
zCells	ZCELLS

Setup: COMPLETED Click run to continue

Fig. 3.74: The Setup Window for The Microring Resonator Example

- *WAVEL_MIN* = smallest wavelength in the wide band signal
- *RESOLUTION* = inverse of the number of cells per wavelength
- *LENGTH_UNIT* = This scales the simulation by the value. DO NOT CHANGE THIS PARAMETER
- *WAVEGUIDE_PERMITTIVITY* = permittivity of the waveguides ensure this is equal to the permittivity set in the CSG.
- *WAVEL_SINGLE* = The wavelength used in the single frequency signal
- ***WAVEL_RESOLVE* = The wavelength used in calculation for the number of** cells in simulation.

General Simulation Parameters (Parameters):

- *RING_RADIUS* = radius of the ring in meters scaled by LENGTH_UNIT.
- *WIDTH_WAVEGUIDE* = Width of waveguides in meters scaled by LENGTH_UNIT.
- *HEIGHT_WAVEGUIDE* = Height of waveguides in meters scaled by LENGTH_UNIT.
- *GAP_WIDTH* = Width of the gap between ring and waveguides in meters scaled by LENGTH_UNIT.

This simulation uses a gaussian distribution in the transverse directions as an approximation to the fundamental spatial mode profile. One can also specify the time signal used to propagate the profile using either a single frequency or FreqWindow Space Time Functions. The SpacetTimeFunctions are assigned to the transverse components of CurrentDistributions, which can be found under the Field Dynamics drop-down of the simulation setup tree.

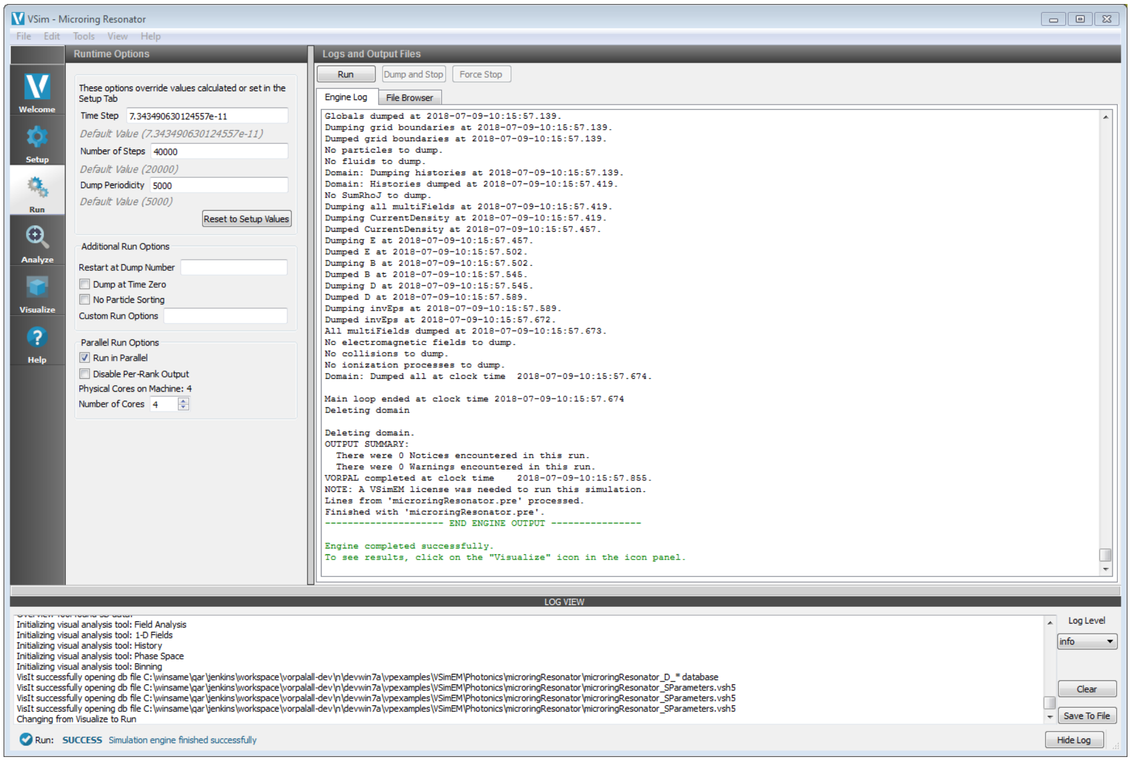
The *Materials* section contains just Silicon and Silica. This section is where one can add or edit materials that get attached to CSG objects. These *Materials* contain the relative permittivity.

In *Field Dynamics* there are *FieldBoundaryConditions* which set the boundary conditions of the simulation. In photonics simulations, Matched Absorbing Layers (MALs), are the most stable boundary conditions for preventing reflections.

Running the Simulation

When the user has saved the setup, continue as follows:

- Proceed to the Run Window by pressing the *Run* button in the left column of buttons.
- When you are finished setting run parameters, click on the *Run* button in the upper left corner of the right pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed” seen in Fig. 3.75



These options override values calculated or set in the Setup Tab

Time Step 7.343490630124557e-11
Default Value (7.343490630124557e-11)
Number of Steps 40000
Default Value (20000)
Dump Periodicity 5000
Default Value (5000)

Additional Run Options
Restart at Dump Number
□ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options
☑ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
Globals dumped at 2018-07-09-10:15:57.139.
Dumping grid boundaries at 2018-07-09-10:15:57.139.
Dumped grid boundaries at 2018-07-09-10:15:57.139.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-07-09-10:15:57.139.
Domain: Histories dumped at 2018-07-09-10:15:57.419.
No SumRhoJ to dump.
Dumping all multiFields at 2018-07-09-10:15:57.419.
Dumping CurrentDensity at 2018-07-09-10:15:57.419.
Dumped CurrentDensity at 2018-07-09-10:15:57.457.
Dumping E at 2018-07-09-10:15:57.457.
Dumped E at 2018-07-09-10:15:57.502.
Dumping B at 2018-07-09-10:15:57.502.
Dumped B at 2018-07-09-10:15:57.545.
Dumping D at 2018-07-09-10:15:57.545.
Dumped D at 2018-07-09-10:15:57.589.
Dumping invEps at 2018-07-09-10:15:57.589.
Dumped invEps at 2018-07-09-10:15:57.672.
All multiFields dumped at 2018-07-09-10:15:57.673.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-09-10:15:57.674.

Main loop ended at clock time 2018-07-09-10:15:57.674
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-07-09-10:15:57.855.
NOTE: A VSimEM license was needed to run this simulation.
Lines from 'microringResonator.pre' processed.
Finished with 'microringResonator.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

```
Initializing visual analysis tool: Field Analysis
Initializing visual analysis tool: 1-D Fields
Initializing visual analysis tool: History
Initializing visual analysis tool: Phase Space
Initializing visual analysis tool: Binning
VisIt successfully opening db file C:\winsame\qar\jenkins\workspace\vorpalall-dev\n\devwin7a\vpexamples\VSimEM\Photonics\microringResonator\microringResonator_D_* database
VisIt successfully opening db file C:\winsame\qar\jenkins\workspace\vorpalall-dev\n\devwin7a\vpexamples\VSimEM\Photonics\microringResonator\microringResonator_SParameters.vsh5
VisIt successfully opening db file C:\winsame\qar\jenkins\workspace\vorpalall-dev\n\devwin7a\vpexamples\VSimEM\Photonics\microringResonator\microringResonator_SParameters.vsh5
VisIt successfully opening db file C:\winsame\qar\jenkins\workspace\vorpalall-dev\n\devwin7a\vpexamples\VSimEM\Photonics\microringResonator\microringResonator_SParameters.vsh5
Changing from Visualize to Run
```

Log Level: info

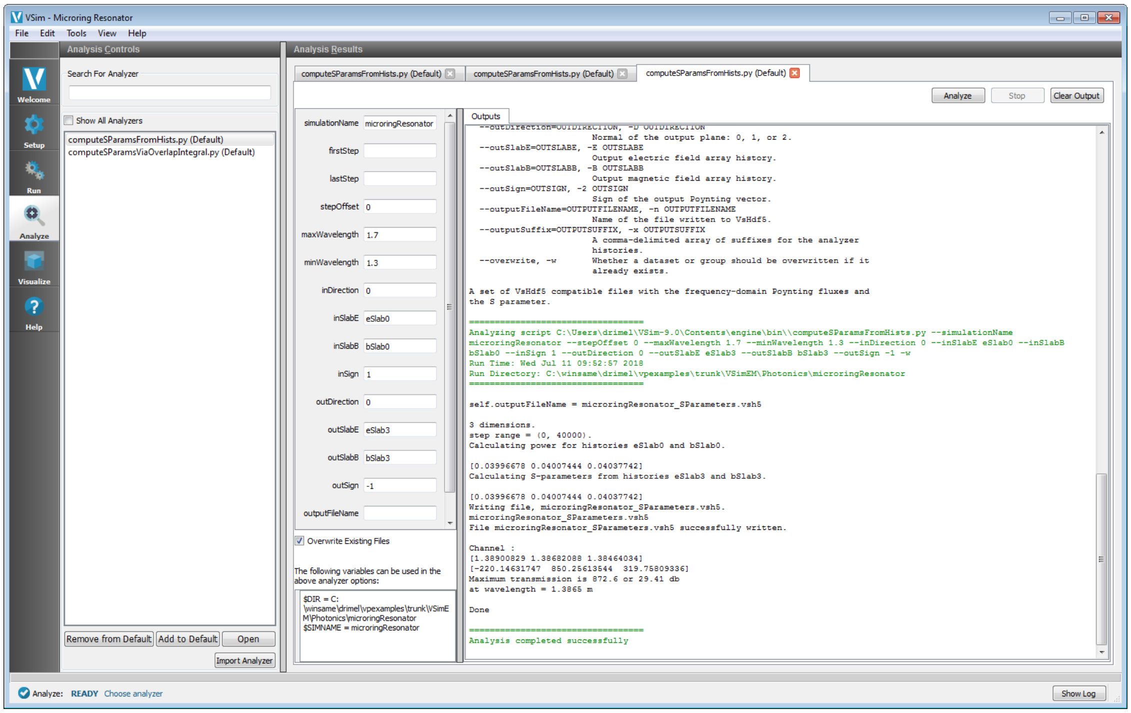
Run: SUCCESS Simulation engine finished successfully

Fig. 3.75: GUI Run Window at completion.

Analyzing the Results

Using Post analysis scripts one can extract transmission coefficients. This is possible due to the field slab histories that are located at each port in the simulation. Each port has an E and B slab history in order to integrate over the poynting flux. This integration is done in a post analysis script called computeSParamsFromHists.py.

Now go to the Analyze pane, select the *computeSParamsFromHists.py* analyzer, and press the *Open* button. Set the maxWavelength to 1.7, outSlabE to be eSlab3, outSlabB to be b. Upon hitting the Analyze button of the Analyze pane, we see that there will be a list of transmission maximum and the wavelength they occur at. This is seen in Fig. 3.76.


VSim - Microring Resonator
File Edit Tools View Help
Analysis Controls
Welcome
Setup
Run
Analyze
Visualize
Help
Search For Analyzer
Show All Analyzers
computeSParamsFromHists.py (Default)
computeSParamsViaOverlapIntegral.py (Default)
Remove from Default
Add to Default
Open
Import Analyzer
Analysis Results
computeSParamsFromHists.py (Default)
computeSParamsFromHists.py (Default)
computeSParamsFromHists.py (Default)
Analyze
Stop
Clear Output
Outputs
Show Log
Analyze: READY Choose analyzer

simulationName: microringResonator
firstStep:
lastStep:
stepOffset: 0
maxWavelength: 1.7
minWavelength: 1.3
inDirection: 0
inSlabE: eSlab0
inSlabB: bSlab0
inSign: 1
outDirection: 0
outSlabE: eSlab3
outSlabB: bSlab3
outSign: -1
outputFileName:

☑ Overwrite Existing Files

The following variables can be used in the above analyzer options:

$DIR = C:\winsame\drimel\vpexamples\trunk\VSimEM\Photonics\microringResonator
$SIMNAME = microringResonator

```
  --outDirection=OUTDIRECTION, -D OUTDIRECTION
                        Normal of the output plane: 0, 1, or 2.
  --outSlabE=OUTSLABE, -E OUTSLABE
                        Output electric field array history.
  --outSlabB=OUTSLABB, -B OUTSLABB
                        Output magnetic field array history.
  --outSign=OUTSIGN, -2 OUTSIGN
                        Sign of the output Poynting vector.
  --outputFileName=OUTPUTFILENAME, -n OUTPUTFILENAME
                        Name of the file written to VsHdf5.
  --outputSuffix=OUTPUTSUFFIX, -x OUTPUTSUFFIX
                        A comma-delimited array of suffixes for the analyzer
                        histories.
  --overwrite, -w       Whether a dataset or group should be overwritten if it
                        already exists.

A set of VsHdf5 compatible files with the frequency-domain Poynting fluxes and
the S parameter.

==================================
Analyzing script C:\Users\drimel\VSim-9.0\Contents\engine\bin\\computeSParamsFromHists.py --simulationName
microringResonator --stepOffset 0 --maxWavelength 1.7 --minWavelength 1.3 --inDirection 0 --inSlabE eSlab0 --inSlabB
bSlab0 --inSign 1 --outDirection 0 --outSlabE eSlab3 --outSlabB bSlab3 --outSign -1 -w
Run Time: Wed Jul 11 09:52:57 2018
Run Directory: C:\winsame\drimel\vpexamples\trunk\VSimEM\Photonics\microringResonator
==================================

self.outputFileName = microringResonator_SParameters.vsh5

3 dimensions.
step range = (0, 40000).
Calculating power for histories eSlab0 and bSlab0.

[0.03996678 0.04007444 0.04037742]
Calculating S-parameters from histories eSlab3 and bSlab3.

[0.03996678 0.04007444 0.04037742]
Writing file, microringResonator_SParameters.vsh5.
microringResonator_SParameters.vsh5
File microringResonator_SParameters.vsh5 successfully written.

Channel :
[1.38900829 1.38682088 1.38464034]
[-220.14631747  850.25613544  319.75809336]
Maximum transmission is 872.6 or 29.41 db
at wavelength = 1.3865 m

Done

==================================
Analysis completed successfully
```

Fig. 3.76: VSim analyzer tab.

Visualizing the results

After performing the above actions proceed to the Visualize Window by pressing the *Visualize* button in the left column of buttons.

One can visualize the transmission coefficients by performing the following:

- Near the top left corner of the window, make sure Data View is set to 1-D Fields.
- In the control Panel select *S_eSlab3bSlab3* for graph1 and set graph 2-4 to *None*.

Once you have performed the above actions one's screen should look like Fig. 3.77.

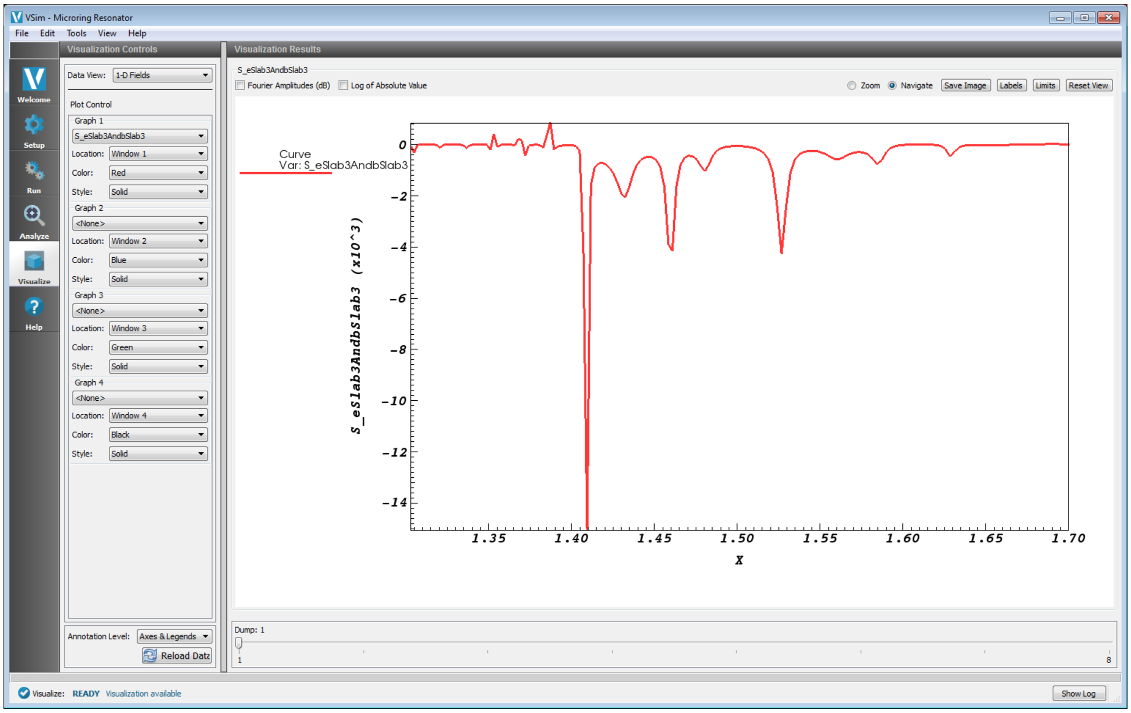
Further Experiments

One can experiment by changing the *GAP_WIDTH* parameters. Then, one can run the computeSParamsViaOverlapIntegral.py script to extract S-Parameters. One can see now changing the *GAP_WIDTH* changes the S-Parameters.

3.5.6 Microring Resonator Mode Calculation (microringResonatorModeCalc.sdf)

Keywords:

`Mode Solver, Microring Resonator, Unidirectional Mode Launcher, MAL, Guided Mode, Photonic`



Data View: 1-D Fields

Plot Control

Graph 1: S_eSlab3AndbSlab3
Location: Window 1
Color: Red
Style: Solid

Graph 2: <None>
Location: Window 2
Color: Blue
Style: Solid

Graph 3: <None>
Location: Window 3
Color: Green
Style: Solid

Graph 4: <None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Dump: 1

Visualize: READY Visualization available

Fig. 3.77: Visualization of Transmission Coefficient

Problem Description

This example demonstrates running a quasi-2D simulation of a dielectric waveguide and extracting the frequency and fields of a guided mode using the filter diagonalization method. This is a quasi-2D simulation of the microringResonator example. The purpose of this example is to calculate the guided mode profile for the microringResonator example. After the mode profile is calculated one can interpolate the profile to the microringResonator example which will propagate the profile.

The waveguide axis runs parallel to the x-axis, and is surrounded by cladding in the y and z directions. The cladding terminates at the y and z simulation boundaries, which are set to perfect conductors (reflecting boundaries). The x simulation boundaries are phase-shifting; i.e., fields on the +x boundary are set to the values of the fields on the -x boundary multiplied by $e^{i\phi}$.

Eigenmodes in such a simulation have the form:

$$\mathbf{E}(\mathbf{x},t) = \mathbf{E}(y,z)e^{i(kx-\omega t)}$$

where the wavenumbers k that may be simulated are given by $kL_x = \phi + 2\pi n$, where n is any integer and L_x is the length of the simulation in the x direction (in this case, two grid cells; or $L_x = 2\Delta x$). Since we will be searching for the lowest-frequency guided modes, we can safely assume $n = 0$ and set $\phi = kL_x$ for a desired k.

The "modal index" or "effective index of refraction" of a waveguide mode is given by $\bar{n} = k/k_0$ where $k_0 = \omega/c$. If the waveguide has index of refraction n_w and the cladding $n_c < n_w$, then a *guided* mode will have a modal index in the range, $n_c < \bar{n} < n_w$.

This simulation can be performed with a VSimEM license.

Opening the Simulation

The Microring Resonator Mode Calculation example is accessed from within VSimComposer by the following actions:

- Select the *New → From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Photonics* option.
- Select *Microring Resonator Mode* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

Simulation Properties

This example contains a number of *Constants* defined to make the simulation easily modifiable. The user should avoid altering constants not mentioned below.

General Simulation Parameters:

RESOLUTION: Specifies the coarseness/fineness of the simulation grid. The cell size is set to be this number multiplied by the smallest simulation feature, i.e., the smaller of the fiber radius or the simulated longitudinal wavelength.

WAVEL_RESOLVE: Specifies wavelength that is used to calculate the grid spacing (DX,DY,DZ). This should be a wavelength between *WAVEL_MIN* and WAVEL_MAX‘.

LENGTH_UNIT: Specifies the length-scale of the simulation. The fiber radius and box size are divided by this number.

Target eigenmode parameters:

NEFF_GUESS: The estimated modal index of the eigenmode of interest; it should have a value between n_c and n_w.

The following excitation parameters specify the frequency-domain characteristics of the current excitation inside the waveguide. Eigenmode extraction works well when only a few modes are present in the simulation; so we excite a narrow band in frequency space (or wavelength space) containing the frequencies (wavelengths) of our eigenmode(s) of interest. The shape of the frequency window is a flat top with Gaussian tails.

Excitation parameters:

WAVEL_MIN: The lower bound of the flat-top portion of the excitation window in wavelength space.

WAVEL_MAX: The upper bound of the flat-top portion of the excitation window in wavelength space.

Mode extraction is a multi-stage process. Initially, an approximation of the spectrum should be obtained via manual inspection of an FFT of the fields resulting from a delta function excitation. Then, an excitation should be crafted that excludes irrelevant modes from an extraction simulation. Finally, the extraction simulation should be excited (without dumping fields), then restarted with dump settings appropriate for the mode extraction analyzer.

The following parameters help the user choose an appropriate set of options in the Basic Settings section related to simulation run time and field dump settings (e.g. “number of steps”, “steps between dumps”, etc.). depending on the type of analysis.

Basic Settings helpers:

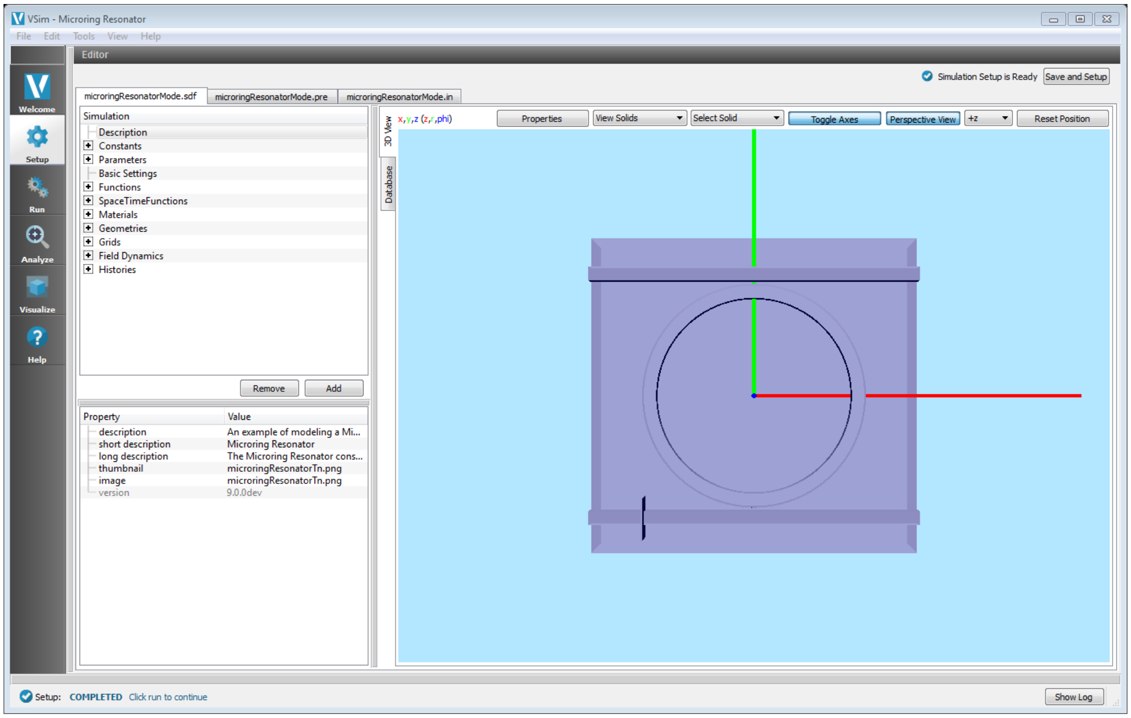
NSTEPS_EXCITE: This is the number of time steps required to encompass the entire simulation excitation. When running the excitation part of the simulation, the “number of steps” should be strictly larger than this parameter.

NSTEPS_EXTRACT: This is the number of time steps required to encompass the entire simulation extraction. When running the extraction part of the simulation, the “number of steps” should be strictly larger than this parameter.

NSTEPSPERDUMP_EXTRACT: This is the number of time steps per dump required for the mode extraction analysis script.

NOTE: When dumping fields for this example, the “dump in groups of” should always be set to 3. This can be accomplished by going to “basic settings” and changing “dump in groups of” to 3.

In the 2D View tab of the right pane of the Setup Window, the device is visible with a smaller grid encompassing the lower left section of the input waveguide.



Property	Value
description	An example of modeling a Mi...
short description	Microring Resonator
long description	The Microring Resonator cons...
thumbnail	microringResonatorTn.png
image	microringResonatorTn.png
version	9.0.0dev

Setup: COMPLETED Click run to continue

Fig. 3.78: Setup Window for the Microring Resonator Device example.

Expanding the Materials section of the Simulation tree shows that the simulation includes silicon and silica. This was created by importing a material from the Database tab of the right pane of the Setup Window.

Perform an Exploratory Simulation

An exploratory simulation is one in which the user wishes to discover the approximate frequencies (or wavelengths) of eigenmodes in a simulation. Such a simulation consists of a wide-band (or short-time) excitation in which we collect the field values (via the EFieldInWaveguide field History; c.f. Setup -> Histories -> EFieldInWaveguide) and subsequent FFT analysis.

In the Setup Window, set up a delta-function (in time) excitation by navigating to: Field Dynamics -> CurrentDistributions -> currentSource; then right-clicking on *J1(x, y, z, t)* and Assign SpaceTimeFunction to *deltaFunc*.

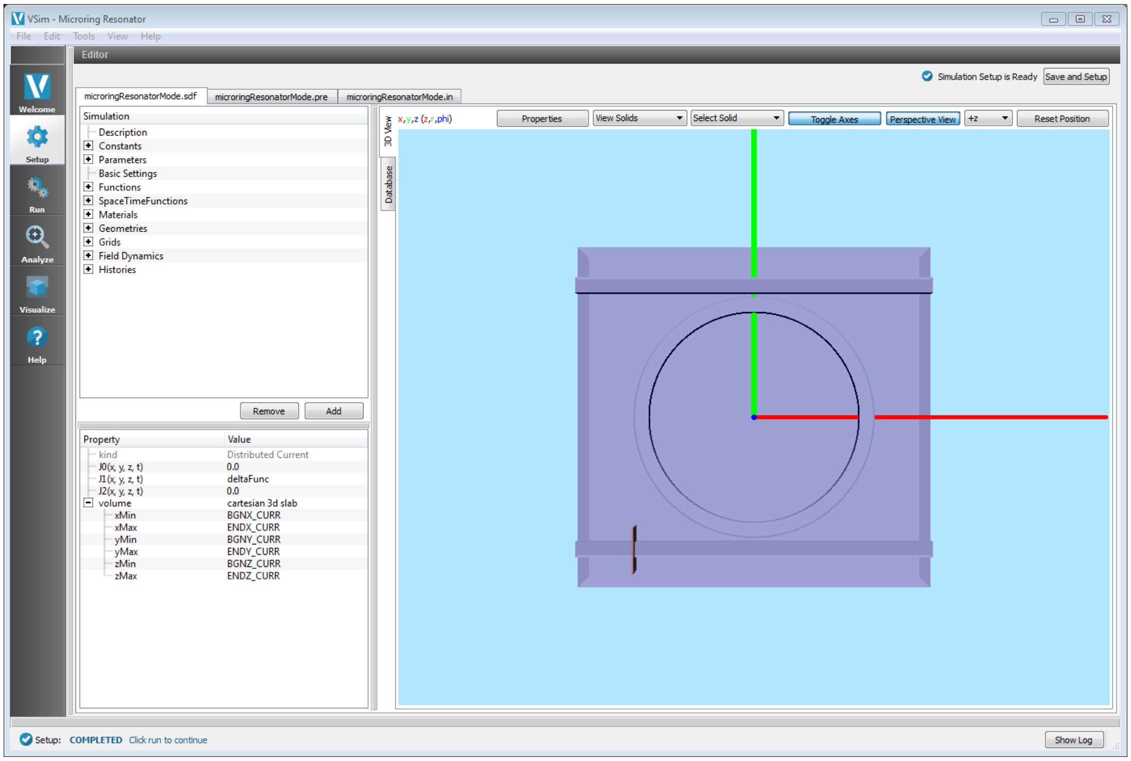
Finally, run the simulation by clicking the *Run* button on the left navigation column and then pressing the *Run* button in the upper left corner of the right pane.

When the simulation finishes, proceed to the visualization screen by clicking the Visualize button on the left navigation column. In the Data View dropdown, select History, then select FFT on the eHist_0 plot, seen in Fig. 3.80.

In the next section, we use the location of the peaks (which are in frequency space) to set up a mode extraction simulation.

Perform extract modes simulation

In this part, we alter the exploratory simulation performed above to extract the lowest-frequency modes identified in the FFT. Navigate back to the Setup Window.

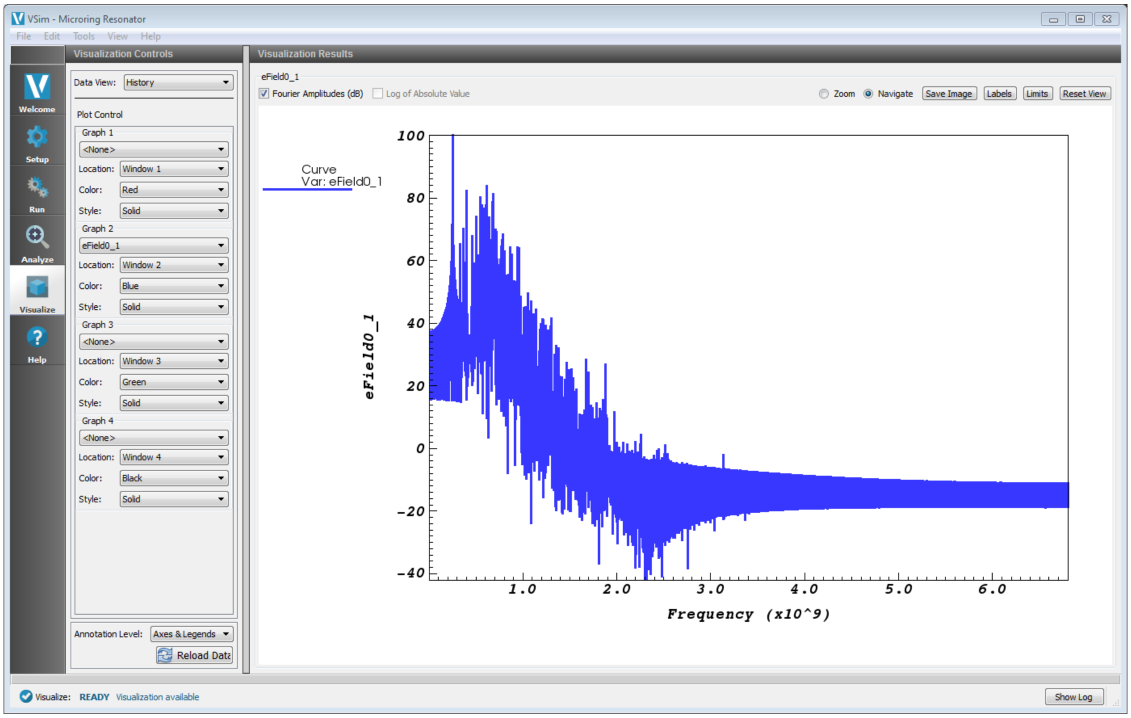


Simulation
- Description
- Constants
- Parameters
- Basic Settings
- Functions
- SpaceTimeFunctions
- Materials
- Geometries
- Grids
- Field Dynamics
- Histories

Property	Value
kind	Distributed Current
J0(x, y, z, t)	0.0
J1(x, y, z, t)	deltaFunc
J2(x, y, z, t)	0.0
volume	cartesian 3d slab
xMin	BGNX_CURR
xMax	ENDX_CURR
yMin	BGNY_CURR
yMax	ENDY_CURR
zMin	BGNZ_CURR
zMax	ENDZ_CURR

Setup: COMPLETED Click run to continue

Fig. 3.79: Choosing a current source



Data View: History

☑ Fourier Amplitudes (dB) □ Log of Absolute Value

Plot Control

Graph 1
<None>
Location: Window 1
Color: Red
Style: Solid

Graph 2
eField0_1
Location: Window 2
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 3
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 3.80: Setup Window for the Cylindrical Dielectric Fiber geometries.

First, set up a simulation excitation with a narrow frequency band. In the Setup Window, navigate to: Field Dynamics -> CurrentDistributions -> currentSource; then right-click on *J1(x, y, z, t)* and Assign SpaceTimeFunction to *gaussFreqWindow*. Then, set the following constants, which were determined by choosing frequency bounds that contain the FFT peaks of interest, then converting frequency to wavelength via $c = f\lambda$.

Now, inspect the NSTEPS_EXCITE parameter and set the following Basic Settings:

- *number of steps*: At least NSTEPS_EXCITE
- *steps between dumps*: 0 (turns off dumping)

Run the simulation.

After the simulation run completes, navigate back to the Setup window to prepare the field-dumping portion of the mode extraction simulation. Set the following Basic Settings:

- *number of steps*: At least NSTEPS_EXTRACT.
- *steps between dumps*: NSTEPSPERDUMP_EXTRACT
- *dump in groups of*: 3

Now, click the *Run* button on the left navigation column. The field-dumping portion of the mode extraction simulation must be run as a restart. Set “Restart at Dump Number” to 1. Run the simulation.

Extracting the Modes with an Analyzer

Once the run finishes and the fields are dumped, proceed to the analysis screen by clicking the Analyze navigation button. In the “Choose an analyzer” dropdown, select extractModesViaOperator.py. Change the following options to

- *electricField*: D,DI
- *magneticField*: B,BI
- *dumpRange*: 2:
- *cellSamples*: ::10,::10,::10

Also check “Overwrite Existing Files”. Run the analyzer by clicking *Analyze* button in the upper right corner. The analyzer output should resemble the following image.

Visualizing the results

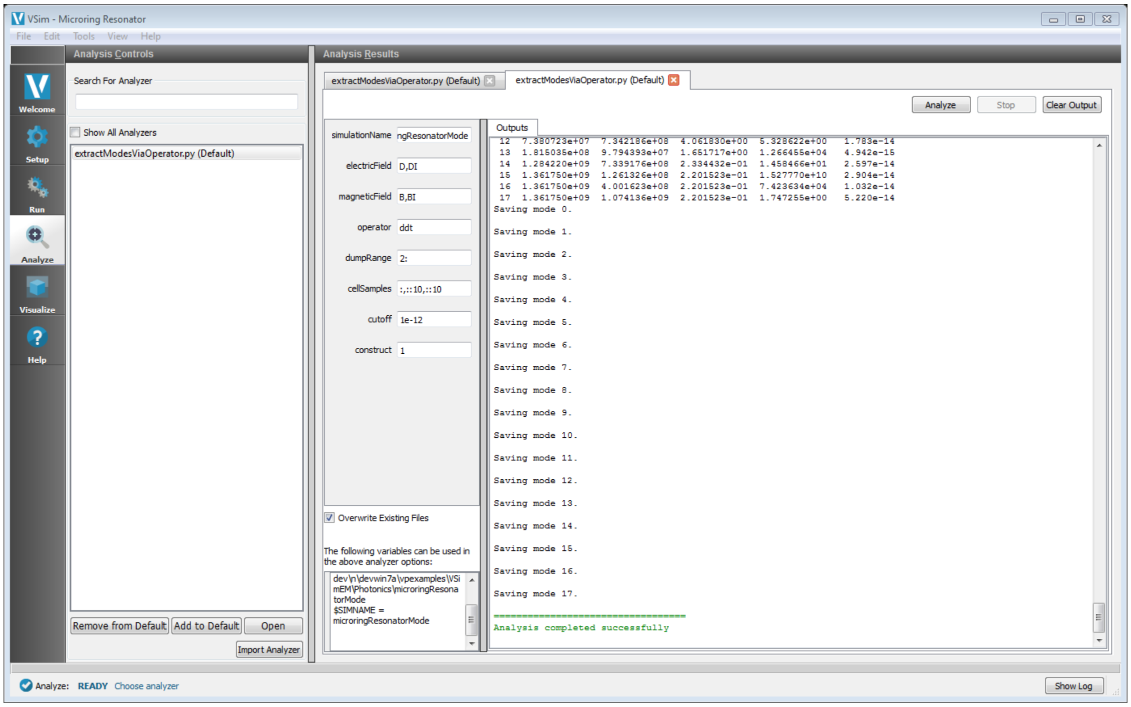
After performing the above actions proceed to the Visualize window by pressing the *Visualize* button in the left column of buttons. Visualize an eigenmode by following these steps:

- Near the top left corner of the window, make sure Data View is set to Data Overview.
- Expand *Scalar Data*, expand *D*, and select *EigenD_1*
- Below the visualization, select the dump number that corresponds to a mode number with the lowest operator error (refer to the analyzer output). The large field values in the visualization that appears should be mostly confined to the waveguide (center of the simulation domain).

Select the *EigenD_1* for another interesting guided mode field pattern:

Further Experiments

One can run a full convergence study of eigenmode frequencies by altering the RESOLUTION parameter in the Setup Window and re-running the simulation and mode extraction script. The simulation grid cell size is set to the RESOLUTION value times the smallest width of the rectangular waveguide.



simulationName ngResonatorMode
electricField D,DI
magneticField B,BI
operator ddt
dumpRange 2:
cellSamples :,::10,::10
cutoff 1e-12
construct 1

☑ Overwrite Existing Files

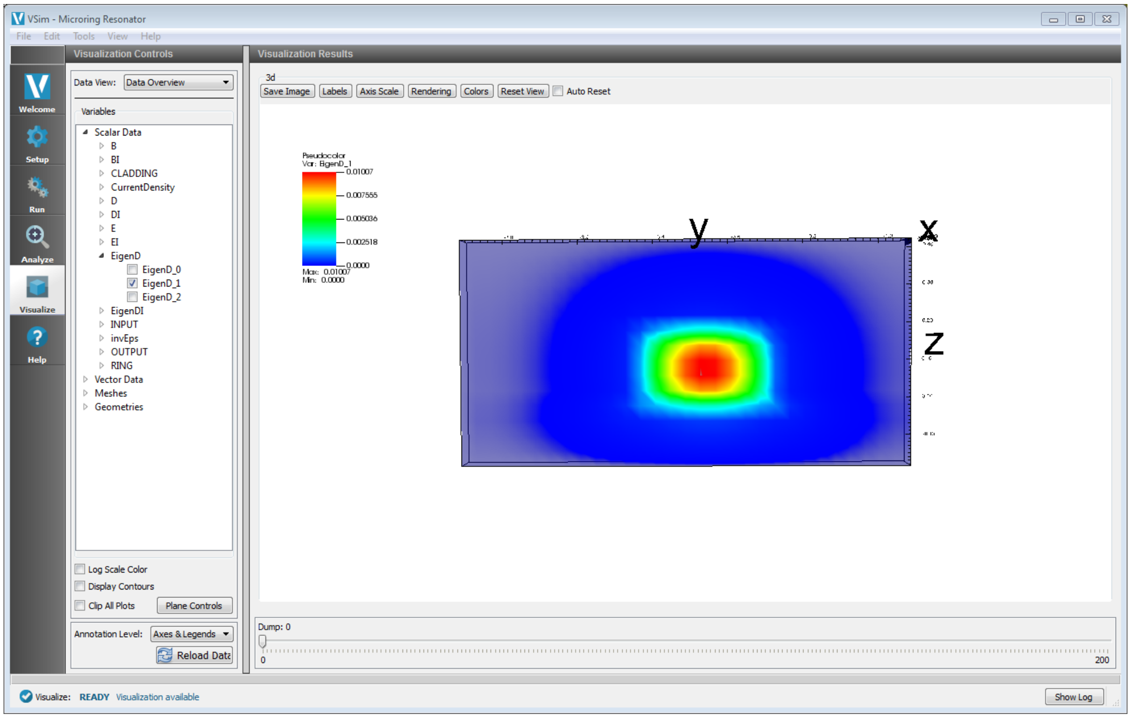
The following variables can be used in the above analyzer options:

dev\n\devwin7a\vpexamples\VSimEM\Photonics\microringResonatorMode
$SIMNAME = microringResonatorMode

```
 12  7.380723e+07  7.342186e+08  4.061830e+00  5.328622e+00   1.783e-14
 13  1.815035e+08  9.794393e+07  1.651717e+00  1.266455e+04   4.942e-15
 14  1.284220e+09  7.339176e+08  2.334432e-01  1.458466e+01   2.597e-14
 15  1.361750e+09  1.261326e+08  2.201523e-01  1.527770e+10   2.904e-14
 16  1.361750e+09  4.001623e+08  2.201523e-01  7.423634e+04   1.032e-14
 17  1.361750e+09  1.074136e+09  2.201523e-01  1.747255e+00   5.220e-14
Saving mode 0.

Saving mode 1.

Saving mode 2.

Saving mode 3.

Saving mode 4.

Saving mode 5.

Saving mode 6.

Saving mode 7.

Saving mode 8.

Saving mode 9.

Saving mode 10.

Saving mode 11.

Saving mode 12.

Saving mode 13.

Saving mode 14.

Saving mode 15.

Saving mode 16.

Saving mode 17.

==================================
Analysis completed successfully
```

Analyze: READY Choose analyzer

Fig. 3.81: Output from the mode extraction script.



Data View: Data Overview

☑ EigenD_1

□ Log Scale Color
□ Display Contours
□ Clip All Plots
□ Auto Reset

Annotation Level: Axes & Legends

Dump: 0

Visualize: READY Visualization available

Fig. 3.82: Output from the mode extraction script.

3.5.7 Gaussian Laser Beam and Photonic Crystal Cavity (photonicCrystal-GaussSrc.sdf)

Keywords:

Gaussian Beam source, photonic crystal, transmission efficiency

Problem description

This example illustrates how to model a Gaussian beam source that is illuminating a cavity inside a hexagonal photonic crystal lattice. The physical setup is shown in Fig. 3.83.

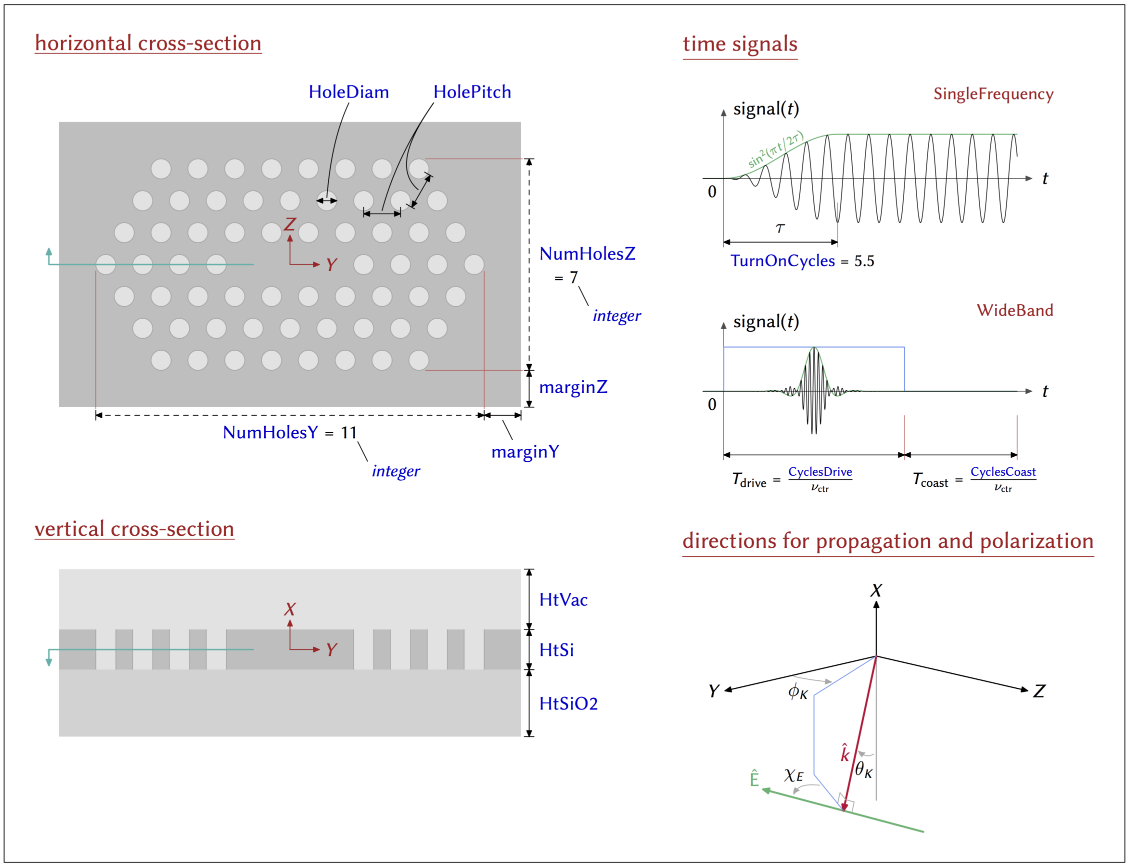


Fig. 3.83: The photonic lattice setup

A Gaussian beam is launched from above into the simulation domain, which comprises three layers: a vacuum region above and a solid dielectric below, which together sandwich a central dielectric layer that contains a lattice of holes. This example includes two possible time signals with which the Gaussian beam will have either WideBand or SingleFrequency.

This simulation can be performed with a VSimEM license.

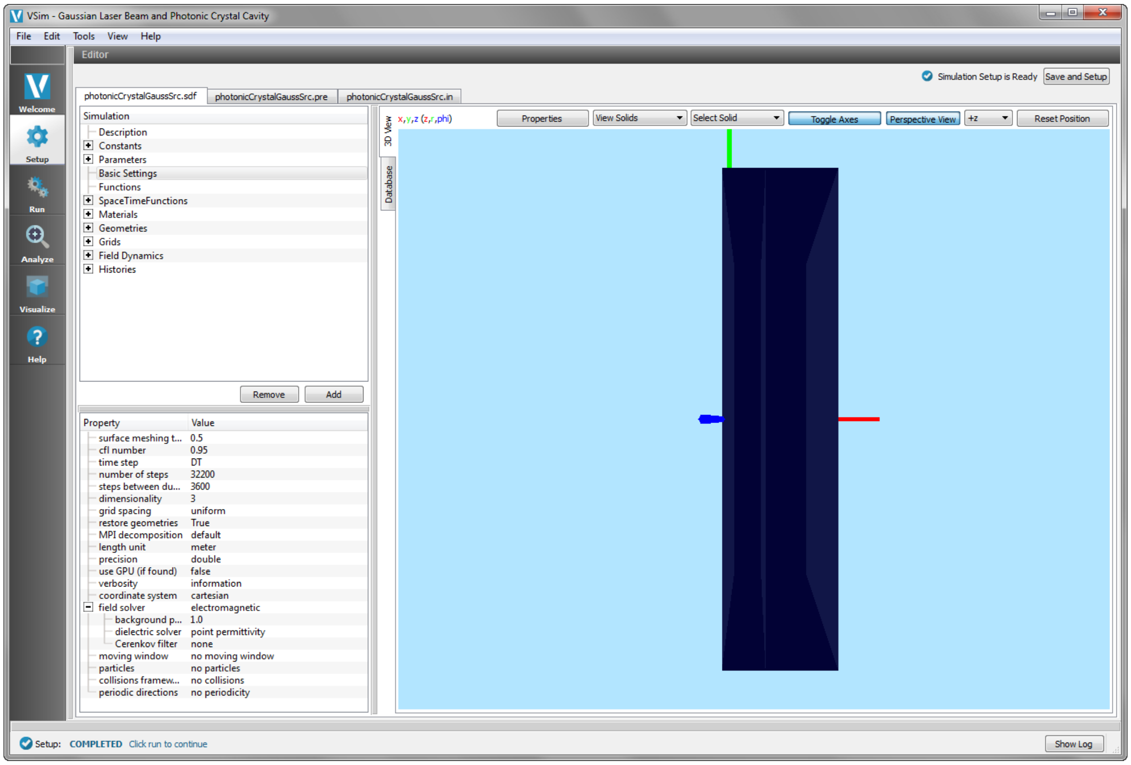
Opening the Simulation

The Photonic Crystal example is accessed from within VSimComposer through the following steps:

- Select the *New* → *From Example…* menu item in the *File* menu.

- In the resulting *Examples* window, expand the *VSim for Electromagnetics* option.
- Expand the *Photonics* option.
- Select "Gaussian Laser Beam and Photonic Crystal Cavity" and press the *Choose* button.
- In the resulting dialog, create a new folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the *Setup Window*, as shown in Fig. 3.84. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



Property	Value
surface meshing t...	0.5
cfl number	0.95
time step	DT
number of steps	32200
steps between du...	3600
dimensionality	3
grid spacing	uniform
restore geometries	True
MPI decomposition	default
length unit	meter
precision	double
use GPU (if found)	false
verbosity	information
coordinate system	cartesian
field solver	electromagnetic
background p...	1.0
dielectric solver	point permittivity
Cerenkov filter	none
moving window	no moving window
particles	no particles
collisions framew...	no collisions
periodic directions	no periodicity

Setup: COMPLETED Click run to continue

Fig. 3.84: The Setup Window for the Gaussian Laser Beam and Photonic Crystal Cavity example

Simulation Properties

This example contains a number of constants that are defined to make the simulation easily modifiable, as can be seen in Fig. 3.85.

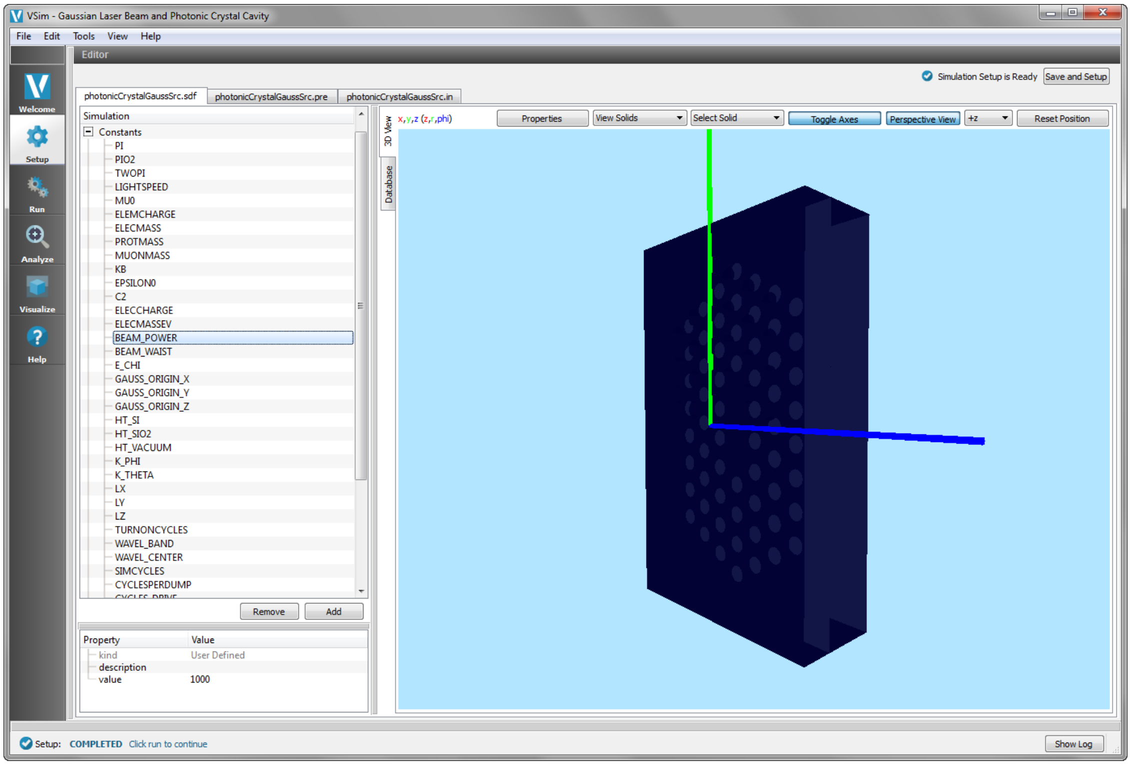
All the following constants should be the only properties you should need to alter in order to specify your simulation domain.

General Simulation Parameters:

- `L{X,Y,Z}` = The length of your simulation domain in the {X,Y,Z} dimension.
- `HT_{VACUUM,SI,SIO2}` = The height of the vacuum, SI and SI02 layers of the photonic crystal.

Source Specifications: (located in the *Parameters* section of the *Elements Tree*)

- `{K_THETA,K_PHI,E_CHI}` = The {polar angle, azimuthal, angle of polarization} respectively.



Property	Value
kind	User Defined
description	
value	1000

Setup: COMPLETED Click run to continue

Fig. 3.85: The Setup Window showing the constants

- WAVEL_CENTER = The central wavelength of your wideband signal. This is also the frequency used in the single frequency simulation type.
- WAVEL_BAND = The wavelength width of your wideband signal, only used in wideBand simType.
- BEAM_WAIST = The width at which your beam power falls off like 1/e.
- BEAM_POWER = The amplitude of your E/M wave.
- GAUSS_ORIGIN_{X,Y,Z} = The point around which your Gaussian profile is centered.
- TURNONCYCLES = The number of cycles you want your single frequency to reach full power.
- SIMCYCLES = Number of wave cycles you want your simulation to run.
- CYCLESPERDUMP = Number of cycles between each dump in the simulation.

The tool used to input the wave into the simulation is a port launcher. It specifies the Electric Displacement Field (D) at a boundary in this case the lower X boundary. The functions defining your D on the boundary are defined in the *SpaceTimeFunctions* element of the *Elements Tree*.

SpaceTimeFunctions:

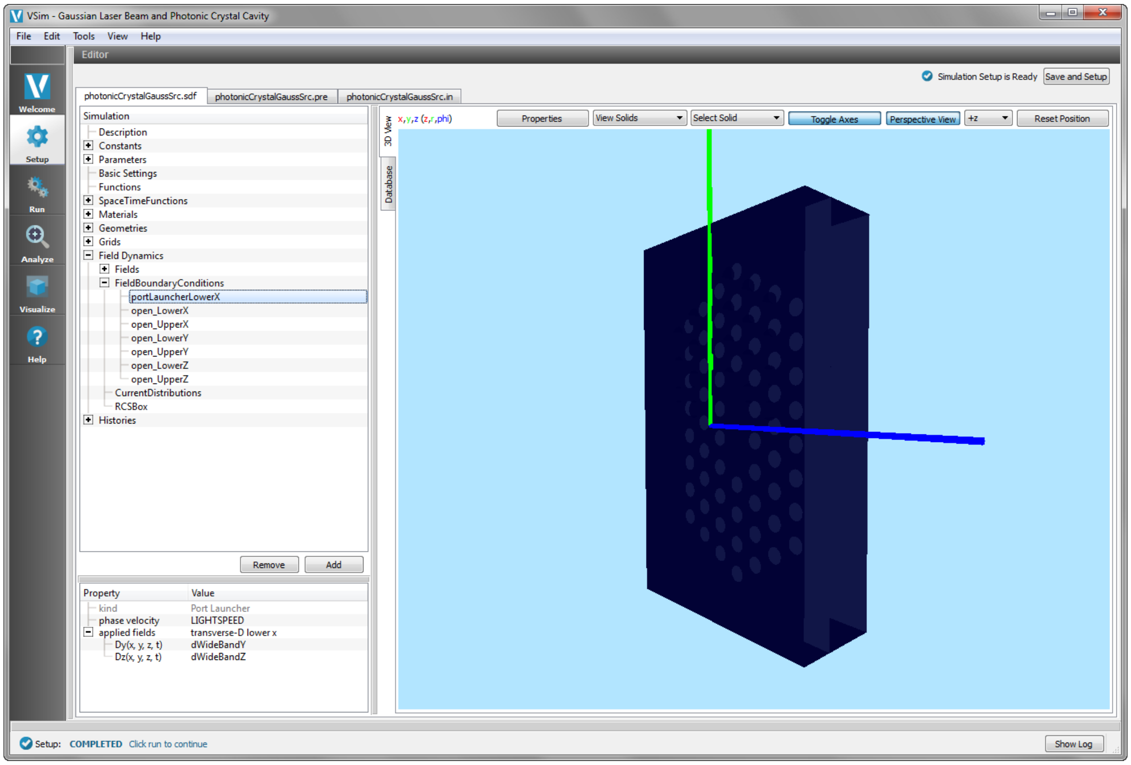
- dSingleFreq{Y,Z} = This is the {x,y,z} component of the single frequency Gaussian beam source, as seen in Fig. 3.85. You put this as a parameter in the current distribution.
- dWideBand{Y,Z} = This is the {x,y,z} component of the wideband Gaussian beam source, as seen in Fig. 3.85. You put this as a parameter in the current distribution.

To choose which signal you want to input into this example simulation:

1. Expand the *FieldBoundaryConditions* element.

2. Left click on the *portLauncherLowerX* condition.

At this point, you should see what is shown in Fig. 3.86.



Simulation
- Description
- Constants
- Parameters
- Basic Settings
- Functions
- SpaceTimeFunctions
- Materials
- Geometries
- Grids
- Field Dynamics
  - Fields
  - FieldBoundaryConditions
    - portLauncherLowerX
    - open_LowerX
    - open_UpperX
    - open_LowerY
    - open_UpperY
    - open_LowerZ
    - open_UpperZ
  - CurrentDistributions
  - RCSBox
- Histories

Property	Value
kind	Port Launcher
phase velocity	LIGHTSPEED
applied fields	transverse-D lower x
Dy(x, y, z, t)	dWideBandY
Dz(x, y, z, t)	dWideBandZ

Setup: COMPLETED Click run to continue

Fig. 3.86: The Setup Window for specifying the signal for the portLauncher

3. To add a signal to the *portLauncherLowerX* just right click on the D{x,y} and select *Assign SpaceTimeFunction*. This will expand another menu that will show you all four defined SpaceTimeFunctions. Select which one you want to input into your simulation. For this documentation, `WideBand{Y,Z}` will be used to demonstrate the functionality of this example.

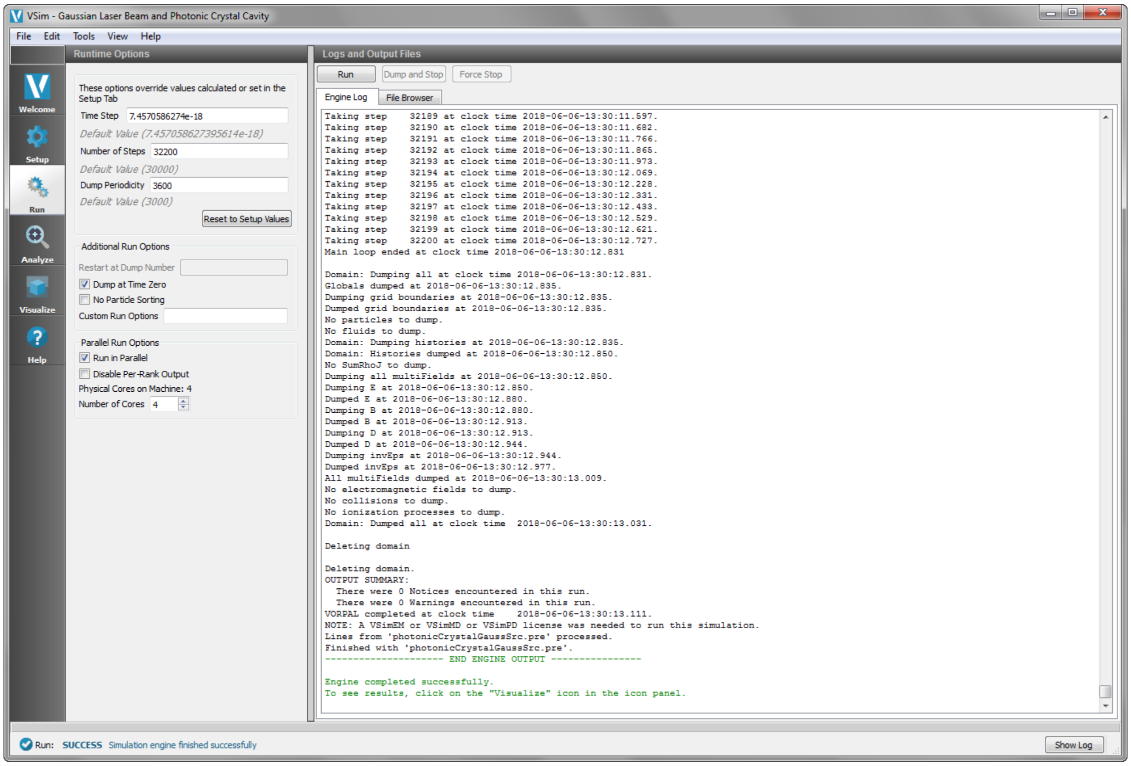
Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the *Run Window* by pressing the *Run* button in the left column of buttons.
- One can enable MPI options to utilize multi-core systems.
- The default values of `Number of Time Steps` and `Dump Periodicity` are taken from the parameters `STEPSTOTAL` and `STEPSPERDUMP`, which use the constants `SIMCYCLES` and `CYCLESPERDUMP`. The formulae for these variables can be found back in the *Setup Window*. These variables are for convenience to calculate good default values and it is important to know that the override option default values ultimately come directly from the numbers in the *Basic Settings* section.

  > *Number of Steps* and *Dump Periodicity*. Just copy these values into the correct fields in the *Run* menu.
  >
  > – *Number of Steps* = `TOTALSTEPS` = 32220
  >
  > – *Dump Periodicity* = `STEPSPERDUMP` = 3600

- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 3.87.



These options override values calculated or set in the Setup Tab

Time Step 7.4570586274e-18

Default Value (7.457058627395614e-18)

Number of Steps 32200

Default Value (30000)

Dump Periodicity 3600

Default Value (3000)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Taking step    32189 at clock time 2018-06-06-13:30:11.597.
Taking step    32190 at clock time 2018-06-06-13:30:11.682.
Taking step    32191 at clock time 2018-06-06-13:30:11.766.
Taking step    32192 at clock time 2018-06-06-13:30:11.865.
Taking step    32193 at clock time 2018-06-06-13:30:11.973.
Taking step    32194 at clock time 2018-06-06-13:30:12.069.
Taking step    32195 at clock time 2018-06-06-13:30:12.228.
Taking step    32196 at clock time 2018-06-06-13:30:12.331.
Taking step    32197 at clock time 2018-06-06-13:30:12.433.
Taking step    32198 at clock time 2018-06-06-13:30:12.529.
Taking step    32199 at clock time 2018-06-06-13:30:12.621.
Taking step    32200 at clock time 2018-06-06-13:30:12.727.
Main loop ended at clock time 2018-06-06-13:30:12.831

Domain: Dumping all at clock time 2018-06-06-13:30:12.831.
Globals dumped at 2018-06-06-13:30:12.835.
Dumping grid boundaries at 2018-06-06-13:30:12.835.
Dumped grid boundaries at 2018-06-06-13:30:12.835.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-06-06-13:30:12.835.
Domain: Histories dumped at 2018-06-06-13:30:12.850.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-06-13:30:12.850.
Dumping E at 2018-06-06-13:30:12.850.
Dumped E at 2018-06-06-13:30:12.880.
Dumping B at 2018-06-06-13:30:12.880.
Dumped B at 2018-06-06-13:30:12.913.
Dumping D at 2018-06-06-13:30:12.913.
Dumped D at 2018-06-06-13:30:12.944.
Dumping invEps at 2018-06-06-13:30:12.944.
Dumped invEps at 2018-06-06-13:30:12.977.
All multiFields dumped at 2018-06-06-13:30:13.009.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-06-13:30:13.031.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-06-13:30:13.111.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.
Lines from 'photonicCrystalGaussSrc.pre' processed.
Finished with 'photonicCrystalGaussSrc.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 3.87: The Run Window for the Gaussian Laser Beam and Photonic Crystal Cavity example

Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the *Visualize Window* by pressing the *Visualize* button in the left column of buttons.

In the simulation, there are specific grid points which store field histories. These histories are placed in various positions of the simulation.

- In the top field of *Visualization Controls*, click the drop down menu and select *History*.

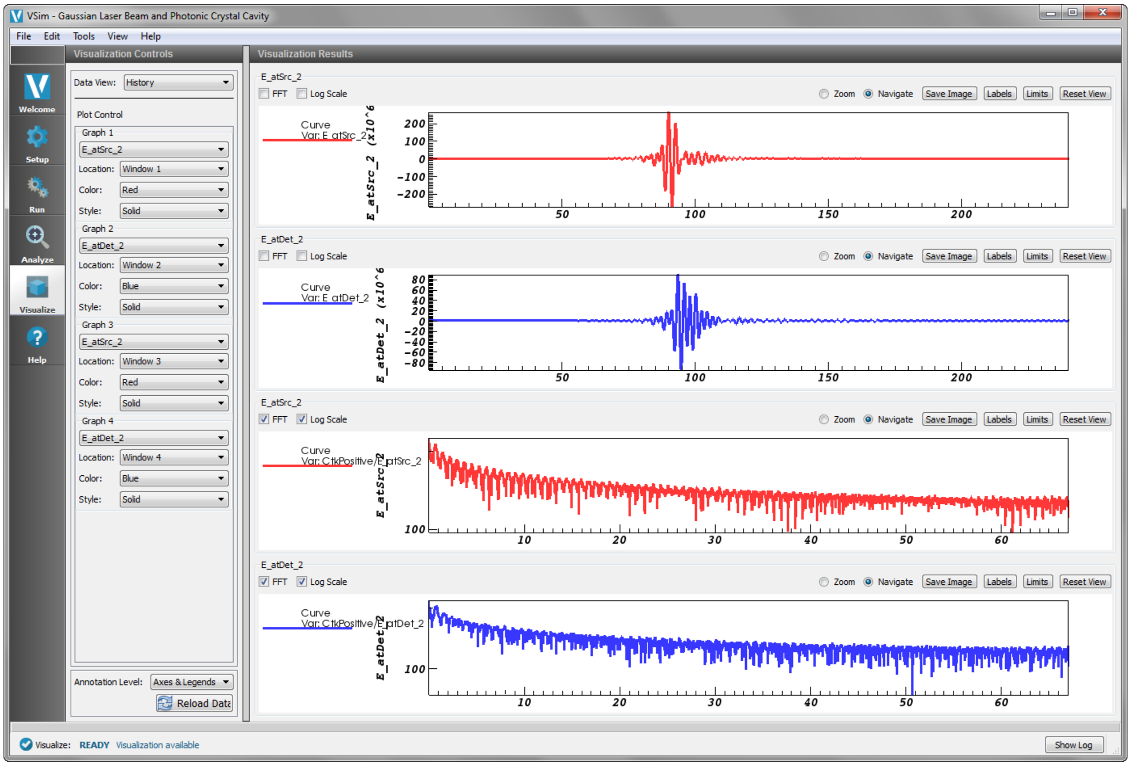
In Fig. 3.88, one can see there are 4 possible graphs to view at one time in the *Visualize Window*. For each graph, one can select the following fields to analyze: (0 = x, 1 = y, 2 = z).

- {E,B}_AtDet_{0,1,2} is in the middle of (y,z) plane and 60 nm above the surface of the crystal.
- {E,B}_AtSrc_{0,1,2} is aligned with the inCav history in the (y,z) plane and is 60 nm below the Si layer, into the SiO2 layer.
- {E,B}_InCav_{0,1,2} is slightly offset from the middle of one of the cavities in the silicon layer (the layer with the lattice).

In each individual graph, one can choose the FFT option to view the frequency domain of your field. This can enable the analysis of the frequency response of the photonic crystal cavity.

Fig. 3.88 depicts four graphs of histories. The first two graphs are amplitude vs time, and the second two are a FFT of the first two on a log scale.

The first and third graphs depict the history `AtSrc_2`, while the second and fourth graphs show the `AtDet_2` history.



Data View: History

Graph 1
E_atSrc_2
Location: Window 1
Color: Red
Style: Solid

Graph 2
E_atDet_2
Location: Window 2
Color: Blue
Style: Solid

Graph 3
E_atSrc_2
Location: Window 3
Color: Red
Style: Solid

Graph 4
E_atDet_2
Location: Window 4
Color: Blue
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 3.88: The Visualize Window for the Gaussian Laser Beam and Photonic Crystal Cavity example

Further Experiments

By using the wideband source and examining the field strength detected below the crystal lattice, one may study the frequency response of this photonic crystal as one changes the device geometry, the dielectric constants, and the location and polarizations of the radiation source and detector.

3.5.8 Dipole Source Illuminating a Photonic Crystal Cavity (photonicCrystalDipoleSrc.sdf)

Keywords:

`dipole source, photonic crystal, transmission efficiency`

Problem description

This example illustrates how to model a dipole source that is illuminating cavities inside a hexagonal photonic crystal lattice. The physical arrangement is shown in Fig. 3.89 and Fig. 3.90.

A point-like dipole lies above the simulation domain, which comprises three layers: a vacuum region above and a solid dielectric below, which together sandwich a central dielectric layer that contains a lattice of holes. This example includes two possible time signals with which to ring the dipole source, as shown in Fig. 3.91.

This simulation can be performed with a VSimEM license.

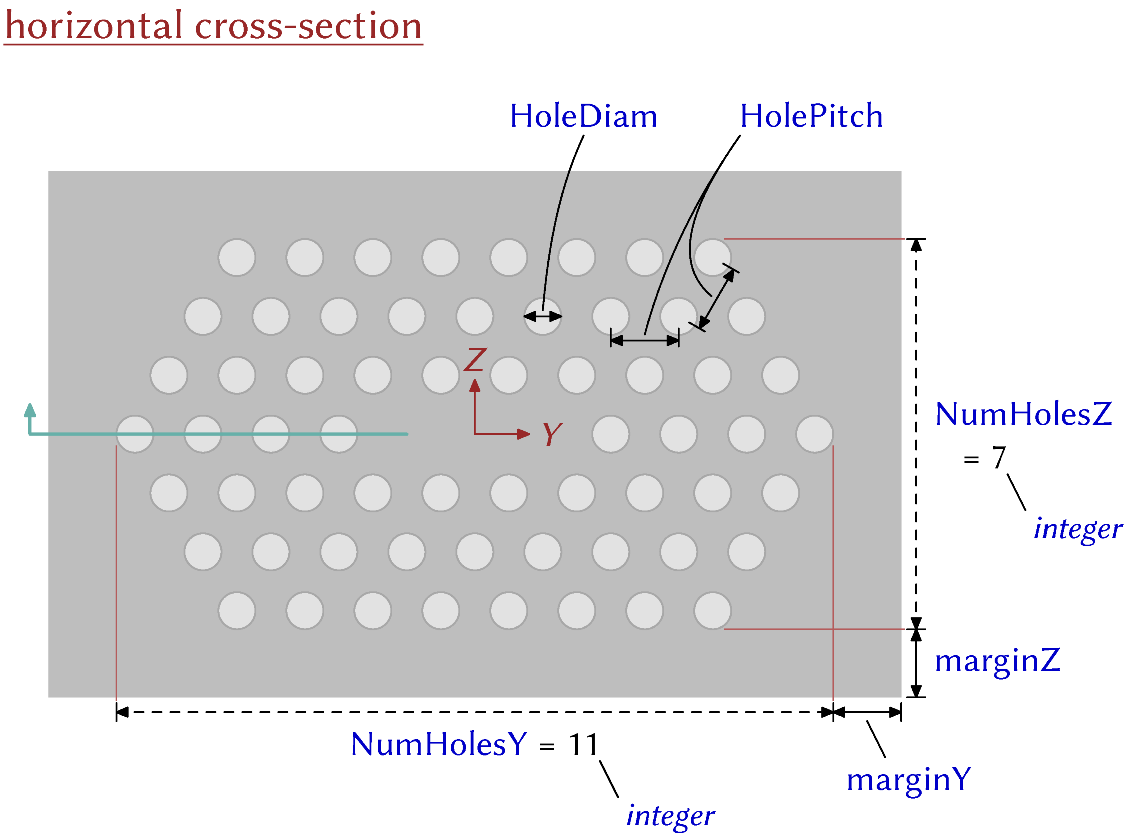


Fig. 3.89: Top view of photonic lattice.

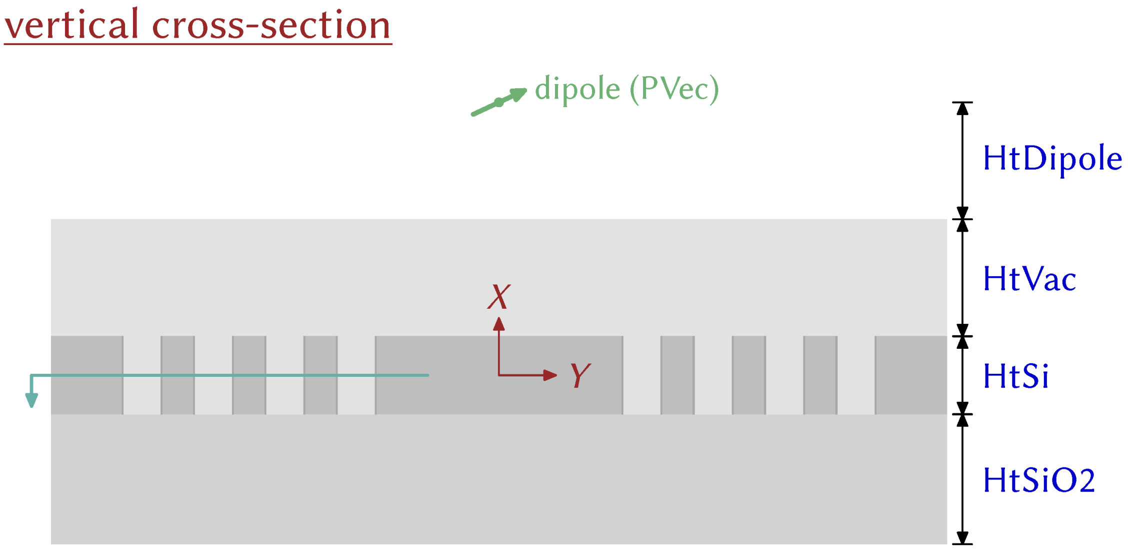


Fig. 3.90: Side view of photonic lattice.

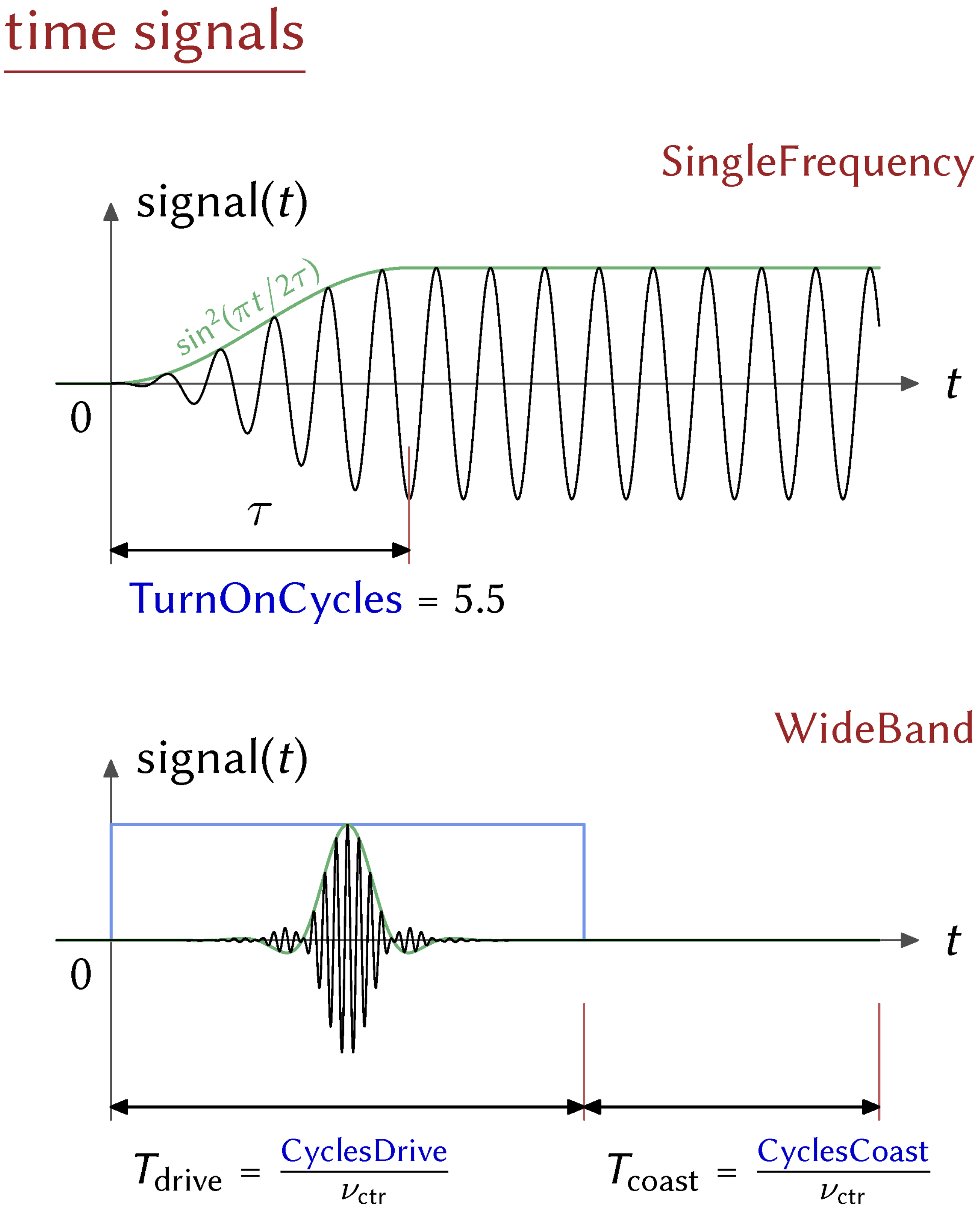

time signals
SingleFrequency
signal(t)
$\sin^2(\pi t/2\tau)$
0
t
τ
TurnOnCycles = 5.5
WideBand
signal(t)
0
t
$T_{\mathrm{drive}} = \frac{\mathrm{CyclesDrive}}{\nu_{\mathrm{ctr}}}$
$T_{\mathrm{coast}} = \frac{\mathrm{CyclesCoast}}{\nu_{\mathrm{ctr}}}$

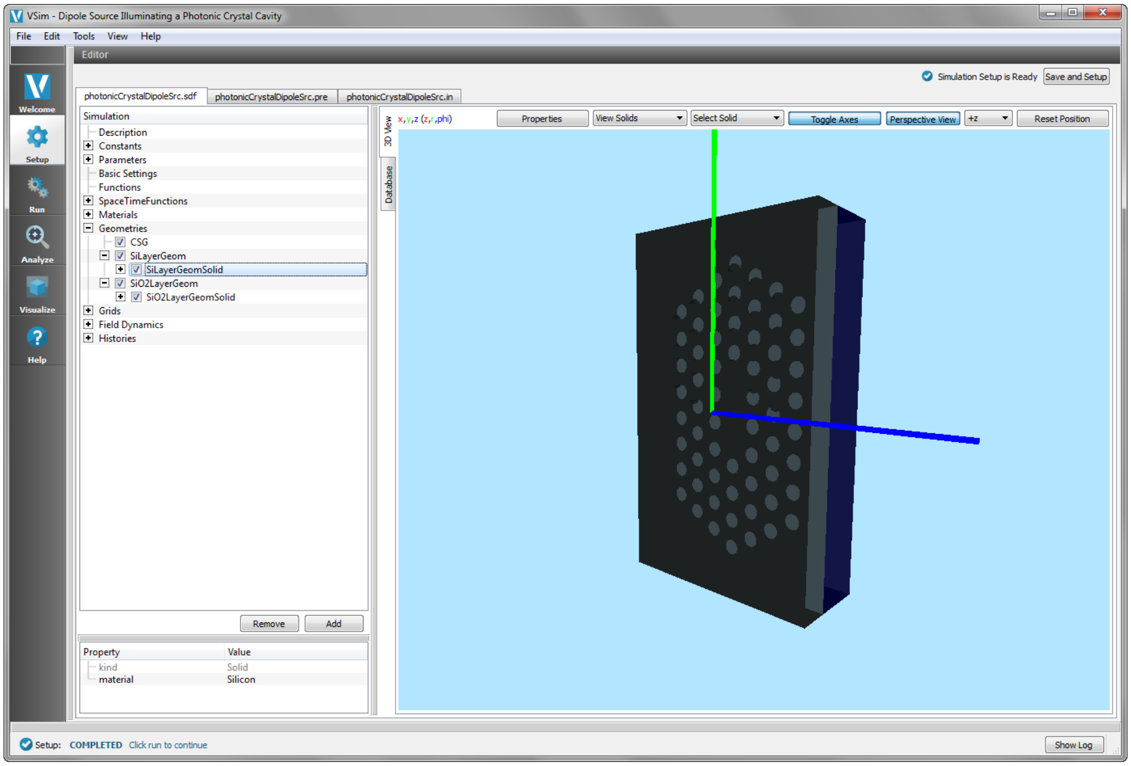
Fig. 3.91: Two possible time signals for ringing the dipole source.

Opening the Simulation

This Photonic Crystal example is accessed from within VSimComposer through the following steps:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window, expand the *VSim for Electromagnetics* option.
- Expand the *Photonics* option.
- Select "Dipole Source Illuminating a Photonic Crystal Cavity" and press the *Choose* button.
- In the resulting dialog, create a new folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the *Setup Window* as shown in Fig. 3.92. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the photonic crystal geometry. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.


VSim - Dipole Source Illuminating a Photonic Crystal Cavity
File Edit Tools View Help
Editor
Welcome
Setup
Run
Analyze
Visualize
Help
Simulation Setup is Ready
Save and Setup
photonicCrystalDipoleSrc.sdf
photonicCrystalDipoleSrc.pre
photonicCrystalDipoleSrc.in
3D View
Database
x,y,z (z,r,phi)
Properties
View Solids
Select Solid
Toggle Axes
Perspective View
+z
Reset Position
Remove
Add
Show Log

Simulation
- Description
- ⊞ Constants
- ⊞ Parameters
- Basic Settings
- Functions
- ⊞ SpaceTimeFunctions
- ⊞ Materials
- ⊟ Geometries
  - ☑ CSG
  - ⊟ ☑ SiLayerGeom
    - ⊞ ☑ SiLayerGeomSolid
  - ⊟ ☑ SiO2LayerGeom
    - ⊞ ☑ SiO2LayerGeomSolid
- ⊞ Grids
- ⊞ Field Dynamics
- ⊞ Histories

Property	Value
kind	Solid
material	Silicon

Setup: COMPLETED Click run to continue

Fig. 3.92: The Setup Window for the Dipole Source Illuminating a Photonic Crystal Cavity example

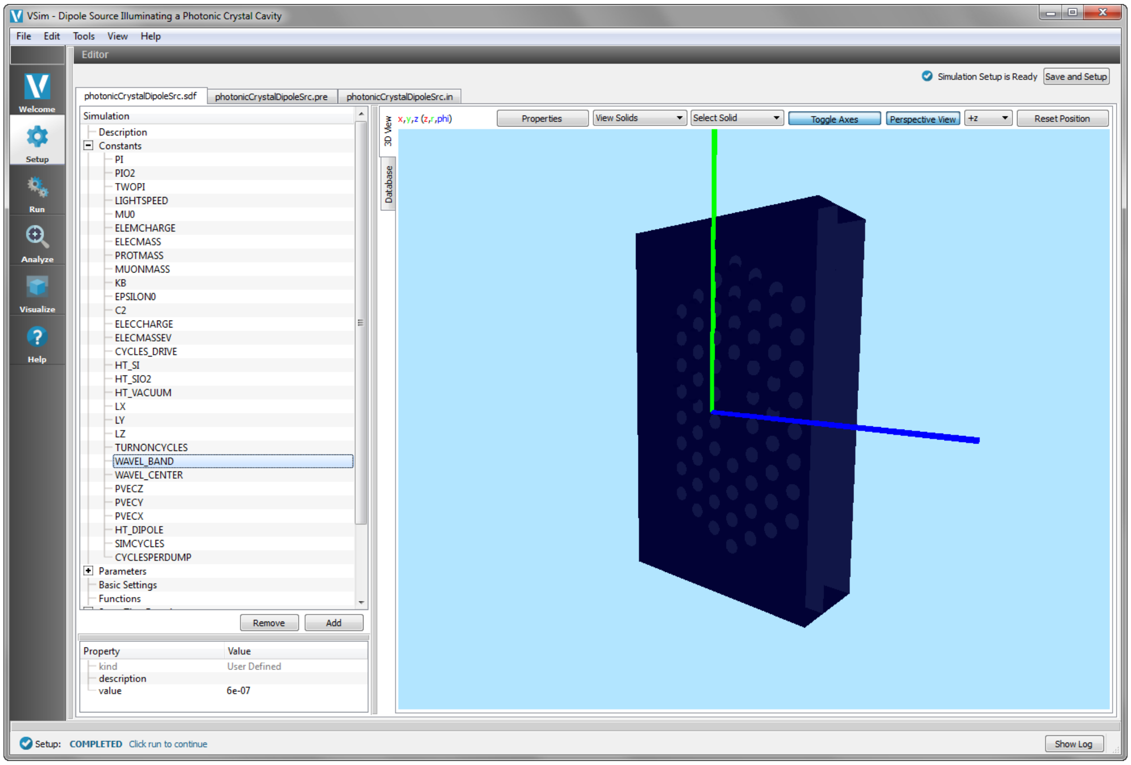
Simulation Properties

This example contains a number of constants defined to make the simulation easily modifiable, as can be seen in Fig. 3.93.

The following constants should be the only properties you should need to alter in order to specify your simulation domain.

General Simulation Parameters:

- `CYCLES_DRIVE` = How many cycles at which the E/M source is driven.



Property	Value
kind	User Defined
description	
value	6e-07

Fig. 3.93: The Setup Window showing the constants

- `HT_{VACUUM,SI,SI02}` = The height of the vacuum, SI and SI02 layers of the photonic crystal.
- `L{X,Y,Z}` = The length of your simulation domain in the {X,Y,Z} dimension.

Source Specifications: (located in the *Parameters* section of the *Elements Tree*)

- `TURNONCYCLES` = The number of cycles you want your single frequency to reach full power.
- `WAVEL_BAND` = The wavelength width of your wideband signal, if doing a wideband simulation.
- `WAVEL_CENTER` = The central wavelength of your wideband signal, and is the frequency used in the single frequency simulation type.
- `PVEC{X,Y,Z}` = The {x,y,z} component of your moment vector for your dipole source.
- `HT_DIPOLE` = The height of the dipole from the lowerX boundary.
- `SIMCYCLES` = Number of wave cycles you want your simulation to run.
- `CYCLESPERDUMP` = Number of cycles between each dump in the simulation.

The tool used to input the wave into the simulation is a port launcher. It specifies the Electric Displacement Field (D) at a boundary in this case the lower X boundary. The functions defining the D on the boundary are defined in the *SpaceTimeFunctions* element of the *Elements Tree*.

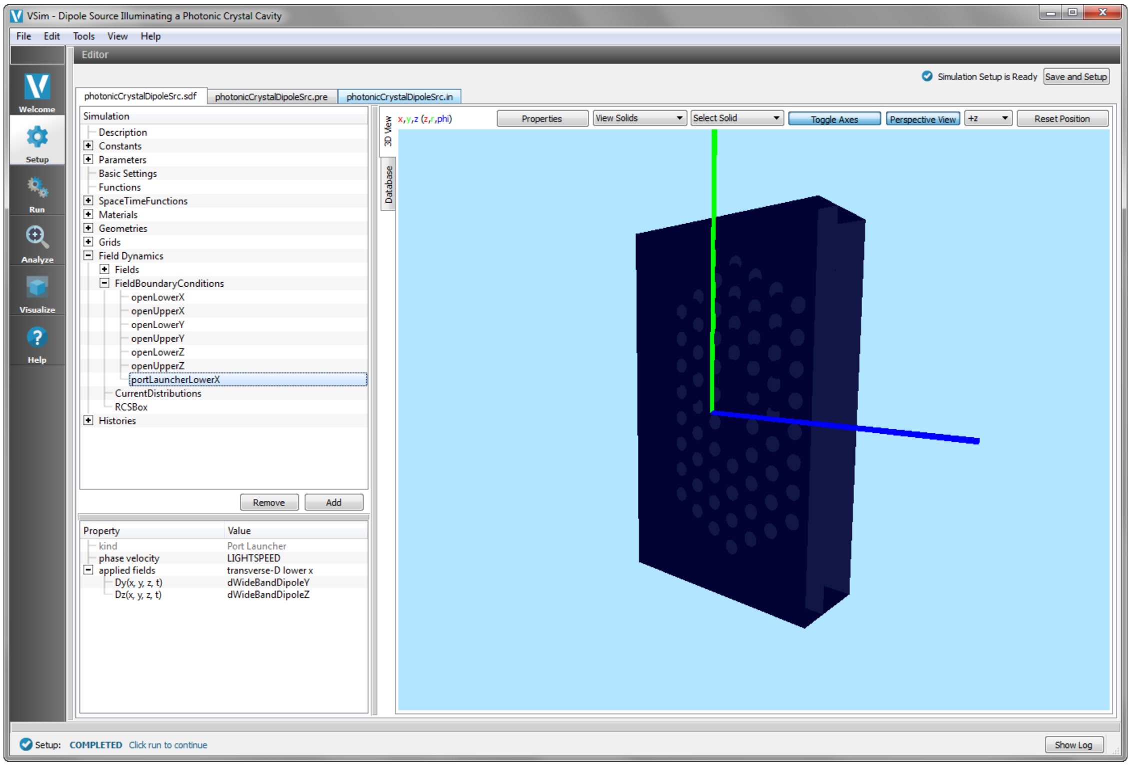
SpaceTimeFunctions:

- `dSingleFreqDipole{Y,Z}` = This is the {x,y,z} component of the single frequency dipole source; as seen in Fig. 3.94, you put this as a parameter in the PortLauncher.
- `dWideBandDipole{Y,Z}` = This is the {x,y,z} component of the wideband dipole source; as seen in Fig. 3.94, you put this as a parameter in the PortLauncher.

To choose which signal you want to input into this example simulation:

1. Expand the *FieldBoundaryConditions* tab.
2. Left click on the *portLauncherLowerX* condition.

At this point, you should see what is shown in Fig. 3.94:



Simulation
- Description
- Constants
- Parameters
- Basic Settings
- Functions
- SpaceTimeFunctions
- Materials
- Geometries
- Grids
- Field Dynamics
  - Fields
  - FieldBoundaryConditions
    - openLowerX
    - openUpperX
    - openLowerY
    - openUpperY
    - openLowerZ
    - openUpperZ
    - portLauncherLowerX
  - CurrentDistributions
  - RCSBox
- Histories

Property	Value
kind	Port Launcher
phase velocity	LIGHTSPEED
applied fields	transverse-D lower x
Dy(x, y, z, t)	dWideBandDipoleY
Dz(x, y, z, t)	dWideBandDipoleZ

Setup: COMPLETED Click run to continue

Fig. 3.94: The Setup Window showing where to specifying the applied field signal for the port launcher

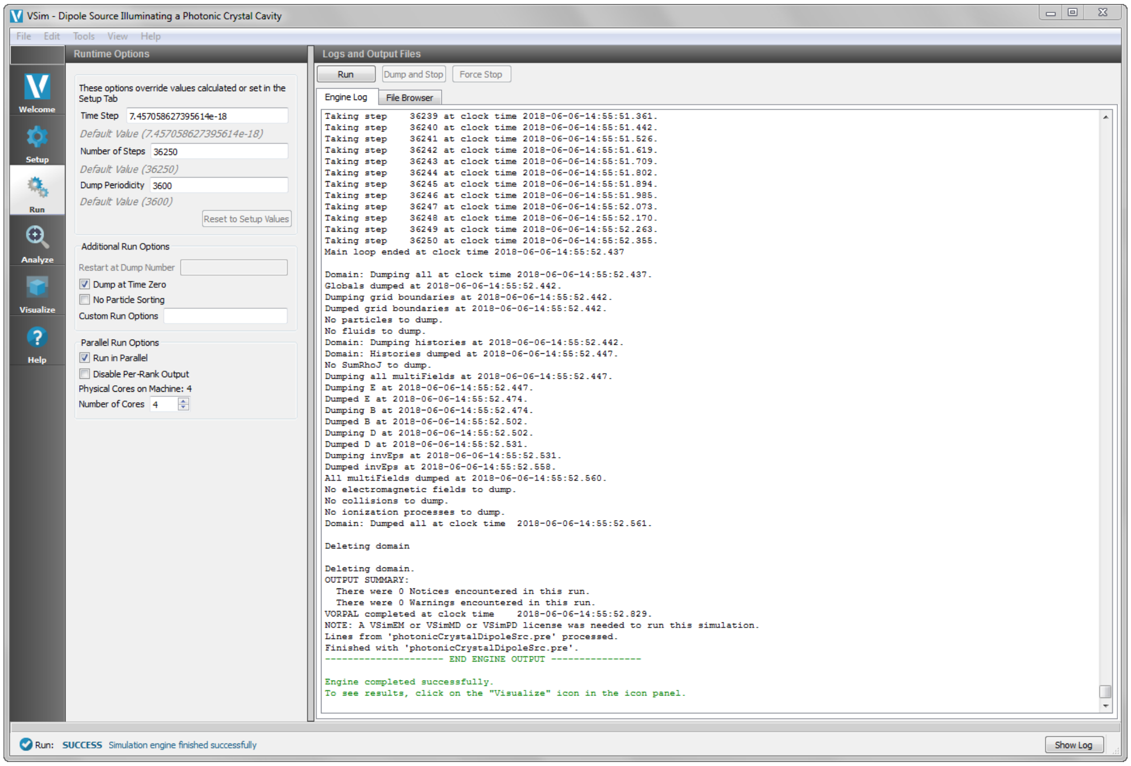
3. To change the signal applied to the *portLauncherLowerX*, right click on the Dx(x, y, z, t) or Dy(x, y, z, t) property and select *Assign SpaceTimeFunction*. This will expand another menu that will show you all four defined SpaceTimeFunctions. Select which one you want to input into your simulation. For this documentation, `dWideBandDipole{Y,Z}` is selected by default to demonstrate the functionality of the example.

Running the Simulation

Once finished with the problem setup, continue as follows:

- Proceed to the *Run Window* by pressing the *Run* button in the left column of buttons.
- One can enable MPI options to utilize multi-core systems.
- The default values of `Number of Time Steps` and `Dump Periodicity` are taken from the parameters `STEPSTOTAL` and `STEPSPERDUMP`, which use the constants `SIMCYCLES` and `CYCLESPERDUMP`. The formulae for these variables can be found back in the *Setup Window*. These variables are for convenience to calculate good default values and it is important to know that the override option default values ultimately come directly from the numbers in the *Basic Settings* section.
  - *Number of Steps* = `STEPSTOTAL` = 36250
  - *Dump Periodicity* = `STEPSPERDUMP` = 3600

- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane. You will see the output of the run in that pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 3.95.



These options override values calculated or set in the Setup Tab

Time Step: 7.457058627395614e-18
Default Value (7.457058627395614e-18)
Number of Steps: 36250
Default Value (36250)
Dump Periodicity: 3600
Default Value (3600)

Additional Run Options
Restart at Dump Number:
☑ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options:

Parallel Run Options
☑ Run in Parallel
☐ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores: 4

```
Taking step    36239 at clock time 2018-06-06-14:55:51.361.
Taking step    36240 at clock time 2018-06-06-14:55:51.442.
Taking step    36241 at clock time 2018-06-06-14:55:51.526.
Taking step    36242 at clock time 2018-06-06-14:55:51.619.
Taking step    36243 at clock time 2018-06-06-14:55:51.709.
Taking step    36244 at clock time 2018-06-06-14:55:51.802.
Taking step    36245 at clock time 2018-06-06-14:55:51.894.
Taking step    36246 at clock time 2018-06-06-14:55:51.985.
Taking step    36247 at clock time 2018-06-06-14:55:52.073.
Taking step    36248 at clock time 2018-06-06-14:55:52.170.
Taking step    36249 at clock time 2018-06-06-14:55:52.263.
Taking step    36250 at clock time 2018-06-06-14:55:52.355.
Main loop ended at clock time 2018-06-06-14:55:52.437

Domain: Dumping all at clock time 2018-06-06-14:55:52.437.
Globals dumped at 2018-06-06-14:55:52.442.
Dumping grid boundaries at 2018-06-06-14:55:52.442.
Dumped grid boundaries at 2018-06-06-14:55:52.442.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-06-06-14:55:52.442.
Domain: Histories dumped at 2018-06-06-14:55:52.447.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-06-14:55:52.447.
Dumping E at 2018-06-06-14:55:52.447.
Dumped E at 2018-06-06-14:55:52.474.
Dumping B at 2018-06-06-14:55:52.474.
Dumped B at 2018-06-06-14:55:52.502.
Dumping D at 2018-06-06-14:55:52.502.
Dumped D at 2018-06-06-14:55:52.531.
Dumping invEps at 2018-06-06-14:55:52.531.
Dumped invEps at 2018-06-06-14:55:52.558.
All multiFields dumped at 2018-06-06-14:55:52.560.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-06-14:55:52.561.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-06-14:55:52.829.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.
Lines from 'photonicCrystalDipoleSrc.pre' processed.
Finished with 'photonicCrystalDipoleSrc.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 3.95: The Run Window for the Dipole Source Illuminating a Photonic Crystal Cavity example

Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the *Visualize Window* by pressing the *Visualize* button in the left column of buttons.

In the simulation, there are specific grid points which store field histories. These histories are placed in various positions of the simulation.

- In the top field of *Visualization Controls*, click the drop down menu and select *History*.

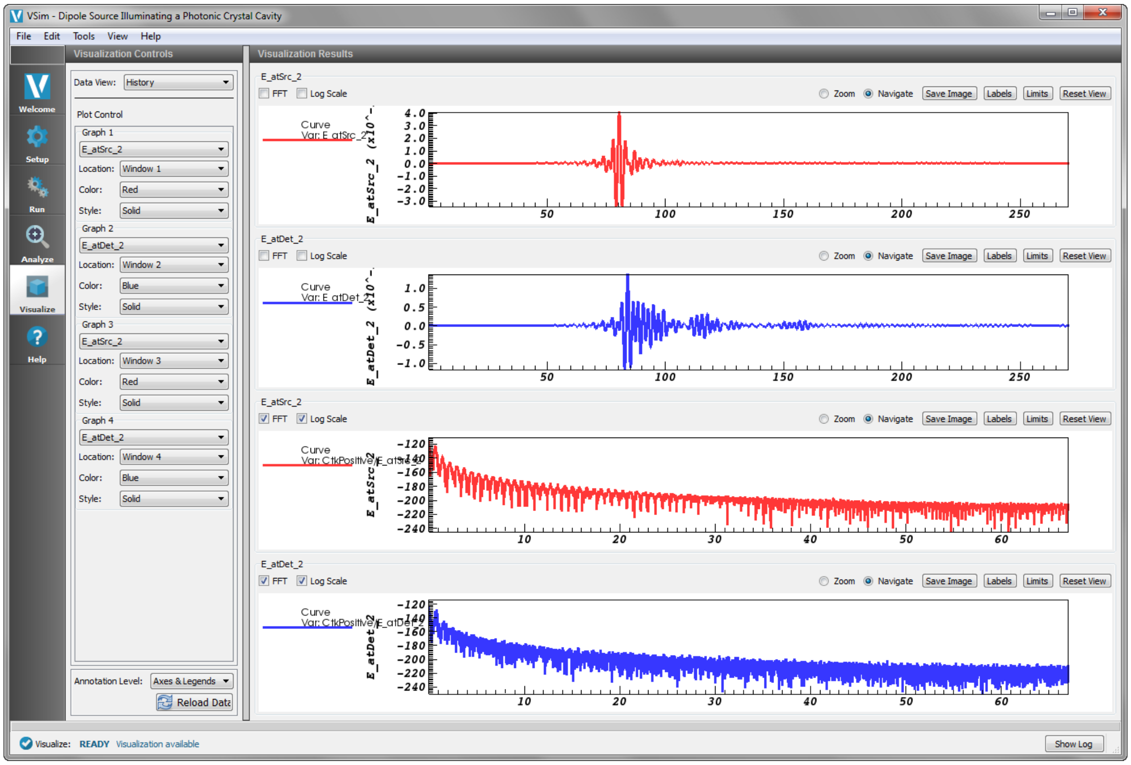
In Fig. 3.96, one can see there are 4 possible graphs to view at one time in the *Visualize Window*. For each graph, one can select the following fields to analyze: (0 = x, 1 = y, 2 = z).

- {E,B}_AtDet_{0,1,2} = In the middle of (y,z) plane and 60 nm above the surface of the crystal.
- {E,B}_AtSrc_{0,1,2} = Is aligned with the inCav history in the (y,z) plane and is 60 nm below the Si layer, into the SiO2 layer.
- {E,B}_InCav_{0,1,2} = Is slightly offset from the middle of one of the cavities in the Silicon layer (the layer with the lattice).

In each individual graph, one can choose the FFT option to view the frequency domain of your field. This can enable the analysis of the frequency response of the photonic crystal cavity.

Fig. 3.96 depicts four graphs of histories. The first two graphs are amplitude vs time, and the second two are a FFT of the first two on a log scale.

The first and third graphs depict the history `AtSrc_2`, while the second and fourth graphs show the *AtDet_2* history.



VSim - Dipole Source Illuminating a Photonic Crystal Cavity

Data View: History

Plot Control

Graph 1: E_atSrc_2; Location: Window 1; Color: Red; Style: Solid

Graph 2: E_atDet_2; Location: Window 2; Color: Blue; Style: Solid

Graph 3: E_atSrc_2; Location: Window 3; Color: Red; Style: Solid

Graph 4: E_atDet_2; Location: Window 4; Color: Blue; Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 3.96: The Visualize Window for the Dipole Source Illuminating a Photonic Crystal Cavity example

Further Experiments

By using the wideband source and examining the field strength detected below the crystal lattice, one may study the frequency response of this photonic crystal as one changes the device geometry, the dielectric constants, and the location and polarizations of the radiation source and detector.

3.6 Photonics (text-based setup)

3.6.1 Dipole Source Illuminating a Photonic Crystal Cavity (photonicCrystalDipoleSrcT.pre)

Keywords:

`dipole source, photonic crystal, transmission efficiency`

Problem description

This example illustrates how to model a dipole source that is illuminating a cavity inside a hexagonal photonic crystal lattice. The physical arrangement is shown in Fig. 3.97 and Fig. 3.98.

A point-like dipole lies above the simulation domain, which is comprised of three layers: a vacuum region above and a solid dielectric below, which together sandwich a central dielectric layer that contains a lattice of holes. This example includes two possible time signals with which to ring the dipole source, as shown in Fig. 3.99.

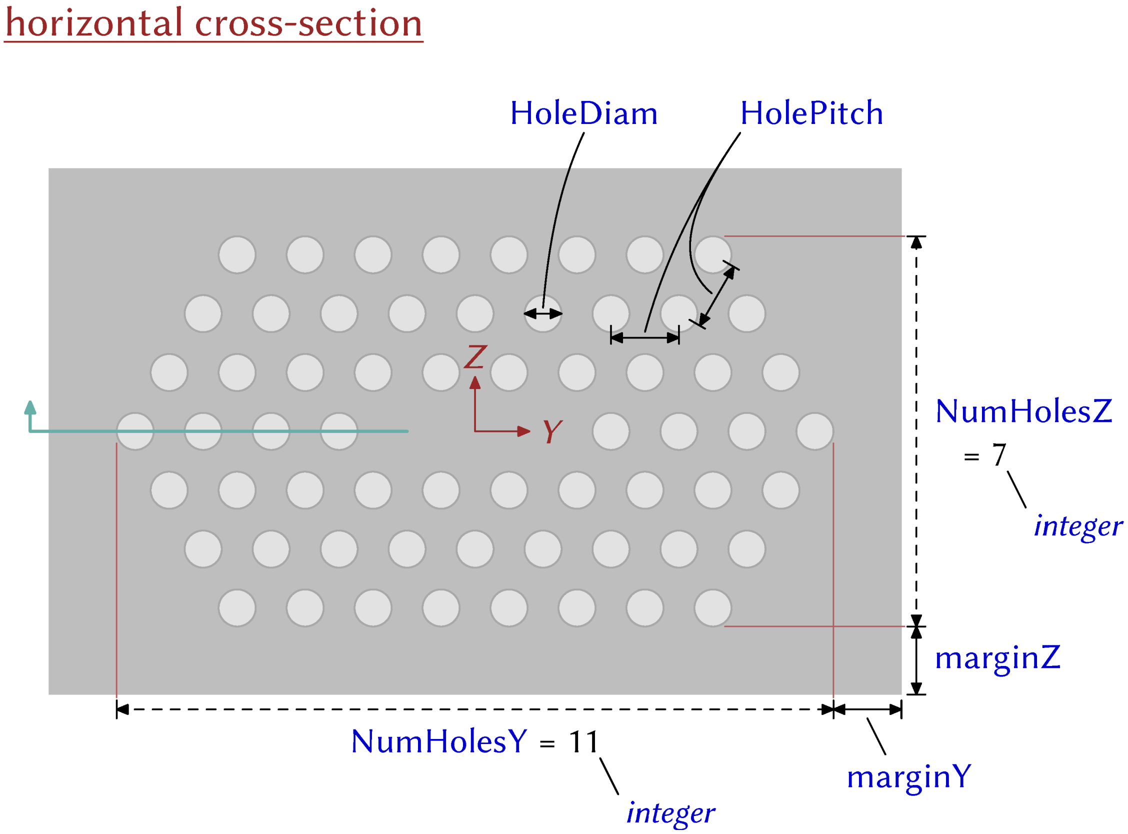


Fig. 3.97: Top view of photonic lattice.

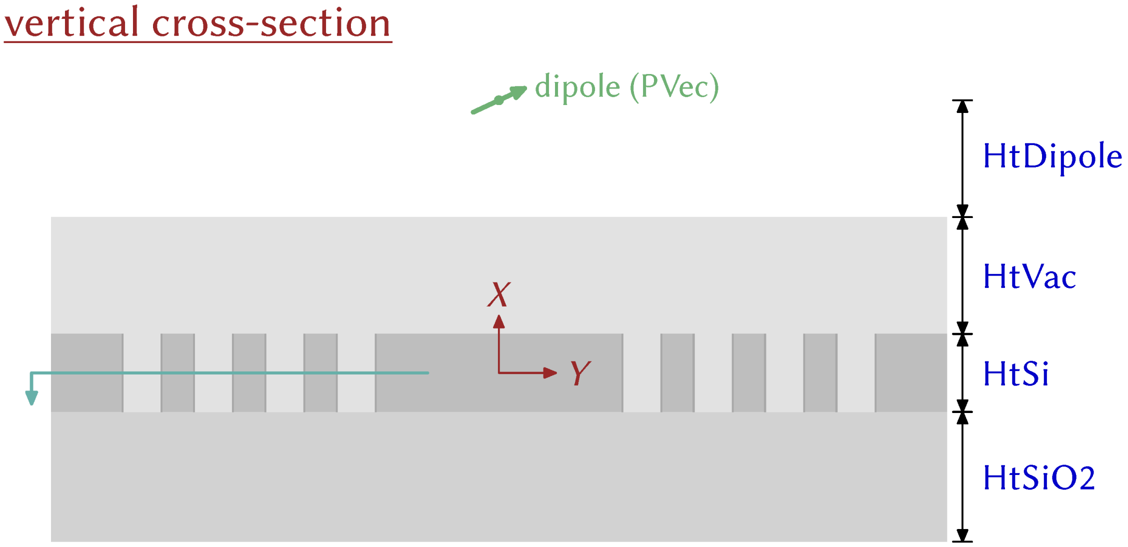


Fig. 3.98: Side view of photonic lattice.

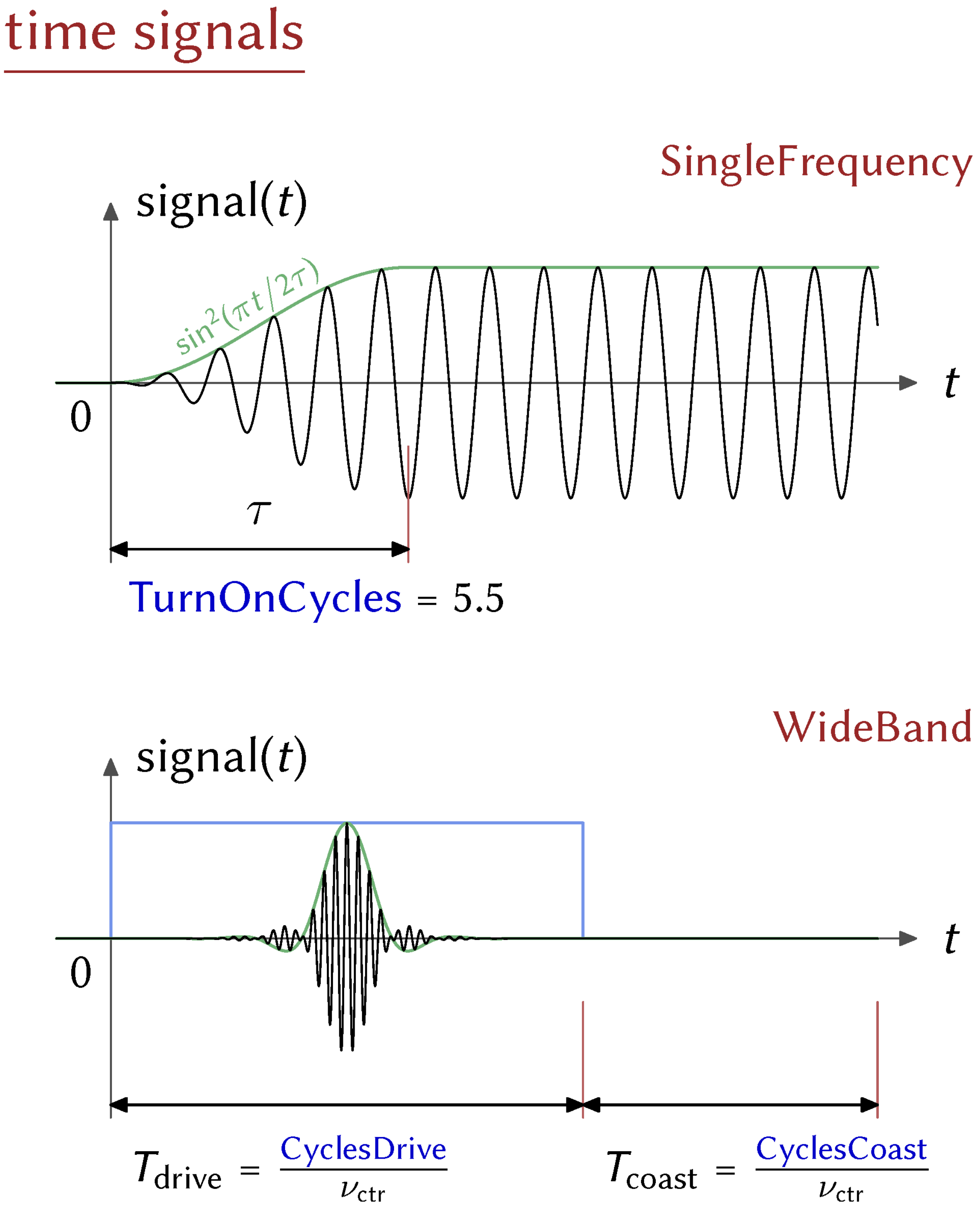

time signals
SingleFrequency
signal(t)
$\sin^2(\pi t/2\tau)$
0
t
τ
TurnOnCycles = 5.5
WideBand
signal(t)
0
t
$T_{\text{drive}} = \frac{\text{CyclesDrive}}{\nu_{\text{ctr}}}$
$T_{\text{coast}} = \frac{\text{CyclesCoast}}{\nu_{\text{ctr}}}$

Fig. 3.99: Two possible time signals for ringing the dipole source.

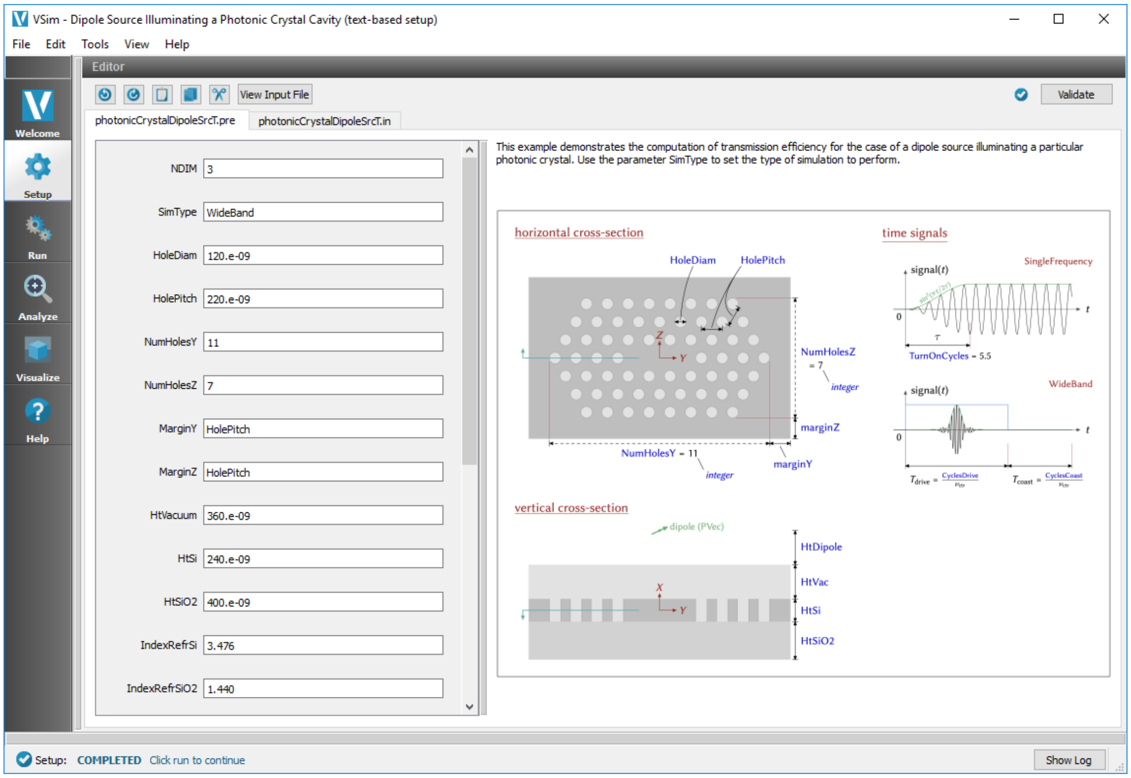
This simulation can be performed with a VSimEM license.

Opening the Simulation

This Photonic Crystal example is accessed from within VSimComposer through the following steps:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window, expand the *VSim for Electromagnetics* option.
- Expand the *Photonics (Text-based setup)* option.
- Select "Dipole Source Illuminating a Photonic Crystal Cavity (Text-based setup)" and press the *Choose* button.
- In the resulting dialog, create a new folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 3.100. A *WideBand* simulation type (see the *SimType* input value) will be used to demonstrate the functionality of this example.



NDIM 3

SimType WideBand

HoleDiam 120.e-09

HolePitch 220.e-09

NumHolesY 11

NumHolesZ 7

MarginY HolePitch

MarginZ HolePitch

HtVacuum 360.e-09

HtSi 240.e-09

HtSiO2 400.e-09

IndexRefrSi 3.476

IndexRefrSiO2 1.440

This example demonstrates the computation of transmission efficiency for the case of a dipole source illuminating a particular photonic crystal. Use the parameter SimType to set the type of simulation to perform.

Setup: COMPLETED Click run to continue

Fig. 3.100: The Setup Window for the Dipole Source Illuminating a Photonic Crystal Cavity (Text-based setup)example

Input File Features

The Input File is shown in Fig. 3.100. This specifies the parameters of the simulation in SI units, see Fig. 3.98 for clarification.

General simulation parameters:

- NDIM = Number of dimensions (3).

- `SimType` = {WideBand, SingleFrequency}
    - `WideBand` = Runs the simulation with a wideBand electromagnetic wave as the source in the simulation. The CyclesDrive parameter dictates how large this pulse is.
    - `SingleFrequency` = Runs the simulation with a single frequency source. This frequency is determined by the waveLengthCtr parameter.
- `SimCycles` = Number of cycles run in simulation.
- `CyclesDrive` = Number of cycles that the signal is driven.
- `CyclesPerDump` = Number of cyles per dump.
- `NominalCellSize` = The size of your cells; should be small enough to resolve your dielectric cavities in the crystal.

Photonic crystal specifications:

- `HoleDiam` = Hole diameter, in meters.
- `HolePitch` = Hole spacing in the hexagonal lattice, in meters.
- `NumHolesY` = Number of rows of holes in the photonic crystal lattice.
- `NumHolesZ` = Number of columns of holes in the photonic crystal lattice.
- `HtVacuum` = Height of vacuum, along x-axis.
- `HtSi` = Height of silicon layer, along x-axis.
- `HtSiO2` = Height of silica layer, along x-axis.
- `IndexRefrSiO2` = Index of refraction for silica.
- `IndexRefrSi` = Index of refraction for silicon.
- `MarginY` = Size of buffer zone on each side of the photonic crystal in the y-direction.
- `MarginZ` = Size of buffer zone on each side of the photonic crystal in the z-direction.

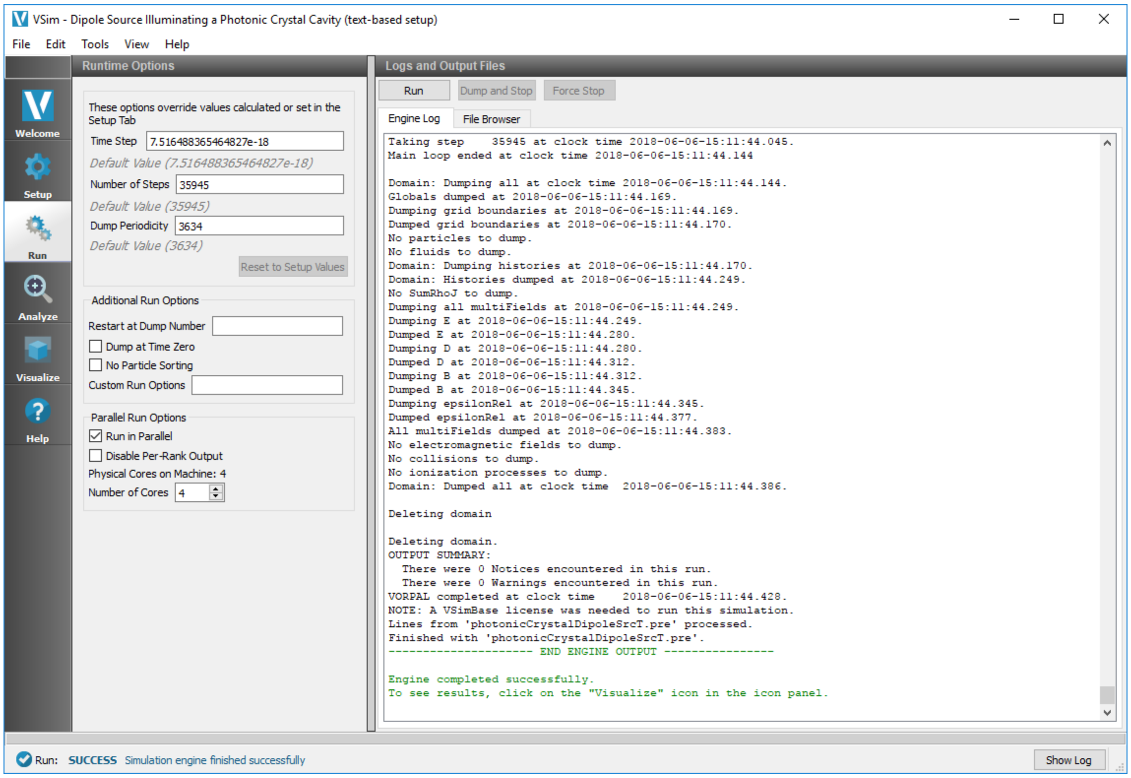
Source specifications:

- `HtDipole` = Height of dipole, along x-axis.
- `PVec{x,y,z}` = The {x,y,z}-component of your dipole vector.
- `WaveLengthCtr` = The central wavelength of your wideband signal, and is the frequency used as the single frequency simulation type.
- `WaveLengthBand` = The wavelength width of your wideband signal, only used in wideBand simType.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the *Run Window* by pressing the *Run* button in the left column of buttons.
- One can enable MPI options to utilize multi-core systems.
- The default values of `Number of Time Steps` and `Dump Periodicity` are calculated from the *CycleDrive* and *CyclesPerDump* in the *Setup Window* shown in Fig. 3.100.
- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 3.101.



These options override values calculated or set in the Setup Tab

Time Step 7.516488365464827e-18

Default Value (7.516488365464827e-18)

Number of Steps 35945

Default Value (35945)

Dump Periodicity 3634

Default Value (3634)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Taking step    35945 at clock time 2018-06-06-15:11:44.045.
Main loop ended at clock time 2018-06-06-15:11:44.144

Domain: Dumping all at clock time 2018-06-06-15:11:44.144.
Globals dumped at 2018-06-06-15:11:44.169.
Dumping grid boundaries at 2018-06-06-15:11:44.169.
Dumped grid boundaries at 2018-06-06-15:11:44.170.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-06-06-15:11:44.170.
Domain: Histories dumped at 2018-06-06-15:11:44.249.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-06-15:11:44.249.
Dumping E at 2018-06-06-15:11:44.249.
Dumped E at 2018-06-06-15:11:44.280.
Dumping D at 2018-06-06-15:11:44.280.
Dumped D at 2018-06-06-15:11:44.312.
Dumping B at 2018-06-06-15:11:44.312.
Dumped B at 2018-06-06-15:11:44.345.
Dumping epsilonRel at 2018-06-06-15:11:44.345.
Dumped epsilonRel at 2018-06-06-15:11:44.377.
All multiFields dumped at 2018-06-06-15:11:44.383.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-06-15:11:44.386.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-06-15:11:44.428.
NOTE: A VSimBase license was needed to run this simulation.
Lines from 'photonicCrystalDipoleSrcT.pre' processed.
Finished with 'photonicCrystalDipoleSrcT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 3.101: The Run Window for the Dipole Source Illuminating a Photonic Crystal Cavity (Text-based setup) example

Visualizing the results

After performing the above actions, continue as follows:

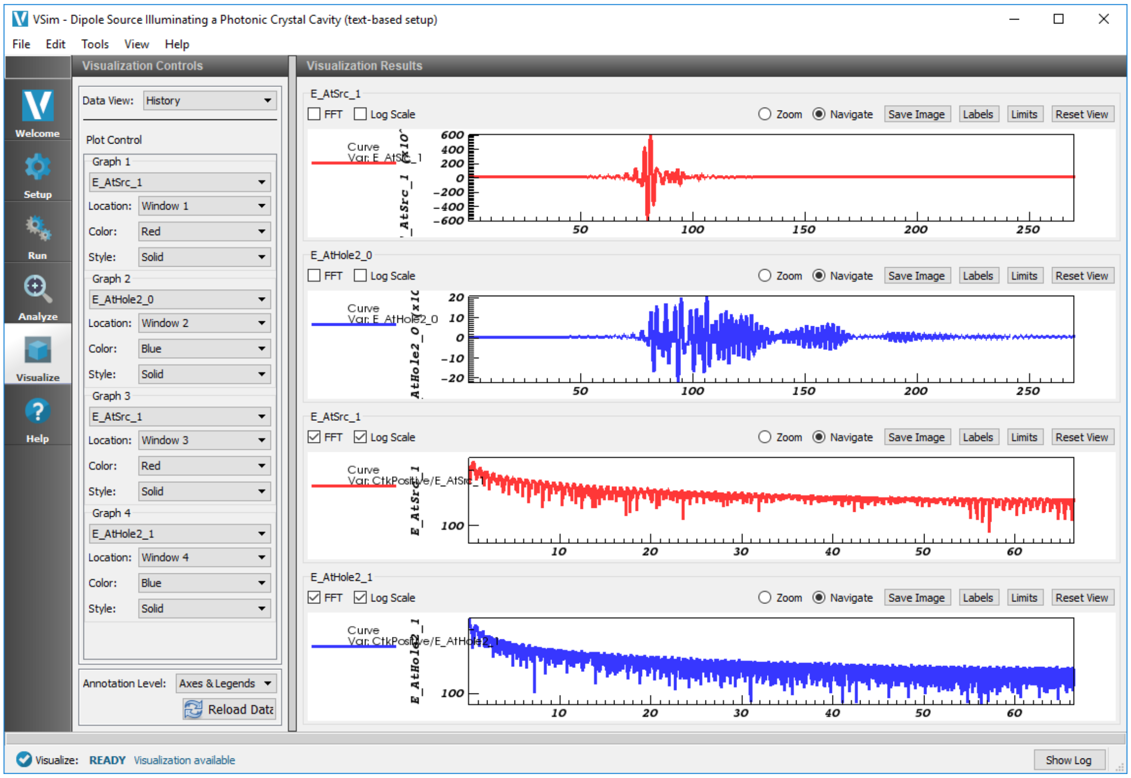
- Proceed to the *Visualize Window* by pressing the **Visualize** button in the left column of buttons.
- In the top field of *Visualization Controls*, click the drop down menu entitled *Data View* and select *History*.

In the simulation, there are specific grid points which store field histories. These histories are placed in various positions of the simulation.

In Fig. 3.102, one can see there are 4 possible graphs to view at one time in the *Visualize Window*. For each graph, one can select the following fields to anaylize: (0 = x, 1 = y, 2 = z).

- {E,B}_AtCtr_{0,1,2} = {0,0} in {y,z} plane in the middle of SiLayer.
- {E,B}_AtDet_{0,1,2} = Just above the upper x boundary.
- {E,B}_AtSrc_{0,1,2} = Just below the source.
- {E,B}_InCav_{0,1,2} = In the cavity off center.
- E_AtHole_{0,1,2} = In the middle of the second hole.
- E_AtMargin{Z,Y}_{0,1,2} = Just inside the Z or Y margin.
- poyntingFluxDet = Just above the upper x boundary.
- poyntingFluxSrc = Just below the source.

In each individual graph, one can choose the FFT option to view the frequency domain of your field. This can enable the analysis of the frequency response of the photonic crystal cavity. Fig. 3.102 depicts four graphs of histories. The first two graphs are amplitude vs time, and the second two are a FFT of the first two on a log scale. The first and third graphs depict the history `AtSrc_1`, while the second and fourth graphs show the `AtHole2_1` history.



Data View: History

Graph 1: E_AtSrc_1; Location: Window 1; Color: Red; Style: Solid

Graph 2: E_AtHole2_0; Location: Window 2; Color: Blue; Style: Solid

Graph 3: E_AtSrc_1; Location: Window 3; Color: Red; Style: Solid

Graph 4: E_AtHole2_1; Location: Window 4; Color: Blue; Style: Solid

Annotation Level: Axes & Legends

E_AtSrc_1: ☐ FFT ☐ Log Scale ○ Zoom ● Navigate

E_AtHole2_0: ☐ FFT ☐ Log Scale ○ Zoom ● Navigate

E_AtSrc_1: ☑ FFT ☑ Log Scale ○ Zoom ● Navigate

E_AtHole2_1: ☑ FFT ☑ Log Scale ○ Zoom ● Navigate

Visualize: READY Visualization available

Fig. 3.102: The Visualize Window for the Dipole Source Illuminating a Photonic Crystal Cavity (Text-based setup) example

Further Experiments

By using the wide-band source and examining the field strength detected below the crystal lattice, one may study the frequency response of this photonic crystal as one changes the device geometry, the dielectric constants, and the location and polarizations of the radiation source and detector.

3.6.2 Gaussian Laser Beam and Photonic Crystal Cavity (photonicCrystalGaussSrcT.pre)

Keywords:

`Gaussian Beam source, photonic crystal, transmission efficiency`

Problem description

This example illustrates how to model a Gaussian beam source that is illuminating a cavity inside a hexagonal photonic crystal lattice. The physical arrangement is shown in Fig. 3.103.

A Gaussian beam is launched from above into the simulation domain, which comprises three layers: a vacuum region above and a solid dielectric below, which together sandwich a central dielectric layer that contains a lattice of holes.

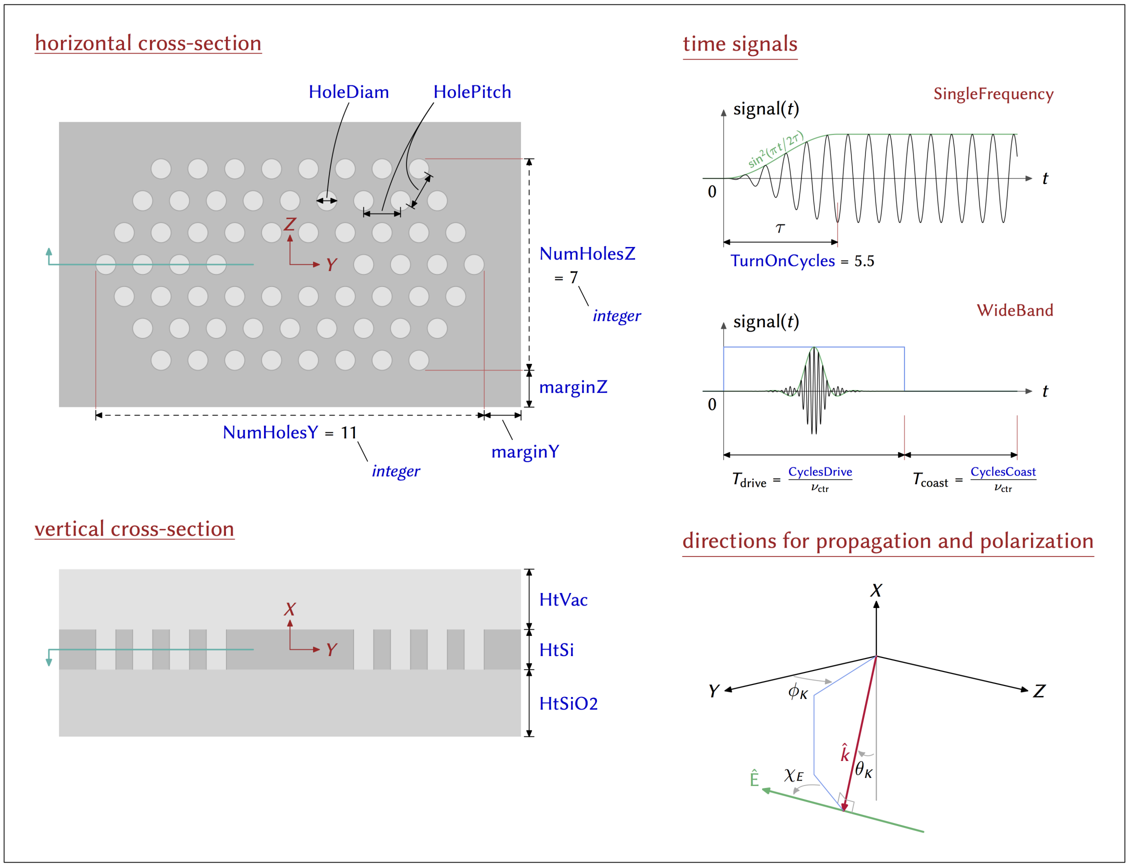


Fig. 3.103: Top view of the photonic lattice.

This example includes two possible time signals for the Gaussian beam: WideBand or SingleFrequency, as shown in Fig. 3.103.

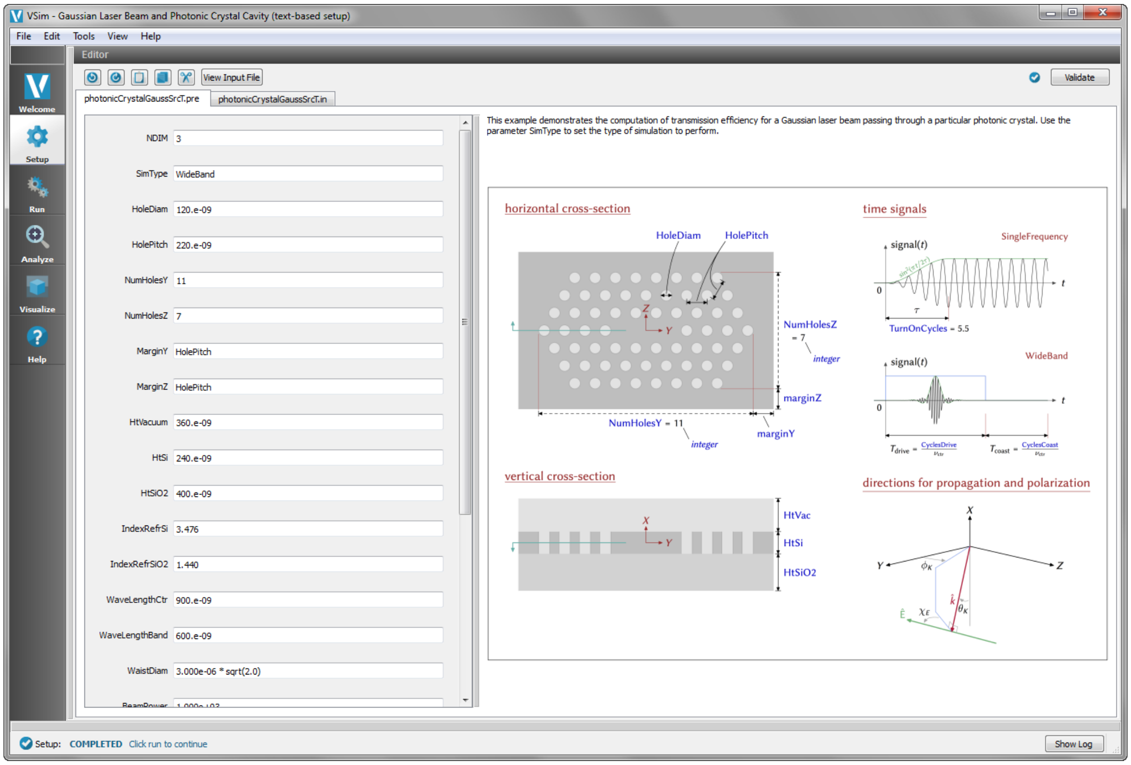
This simulation can be performed with a VSimEM license.

Opening the Simulation

This example is accessed from within VSimComposer through the following steps:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window, expand the *VSim for Electromagnetics* option.
- Expand the *Photonics (Text-based setup)* option.
- Select "Gaussian Laser Beam and Photonic Crystal Cavity (Text-based setup)" and press the *Choose* button.
- In the resulting dialog, create a new folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 3.104. For this documentation, a `WideBand SimType` will be used to demonstrate the functionality of this example.



NDIM 3
SimType WideBand
HoleDiam 120.e-09
HolePitch 220.e-09
NumHolesY 11
NumHolesZ 7
MarginY HolePitch
MarginZ HolePitch
HtVacuum 360.e-09
HtSi 240.e-09
HtSiO2 400.e-09
IndexRefrSi 3.476
IndexRefrSiO2 1.440
WaveLengthCtr 900.e-09
WaveLengthBand 600.e-09
WaistDiam 3.000e-06 * sqrt(2.0)
BeamPower 1.000e+03

This example demonstrates the computation of transmission efficiency for a Gaussian laser beam passing through a particular photonic crystal. Use the parameter SimType to set the type of simulation to perform.

Setup: COMPLETED Click run to continue

Fig. 3.104: The Setup Window for the Gaussian Laser Beam and Photonic Crystal Cavity (Text-based setup) example

Input File Features

The setup shown in Fig. 3.104 specifies the parameters of the simulation in SI units. For a visual reference of what these variables do, consult Fig. 3.103.

General simulation parameters:

- `NDIM` = Number of dimensions (3).
- `SimType` = {WideBand, SingleFrequency}.
  - `WideBand` = Runs the simulation with a wideBand electromagnetic wave as the source in the simulation. The CyclesDrive parameter dictates how large this pulse is.
  - `SingleFrequency` = Runs the simulation with a single frequency source. This frequency is determined by the waveLengthCtr parameter.
- `SimCycles` = Number of cycles run in simulation.
- `CyclesDrive` = Number of cycles to drive the wideband source.
- `CyclesPerDump` = Number of cyles per dump.
- `NominalCellSize` = The size of your cells, should be small enough to resolve your dielectric cavities in the crystal.

Photonic crystal specifications:

- `HoleDiam` = Diameter of each hole in crystal lattice.
- `HolePitch` = Distance between the center of each hole, all distances equal due to lattice.
- `NumHolesY` = Number of rows of holes.
- `NumHolesZ` = Number of columns of holes.
- `HtVacuum` = Height of vacuum, along x-axis.
- `HtSi` = Height of silicon layer, along x-axis.
- `HtSiO2` = Height of silica layer, along x-axis.
- `IndexRefrSiO2` = Index of refraction for silica.
- `IndexRefrSi` = Index of refraction for silicon.
- `MarginY` = How far the edge of the crystal is from your lattice in the x-direction.
- `MarginZ` = how far the edge of the crystal is from your lattice in the z-direction.

Source specifications:

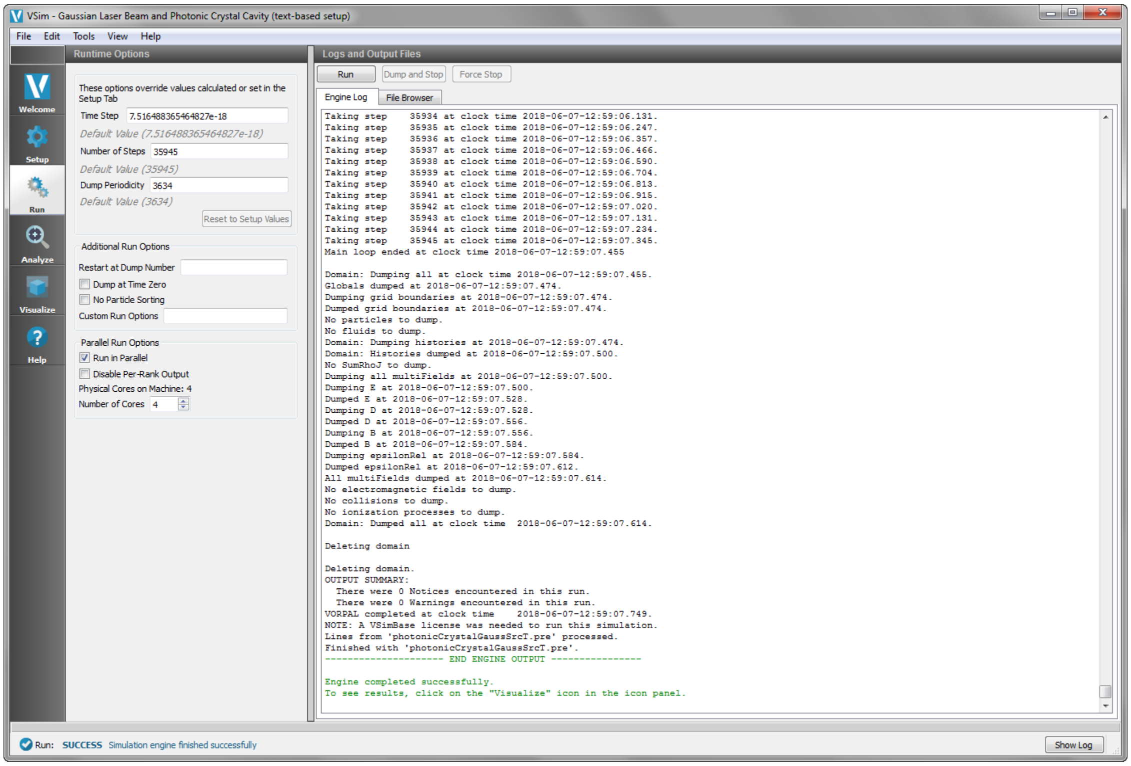
- `BeamPowr` = Power of the beam.
- `WaistDiam` = Diameter of your Gaussian beam waist.
- `{ThetaK,phiK,chiE}` = The {polar angle, azimuthal, angle of polarization} respectively.
- `WaveLengthCtr` = The central wavelength of your wideband singal, and is the frequency used as the single frequency simulation type.
- `WaveLengthBand` = The wavelength width of your wide band signal, only used in wideBand simType.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the *Run Window* by pressing the *Run* button in the left column of buttons.
- MPI can be enabled to utilize multi-core systems.
- The default values of `Number of Time Steps` and `Dump Periodicity` are calculated using by the CyclesDrive and CyclesPerDump inputs in the *Setup Window*.

- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 3.105.



These options override values calculated or set in the Setup Tab

Time Step 7.516488365464827e-18
Default Value (7.516488365464827e-18)
Number of Steps 35945
Default Value (35945)
Dump Periodicity 3634
Default Value (3634)

Additional Run Options
Restart at Dump Number
□ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options
☑ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
Taking step    35934 at clock time 2018-06-07-12:59:06.131.
Taking step    35935 at clock time 2018-06-07-12:59:06.247.
Taking step    35936 at clock time 2018-06-07-12:59:06.357.
Taking step    35937 at clock time 2018-06-07-12:59:06.466.
Taking step    35938 at clock time 2018-06-07-12:59:06.590.
Taking step    35939 at clock time 2018-06-07-12:59:06.704.
Taking step    35940 at clock time 2018-06-07-12:59:06.813.
Taking step    35941 at clock time 2018-06-07-12:59:06.915.
Taking step    35942 at clock time 2018-06-07-12:59:07.020.
Taking step    35943 at clock time 2018-06-07-12:59:07.131.
Taking step    35944 at clock time 2018-06-07-12:59:07.234.
Taking step    35945 at clock time 2018-06-07-12:59:07.345.
Main loop ended at clock time 2018-06-07-12:59:07.455

Domain: Dumping all at clock time 2018-06-07-12:59:07.455.
Globals dumped at 2018-06-07-12:59:07.474.
Dumping grid boundaries at 2018-06-07-12:59:07.474.
Dumped grid boundaries at 2018-06-07-12:59:07.474.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-06-07-12:59:07.474.
Domain: Histories dumped at 2018-06-07-12:59:07.500.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-07-12:59:07.500.
Dumping E at 2018-06-07-12:59:07.500.
Dumped E at 2018-06-07-12:59:07.528.
Dumping D at 2018-06-07-12:59:07.528.
Dumped D at 2018-06-07-12:59:07.556.
Dumping B at 2018-06-07-12:59:07.556.
Dumped B at 2018-06-07-12:59:07.584.
Dumping epsilonRel at 2018-06-07-12:59:07.584.
Dumped epsilonRel at 2018-06-07-12:59:07.612.
All multiFields dumped at 2018-06-07-12:59:07.614.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-07-12:59:07.614.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-07-12:59:07.749.
NOTE: A VSimBase license was needed to run this simulation.
Lines from 'photonicCrystalGaussSrcT.pre' processed.
Finished with 'photonicCrystalGaussSrcT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 3.105: The Run Window for the Gaussian Laser Beam and Photonic Crystal Cavity (Text-based setup) example

Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the *Visualize Window* by pressing the *Visualize* button in the left column of buttons.

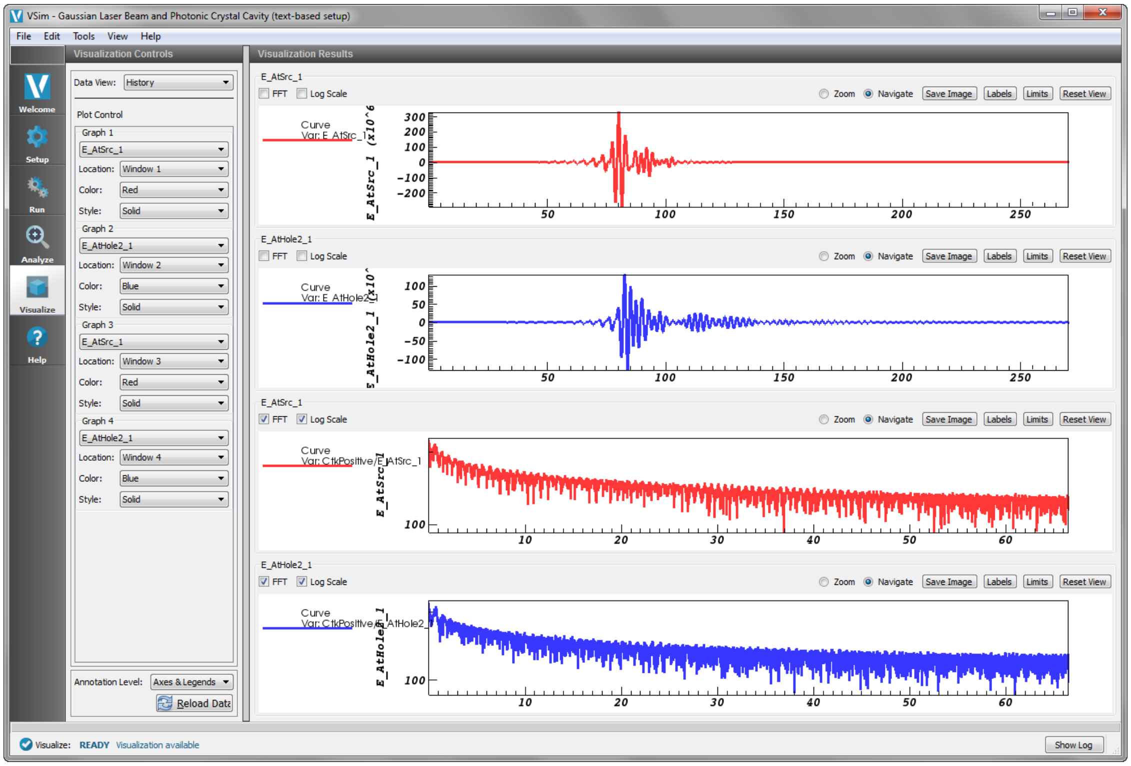
The input file specifies grid points at which to record field histories.

- In the top field of *Visualization Controls*, click the drop down menu and select *History*.

By default, there are 4 possible graphs viewable at one time in the *Visualize Window*. For each graph, one can select from the following fields: (0 = x, 1 = y, 2 = z).

- {E,B}_AtCtr_{0,1,2} = {0,0} in {y,z} plane in the middle of SiLayer.
- {E,B}_AtDet_{0,1,2} = Just above the upper x boundary.
- {E,B}_AtSrc_{0,1,2} = Just below the source.
- {E,B}_InCav_{0,1,2} = In the cavity off center.
- E_AtHole_{0,1,2} = In the middle of the second hole.
- E_AtMargin{Z,Y}_{0,1,2} = Just inside the z or y margin.
- poyntingFluxDet = Just above the upper x boundary.
- poyntingFluxSrc = Just below the source.

In each individual graph, one can choose the FFT option to view the frequency domain of your field. This can enable the analysis of the frequency response of the photonic crystal cavity. Below, Fig. 3.106 depicts four graphs of histories. The first two graphs are amplitude vs time, and the second two are a FFT of the first two on a log scale. The first and third graphs depict the history `AtSrc_2`, while the second and fourth graphs show the `AtDet_2` history.


VSim - Gaussian Laser Beam and Photonic Crystal Cavity (text-based setup)

Data View: History

Plot Control

Graph 1: E_AtSrc_1; Location: Window 1; Color: Red; Style: Solid

Graph 2: E_AtHole2_1; Location: Window 2; Color: Blue; Style: Solid

Graph 3: E_AtSrc_1; Location: Window 3; Color: Red; Style: Solid

Graph 4: E_AtHole2_1; Location: Window 4; Color: Blue; Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 3.106: The Visualize Window for the Gaussian Laser Beam and Photonic Crystal Cavity (Text-based setup) example

Further Experiments

By using the wide-band source and examining the field strength detected below the crystal lattice, one may study the frequency response of this photonic crystal as the device geometry, dielectric constants, location and polarizations of the radiation source and detector change.

3.6.3 Dielectric Waveguide with Mode Launcher using Point Permittivity (dielectricWaveguideModeLaunchPPT.pre)

Keywords:

`Mode Loading, Photonic Waveguide, Unidirectional Mode Launcher, MAL, Guided Mode, Semiconductor`

Problem description

The dielectric waveguide consists of a single, straight silicon waveguide that is parallel to the x-axis and centered at the origin. The waveguide is surrounded by silica. Matched Absorbing Layers (MALs) are used to dampen the E and B fields near the boundary of the simulation. This is a way to dampen reflected fields from the simulation boundaries.

The fundamental guided mode is launched in the silicon waveguide in the +x direction. The fundamental mode was extracted in the “Dielectric Waveguide Mode Calculation using Point Permittivty” example. The extractModesViaOperator.py analyzer produced the eigenmode vsh5 which is loaded into this simulation.

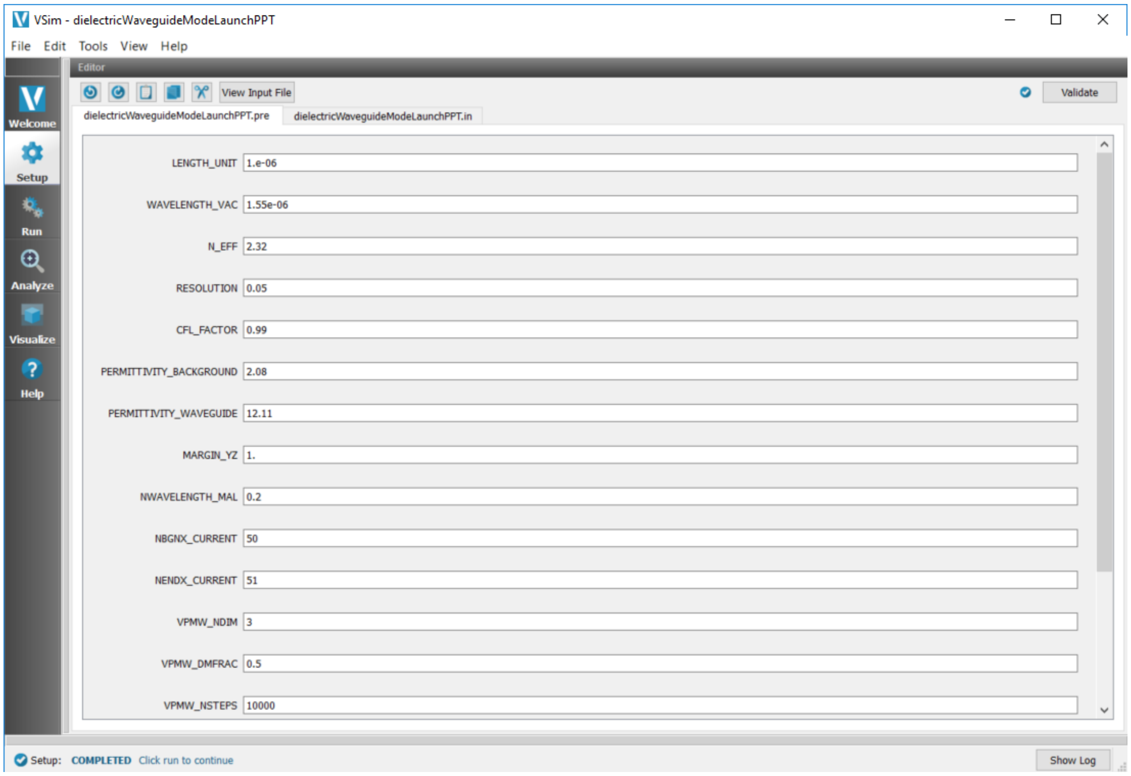
This simulation can be performed with a VSimEM license.

Opening the Simulation

The dielectric waveguide example can be accessed from within VSimComposer through the following steps:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window, expand the *VSim for Electromagnetics* option.
- Expand the *Photonics (text-based setup)* option.
- Select *Dielectric Waveguide with Mode Launcher using Point Permittivity (text-based setup)* and press the *Choose* button.
- In the resulting dialog, create a *New Folder* if desired, and press the *Save* button to create a copy of this example.

Some relvant parameters should now be visible as seen in Fig. 3.107. You can access more of the variables, functions, and geometries by clicking *View Input File* in the tool bar.


VSim - dielectricWaveguideModeLaunchPPT
File
Edit
Tools
View
Help
Editor
View Input File
Validate
dielectricWaveguideModeLaunchPPT.pre
dielectricWaveguideModeLaunchPPT.in
Welcome
Setup
Run
Analyze
Visualize
Help
Show Log

LENGTH_UNIT	1.e-06
WAVELENGTH_VAC	1.55e-06
N_EFF	2.32
RESOLUTION	0.05
CFL_FACTOR	0.99
PERMITTIVITY_BACKGROUND	2.08
PERMITTIVITY_WAVEGUIDE	12.11
MARGIN_YZ	1.
NWAVELENGTH_MAL	0.2
NBGNX_CURRENT	50
NENDX_CURRENT	51
VPMW_NDIM	3
VPMW_DMFRAC	0.5
VPMW_NSTEPS	10000

Setup: COMPLETED Click run to continue

Fig. 3.107: The Setup window for the dielectric waveguide example showing some relevant constants.

Simulation Properties

This example contains a number of constants defined to make the simulation easily modifiable. Some relevant constants are listed below.

PERMITTVITY_WAVEGUIDE and PERMITTVITY_BACKGROUND: Relative permittivities of silicon and silica. These constants are used in multiple parameters and in the accompanying Python file for solving the waveguide modes.

LENGTH_UNIT: The constant factor by which VSim will scale all simulation lengths.

WAVELENGTH_VAC: Wavelength of the input signal. This wavelength is also used for the calculation of the fundamental guided mode of the device.

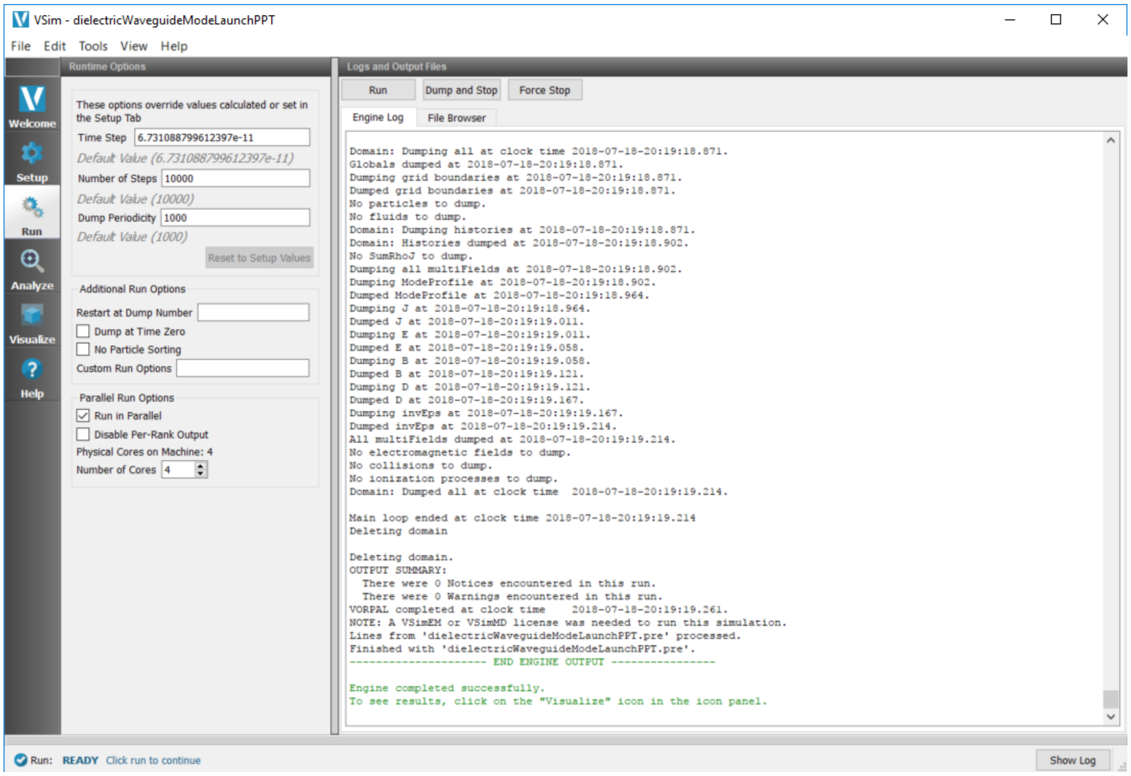
NWAVELENGTH_MAL: Approximate number of wavelengths that can fit in a MAL region. The thickness of the MAL regions in this example are measured in wavelengths.

In photonics simulations, Matched Absorbing Layers (MALs) are the most stable boundary conditions for preventing reflections. The eigen mode is imported form a vsh5 file. This can be seen in the input file.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the *Run* button in the left column. You will be asked to Save. Click *Save*.
- To run the file, click on the *Run* button in the upper left corner of the right pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” The result is shown in Fig. 3.108.



These options override values calculated or set in the Setup Tab

Time Step 6.731088799612397e-11

Default Value (6.731088799612397e-11)

Number of Steps 10000

Default Value (10000)

Dump Periodicity 1000

Default Value (1000)

Additional Run Options

Restart at Dump Number

☐ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Domain: Dumping all at clock time 2018-07-18-20:19:18.871.
Globals dumped at 2018-07-18-20:19:18.871.
Dumping grid boundaries at 2018-07-18-20:19:18.871.
Dumped grid boundaries at 2018-07-18-20:19:18.871.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-07-18-20:19:18.871.
Domain: Histories dumped at 2018-07-18-20:19:18.902.
No SumRhoJ to dump.
Dumping all multiFields at 2018-07-18-20:19:18.902.
Dumping ModeProfile at 2018-07-18-20:19:18.902.
Dumped ModeProfile at 2018-07-18-20:19:18.964.
Dumping J at 2018-07-18-20:19:18.964.
Dumped J at 2018-07-18-20:19:19.011.
Dumping E at 2018-07-18-20:19:19.011.
Dumped E at 2018-07-18-20:19:19.058.
Dumping B at 2018-07-18-20:19:19.058.
Dumped B at 2018-07-18-20:19:19.121.
Dumping D at 2018-07-18-20:19:19.121.
Dumped D at 2018-07-18-20:19:19.167.
Dumping invEps at 2018-07-18-20:19:19.167.
Dumped invEps at 2018-07-18-20:19:19.214.
All multiFields dumped at 2018-07-18-20:19:19.214.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-18-20:19:19.214.

Main loop ended at clock time 2018-07-18-20:19:19.214
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-07-18-20:19:19.261.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'dielectricWaveguideModeLaunchPPT.pre' processed.
Finished with 'dielectricWaveguideModeLaunchPPT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: READY Click run to continue

Fig. 3.108: The output after a successful run.

Visualizing the Results

Then proceed to the Visualize window by pressing the *Visualize* button in the left column.

A useful visualization of the dielectric waveguide would be to view the y-component of the E field to qualitatively see the mode propagate down the waveguide.

- Near the top left corner of the window, make sure *Data View* is set to *Data Overview*.
- Expand *Scalar Data*, expand *E*, and select *E_y*
- In the controls below the variables frame, select *Clip All Plots*.
- In the top of the screen, press the button that's titled *Colors*, check the Fix Minimum and Fix Maximum buttons, and input {-.03, .03} for the min and max, respectively.

Fig. 3.109 shows an example of what one should expect if one has run the simulation for enough cycles.

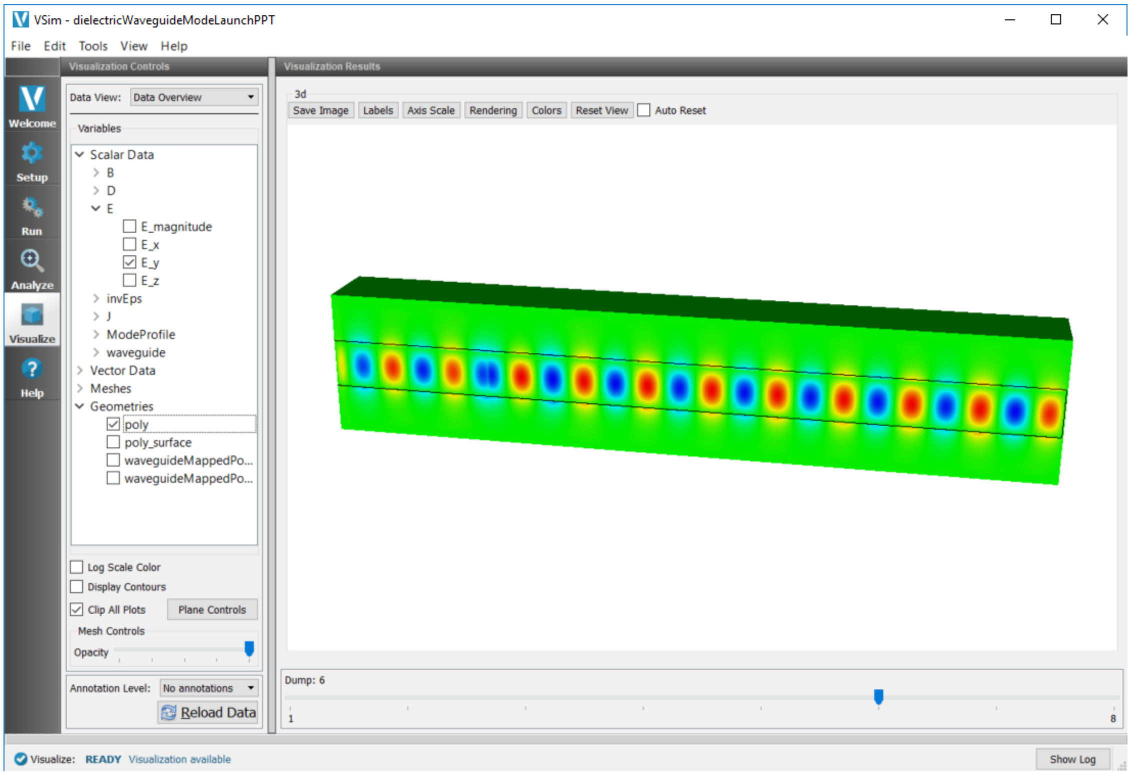


Fig. 3.109: Visualization of the B field's Z component

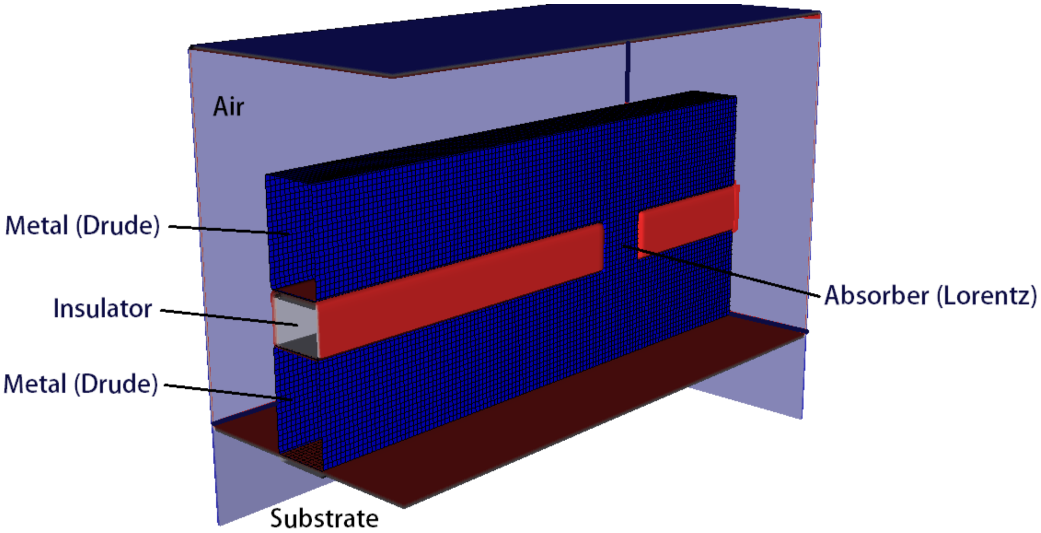
Further Experiments

One can experiment by changing constants or introducing a different signal to drive the waveguide.

3.6.4 Drude-Lorentz MIM Waveguide (MIMwaveguideT.pre)

Problem Description

A metal-insulator-metal (MIM) waveguide can propagate optical frequency electromagnetic radiation due to the effective negative index material property of the metal at those frequencies. This negative index material is represented with a time-domain Drude model dielectric, which can support a wide range of frequencies and wide bandwidth.



In addition to the MIM waveguide, a section of the insulator is removed, and replaced with a resonant absorber material, using a time-domain version of the traditional Lorentz material.

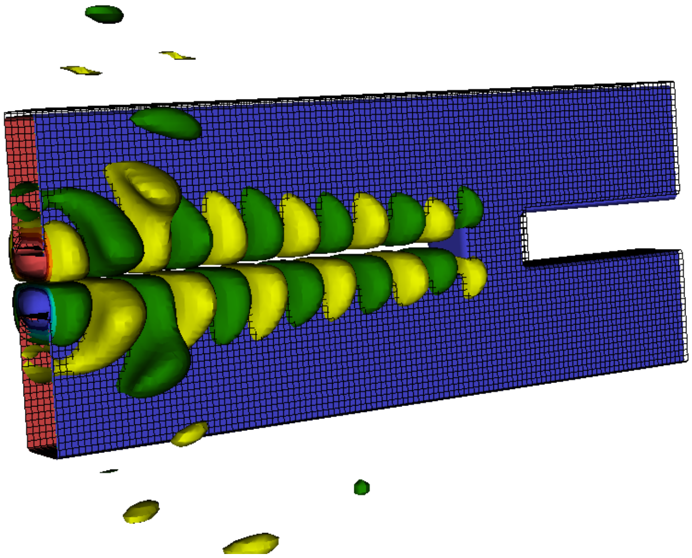


Fig. 3.110: Longitudinal electric field in the MIM waveguide.

A spatial gaussian waveform is incident on the edge of the MIM waveguide, coupling to it, and propagating down the length of the waveguide until it encounters the Lorentz material inclusion, where the wave is absorbed. For the incident wave to couple effectively to the MIM waveguide, the spatial size of the gaussian waveform must be a good match to the size of the waveguide, or a large portion of the incident wave will scatter off the structure, rather than coupling to it.

Also, the width, strength, and natural frequency of the Lorentz material inclusion determines whether the wave is reflected, absorbed, or transmitted when it encounters the inclusion.

The length of the MIM waveguide, and the direction of wave propagation is in the x-direction. The width of the waveguide is in the z-direction, and the height of the waveguide is in the y-direction. The waveguide sits atop an insulator substrate, and is surrounded by air. The boundaries of the simulation are ports, allowing for incoming and outgoing waves.

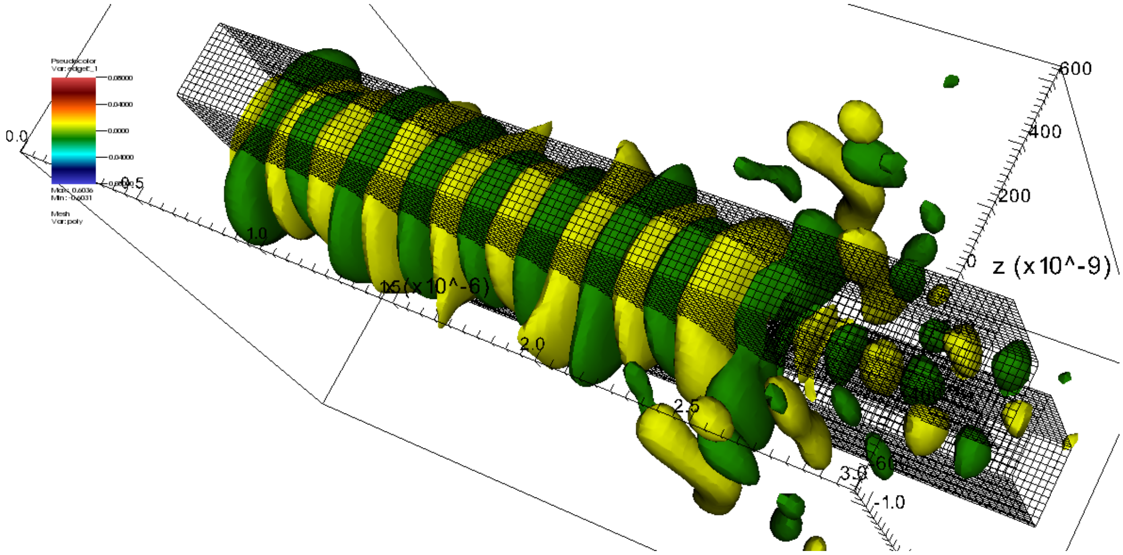


Fig. 3.111: Transverse electric field in the MIM waveguide.

This simulation is the primary example demonstrating the use of the general purpose Drude-Debye-Lorentz-Dielectric macro file, DrudeDebyeLorentzDielectric. Calls to these macros can be found by searching the input file for the string "DDLD".

This simulation can be performed with a VSimEM license.

Opening the Simulation

The MIM waveguide example is accessed from within VSimComposer by the following actions:

- Select the *New → From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Cavities and Waveguides (text-based setup)* option.
- Select "Drude-Lorentz MIM Waveguide (text-based setup)" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

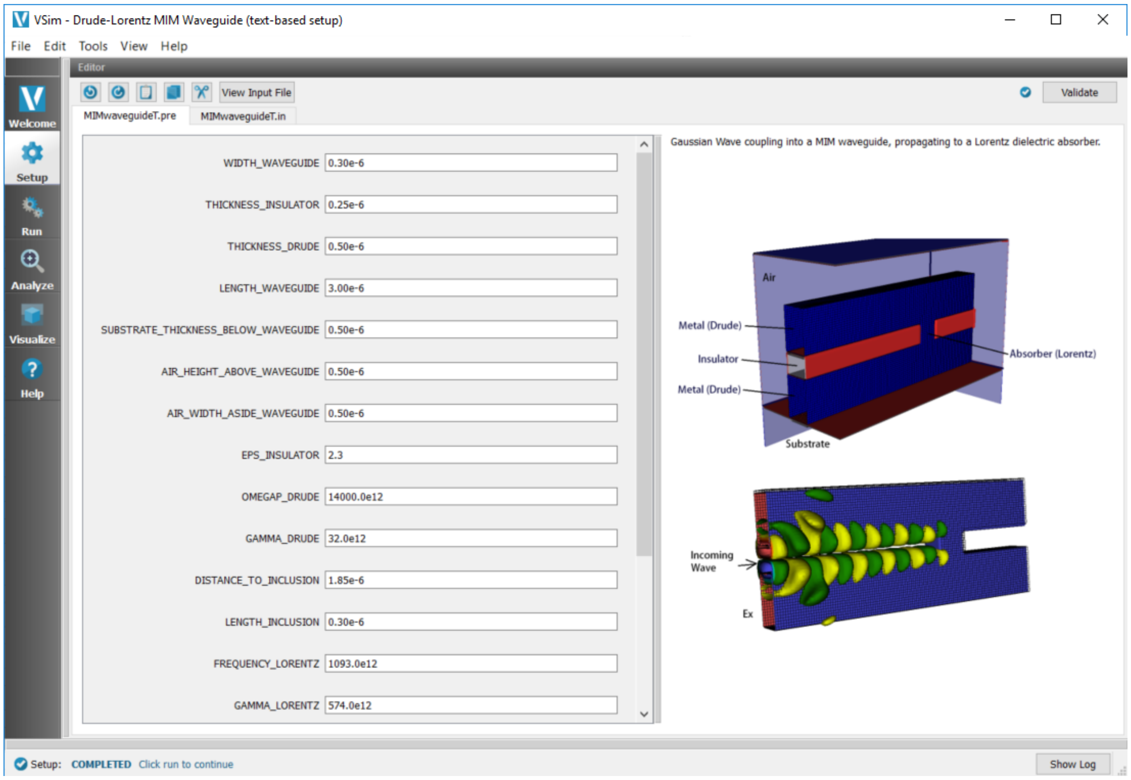
The basic variables of this problem will now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 3.112.

Input File Features

The input file allows the user to choose the waveguide geometry parameters, the material properties of the Drude metal, insulator, and Lorentz inclusion, and the frequency and spatial size of the incident wave.

The input file also contains a parameter to adjust the spatial resolution of the mesh.

Default parameters are selected to correspond to violet light, a Drude material corresponding to silver, SiO2 insulator (and substrate), and a Lorentz material corresponding to AlAs. The default variable values can be compared to the well known material properties of these materials to establish the exact correspondence to the well-known mathematical descriptions of the Drude and Lorentz models.



WIDTH_WAVEGUIDE 0.30e-6

THICKNESS_INSULATOR 0.25e-6

THICKNESS_DRUDE 0.50e-6

LENGTH_WAVEGUIDE 3.00e-6

SUBSTRATE_THICKNESS_BELOW_WAVEGUIDE 0.50e-6

AIR_HEIGHT_ABOVE_WAVEGUIDE 0.50e-6

AIR_WIDTH_ASIDE_WAVEGUIDE 0.50e-6

EPS_INSULATOR 2.3

OMEGAP_DRUDE 14000.0e12

GAMMA_DRUDE 32.0e12

DISTANCE_TO_INCLUSION 1.85e-6

LENGTH_INCLUSION 0.30e-6

FREQUENCY_LORENTZ 1093.0e12

GAMMA_LORENTZ 574.0e12

Gaussian Wave coupling into a MIM waveguide, propagating to a Lorentz dielectric absorber.

Setup: COMPLETED Click run to continue

Fig. 3.112: Setup Window for the MIMwaveguideT example.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 3.113.

Visualizing the results

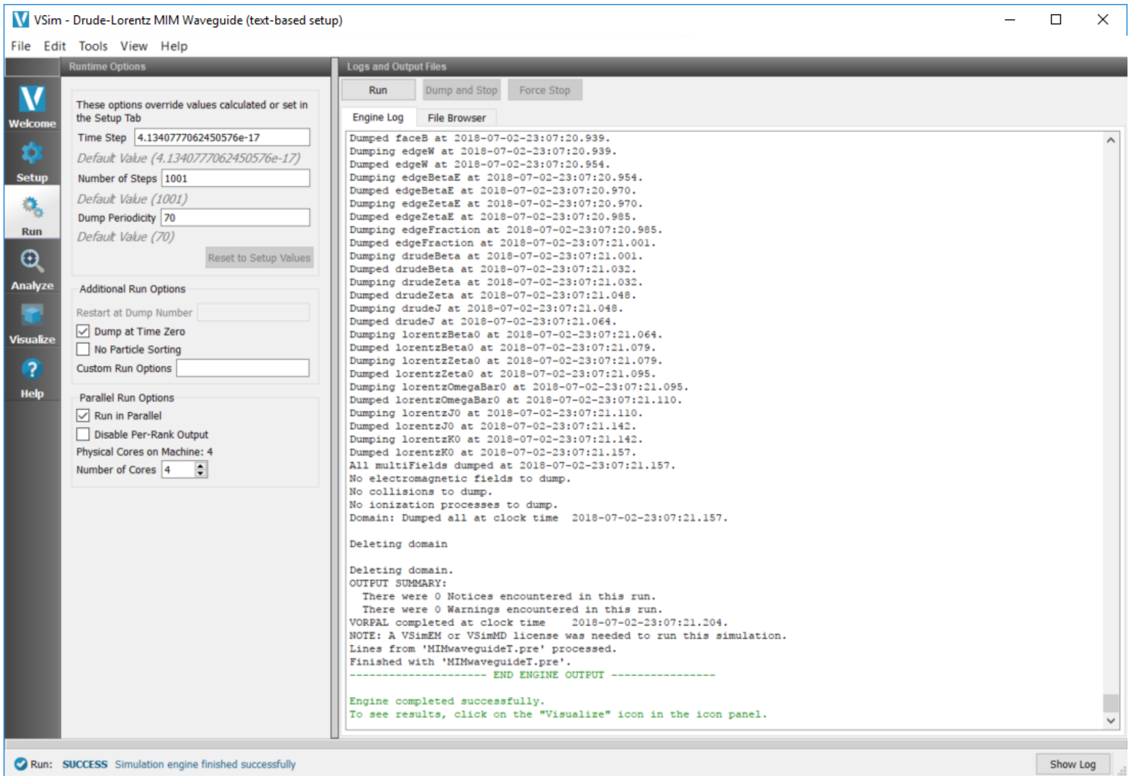
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The results are best viewed by looking at the y component of the electric field. To view the fields:

- Expand *Scalar Data*
- Expand *edgeE*
- Select *edgeE_y*
- Select the *Clip All Plots* checkbox
- Move the dump slider forward in time

You can add the structure by expanding *Geometries* and selecting the *poly* box. The field is shown in Fig. 3.114.



These options override values calculated or set in the Setup Tab

Time Step 4.1340777062450576e-17

Default Value (4.1340777062450576e-17)

Number of Steps 1001

Default Value (1001)

Dump Periodicity 70

Default Value (70)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

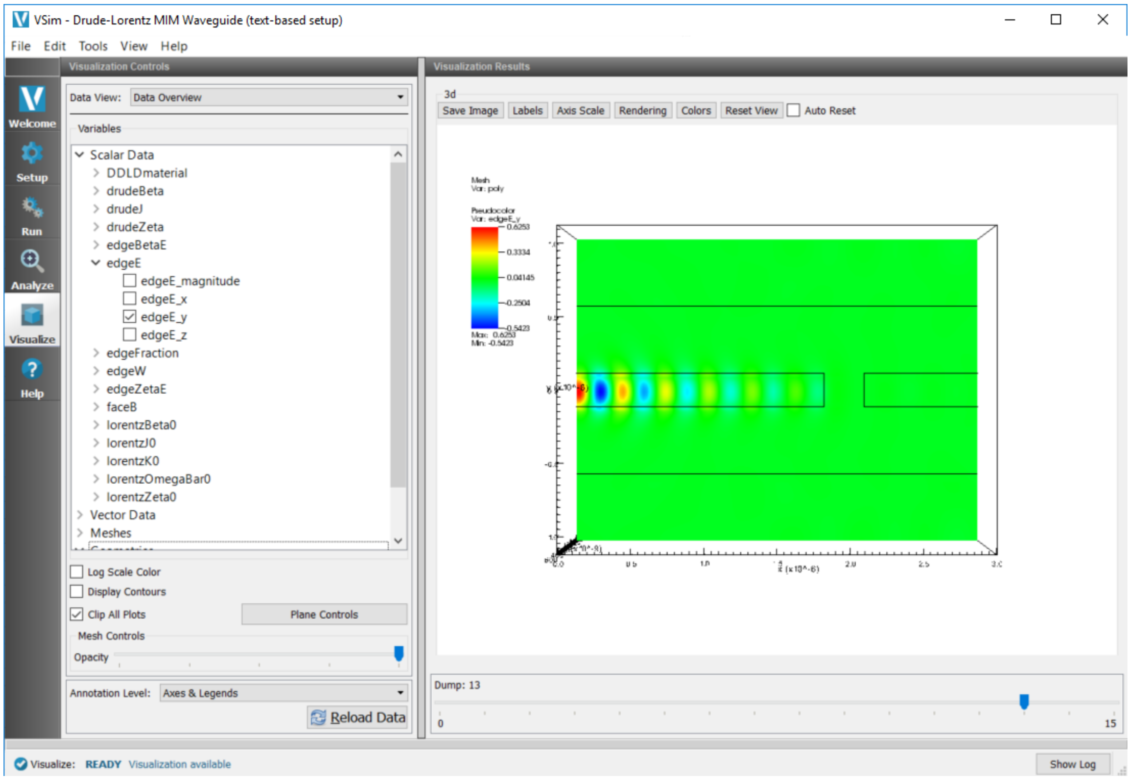
Physical Cores on Machine: 4

Number of Cores 4

```
Dumped faceB at 2018-07-02-23:07:20.939.
Dumping edgeW at 2018-07-02-23:07:20.939.
Dumped edgeW at 2018-07-02-23:07:20.954.
Dumping edgeBetaE at 2018-07-02-23:07:20.954.
Dumped edgeBetaE at 2018-07-02-23:07:20.970.
Dumping edgeZetaE at 2018-07-02-23:07:20.970.
Dumped edgeZetaE at 2018-07-02-23:07:20.985.
Dumping edgeFraction at 2018-07-02-23:07:20.985.
Dumped edgeFraction at 2018-07-02-23:07:21.001.
Dumping drudeBeta at 2018-07-02-23:07:21.001.
Dumped drudeBeta at 2018-07-02-23:07:21.032.
Dumping drudeZeta at 2018-07-02-23:07:21.032.
Dumped drudeZeta at 2018-07-02-23:07:21.048.
Dumping drudeJ at 2018-07-02-23:07:21.048.
Dumped drudeJ at 2018-07-02-23:07:21.064.
Dumping lorentzBeta0 at 2018-07-02-23:07:21.064.
Dumped lorentzBeta0 at 2018-07-02-23:07:21.079.
Dumping lorentzZeta0 at 2018-07-02-23:07:21.079.
Dumped lorentzZeta0 at 2018-07-02-23:07:21.095.
Dumping lorentzOmegaBar0 at 2018-07-02-23:07:21.095.
Dumped lorentzOmegaBar0 at 2018-07-02-23:07:21.110.
Dumping lorentzJ0 at 2018-07-02-23:07:21.110.
Dumped lorentzJ0 at 2018-07-02-23:07:21.142.
Dumping lorentzK0 at 2018-07-02-23:07:21.142.
Dumped lorentzK0 at 2018-07-02-23:07:21.157.
All multiFields dumped at 2018-07-02-23:07:21.157.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-02-23:07:21.157.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-07-02-23:07:21.204.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'MIMwaveguideT.pre' processed.
Finished with 'MIMwaveguideT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 3.113: The Run Window at the end of execution.



Data View: Data Overview

Variables

- Scalar Data
  - DDLDmaterial
  - drudeBeta
  - drudeJ
  - drudeZeta
  - edgeBetaE
  - edgeE
    - □ edgeE_magnitude
    - □ edgeE_x
    - ☑ edgeE_y
    - □ edgeE_z
  - edgeFraction
  - edgeW
  - edgeZetaE
  - faceB
  - lorentzBeta0
  - lorentzJ0
  - lorentzK0
  - lorentzOmegaBar0
  - lorentzZeta0
- Vector Data
- Meshes

□ Log Scale Color

□ Display Contours

☑ Clip All Plots

Mesh Controls

Opacity

Annotation Level: Axes & Legends

Dump: 13

0 15

Visualize: READY Visualization available

Fig. 3.114: Visualization of the E_y field component.

We can see that fields are well coupled between the two metal layers of the waveguide, with only some small leakage, and transient behavior at the entrance. The fields then diminish abruptly at the inclusion, where the wave is mostly absorbed.

3.7 Scattering

3.7.1 Scattering off Multiple Objects (dielecPlusMetalObjs.sdf)

Keywords:

electromagnetics, pulse, dielectric

Problem Description

The Scattering off Multiple Objects simulation illustrates the ability to define different materials with different dielectric properties (perfect electric conductor, sapphire, alumina) and have an electromagnetic pulse reflect off of both a complex metal surface and dielectric medium. It also illustrates a wave launcher to be used with different dielectric materials. This example can also be modified to calculate Radar Cross Sections.

This simulation can be performed with a VSimEM, VSimMD or VSimPD license.

Opening the Simulation

The Scattering off Multiple Objects example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Scattering* option.
- Select “Scattering off Multiple Objects” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

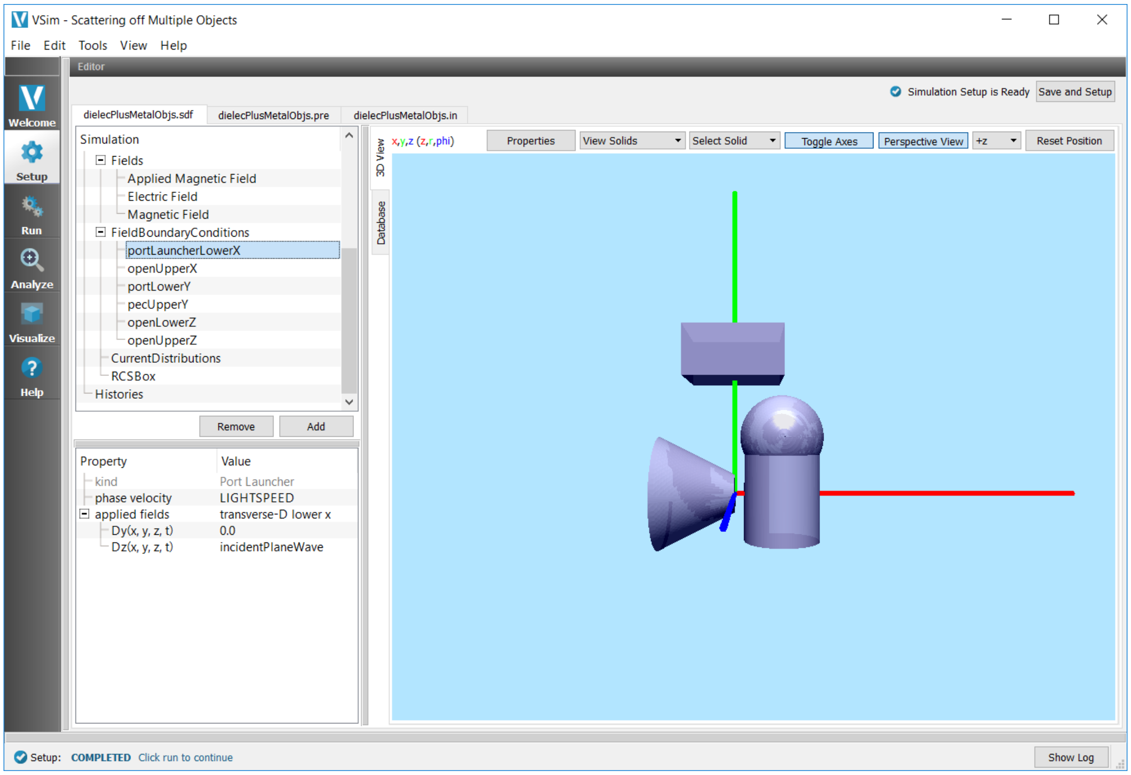
All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 3.115. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.

Simulation Properties

This simulation includes just 1 user defined constant, WAVELENGTH, and just 2 user defined parameters, FREQUENCY and OMEGA. These three terms will define the incoming wave which is defined in the SpaceTimeFunctions element.

CSG shapes are used to define the geometries of the simulation. A sphere is unioned with a cylinder and given a material of sapphire. The box is an alumina structure and the truncated cone serves as a perfect electric conductor.

Placing all of these shapes and various materials in the same simulation shows the how the electromagnetic wave can scatter off of different materials.



- Simulation
  - Fields
    - Applied Magnetic Field
    - Electric Field
    - Magnetic Field
  - FieldBoundaryConditions
    - portLauncherLowerX
    - openUpperX
    - portLowerY
    - pecUpperY
    - openLowerZ
    - openUpperZ
  - CurrentDistributions
  - RCSBox
- Histories

Property	Value
kind	Port Launcher
phase velocity	LIGHTSPEED
applied fields	transverse-D lower x
Dy(x, y, z, t)	0.0
Dz(x, y, z, t)	incidentPlaneWave

Setup: COMPLETED Click run to continue

Fig. 3.115: Setup Window for the Scattering off Multiple Objects example.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- Here you can set run parameters, including how many cores to run with (under the MPI tab).
- When you are finished setting run parameters, click on the *Run* button in the upper left corner of the *Logs and output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 3.116.

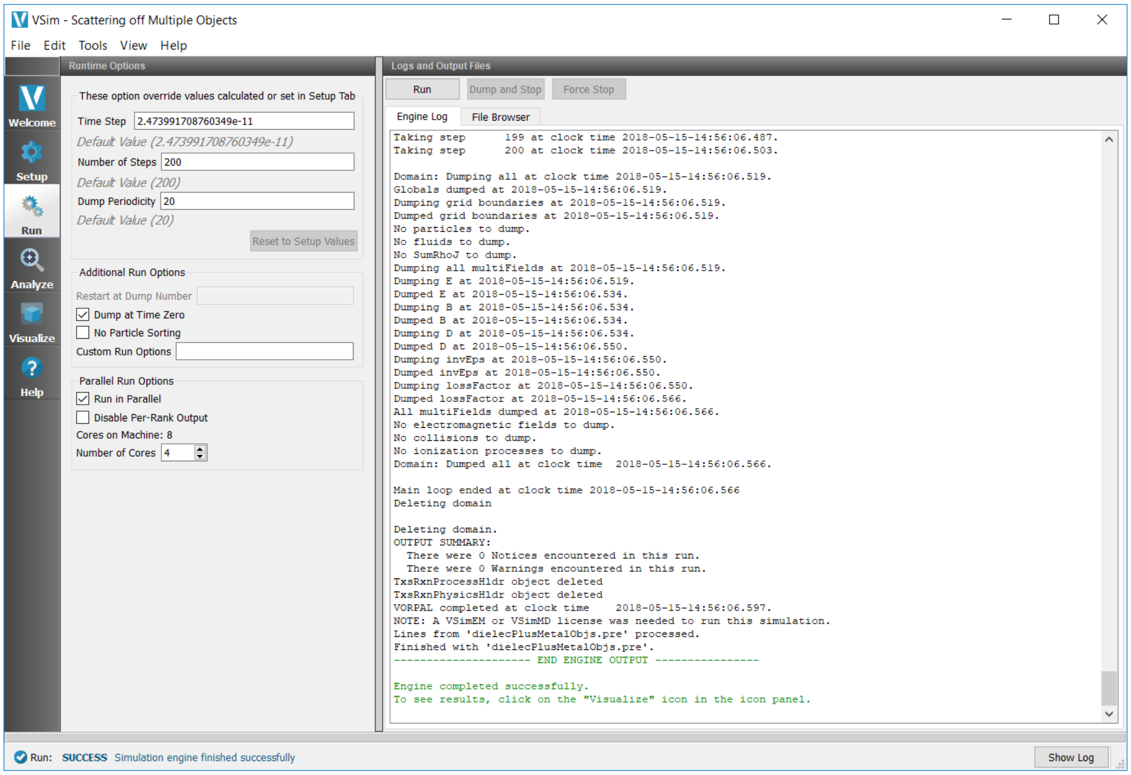
Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the electric field, as well as the geometries as shown in Fig. 3.117, do the following:

- Expand *Scalar Data*
- Expand *E*
- Select *E_z*
- Expand *Geometries*
- Select *poly (AluminaObject) poly (PECObject) poly (sapphireObject)*
- Select *Clip All Plots*



These option override values calculated or set in Setup Tab

Time Step 2.473991708760349e-11
Default Value (2.473991708760349e-11)
Number of Steps 200
Default Value (200)
Dump Periodicity 20
Default Value (20)

Additional Run Options

Restart at Dump Number
☑ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
☐ Disable Per-Rank Output
Cores on Machine: 8
Number of Cores 4

```
Taking step      199 at clock time 2018-05-15-14:56:06.487.
Taking step      200 at clock time 2018-05-15-14:56:06.503.

Domain: Dumping all at clock time 2018-05-15-14:56:06.519.
Globals dumped at 2018-05-15-14:56:06.519.
Dumping grid boundaries at 2018-05-15-14:56:06.519.
Dumped grid boundaries at 2018-05-15-14:56:06.519.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-15-14:56:06.519.
Dumping E at 2018-05-15-14:56:06.519.
Dumped E at 2018-05-15-14:56:06.534.
Dumping B at 2018-05-15-14:56:06.534.
Dumped B at 2018-05-15-14:56:06.534.
Dumping D at 2018-05-15-14:56:06.534.
Dumped D at 2018-05-15-14:56:06.550.
Dumping invEps at 2018-05-15-14:56:06.550.
Dumped invEps at 2018-05-15-14:56:06.550.
Dumping lossFactor at 2018-05-15-14:56:06.550.
Dumped lossFactor at 2018-05-15-14:56:06.566.
All multiFields dumped at 2018-05-15-14:56:06.566.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-15-14:56:06.566.

Main loop ended at clock time 2018-05-15-14:56:06.566
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-05-15-14:56:06.597.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'dielecPlusMetalObjs.pre' processed.
Finished with 'dielecPlusMetalObjs.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 3.116: The Run Window at the end of execution.

- Above the image, select *Colors*
- In the *Color Options* window set the color table to *hot_desaturated* and fix the minimum and maximum values to -2 and 2, respectively.
- Click *Apply*, followed by *OK*.

Initially, no field will be seen, as one is looking at Dump 1 when no fields are yet in the simulation. Move the slider at the bottom of the right pane to see the electric field at different times.

In this example, it is beneficial to change the color table to better see the fields penetrating the dielectric objects. To do this, click on the *Colors* button at the top of the visualization and select “hot_desaturated”, for example.

Further Experiments

One idea is to include radar cross section histories at setup time to be able to visualize the far fields.

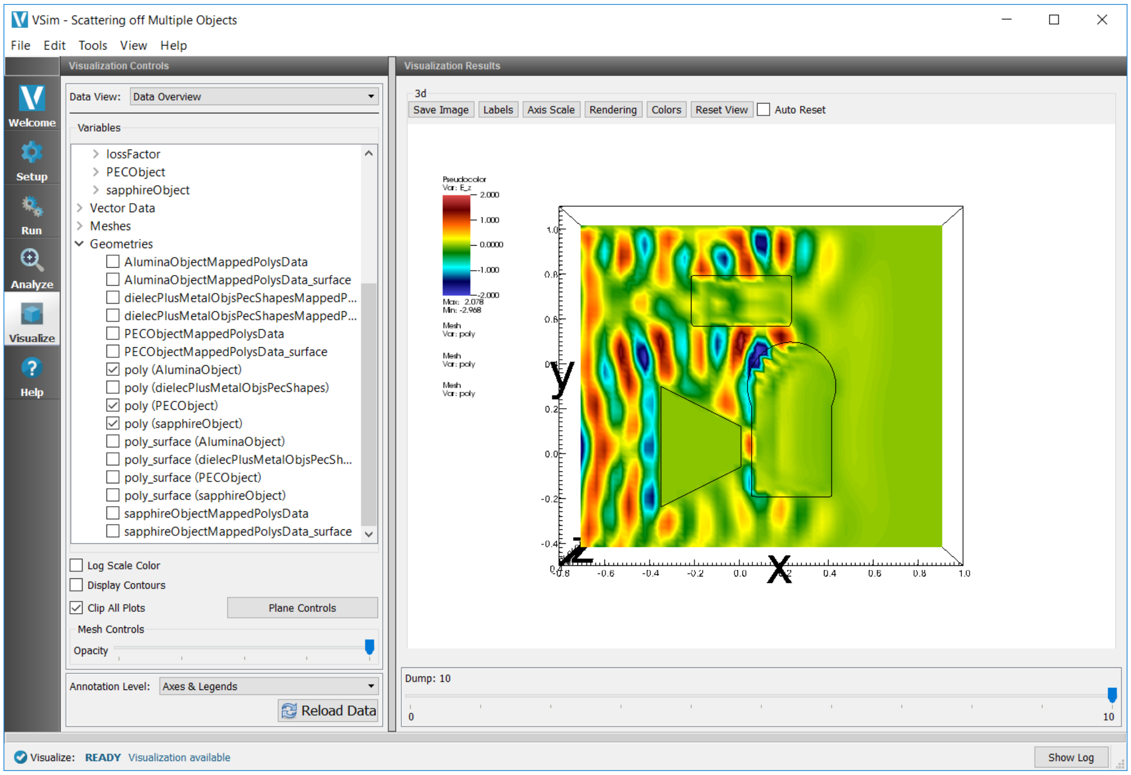
This example is easily modifiable to include a different geometry and wave form.

Try changing the materials to see how it affects the wave.

3.8 Scattering (text-based setup)

3.8.1 Ground Penetrating Radar (groundPenetratingRadarT.pre)

Keywords:



Data View: Data Overview

Variables

- lossFactor
- PECObject
- sapphireObject
- Vector Data
- Meshes
- Geometries
  - ☐ AluminaObjectMappedPolysData
  - ☐ AluminaObjectMappedPolysData_surface
  - ☐ dielecPlusMetalObjsPecShapesMappedP...
  - ☐ dielecPlusMetalObjsPecShapesMappedP...
  - ☐ PECObjectMappedPolysData
  - ☐ PECObjectMappedPolysData_surface
  - ☑ poly (AluminaObject)
  - ☐ poly (dielecPlusMetalObjsPecShapes)
  - ☑ poly (PECObject)
  - ☑ poly (sapphireObject)
  - ☐ poly_surface (AluminaObject)
  - ☐ poly_surface (dielecPlusMetalObjsPecSh...
  - ☐ poly_surface (PECObject)
  - ☐ poly_surface (sapphireObject)
  - ☐ sapphireObjectMappedPolysData
  - ☐ sapphireObjectMappedPolysData_surface

☐ Log Scale Color

☐ Display Contours

☑ Clip All Plots

Opacity

Annotation Level: Axes & Legends

☐ Auto Reset

Dump: 10

Visualize: READY Visualization available

Fig. 3.117: Visualization of wave as it hits the objects.

`GPR, ground penetrating radar, lossy dielectric`

Problem description

This simulation launches a plane wave, polarized in the Z-direction into a lossy dielectric. Embedded within the lossy dielectric is a mine, modelled as a pure electric conductor. The return wave at the surface can be monitored with histories. The simulation is 2D, but with minor effort can be expanded to 3 dimensions.

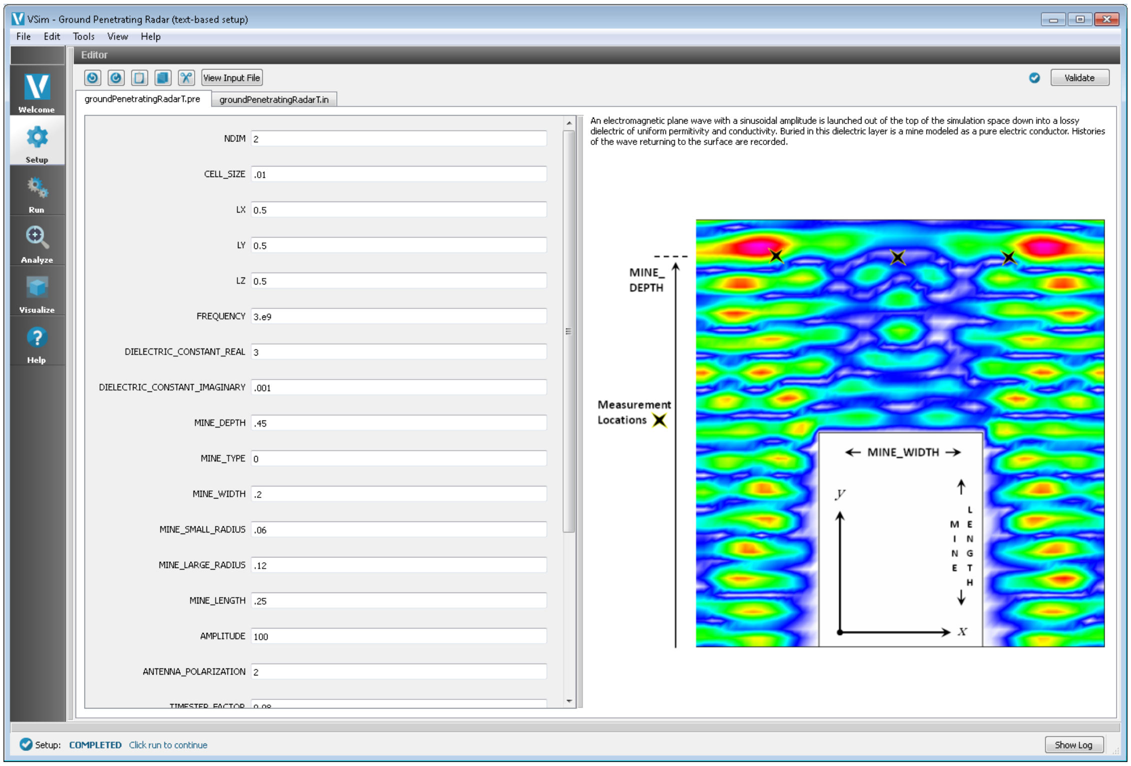
This simulation can be performed with a VSimEM license.

Opening the Simulation

The Ground Penetrating Radar example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Scattering (text-based setup)* option.
- Select “Ground Penetrating Radar (text-based setup)” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The key parameters of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 3.118.



VSim - Ground Penetrating Radar (text-based setup)
File Edit Tools View Help
Editor
View Input File
Validate
groundPenetratingRadarT.pre
groundPenetratingRadarT.in
Welcome
Setup
Run
Analyze
Visualize
Help
MINE_ DEPTH
Measurement Locations
MINE_WIDTH
MINE_LENGTH
y
x
Show Log

NDIM	2
CELL_SIZE	.01
LX	0.5
LY	0.5
LZ	0.5
FREQUENCY	3.e9
DIELECTRIC_CONSTANT_REAL	3
DIELECTRIC_CONSTANT_IMAGINARY	.001
MINE_DEPTH	.45
MINE_TYPE	0
MINE_WIDTH	.2
MINE_SMALL_RADIUS	.06
MINE_LARGE_RADIUS	.12
MINE_LENGTH	.25
AMPLITUDE	100
ANTENNA_POLARIZATION	2
TIMESTEP_FACTOR	0.98

An electromagnetic plane wave with a sinusoidal amplitude is launched out of the top of the simulation space down into a lossy dielectric of uniform permitivity and conductivity. Buried in this dielectric layer is a mine modeled as a pure electric conductor. Histories of the wave returning to the surface are recorded.

Setup: COMPLETED Click run to continue

Fig. 3.118: Setup Window for the Ground Penetrating Radar example.

Input File Features

This file allows for the modification of plane wave operating frequency, simulation domain size, resolution, dielectric permittivity, size and conductivity, mine size and location as well as history location.

Running the Simulation

After performing the above actions, continue as follows:

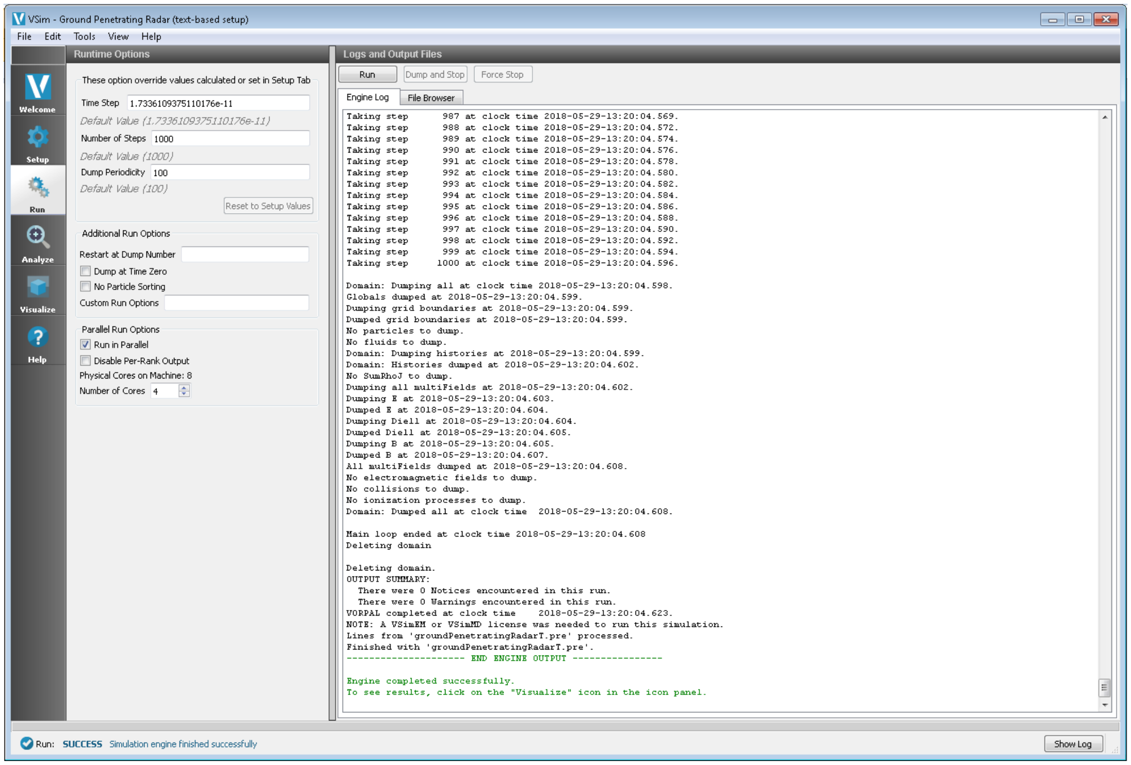
- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner. of the window. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in the window below.

Visualizing the Results

- After the simulation has completed running, click on the Visualize Window.

The electric field can be viewed by:

- Expand *Scalar Data*
- Expand *E*
- Select *E_z*
- Expand *Geometries*



These option override values calculated or set in Setup Tab

Time Step 1.7336109375110176e-11

Default Value (1.7336109375110176e-11)

Number of Steps 1000

Default Value (1000)

Dump Periodicity 100

Default Value (100)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 8

Number of Cores 4

```
Taking step      987 at clock time 2018-05-29-13:20:04.569.
Taking step      988 at clock time 2018-05-29-13:20:04.572.
Taking step      989 at clock time 2018-05-29-13:20:04.574.
Taking step      990 at clock time 2018-05-29-13:20:04.576.
Taking step      991 at clock time 2018-05-29-13:20:04.578.
Taking step      992 at clock time 2018-05-29-13:20:04.580.
Taking step      993 at clock time 2018-05-29-13:20:04.582.
Taking step      994 at clock time 2018-05-29-13:20:04.584.
Taking step      995 at clock time 2018-05-29-13:20:04.586.
Taking step      996 at clock time 2018-05-29-13:20:04.588.
Taking step      997 at clock time 2018-05-29-13:20:04.590.
Taking step      998 at clock time 2018-05-29-13:20:04.592.
Taking step      999 at clock time 2018-05-29-13:20:04.594.
Taking step     1000 at clock time 2018-05-29-13:20:04.596.

Domain: Dumping all at clock time 2018-05-29-13:20:04.598.
Globals dumped at 2018-05-29-13:20:04.599.
Dumping grid boundaries at 2018-05-29-13:20:04.599.
Dumped grid boundaries at 2018-05-29-13:20:04.599.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-05-29-13:20:04.599.
Domain: Histories dumped at 2018-05-29-13:20:04.602.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-29-13:20:04.602.
Dumping E at 2018-05-29-13:20:04.603.
Dumped E at 2018-05-29-13:20:04.604.
Dumping Diell at 2018-05-29-13:20:04.604.
Dumped Diell at 2018-05-29-13:20:04.605.
Dumping B at 2018-05-29-13:20:04.605.
Dumped B at 2018-05-29-13:20:04.607.
All multiFields dumped at 2018-05-29-13:20:04.608.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-29-13:20:04.608.

Main loop ended at clock time 2018-05-29-13:20:04.608
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-05-29-13:20:04.623.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'groundPenetratingRadarT.pre' processed.
Finished with 'groundPenetratingRadarT.pre'.
-------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 3.119: The Run Window at the end of execution.

- Select *poly*
- Move the dump slider forward in time

The wave reflection measured at the three surface locations can be viewed under the *Histories* Data View

Further Experiments

The parameters of the dielectric can be easily modified. It would also be possible to modify the sources to be horn antennas instead of point sources; this would more accurately model a real world ground penetrating radar situation.

3.8.2 Radar Cross Section of a Cylinder (radarCrossSectionT.pre)

Keywords:

`RCS, far field, radar cross section`

Problem description

This simulation launches a plane wave, polarized in the Y-direction at a conducting cylinder in free space. After the plane wave has been launched the Radar Cross section is computed. This problem is a template to solve any Bistatic radar cross section problem.

This simulation can be performed with a VSimEM license.

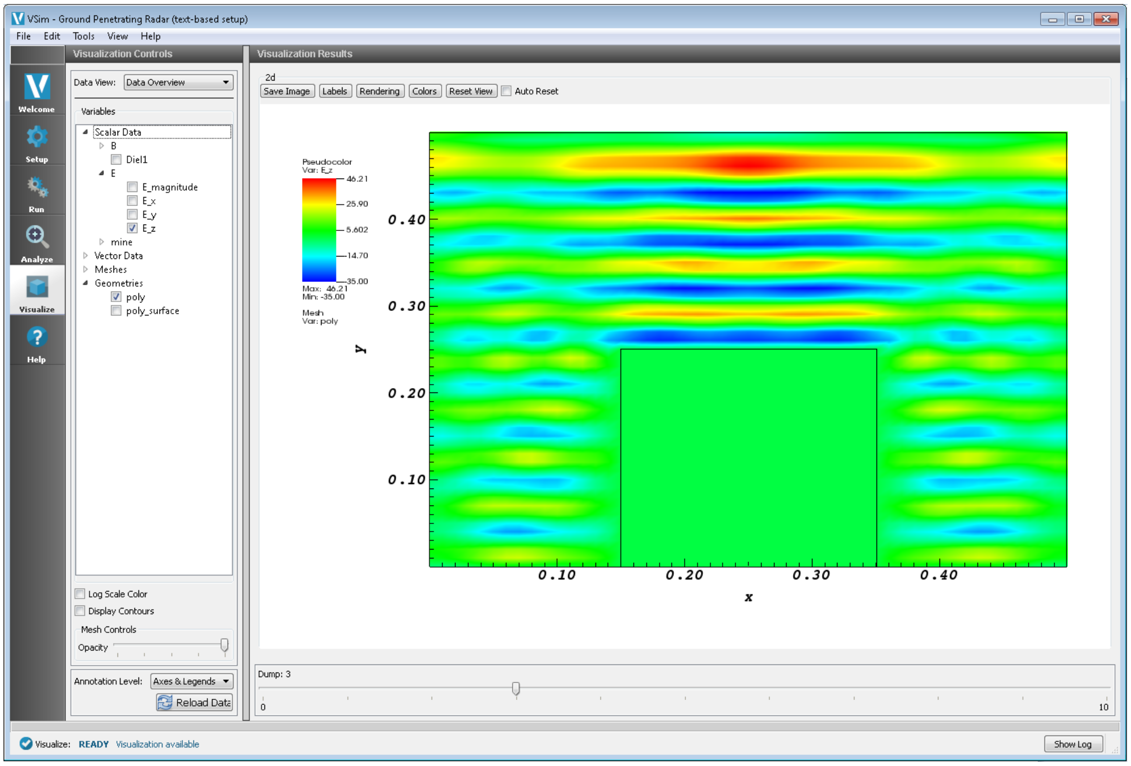


Fig. 3.120: The electric field in the simulation space. Seen here: the cylindrical cone mine-shape.

Opening the Simulation

The radarCrossSectionT example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Advanced Examples (Text-based setup)* option.
- Select “Radar Cross Section” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The key parameters of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in the figure Fig. 3.121.

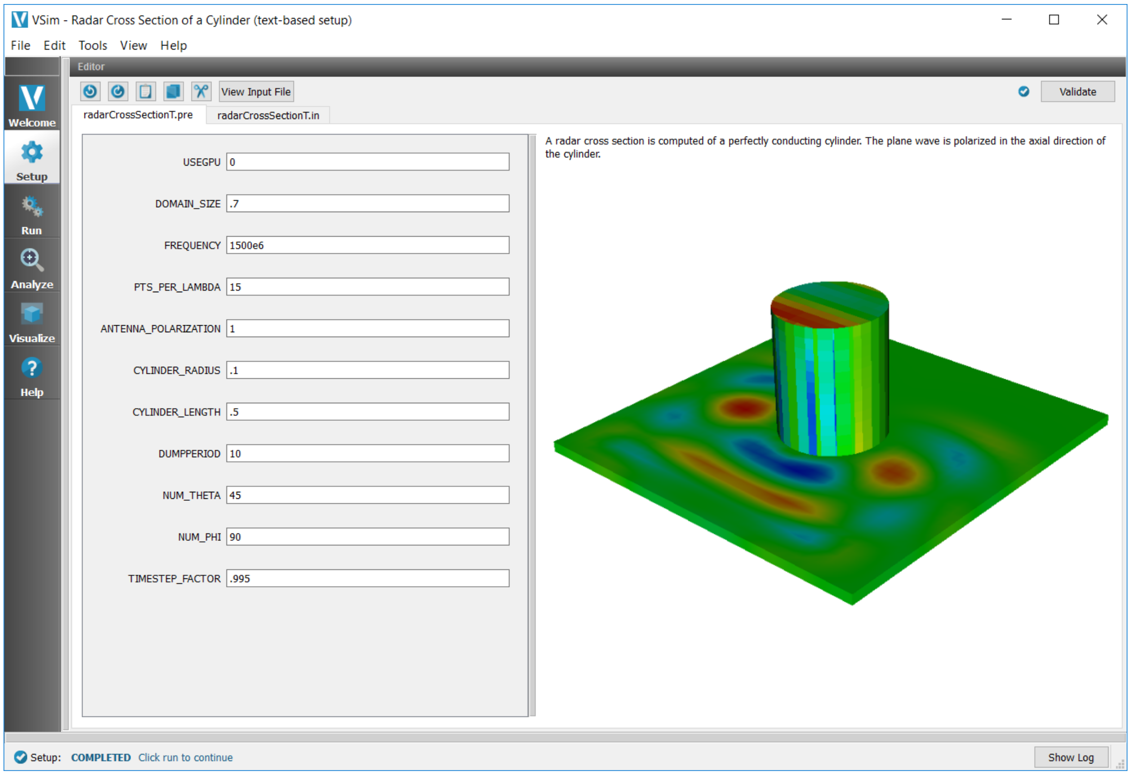
Input File Features

This file allows the modification of plane wave operating frequency, orientation, simulation domain size and far field resolution. This file has had it’s accuracy reduced marginally in order to reduce run time. It is generally recommended that between 10 and 20 points per wavelength are used for full accuracy.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.



USEGPU 0

DOMAIN_SIZE .7

FREQUENCY 1500e6

PTS_PER_LAMBDA 15

ANTENNA_POLARIZATION 1

CYLINDER_RADIUS .1

CYLINDER_LENGTH .5

DUMPPERIOD 10

NUM_THETA 45

NUM_PHI 90

TIMESTEP_FACTOR .995

A radar cross section is computed of a perfectly conducting cylinder. The plane wave is polarized in the axial direction of the cylinder.

Setup: COMPLETED Click run to continue

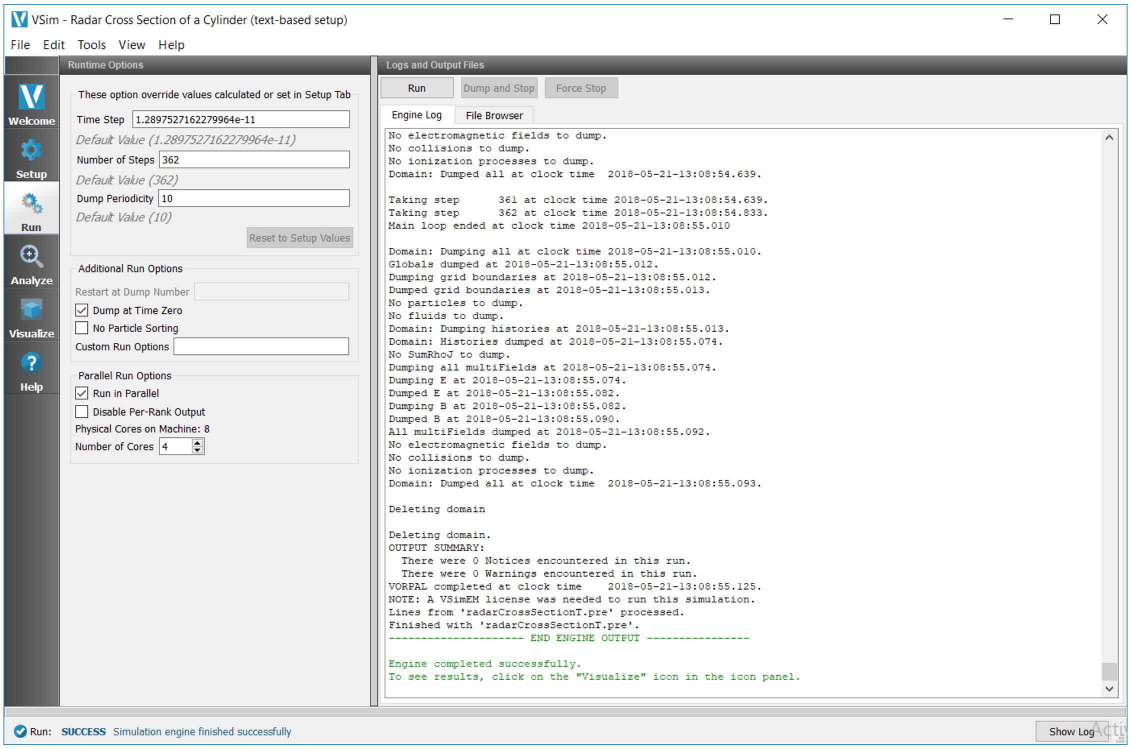
Fig. 3.121: Setup Window for the Radar Cross Section example.

- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane. You will see the output of the run in that pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 3.122

Analyzing the data

To calculate the radar cross section at far field, complete the following steps:

- Proceed to the Analysis window by pressing the Analyze button in the left column of buttons.
- Click 'Show All Analyzers'
- In the resulting dialog, select computeFarFieldFromKirhhoffBox.py (Fig. 3.123) and press Open.
- Input values for the analyzer parameters. The analyzer may be run multiple times, allowing the user to experiment with different values.
  - simulationName - radarCrossSectionT (name of the input file)
  - fieldLabel - E (name of the electric field)
  - farFieldRadius - 10.0 (distance to far field in m, 10.0 is a good value)
  - timeStepStride - 20 (number of timesteps between far field calculations; determines how many far fields are output; 20 steps should yield 4 far fields in this case)
  - getFourierComponent - 0 (whether to fourier analyze for a particular frequency)
  - frequency - the frequency to use in the fourier analysis (not needed here).
  - numTheta - 60 (number of theta points in the far field, 18 for a quick calculation, 45 for finer resolution)



These option override values calculated or set in Setup Tab

Time Step 1.2897527162279964e-11

Default Value (1.2897527162279964e-11)

Number of Steps 362

Default Value (362)

Dump Periodicity 10

Default Value (10)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

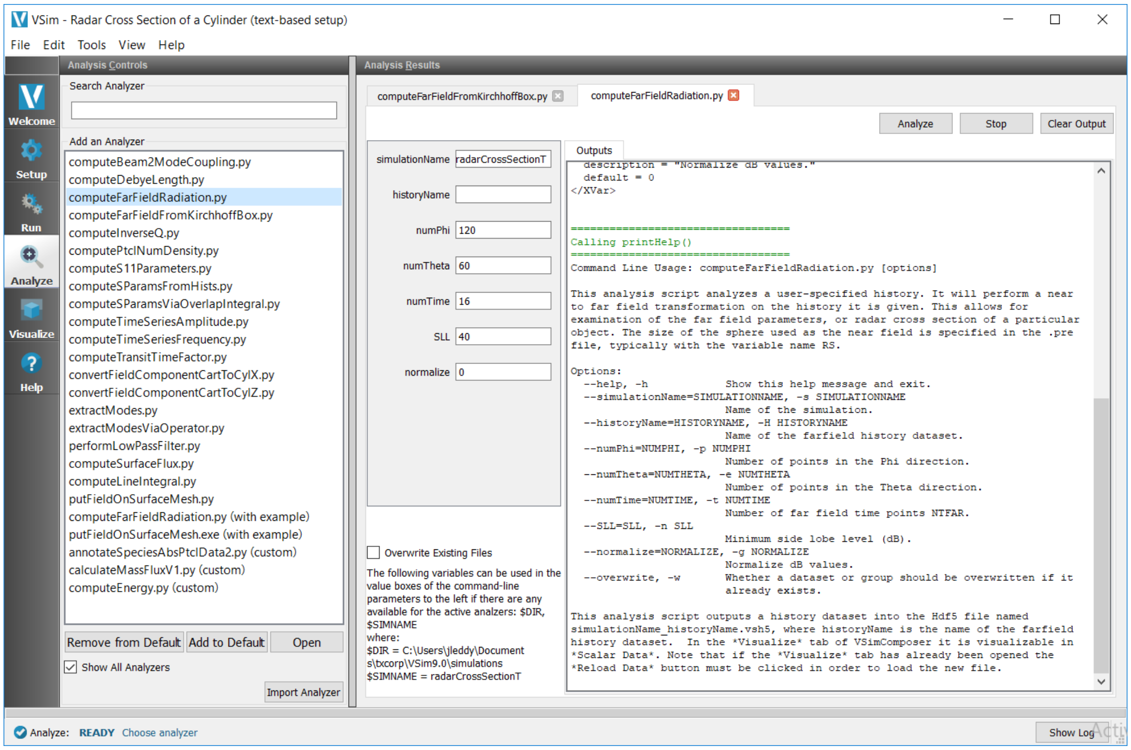
Physical Cores on Machine: 8

Number of Cores 4

```
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-21-13:08:54.639.

Taking step      361 at clock time 2018-05-21-13:08:54.639.
Taking step      362 at clock time 2018-05-21-13:08:54.833.
Main loop ended at clock time 2018-05-21-13:08:55.010

Domain: Dumping all at clock time 2018-05-21-13:08:55.010.
Globals dumped at 2018-05-21-13:08:55.012.
Dumping grid boundaries at 2018-05-21-13:08:55.012.
Dumped grid boundaries at 2018-05-21-13:08:55.013.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-05-21-13:08:55.013.
Domain: Histories dumped at 2018-05-21-13:08:55.074.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-21-13:08:55.074.
Dumping E at 2018-05-21-13:08:55.074.
Dumped E at 2018-05-21-13:08:55.082.
Dumping B at 2018-05-21-13:08:55.082.
Dumped B at 2018-05-21-13:08:55.090.
All multiFields dumped at 2018-05-21-13:08:55.092.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-21-13:08:55.093.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-05-21-13:08:55.125.
NOTE: A VSimEM license was needed to run this simulation.
Lines from 'radarCrossSectionT.pre' processed.
Finished with 'radarCrossSectionT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 3.122: The Run Window at the end of execution.



Search Analyzer

Add an Analyzer

computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
computeSurfaceFlux.py
computeLineIntegral.py
putFieldOnSurfaceMesh.py
computeFarFieldRadiation.py (with example)
putFieldOnSurfaceMesh.exe (with example)
annotateSpeciesAbsPtclData2.py (custom)
calculateMassFluxV1.py (custom)
computeEnergy.py (custom)

☑ Show All Analyzers

simulationName radarCrossSectionT

historyName

numPhi 120

numTheta 60

numTime 16

SLL 40

normalize 0

☐ Overwrite Existing Files

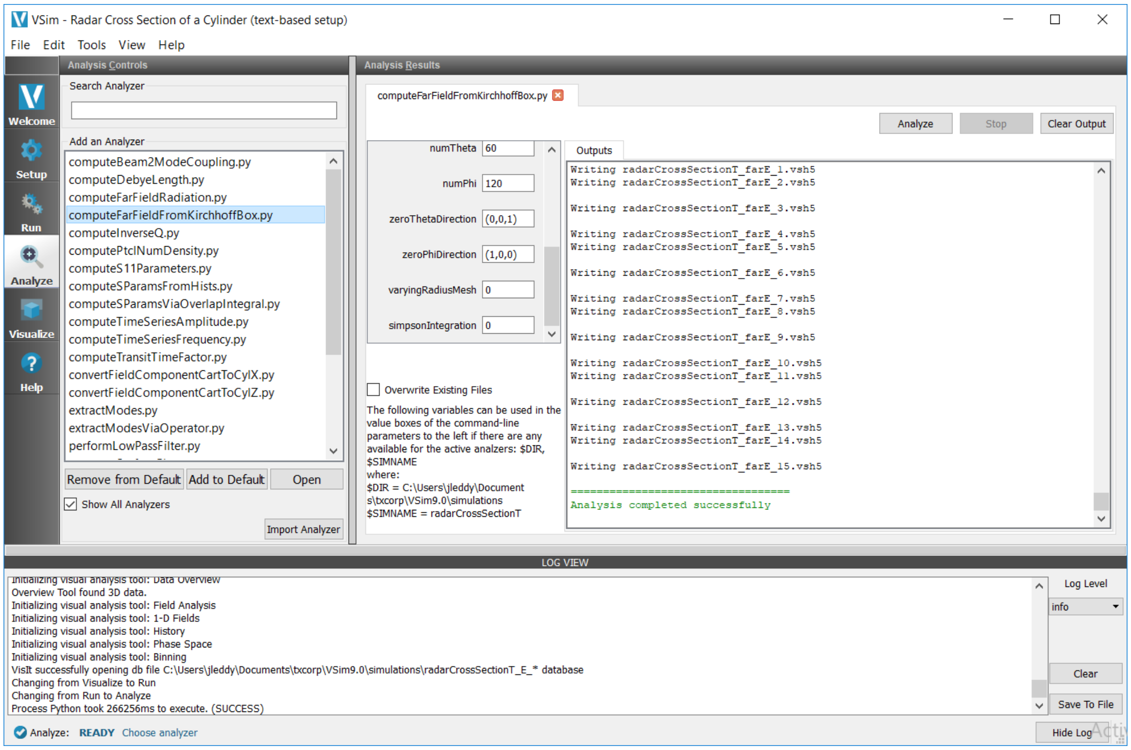
The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\jleddy\Documents\txcorp\VSim9.0\simulations
$SIMNAME = radarCrossSectionT

```
  description = "Normalize dB values."
  default = 0
</XVar>


====================================
Calling printHelp()
====================================
Command Line Usage: computeFarFieldRadiation.py [options]

This analysis script analyzes a user-specified history. It will perform a near
to far field transformation on the history it is given. This allows for
examination of the far field parameters, or radar cross section of a particular
object. The size of the sphere used as the near field is specified in the .pre
file, typically with the variable name RS.

Options:
  --help, -h            Show this help message and exit.
  --simulationName=SIMULATIONNAME, -s SIMULATIONNAME
                        Name of the simulation.
  --historyName=HISTORYNAME, -H HISTORYNAME
                        Name of the farfield history dataset.
  --numPhi=NUMPHI, -p NUMPHI
                        Number of points in the Phi direction.
  --numTheta=NUMTHETA, -e NUMTHETA
                        Number of points in the Theta direction.
  --numTime=NUMTIME, -t NUMTIME
                        Number of far field time points NTFAR.
  --SLL=SLL, -n SLL
                        Minimum side lobe level (dB).
  --normalize=NORMALIZE, -g NORMALIZE
                        Normalize dB values.
  --overwrite, -w       Whether a dataset or group should be overwritten if it
                        already exists.

This analysis script outputs a history dataset into the Hdf5 file named
simulationName_historyName.vsh5, where historyName is the name of the farfield
history dataset.  In the *Visualize* tab of VSimComposer it is visualizable in
*Scalar Data*. Note that if the *Visualize* tab has already been opened the
*Reload Data* button must be clicked in order to load the new file.
```

Analyze: **READY** Choose analyzer

Fig. 3.123: The Analysis Window after running

- numPhi - 120 (number of phi points in the far field, 36 for a quick calculation, 90 for finer resolution)
- zeroThetaDirection - (0,0,1) (determines orientation of far field coordinate system)
- zeroPhiDirection - (1,0,0) (determines orientation of far field coordinate system
- varyingRadiusMesh - 0 (Set to 1 in order to make far field mesh adapt to magnitude of far field solution: the classic lobe view)
- simpsonIntegration - 0 (Set to 1 for more accurate integration)



computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py

☑ Show All Analyzers

numTheta	60
numPhi	120
zeroThetaDirection	(0,0,1)
zeroPhiDirection	(1,0,0)
varyingRadiusMesh	0
simpsonIntegration	0

□ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\jleddy\Documents\txcorp\VSim9.0\simulations
$SIMNAME = radarCrossSectionT

```
Writing radarCrossSectionT_farE_1.vsh5
Writing radarCrossSectionT_farE_2.vsh5

Writing radarCrossSectionT_farE_3.vsh5

Writing radarCrossSectionT_farE_4.vsh5
Writing radarCrossSectionT_farE_5.vsh5

Writing radarCrossSectionT_farE_6.vsh5

Writing radarCrossSectionT_farE_7.vsh5
Writing radarCrossSectionT_farE_8.vsh5

Writing radarCrossSectionT_farE_9.vsh5

Writing radarCrossSectionT_farE_10.vsh5
Writing radarCrossSectionT_farE_11.vsh5

Writing radarCrossSectionT_farE_12.vsh5

Writing radarCrossSectionT_farE_13.vsh5
Writing radarCrossSectionT_farE_14.vsh5

Writing radarCrossSectionT_farE_15.vsh5

==================================
Analysis completed successfully
```

```
Initializing visual analysis tool: Data Overview
Overview Tool found 3D data.
Initializing visual analysis tool: Field Analysis
Initializing visual analysis tool: 1-D Fields
Initializing visual analysis tool: History
Initializing visual analysis tool: Phase Space
Initializing visual analysis tool: Binning
VisIt successfully opening db file C:\Users\jleddy\Documents\txcorp\VSim9.0\simulations\radarCrossSectionT_E_* database
Changing from Visualize to Run
Changing from Run to Analyze
Process Python took 266256ms to execute. (SUCCESS)
```

Analyze: READY Choose analyzer

Fig. 3.124: The Analysis Window after running

Visualizing the Results

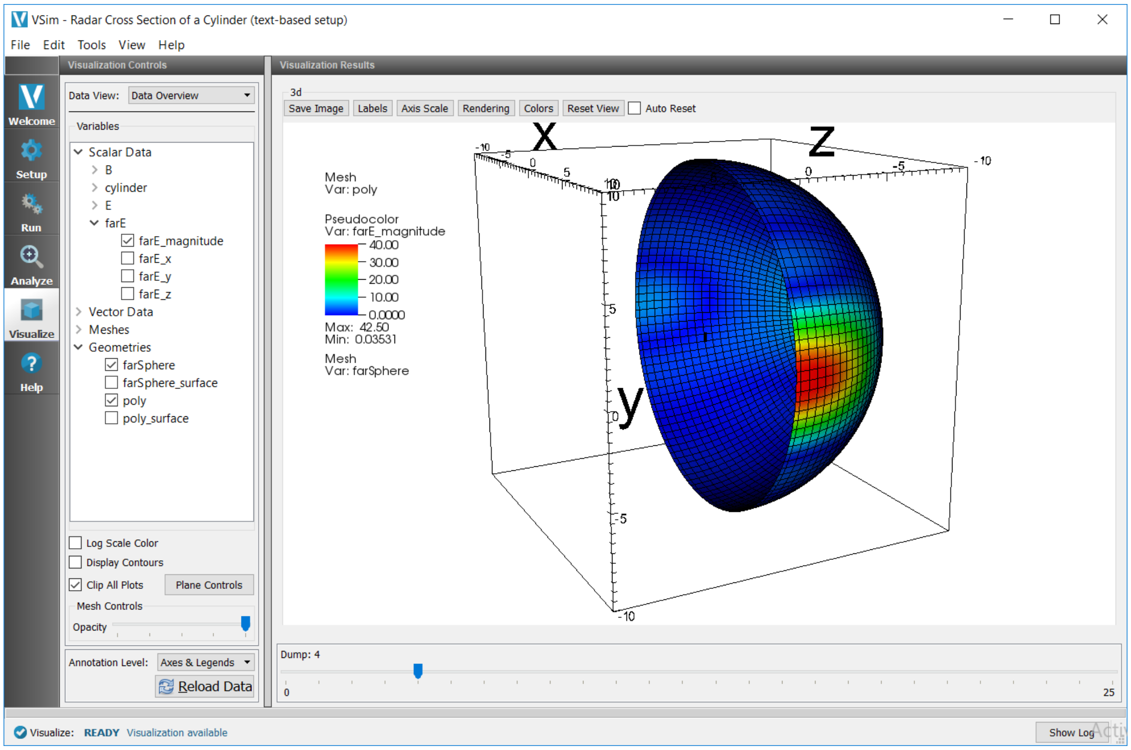
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The Radar cross section as shown in Fig. 3.125, do the following:

- Expand *Scalar Data*
- Select *farE*
- Expand *Geometries*
- Select *farSphere*
- Select *poly*
- Select *Clip All Plots*

You should see the far field, time-dependent signal on the sphere with the cylinder in the center.



Data View: Data Overview

Variables

- Scalar Data
  - B
  - cylinder
  - E
  - farE
    - ☑ farE_magnitude
    - ☐ farE_x
    - ☐ farE_y
    - ☐ farE_z
- Vector Data
- Meshes
- Geometries
  - ☑ farSphere
  - ☐ farSphere_surface
  - ☑ poly
  - ☐ poly_surface

☐ Log Scale Color

☐ Display Contours

☑ Clip All Plots   Plane Controls

Mesh Controls

Opacity

Annotation Level: Axes & Legends

Reload Data

Visualize: READY Visualization available

Fig. 3.125: The radar cross section

Further Experiments

The physical dimensions of the cylinder can be modified from the parameters window.

3.9 Other EM

3.9.1 Spherical Lens (sphericalLens.sdf)

Keywords:

refraction, focusing, dielectrics

Problem Description

The Spherical Lens is a full wave solution to a simple, thin lens with spherical surfaces. Focusing occurs because light rays farther from the center hit the surface at a more oblique angle, resulting in more bending, according to Snell's law. The focusing length of a spherical lens is given by $f = R/(2 - 2/eps_r^{1/2})$, where math:*eps_r* is the relative permittivity of the material making up the lens.

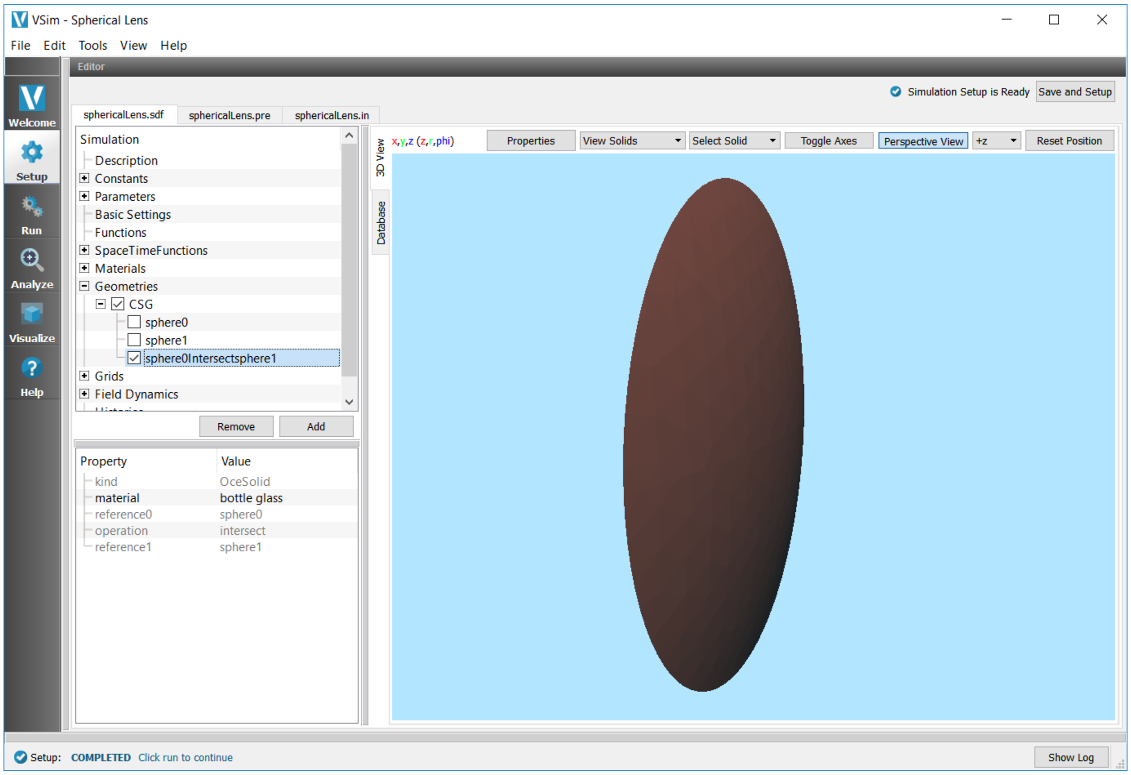
This simulation can be performed with a VSimEM license.

Opening the Simulation

The Spherical Lens example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electro* option.
- Expand the *Other EM* option.
- Select "Spherical Lens" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the setup window as shown in Fig. 3.126. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



Simulation Setup is Ready

Property	Value
kind	OceSolid
material	bottle glass
reference0	sphere0
operation	intersect
reference1	sphere1

Setup: COMPLETED Click run to continue

Fig. 3.126: Setup window for the Spherical Lens example.

Simulation Properties

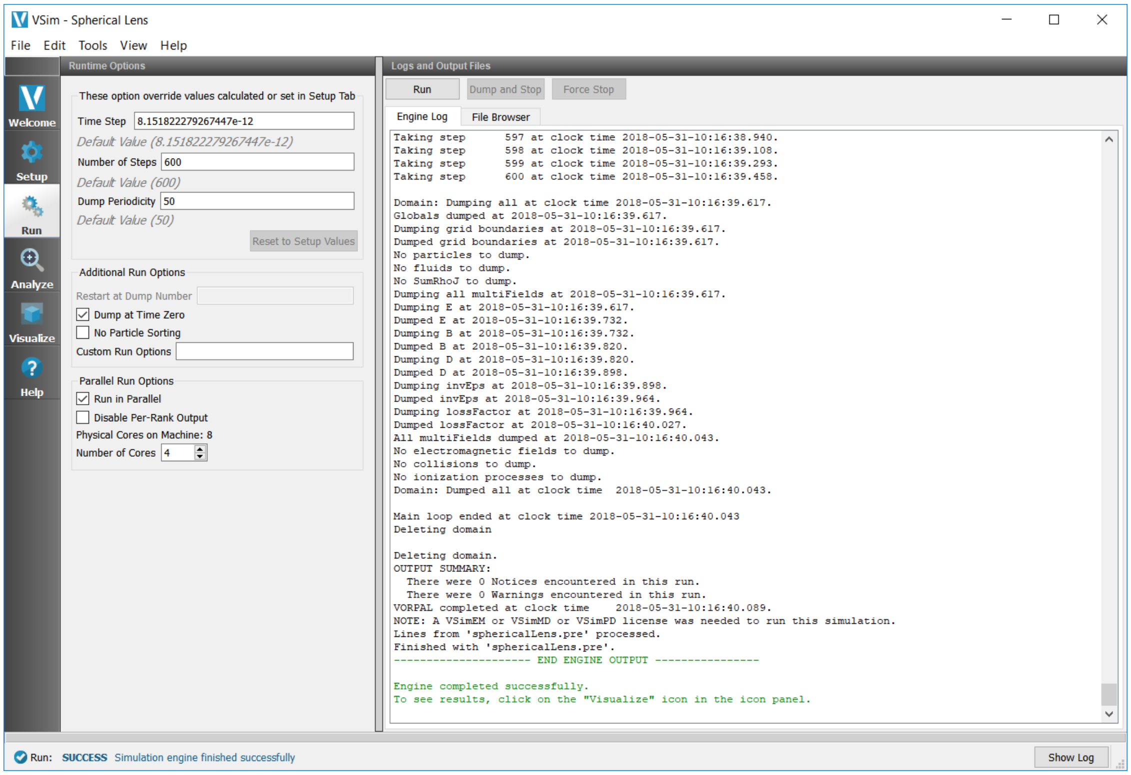
The spherical lens is constructed in CSG using the intersection of two spheres. You can pull the spheres apart to get a taller lens, and you can change the radius of the spheres to have a lens with more curvature. The grid is set so that it will capture the focus at the right for the initial setup.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.

- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 3.127.



These option override values calculated or set in Setup Tab

Time Step 8.151822279267447e-12

Default Value (8.151822279267447e-12)

Number of Steps 600

Default Value (600)

Dump Periodicity 50

Default Value (50)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 8

Number of Cores 4

```
Taking step      597 at clock time 2018-05-31-10:16:38.940.
Taking step      598 at clock time 2018-05-31-10:16:39.108.
Taking step      599 at clock time 2018-05-31-10:16:39.293.
Taking step      600 at clock time 2018-05-31-10:16:39.458.

Domain: Dumping all at clock time 2018-05-31-10:16:39.617.
Globals dumped at 2018-05-31-10:16:39.617.
Dumping grid boundaries at 2018-05-31-10:16:39.617.
Dumped grid boundaries at 2018-05-31-10:16:39.617.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-31-10:16:39.617.
Dumping E at 2018-05-31-10:16:39.617.
Dumped E at 2018-05-31-10:16:39.732.
Dumping B at 2018-05-31-10:16:39.732.
Dumped B at 2018-05-31-10:16:39.820.
Dumping D at 2018-05-31-10:16:39.820.
Dumped D at 2018-05-31-10:16:39.898.
Dumping invEps at 2018-05-31-10:16:39.898.
Dumped invEps at 2018-05-31-10:16:39.964.
Dumping lossFactor at 2018-05-31-10:16:39.964.
Dumped lossFactor at 2018-05-31-10:16:40.027.
All multiFields dumped at 2018-05-31-10:16:40.043.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-31-10:16:40.043.

Main loop ended at clock time 2018-05-31-10:16:40.043
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-05-31-10:16:40.089.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.
Lines from 'sphericalLens.pre' processed.
Finished with 'sphericalLens.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 3.127: The Run window at the end of execution.

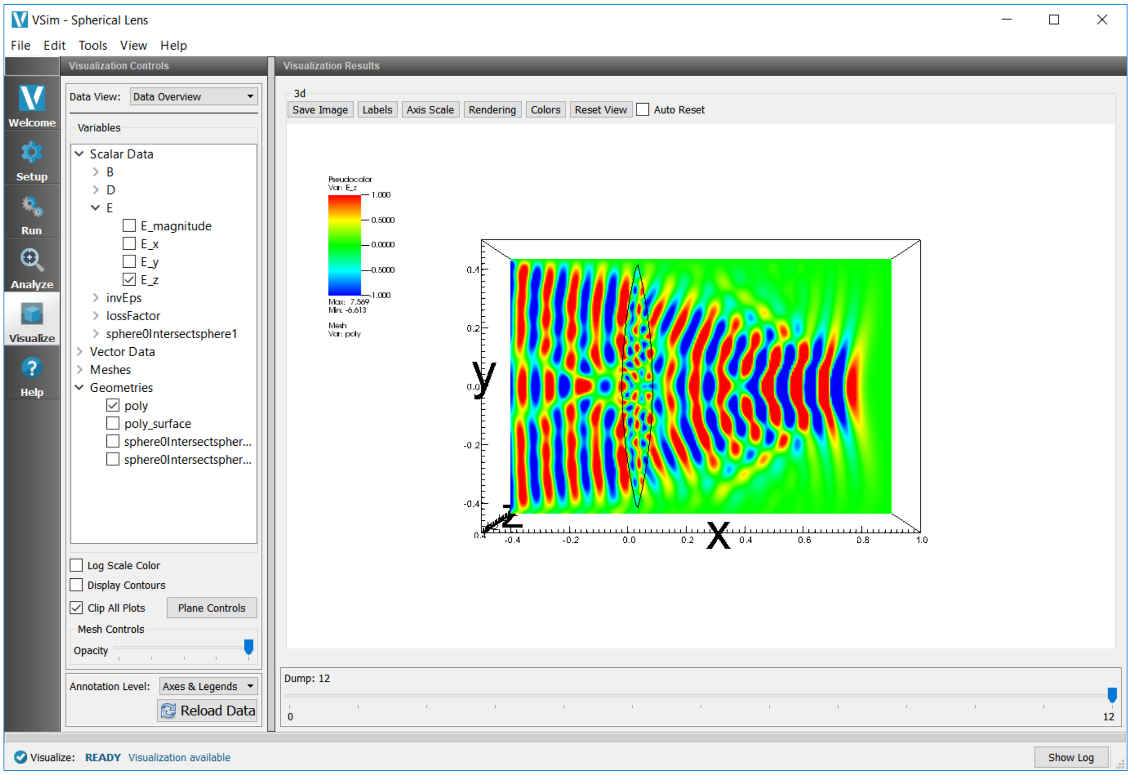
Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize window by pressing the Visualize button in the left column of buttons.

To see the field focus after the lens as shown in Fig. 3.128, do the following:

- Expand *Scalar Data*, expand *E*
- Select *E_z*
- Click the *Colors* button and set the minimum to -1 and the maximum to +1.
- Expand *Geometries*
- Select *poly*
- Select *Clip all plots*
- Move the dump slide to the right to see the wave come in, focus after the lens, and then diverge again after approximately x=0.4. One can see interference of the incoming wave with the reflection off the face of the lens. One can also see interference patterns within the lens.



Data View: Data Overview

☐ E_magnitude
☐ E_x
☐ E_y
☑ E_z

☑ poly
☐ poly_surface

☐ Log Scale Color
☐ Display Contours
☑ Clip All Plots

Annotation Level: Axes & Legends

Dump: 12

Visualize: READY Visualization available

Fig. 3.128: Visualization of the lens focusing

Further Experiments

Use a material of larger dielectric constant to see more focusing.

Reduce the sphere radii to have more focusing.

3.10 Other EM (text-based setup)

3.10.1 Specific Absorption Rate (humanHeadT.pre)

Keywords:

`dielectrics, power calculations, stl files`

Problem Description

The Specific Absorption Rate simulation computes the power absorption in a human head where the brain tissue is approximated using the salt water model. A dipole source is included to imitate a simple antenna source from a cell phone. This example can serve as the basis for a true specific absorption rate calculation for a human head with a source coming from a cell phone antenna.

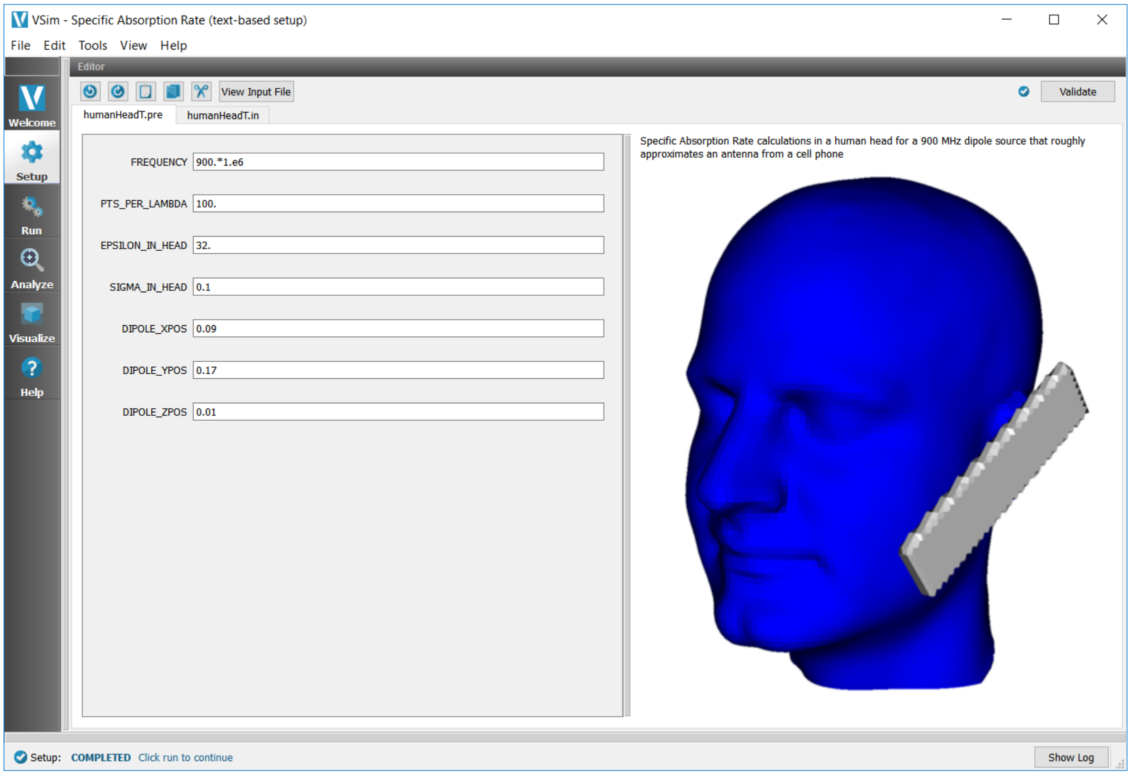
This simulation can only be performed with a VSimEM, VSimMD or VSimPD license.

Opening the Simulation

The Human Head example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Other EM (text-based setup)* option.
- Select “Specific Absorption Rate (text-based setup)” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the right pane of the “Setup” window, as shown in *humanheadtsetupwin*.



VSim - Specific Absorption Rate (text-based setup)

FREQUENCY 900.*1.e6

PTS_PER_LAMBDA 100.

EPSILON_IN_HEAD 32.

SIGMA_IN_HEAD 0.1

DIPOLE_XPOS 0.09

DIPOLE_YPOS 0.17

DIPOLE_ZPOS 0.01

Specific Absorption Rate calculations in a human head for a 900 MHz dipole source that roughly approximates an antenna from a cell phone

Setup: COMPLETED Click run to continue

Fig. 3.129: Setup window for the Human Head example.

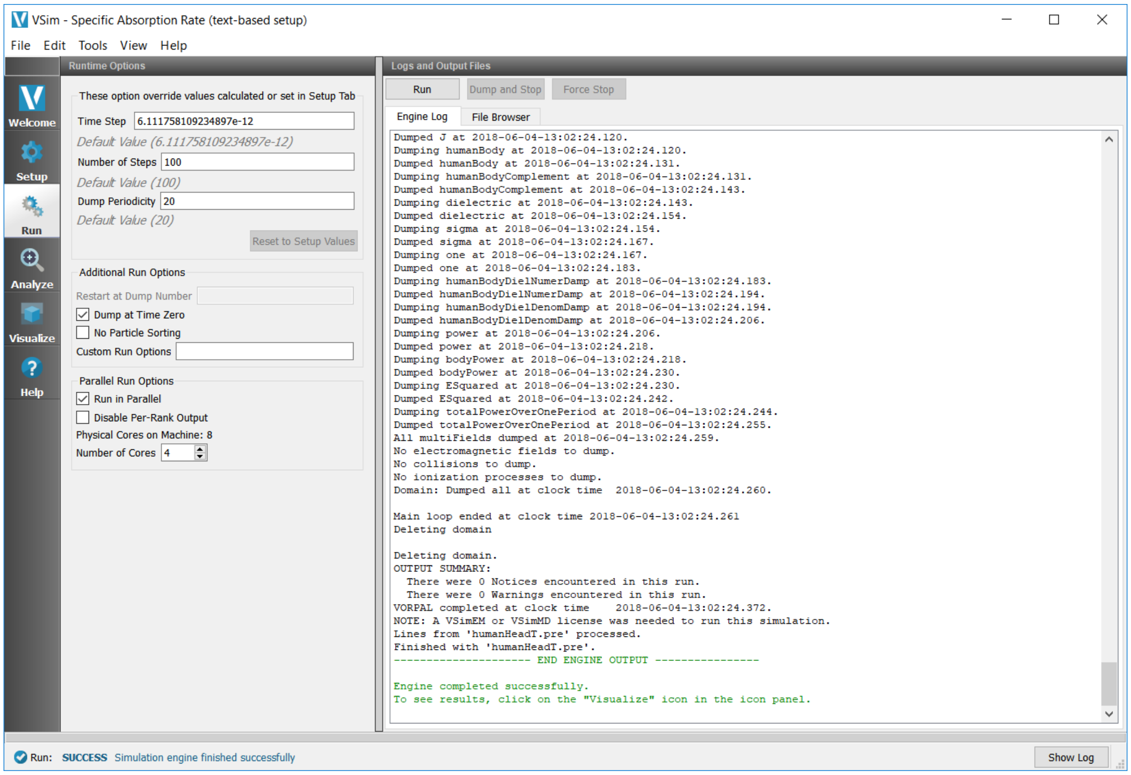
Input File Features

The input file allows one to select the frequency of the dipole source as well as the number of grid points to include per wavelength for the wave in a vacuum. One can also set the dielectric value and conductivity value in the human head. We have also included the ability to select the position of the dipole source approximating the cell phone antenna. A voxel representation of the human head can also be used for more specific tissue values via a voxel dat file with python.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 3.130.



These option override values calculated or set in Setup Tab

Time Step 6.111758109234897e-12
Default Value (6.111758109234897e-12)
Number of Steps 100
Default Value (100)
Dump Periodicity 20
Default Value (20)

Additional Run Options
Restart at Dump Number
☑ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options
☑ Run in Parallel
☐ Disable Per-Rank Output
Physical Cores on Machine: 8
Number of Cores 4

```
Dumped J at 2018-06-04-13:02:24.120.
Dumping humanBody at 2018-06-04-13:02:24.120.
Dumped humanBody at 2018-06-04-13:02:24.131.
Dumping humanBodyComplement at 2018-06-04-13:02:24.131.
Dumped humanBodyComplement at 2018-06-04-13:02:24.143.
Dumping dielectric at 2018-06-04-13:02:24.143.
Dumped dielectric at 2018-06-04-13:02:24.154.
Dumping sigma at 2018-06-04-13:02:24.154.
Dumped sigma at 2018-06-04-13:02:24.167.
Dumping one at 2018-06-04-13:02:24.167.
Dumped one at 2018-06-04-13:02:24.183.
Dumping humanBodyDielNumerDamp at 2018-06-04-13:02:24.183.
Dumped humanBodyDielNumerDamp at 2018-06-04-13:02:24.194.
Dumping humanBodyDielDenomDamp at 2018-06-04-13:02:24.194.
Dumped humanBodyDielDenomDamp at 2018-06-04-13:02:24.206.
Dumping power at 2018-06-04-13:02:24.206.
Dumped power at 2018-06-04-13:02:24.218.
Dumping bodyPower at 2018-06-04-13:02:24.218.
Dumped bodyPower at 2018-06-04-13:02:24.230.
Dumping ESquared at 2018-06-04-13:02:24.230.
Dumped ESquared at 2018-06-04-13:02:24.242.
Dumping totalPowerOverOnePeriod at 2018-06-04-13:02:24.244.
Dumped totalPowerOverOnePeriod at 2018-06-04-13:02:24.255.
All multiFields dumped at 2018-06-04-13:02:24.259.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-04-13:02:24.260.

Main loop ended at clock time 2018-06-04-13:02:24.261
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-04-13:02:24.372.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'humanHeadT.pre' processed.
Finished with 'humanHeadT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 3.130: The Run window at the end of execution.

Visualizing the Results

After performing the above actions, continue as follows:

- **Proceed to the Visualize window by pressing the** Visualize button in the left column of buttons.

To create the image shown in Fig. 3.131:

- Expand *Scalar Data*
- Select *ESquared*
- Expand *Geometries*
- Select *poly*
- Select *Log Scale Color*
- Select *Display Contours* and set the # of contours to 7
- Select *Clip All Plots*
- Click the *Plane Controls* button and set the *Clip Plane Normal* to X
- Set the *Origin Of Normal Vector* to X = 0.1, Y = 0, Z = 0

- Move the dump slider forward in time
- Click and drag with the mouse to rotate the image

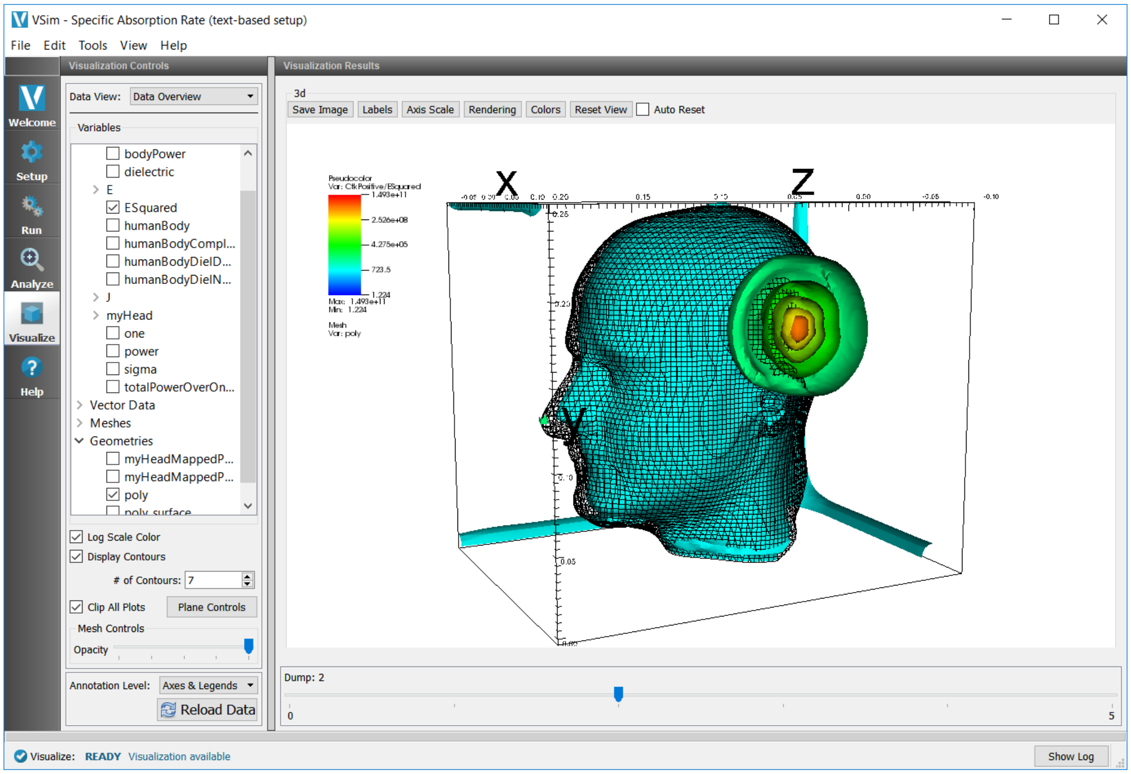


Fig. 3.131: Visualization of the absorption of power by a human head via a clip.

Further Experiments

We suggest the user change the frequency of the dipole source to imitate different cell phone models at different frequencies.

We also suggest the user change the position of the dipole implying a change in location of the cell phone antenna.

It would also be interesting to change the dielectric and conductivity value to model a dipole source in a vacuum.

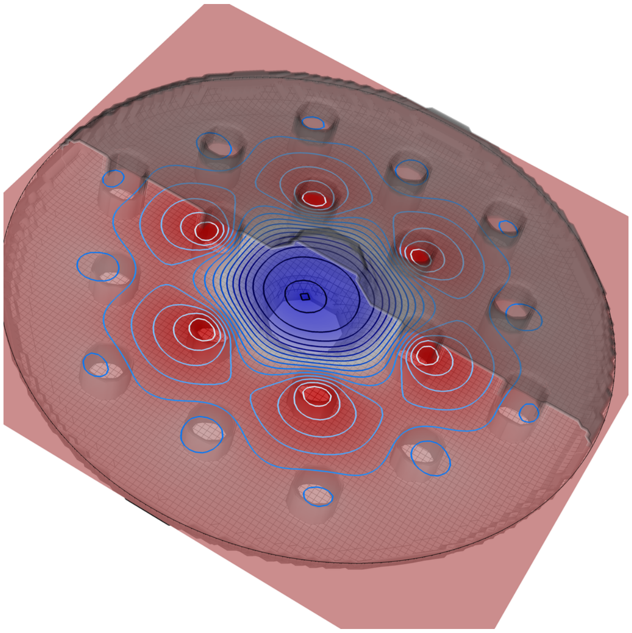
3.10.2 Photonic Crystal in Metal Cavity (phcInMetalCavityT.pre)

Problem Description

A photonic crystal (PhC) is capable of confining electromagnetic fields in waveguides and cavities using a periodic geometry. This simulation features a dielectric photonic crystal cavity—a triangular lattice of dielectric rods, with one rod removed—inside a metal cavity. The cavity axis, and the dielectric rods, are in the z direction.

The photonic crystal structure is similar to that described in *[BWC08]*, truncated after two layers of the lattice structure. The metal cavity resembles an elliptical (or rounded pillbox) cavity, with short beam tubes.

Modeling dielectric and metal can be difficult: at dielectric corners or triple points (where dielectric, metal, and vacuum meet), the electromagnetic fields generally must diverge (to infinity) to preserve continuity dictated by Maxwell's



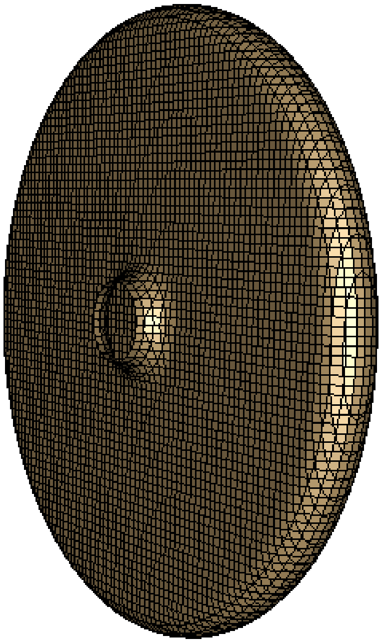


Fig. 3.132: The metal cavity surrounding the PhC structure.

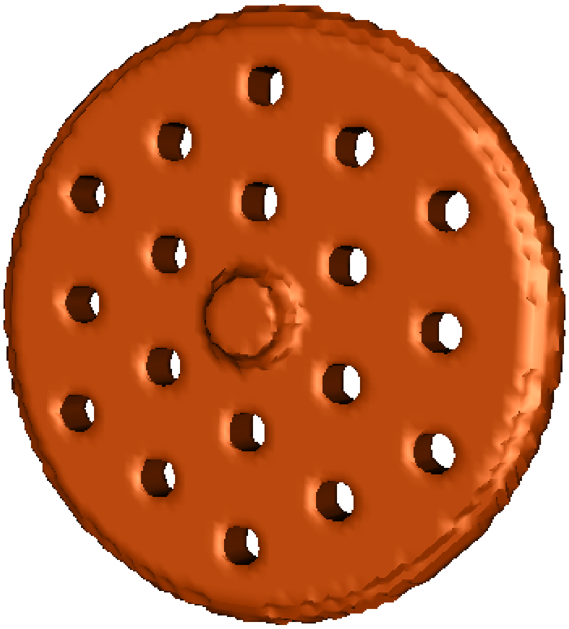


Fig. 3.133: The vacuum region (inside the metal cavity, outside the dielectric rods).

equations *[Had02]*. However, when the interface between dielectric and vacuum is always perpendicular to the metal surface, as in this simulation, the fields remain finite.

This simulation demonstrates a method for combining dielectric and metal, as long as the metal surface is perpendicular to x, y, or z whenever it intersects dielectric (and the vacuum/dielectric interface remains perpendicular to the metal surface at those points).

When the PhC cavity mode is excited, the fields are trapped radially mainly by the dielectric rods.

This simulation can be performed with a VSimEM, VSimMD or VSimPA license.

Opening the Simulation

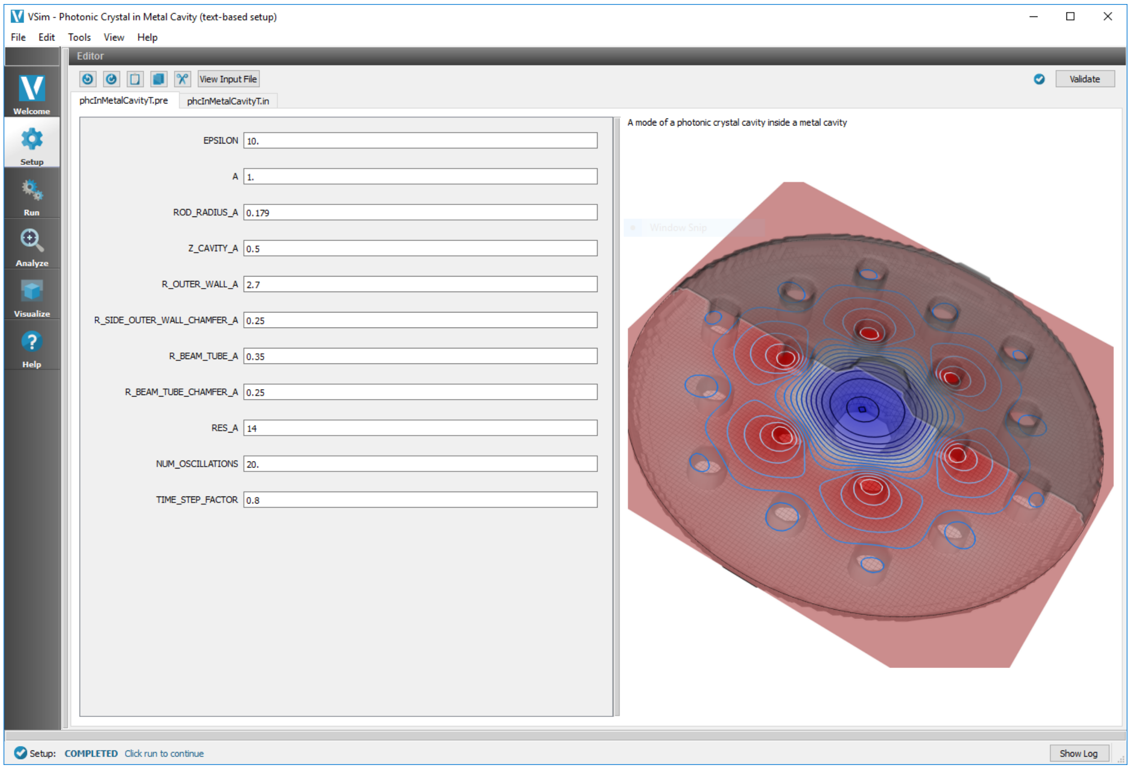
The PhC in Metal Cavity example is accessed from within VSimComposer by the following actions

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Electromagnetics* option.
- Expand the *Other EM (text-based setup)* option.
- Select "Photonic Crystal in Metal Cavity (text-based setup)" and press the *Choose* button.
- In the resulting dialog, create a new folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem will now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 3.134.

Input File Features

The input file allows the user to choose the dielectric contrast and radius of the rods, the shapes and sizes of the cavity and beam tubes (to some extent), the grid resolution, and the number of oscillations to simulate after excitation. The entire simulation is scaled to the lattice constant, which is set to 1 by default.



A mode of a photonic crystal cavity inside a metal cavity

EPSILON	10.
A	1.
ROD_RADIUS_A	0.179
Z_CAVITY_A	0.5
R_OUTER_WALL_A	2.7
R_SIDE_OUTER_WALL_CHAMFER_A	0.25
R_BEAM_TUBE_A	0.35
R_BEAM_TUBE_CHAMFER_A	0.25
RES_A	14
NUM_OSCILLATIONS	20.
TIME_STEP_FACTOR	0.8

Setup: COMPLETED Click run to continue

Fig. 3.134: Setup Window for the phcInMetalCavityT example.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the *Run* button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right pane of the window. You will see the output of the run in that pane. The run has completed when you see the output “Engine completed successfully.” This is shown in Fig. 3.135.

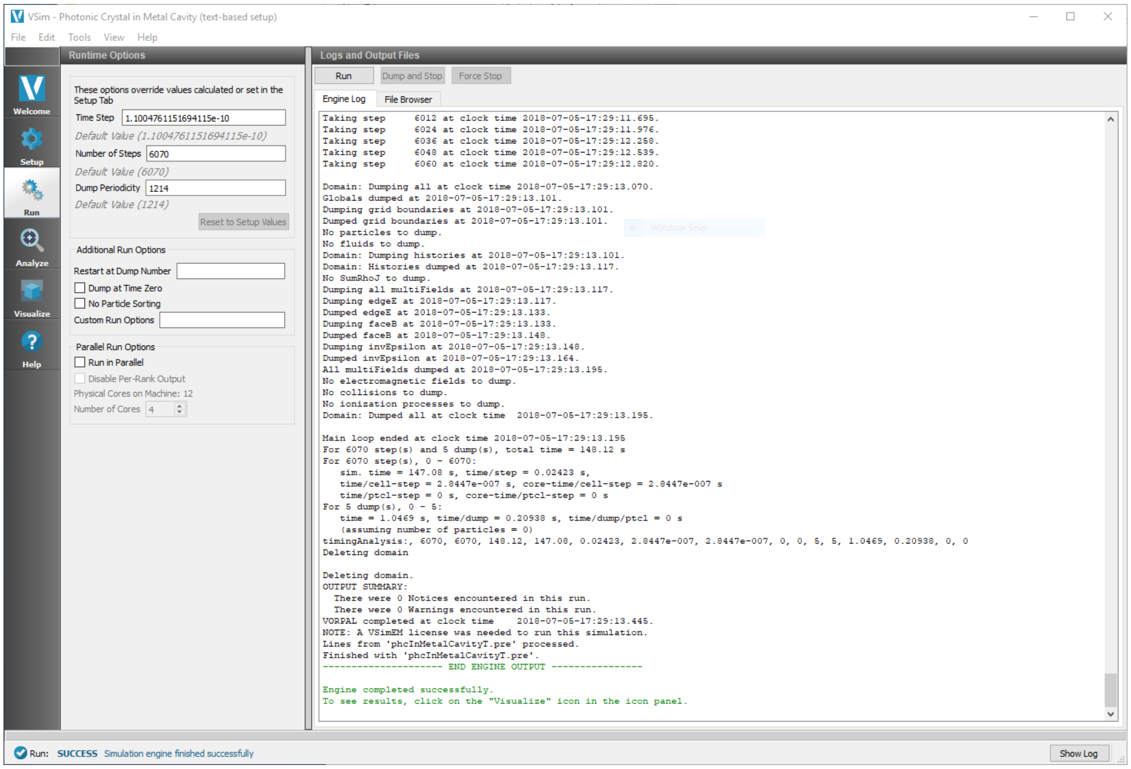
Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the *Visualize* button in the left column of buttons.

Due to the symmetry of this system, the results are best viewed by looking at the z component of the electric field as follows:

- Expand *Scalar Data*
- Expand *edgeE*
- Select *edgeE_z*
- Select *Clip All Plots*
- Click *Plane Controls* and set the Normal in the z-direction
- Expand *Geometries*



These options override values calculated or set in the Setup Tab

Time Step 1.1004761151694115e-10

Default Value (1.1004761151694115e-10)

Number of Steps 6070

Default Value (6070)

Dump Periodicity 1214

Default Value (1214)

Additional Run Options

Restart at Dump Number

☐ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☐ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 12

Number of Cores 4

```
Taking step     6012 at clock time 2018-07-05-17:29:11.695.
Taking step     6024 at clock time 2018-07-05-17:29:11.976.
Taking step     6036 at clock time 2018-07-05-17:29:12.258.
Taking step     6048 at clock time 2018-07-05-17:29:12.539.
Taking step     6060 at clock time 2018-07-05-17:29:12.820.

Domain: Dumping all at clock time 2018-07-05-17:29:13.070.
Globals dumped at 2018-07-05-17:29:13.101.
Dumping grid boundaries at 2018-07-05-17:29:13.101.
Dumped grid boundaries at 2018-07-05-17:29:13.101.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-07-05-17:29:13.101.
Domain: Histories dumped at 2018-07-05-17:29:13.117.
No SumRhoJ to dump.
Dumping all multiFields at 2018-07-05-17:29:13.117.
Dumping edgeE at 2018-07-05-17:29:13.117.
Dumped edgeE at 2018-07-05-17:29:13.133.
Dumping faceB at 2018-07-05-17:29:13.133.
Dumped faceB at 2018-07-05-17:29:13.148.
Dumping invEpsilon at 2018-07-05-17:29:13.148.
Dumped invEpsilon at 2018-07-05-17:29:13.164.
All multiFields dumped at 2018-07-05-17:29:13.195.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-05-17:29:13.195.

Main loop ended at clock time 2018-07-05-17:29:13.195
For 6070 step(s) and 5 dump(s), total time = 148.12 s
For 6070 step(s), 0 - 6070:
   sim. time = 147.08 s, time/step = 0.02423 s,
   time/cell-step = 2.8447e-007 s, core-time/cell-step = 2.8447e-007 s
   time/ptcl-step = 0 s, core-time/ptcl-step = 0 s
For 5 dump(s), 0 - 5:
   time = 1.0469 s, time/dump = 0.20938 s, time/dump/ptcl = 0 s
   (assuming number of particles = 0)
timingAnalysis:, 6070, 6070, 148.12, 147.08, 0.02423, 2.8447e-007, 2.8447e-007, 0, 0, 5, 5, 1.0469, 0.20938, 0, 0
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-07-05-17:29:13.445.
NOTE: A VSimEM license was needed to run this simulation.
Lines from 'phcInMetalCavityT.pre' processed.
Finished with 'phcInMetalCavityT.pre'.
-------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

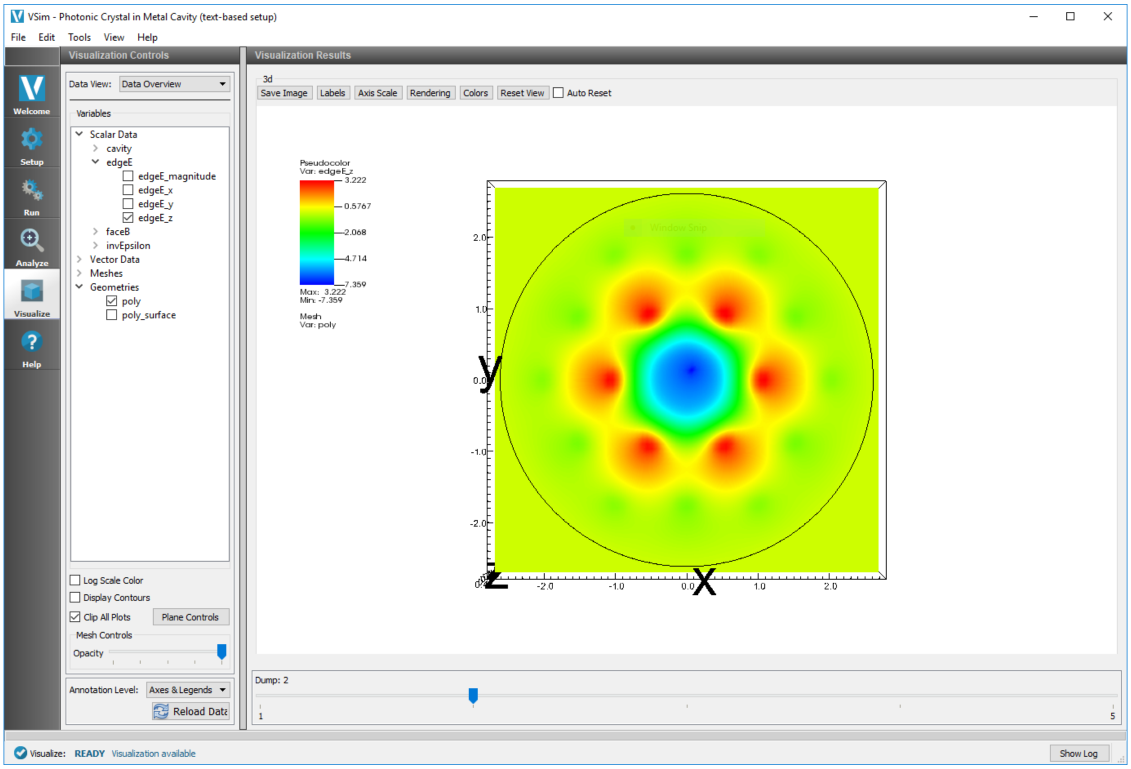
Run: SUCCESS Simulation engine finished successfully

Fig. 3.135: The Run Window at the end of execution.

- Select *poly*

The field at dump 2 is shown in Fig. 3.136.

We can see that fields are trapped by the two layers of dielectric rods, and to a lesser (but final) extent by the surrounding metal cavity.



Data View: Data Overview

Annotation Level: Axes & Legends

Dump: 2

Visualize: READY Visualization available

Fig. 3.136: Visualization of the E_z field component.

CHAPTER

FOUR

VSIM FOR MICROWAVE DEVICE EXAMPLES

These examples illustrate how to solve complex problems in microwave devices.

These examples can be run with a VSimMD license.

4.1 Cavities and Waveguides

4.1.1 Coaxial Cylinder (coax.sdf)

Keywords:

`coax, coaxial geometry, cylinder, current pulse, rlc circuit, step potential`

Problem description

This example probes the electromagnetic properties of a semi-infinite coaxial cylinder. One end of the cylinder lies in the simulation space. The length of the cable is large compared to its diameter. The outer radius is 8 cm, the inner radius is 2 cm, and the section considered is 20 cm long. The inner cylinder is shorter than the outer cylinder and there is an electron absorbing cap on the end of the outer cylinder. When the simulation initiates, a single EM pulse is launched into the open, continuous end of the geometry and propagates to the capped tip. Electrons are ejected from the tip of the inner cylinder when the pulse reaches it.

This computational model is equivalent to applying a step-potential to one end of a coaxial cable. The step-potential propagates at the speed of light until it reaches the tip of the inner cylinder. The RLC nature of the coax cable causes overshoot and ringing of the potential. At the inner tip, an attenuating series of oscillations occurs accompanied by electron emissions. Gradually the tip potential stabilizes at the applied potential.

This simulation can be performed with a VSimMD license.

Opening the simulation

The coax example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Cavities and Waveguides* option.
- Select *Coaxial Cylinder* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

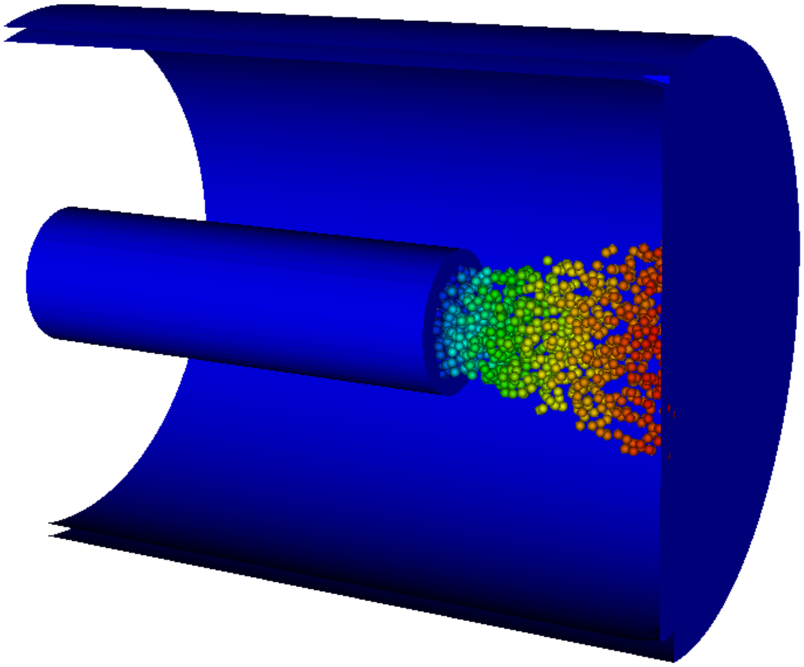


Fig. 4.1: The electrons are emitted from the tip of the inner cylinder after the pulse reaches it.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 4.2. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.

Simulation properties

The coax example includes several Constants for easy adjustment of simulation properties. Those include:

- TFACTOR: A ramping factor of the applied field
- EFACTOR: The amplitude of the applied field
- EMITTED_CURRENT: The current emitted from the tip of the inner cylinder

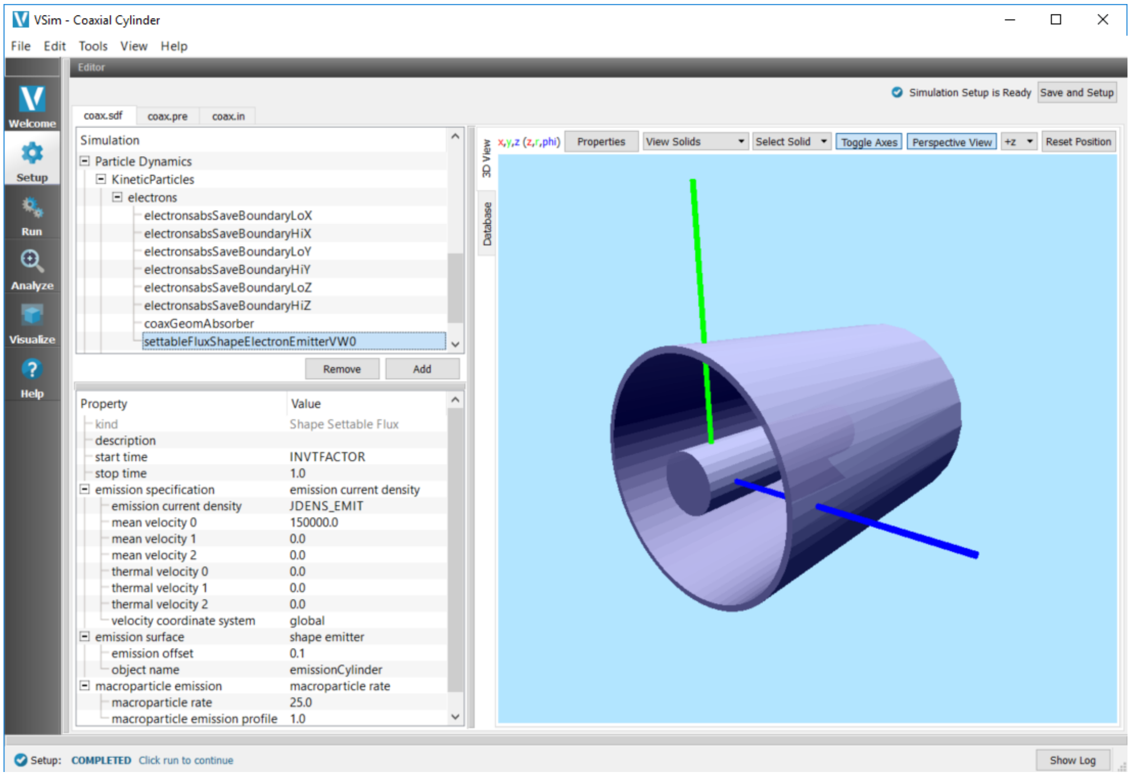
There are also several SpaceTimeFunctions defined for easy application to wave launchers and particle emitters. Those include:

- edgeDy: The applied field in the y-direction
- edgeDz: The applied field in the z-direction
- nomask: This allows emission from the entire geometry of the flux emitter

Other Properties of the simulation include CSG defined geometries, a wave launcher on the lower x boundary, and a settable flux emitter on the tip of the inner cylinder.

Running the simulation

After performing the above actions, continue as follows:



```
Simulation
⊟ Particle Dynamics
  ⊟ KineticParticles
    ⊟ electrons
        electronsabsSaveBoundaryLoX
        electronsabsSaveBoundaryHiX
        electronsabsSaveBoundaryLoY
        electronsabsSaveBoundaryHiY
        electronsabsSaveBoundaryLoZ
        electronsabsSaveBoundaryHiZ
        coaxGeomAbsorber
        settableFluxShapeElectronEmitterVW0
```

Property	Value
kind	Shape Settable Flux
description	
start time	INVTFACTOR
stop time	1.0
⊟ emission specification	emission current density
emission current density	JDENS_EMIT
mean velocity 0	150000.0
mean velocity 1	0.0
mean velocity 2	0.0
thermal velocity 0	0.0
thermal velocity 1	0.0
thermal velocity 2	0.0
velocity coordinate system	global
⊟ emission surface	shape emitter
emission offset	0.1
object name	emissionCylinder
⊟ macroparticle emission	macroparticle rate
macroparticle rate	25.0
macroparticle emission profile	1.0

Setup: COMPLETED Click run to continue

Fig. 4.2: Setup Window for the Coaxial Cylinder example.

- Proceed to the Run Window by pressing the *Run* button in the left column.
- To run the file, click the *Run* button in the upper left corner of the right pane. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 4.3.

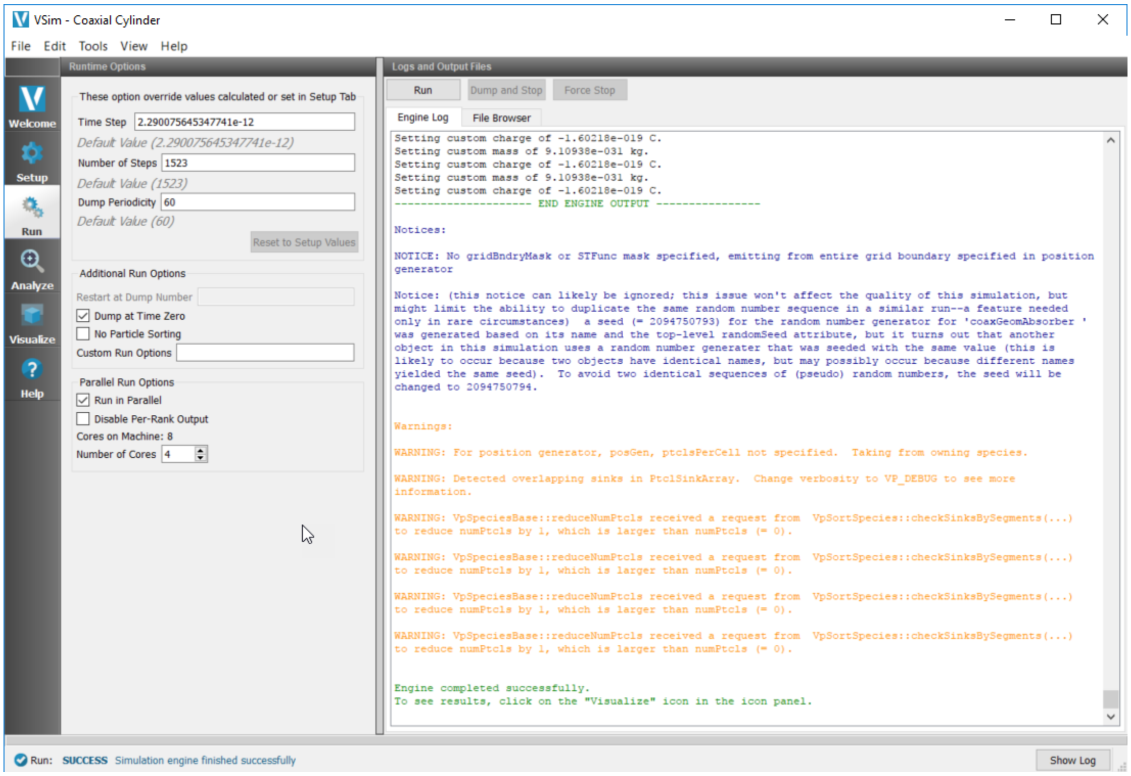
Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by clicking the *Visualize* button in the left column.

To create the image seen in Fig. 4.4, proceed as follows:

- In the variables tree expand *Scalar Data*
- Expand *E*
- Select *E_y*
- Expand *Geometries*
- Select *poly (coaxGeom)*
- Beneath the variables tree, select *Display Contours* and set the # of contours to 10
- Select *Clip All Plots*
- In the Particle Style section, set the size to 6 and choose Sphere for Symbol
- In the variables tree expand *Particle Data*



These option override values calculated or set in Setup Tab

Time Step 2.290075645347741e-12

Default Value (2.290075645347741e-12)

Number of Steps 1523

Default Value (1523)

Dump Periodicity 60

Default Value (60)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Cores on Machine: 8

Number of Cores 4

```
Setting custom charge of -1.60218e-019 C.
Setting custom mass of 9.10938e-031 kg.
Setting custom charge of -1.60218e-019 C.
Setting custom mass of 9.10938e-031 kg.
Setting custom charge of -1.60218e-019 C.
-------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: No gridBndryMask or STFunc mask specified, emitting from entire grid boundary specified in position
generator

Notice: (this notice can likely be ignored; this issue won't affect the quality of this simulation, but
might limit the ability to duplicate the same random number sequence in a similar run--a feature needed
only in rare circumstances)  a seed (= 2094750793) for the random number generator for 'coaxGeomAbsorber '
was generated based on its name and the top-level randomSeed attribute, but it turns out that another
object in this simulation uses a random number generater that was seeded with the same value (this is
likely to occur because two objects have identical names, but may possibly occur because different names
yielded the same seed).  To avoid two identical sequences of (pseudo) random numbers, the seed will be
changed to 2094750794.


Warnings:

WARNING: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more
information.

WARNING: VpSpeciesBase::reduceNumPtcls received a request from  VpSortSpecies::checkSinksBySegments(...)
to reduce numPtcls by 1, which is larger than numPtcls (= 0).

WARNING: VpSpeciesBase::reduceNumPtcls received a request from  VpSortSpecies::checkSinksBySegments(...)
to reduce numPtcls by 1, which is larger than numPtcls (= 0).

WARNING: VpSpeciesBase::reduceNumPtcls received a request from  VpSortSpecies::checkSinksBySegments(...)
to reduce numPtcls by 1, which is larger than numPtcls (= 0).

WARNING: VpSpeciesBase::reduceNumPtcls received a request from  VpSortSpecies::checkSinksBySegments(...)
to reduce numPtcls by 1, which is larger than numPtcls (= 0).


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.3: The Run Window at the end of a successful execution.

- Expand *electrons*
- Select *electrons_ux*
- Now in the right pane move the dump slider forward in time
- The axis and legends can be hidden using the dropdown menu in the lower left corner of the window

To obtain a clearer picture of what is happening at the cylinder tip, switch the Data View (in the left pane) to *History*. One dimensional plots of the number of electrons (called numMacroPtcls), the electric potential (phi), and the emitted and absorbed current should come up automatically.

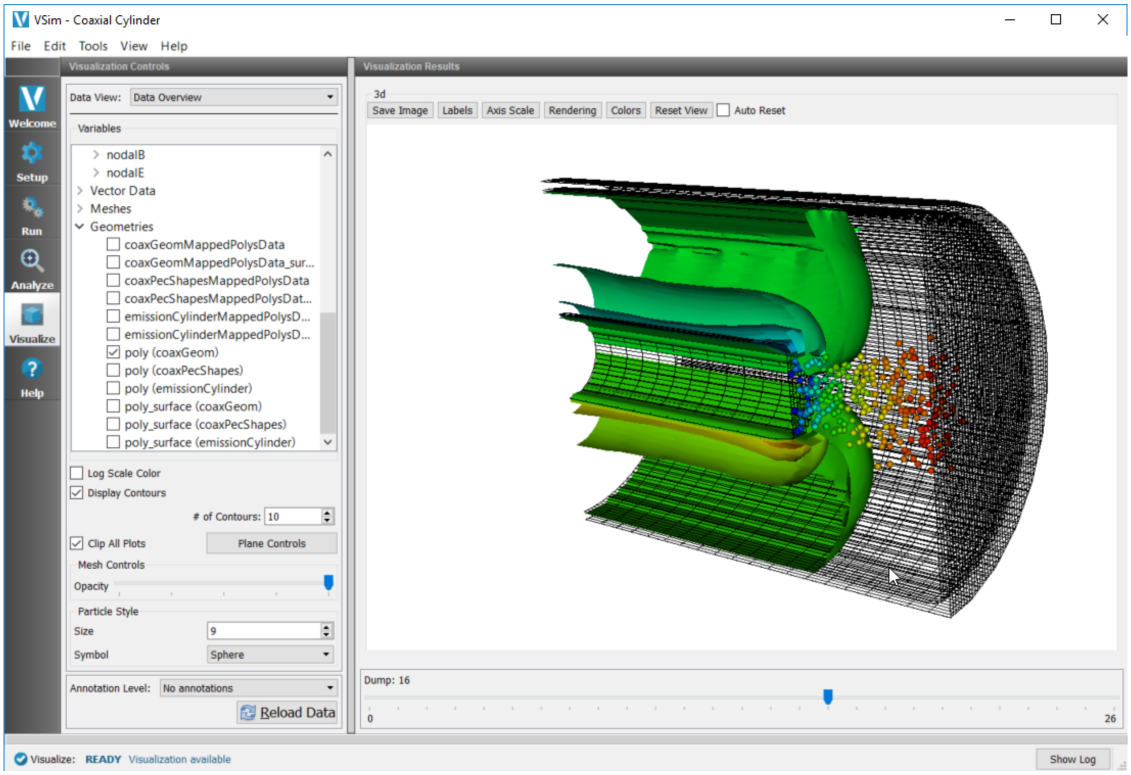
You can set the location of Graph 2 to Window 1 as in Fig. 4.5.

The potential is measured between the interior of the inner cylinder and the capped end of the outer cylinder. The plot of the potential is noisy due to the emission of electrons from the tip. It may be insightful to run the simulation once without electrons so you can see the ringing on the waveform of phi. A similar signal is obtained by hooking up an oscilloscope to a coaxial cable. Electrons can be suppressed by setting the EMITTED_CURRENT parameter to 0 during setup.

The coaxial cylinder behaves like an RLC circuit: the cylinders provide a series resistance along their length, they are coupled capacitively, and generate self-inductance due to the current. By default, the rise-time of the pulse is near the resonance of the circuit, resulting in an acceptable rise time, low overshoot, and quick damping. This makes it a good driver of the circuit.

Further Experiments

Try experimenting with different dimensions of coax. In particular, note how the radii and pulse profile affect the potential response on the phi History plot.



VSim - Coaxial Cylinder

Data View: Data Overview

Variables
- nodalB
- nodalE
- Vector Data
- Meshes
- Geometries
  - □ coaxGeomMappedPolysData
  - □ coaxGeomMappedPolysData_sur...
  - □ coaxPecShapesMappedPolysData
  - □ coaxPecShapesMappedPolysDat...
  - □ emissionCylinderMappedPolysD...
  - □ emissionCylinderMappedPolysD...
  - ☑ poly (coaxGeom)
  - □ poly (coaxPecShapes)
  - □ poly (emissionCylinder)
  - □ poly_surface (coaxGeom)
  - □ poly_surface (coaxPecShapes)
  - □ poly_surface (emissionCylinder)

□ Log Scale Color

☑ Display Contours

# of Contours: 10

☑ Clip All Plots

Particle Style

Size: 9

Symbol: Sphere

Annotation Level: No annotations

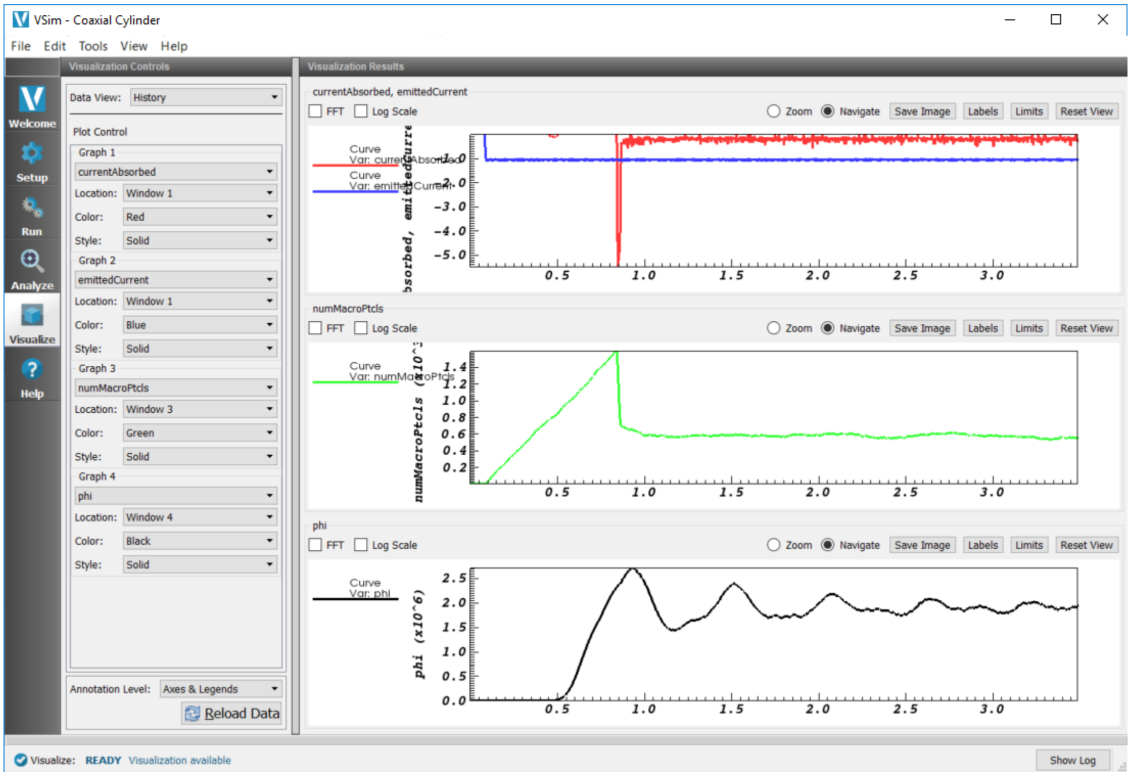
3d

□ Auto Reset

Dump: 16

Visualize: READY Visualization available

Fig. 4.4: Visualization of the coaxial cylinder as a color contour plot.



VSim - Coaxial Cylinder

Data View: History

Plot Control

Graph 1: currentAbsorbed
Location: Window 1
Color: Red
Style: Solid

Graph 2: emittedCurrent
Location: Window 1
Color: Blue
Style: Solid

Graph 3: numMacroPtcls
Location: Window 3
Color: Green
Style: Solid

Graph 4: phi
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

currentAbsorbed, emittedCurrent: □ FFT □ Log Scale ○ Zoom ● Navigate

numMacroPtcls: □ FFT □ Log Scale ○ Zoom ● Navigate

phi: □ FFT □ Log Scale ○ Zoom ● Navigate

Visualize: READY Visualization available

Fig. 4.5: The History visualization window with the electrons.

4.1.2 Cylindrical Waveguide (cylindricalWaveguide.sdf)

Keywords:

electromagnetics, waveguide, dispersion

Problem Description

This VSimMD example illustrates how to find the modes of a cylindrical waveguide.

This simulation can be performed with a VSimMD license.

Simulation Properties

A section of cylindrical waveguide is simulated with the goal of extracting its propagating mode frequencies. The simulation is only two cells wide in X, but through the use of a phase-shifting periodic boundary condition, a much longer waveguide is simulated. The modes are extracted for longitudinal k-vectors, $\frac{2\pi n}{L_x}$. The maximum current is $I_0 = I(\tau/2)$. The waveguide is first excited with a transverse current that is off axis so as to excite modes of any symmetry. The temporal excitation is chosen to excite only a range of frequencies, from somewhat below the lowest cutoff up to the modes corresponding to $n = 1$. The Fourier transform of a history recording the electric field shows a clean output with a modest number of modes. Precise values for those frequencies can be obtained using the extractModes analyzer.

Opening the Simulation

The Cylindrical Waveguide example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Cavities and Waveguides* option.
- Select *Cylindrical Waveguide* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 4.6. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. (To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.) For the current view, the setup has been rotated to be able to see down the waveguide, and the view of the grid has been turned off. The box inside the waveguide is the location of the current source that will drive the waveguide.

Opening `Functions` of the tree shows a function, freqBandPulse, which is

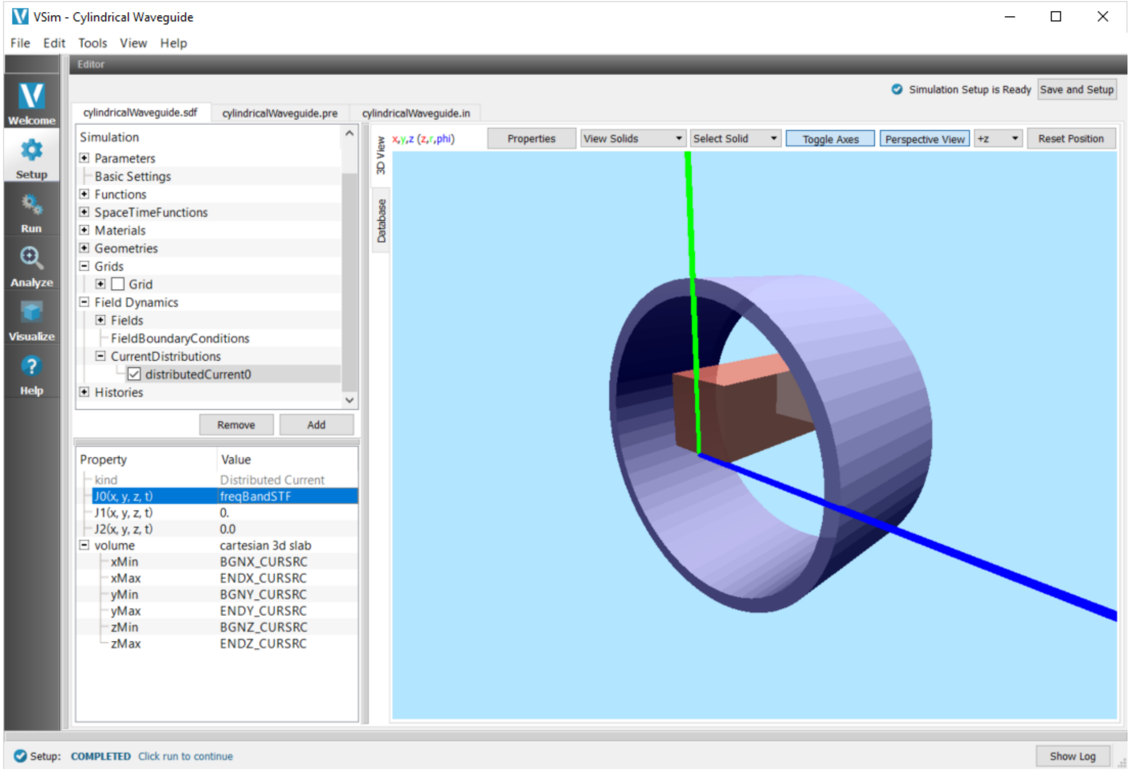
$$freqBandPulse(t, f_l, f_h, \delta_f, t_{off}) = H(t_{off} - t)exp(-\delta_f^2(t - 0.5t_{off})^2)\frac{sin(2\pi f_h(t - 0.5t_{off})) - sin(2\pi f_l(t - 0.5t_{off}))}{(2\pi f_h - 2\pi f_l)(t - 0.5t_{off})}$$

This function has a Fourier spectrum that is fairly flat for $f_l < f < f_h$ and falls off rapidly over a frequency width of δ_f, so that it is nearly zero for $f < f_l - \delta_f$ or $f < f_h + \delta_f$. This excitation gives a range of modes to be analyzed.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run icon in the left panel.



- Simulation
  - ⊞ Parameters
  - Basic Settings
  - ⊞ Functions
  - ⊞ SpaceTimeFunctions
  - ⊞ Materials
  - ⊞ Geometries
  - ⊟ Grids
    - ⊞ □ Grid
  - ⊟ Field Dynamics
    - ⊞ Fields
    - FieldBoundaryConditions
    - ⊟ CurrentDistributions
      - ☑ distributedCurrent0
  - ⊞ Histories

Property	Value
kind	Distributed Current
J0(x, y, z, t)	freqBandSTF
J1(x, y, z, t)	0.
J2(x, y, z, t)	0.0
⊟ volume	cartesian 3d slab
xMin	BGNX_CURSRC
xMax	ENDX_CURSRC
yMin	BGNY_CURSRC
yMax	ENDY_CURSRC
zMin	BGNZ_CURSRC
zMax	ENDZ_CURSRC

Setup: COMPLETED Click run to continue

Fig. 4.6: Initial Setup Window for the Cylindrical Waveguide example.

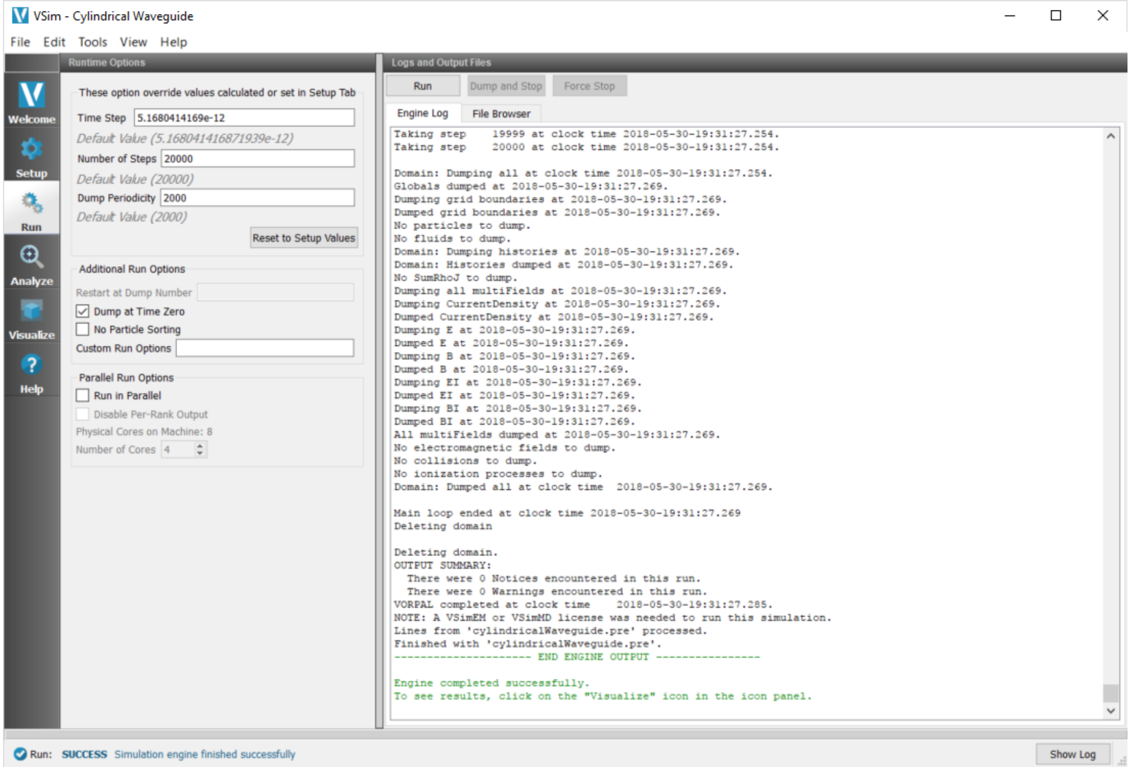
- Check the center panel that you will run for 20,000 steps, dumping every 2,000.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.7.

Visualizing the spectrum

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize icon in the left panel.
- Select History under Data View.
- For Graph 4, select <None> to be plotted.
- Then for each plot select the FFT checkbox
- In the upper right corner of each plot, select Limits and set X-Axis max to 2e9.
- The result should be that shown in Fig. 4.8.

One can see the TM mode in this spectrum. One can measure the mode frequency by projecting the spectrum down on the axis. With this simulation of 20,000 steps, for a total time of 103 ns, one expects the peak to have a width of roughly 1/103 ns or 0.01 GHz. This gives the error in the frequency from this method.



These option override values calculated or set in Setup Tab

Time Step 5.1680414169e-12
Default Value (5.168041416871939e-12)
Number of Steps 20000
Default Value (20000)
Dump Periodicity 2000
Default Value (2000)

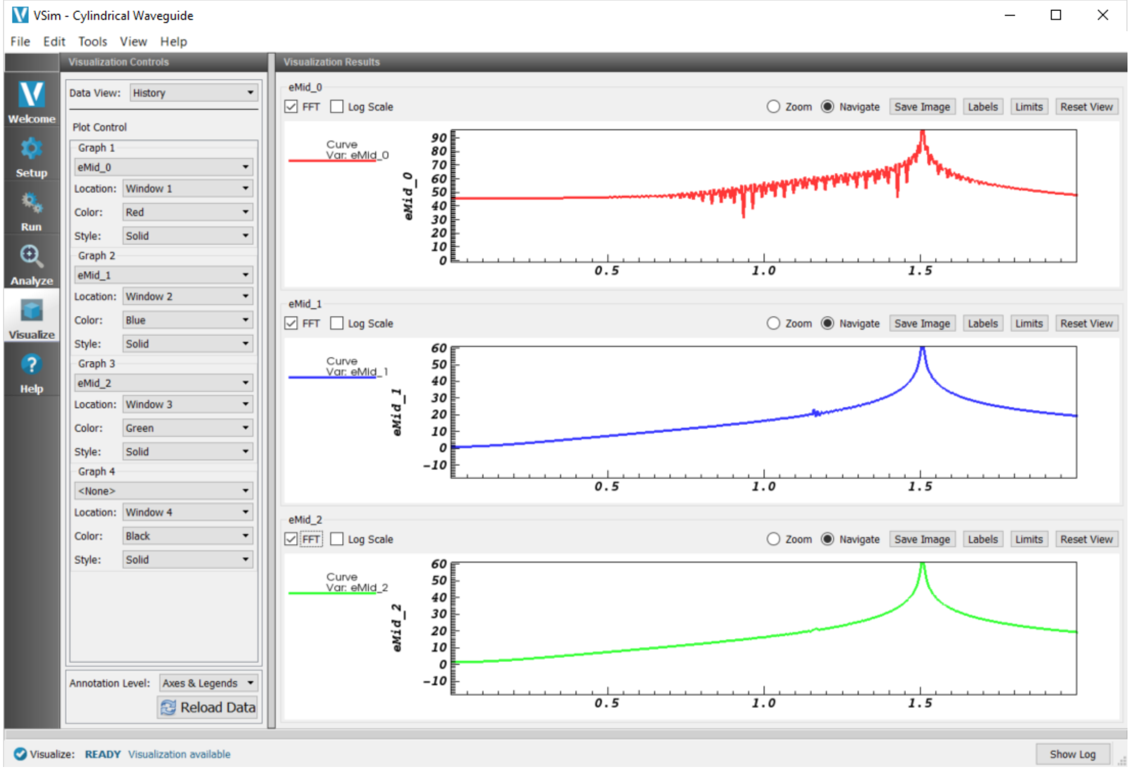
Additional Run Options
Restart at Dump Number
☑ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options
☐ Run in Parallel
☐ Disable Per-Rank Output
Physical Cores on Machine: 8
Number of Cores 4

```
Taking step    19999 at clock time 2018-05-30-19:31:27.254.
Taking step    20000 at clock time 2018-05-30-19:31:27.254.

Domain: Dumping all at clock time 2018-05-30-19:31:27.254.
Globals dumped at 2018-05-30-19:31:27.269.
Dumping grid boundaries at 2018-05-30-19:31:27.269.
Dumped grid boundaries at 2018-05-30-19:31:27.269.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-05-30-19:31:27.269.
Domain: Histories dumped at 2018-05-30-19:31:27.269.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-30-19:31:27.269.
Dumping CurrentDensity at 2018-05-30-19:31:27.269.
Dumped CurrentDensity at 2018-05-30-19:31:27.269.
Dumping E at 2018-05-30-19:31:27.269.
Dumped E at 2018-05-30-19:31:27.269.
Dumping B at 2018-05-30-19:31:27.269.
Dumped B at 2018-05-30-19:31:27.269.
Dumping EI at 2018-05-30-19:31:27.269.
Dumped EI at 2018-05-30-19:31:27.269.
Dumping BI at 2018-05-30-19:31:27.269.
Dumped BI at 2018-05-30-19:31:27.269.
All multiFields dumped at 2018-05-30-19:31:27.269.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-30-19:31:27.269.

Main loop ended at clock time 2018-05-30-19:31:27.269
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-05-30-19:31:27.285.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'cylindricalWaveguide.pre' processed.
Finished with 'cylindricalWaveguide.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 4.7: Run Window for the Cylindrical Waveguide example after the initial run.



Data View: History

Plot Control
Graph 1
eMid_0
Location: Window 1
Color: Red
Style: Solid
Graph 2
eMid_1
Location: Window 2
Color: Blue
Style: Solid
Graph 3
eMid_2
Location: Window 3
Color: Green
Style: Solid
Graph 4
<None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

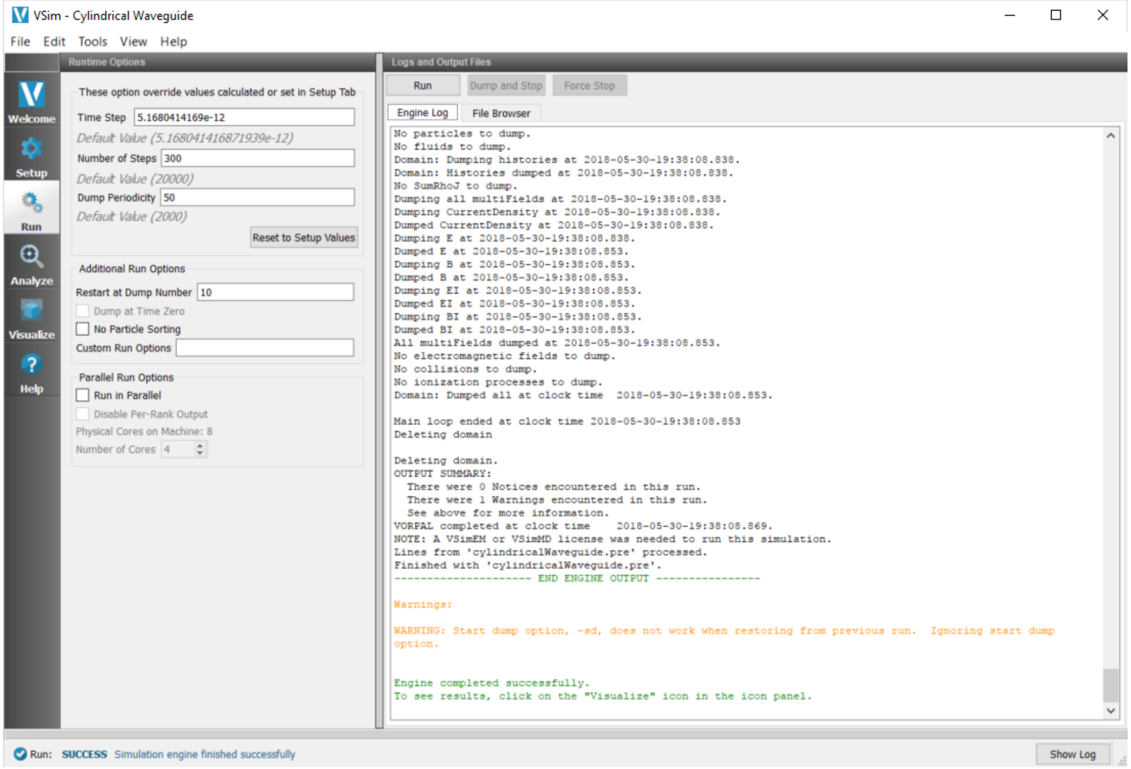
eMid_0: ☑ FFT ☐ Log Scale
eMid_1: ☑ FFT ☐ Log Scale
eMid_2: ☑ FFT ☐ Log Scale

Visualize: **READY** Visualization available

Fig. 4.8: Spectrum for the Cylindrical Waveguide example after the initial run.

Analyzing the Spectrum

We can obtain more accurate frequencies using the Filter Diagonalization Method. To do this, we need to take a bit more data. We need to have the number of dumps equal to three times the number of modes, so we run again, restarting from dump 10 for another 300 steps, dumping every 50 time steps. This will give us an additional 6 dumps. The Run Window for this part of the simulation is shown in Fig. 4.9.



These option override values calculated or set in Setup Tab

Time Step 5.1680414169e-12
Default Value (5.168041416871939e-12)
Number of Steps 300
Default Value (20000)
Dump Periodicity 50
Default Value (2000)

Additional Run Options

Restart at Dump Number 10
□ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options

□ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 8
Number of Cores 4

```
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-05-30-19:38:08.838.
Domain: Histories dumped at 2018-05-30-19:38:08.838.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-30-19:38:08.838.
Dumping CurrentDensity at 2018-05-30-19:38:08.838.
Dumped CurrentDensity at 2018-05-30-19:38:08.838.
Dumping E at 2018-05-30-19:38:08.838.
Dumped E at 2018-05-30-19:38:08.853.
Dumping B at 2018-05-30-19:38:08.853.
Dumped B at 2018-05-30-19:38:08.853.
Dumping EI at 2018-05-30-19:38:08.853.
Dumped EI at 2018-05-30-19:38:08.853.
Dumping BI at 2018-05-30-19:38:08.853.
Dumped BI at 2018-05-30-19:38:08.853.
All multiFields dumped at 2018-05-30-19:38:08.853.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-30-19:38:08.853.

Main loop ended at clock time 2018-05-30-19:38:08.853
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 1 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-05-30-19:38:08.869.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'cylindricalWaveguide.pre' processed.
Finished with 'cylindricalWaveguide.pre'.
-------------------- END ENGINE OUTPUT ----------------

Warnings:

WARNING: Start dump option, -sd, does not work when restoring from previous run.  Ignoring start dump
option.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.9: Run Window for the Cylindrical Waveguide example for the second run.

We now move to the Analyze Window, open extractModes, and set the field to be E. Then set the number of modes to be 2, and the begin and end dumps to be 10 and 16, respectively. Also set sampleType to 1. Upon hitting the `Analyze` button in the upper right, one sees the analysis output as shown in Fig. 4.10.

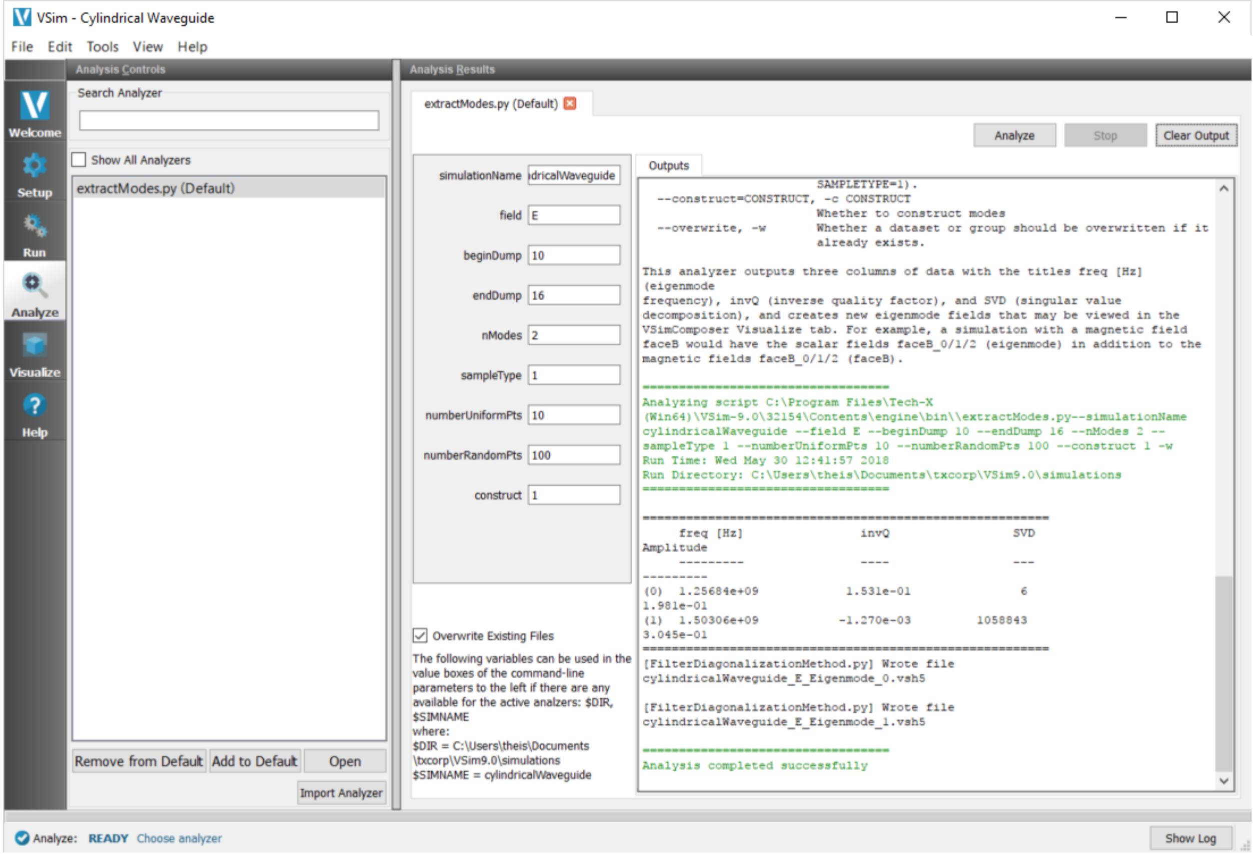
The computed mode frequencies are shown along with the inverse-Q values. Since this system is not lossy, the values of invQ, when significant, indicate that the mode calculations are dubious. However, we see that the 2nd mode has been well obtained.

These modes will now show up in the visualize panel, where one can reload the data, and modes will show up as seen in Fig. 4.11. The well obtained mode occupies dumps 1-16.

4.1.3 Pillbox Cavity (pillboxCavity.sdf)

Keywords:

`Pillbox cavity, Figures of merit, Transit time factor, Geometry factor`



simulationName ıdricalWaveguide

field E

beginDump 10

endDump 16

nModes 2

sampleType 1

numberUniformPts 10

numberRandomPts 100

construct 1

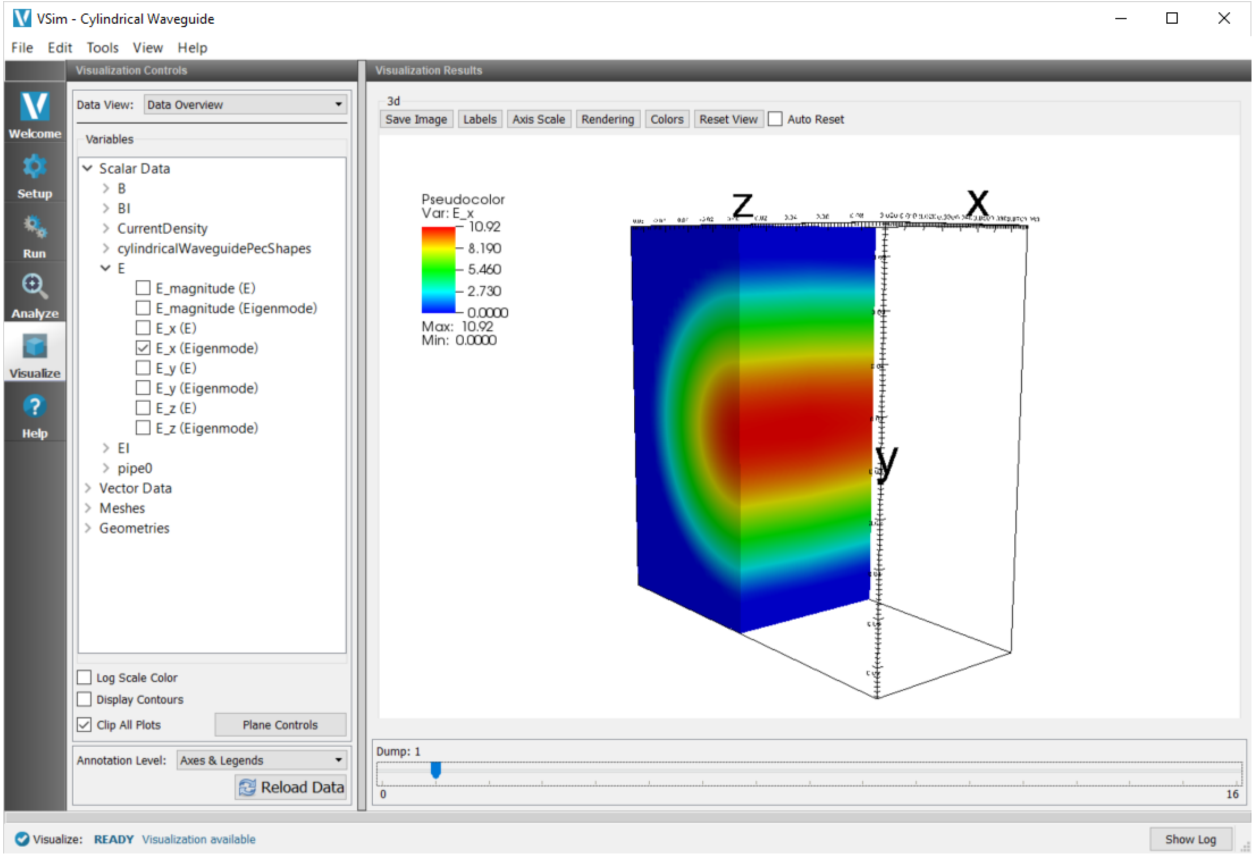
☑ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\theis\Documents\txcorp\VSim9.0\simulations
$SIMNAME = cylindricalWaveguide

```
                      SAMPLETYPE=1).
  --construct=CONSTRUCT, -c CONSTRUCT
                      Whether to construct modes
  --overwrite, -w     Whether a dataset or group should be overwritten if it
                      already exists.

This analyzer outputs three columns of data with the titles freq [Hz]
(eigenmode
frequency), invQ (inverse quality factor), and SVD (singular value
decomposition), and creates new eigenmode fields that may be viewed in the
VSimComposer Visualize tab. For example, a simulation with a magnetic field
faceB would have the scalar fields faceB_0/1/2 (eigenmode) in addition to the
magnetic fields faceB_0/1/2 (faceB).

==================================
Analyzing script C:\Program Files\Tech-X
(Win64)\VSim-9.0\32154\Contents\engine\bin\\extractModes.py--simulationName
cylindricalWaveguide --field E --beginDump 10 --endDump 16 --nModes 2 --
sampleType 1 --numberUniformPts 10 --numberRandomPts 100 --construct 1 -w
Run Time: Wed May 30 12:41:57 2018
Run Directory: C:\Users\theis\Documents\txcorp\VSim9.0\simulations
==================================

========================================================
     freq [Hz]                invQ                 SVD
Amplitude
     ---------                ----                 ---
---------
(0)  1.25684e+09            1.531e-01                6
1.981e-01
(1)  1.50306e+09           -1.270e-03          1058843
3.045e-01
========================================================
[FilterDiagonalizationMethod.py] Wrote file
cylindricalWaveguide_E_Eigenmode_0.vsh5

[FilterDiagonalizationMethod.py] Wrote file
cylindricalWaveguide_E_Eigenmode_1.vsh5

==================================
Analysis completed successfully
```

Analyze: READY Choose analyzer

Fig. 4.10: Analysis window for the Cylindrical Waveguide example for mode extraction.



Data View: Data Overview

Variables

- Scalar Data
  - B
  - BI
  - CurrentDensity
  - cylindricalWaveguidePecShapes
  - E
    - □ E_magnitude (E)
    - □ E_magnitude (Eigenmode)
    - □ E_x (E)
    - ☑ E_x (Eigenmode)
    - □ E_y (E)
    - □ E_y (Eigenmode)
    - □ E_z (E)
    - □ E_z (Eigenmode)
  - El
  - pipe0
- Vector Data
- Meshes
- Geometries

□ Log Scale Color
□ Display Contours
☑ Clip All Plots

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 1

Visualize: READY Visualization available

Fig. 4.11: Analysis window for the Cylindrical Waveguide example for mode extraction.

Problem description

This VSimMD example demonstrates the usage of VSim in computing the eigenmodes and figures of merit of two simple cavities. One may select either the closed pillbox cavity for which the analytic solution is well known, or a cavity based on the closed pillbox, but having outlets leading to the periodic domain boundaries. Like other examples utilizing the extractModes.py analyzer, the simulation run is done in two steps. In the first step, the cavity is excited by a sinc pulse current source and output is dumped only at the end of this excitation run. Then in the second step, output is dumped at intervals which are sufficiently short compared to the frequencies of interest. The output from the second run is used by the extractModes.py analyzer to compute the eigenmodes. Then, the computeTransitTimeFactor.py and computeCavityG analyzers are used to compute the transit time factors and geometry factors of the eigenmodes.

This simulation can be performed with a VSimMD or VSimEM license.

Opening the Simulation

The pillbox cavity example is accessed from within VSimComposer by the following actions:

- Go to *File* → *New* → *From Example…*
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Cavities and Waveguides* option.
- Select “Pillbox Cavity” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The properties and values that create the simulation are accessible in the left pane when the Setup Window is selected. The right pane shows a 3D view of the selected geometry components, grids and current distributions.

The geometry of the closed pillbox cavity is called *pillboxCavityAnalytical* and the geometry of the periodic cavity with outlets on either end is called *pillboxCavityWithTube*. These can be visualized individually by expanding *Geometries*, de-selecting and then expanding *CSG*, and then selecting either *pillboxCavityAnalytical* or *pillboxCavityWithTube*.

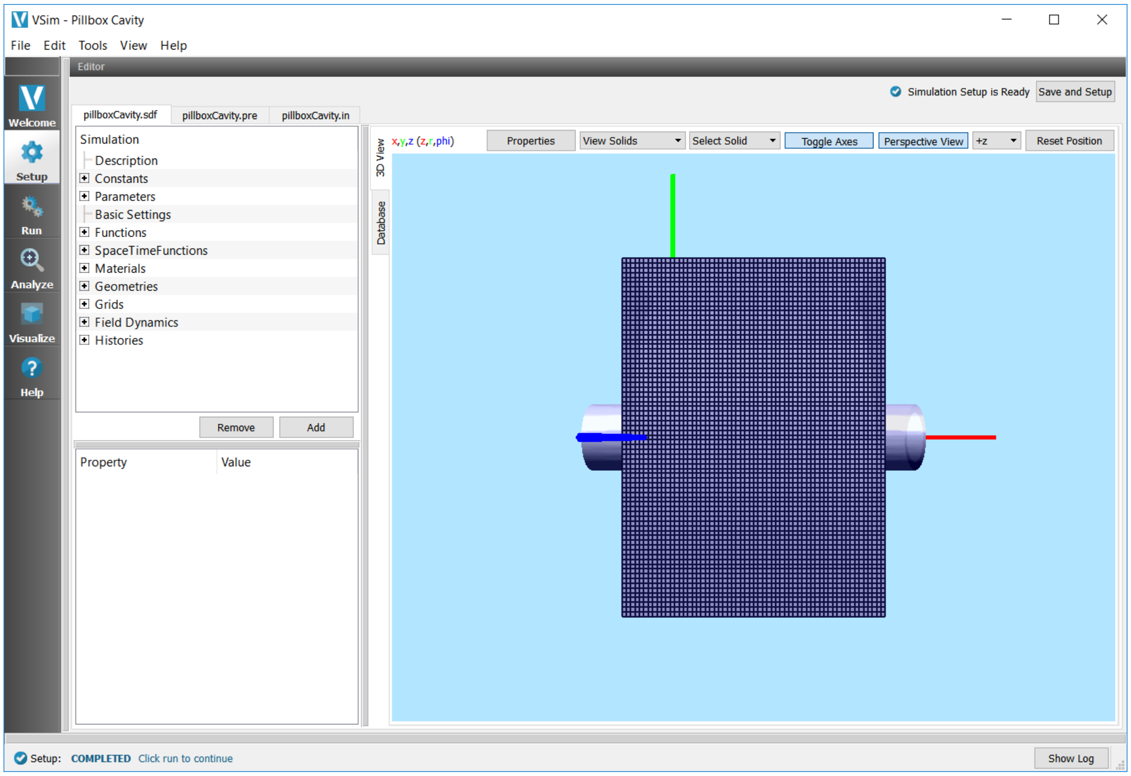
Running the Simulation and Analyzing Results

Step 1: Cavity selection

- If you want to model the closed cavity, skip **Step 1** and go to **Step 2**. The closed cavity is set by default.
- To model the periodic cavity, go to the Setup Window.
- Go to *Geometries* → *CSG*.
- Click on *pillboxCavityAnalytical* under *CSG*.
- The bottom left pane will show properties of the selected geometry. At this time, the material should be set to *PEC* (perfect electric conductor). Double click on *PEC* and select the blank line.
- Now click on *pillboxCavityWithTube* under *CSG*.
- Select *PEC* as the material for *pillboxCavityWithTube*.

Step 2: Excitation

- Go to the Run Window by pressing the Run button in the left column of buttons.
- This simulation may be accelerated by running on multiple MPI ranks. The parallel options are in the *Parallel Run Options* box below *Additional Run Options* in the *Runtime Options* pane.



Simulation Setup is Ready

Setup: COMPLETED Click run to continue

Fig. 4.12: Visualizing the periodic cavity geometry in the Setup Window.

- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in this pane. The simulation will run for 30000 time steps and dump output once at the end. The run has completed when you see the output, “Engine completed successfully.” A snapshot of the simulation run completion is shown in Fig. 4.13.

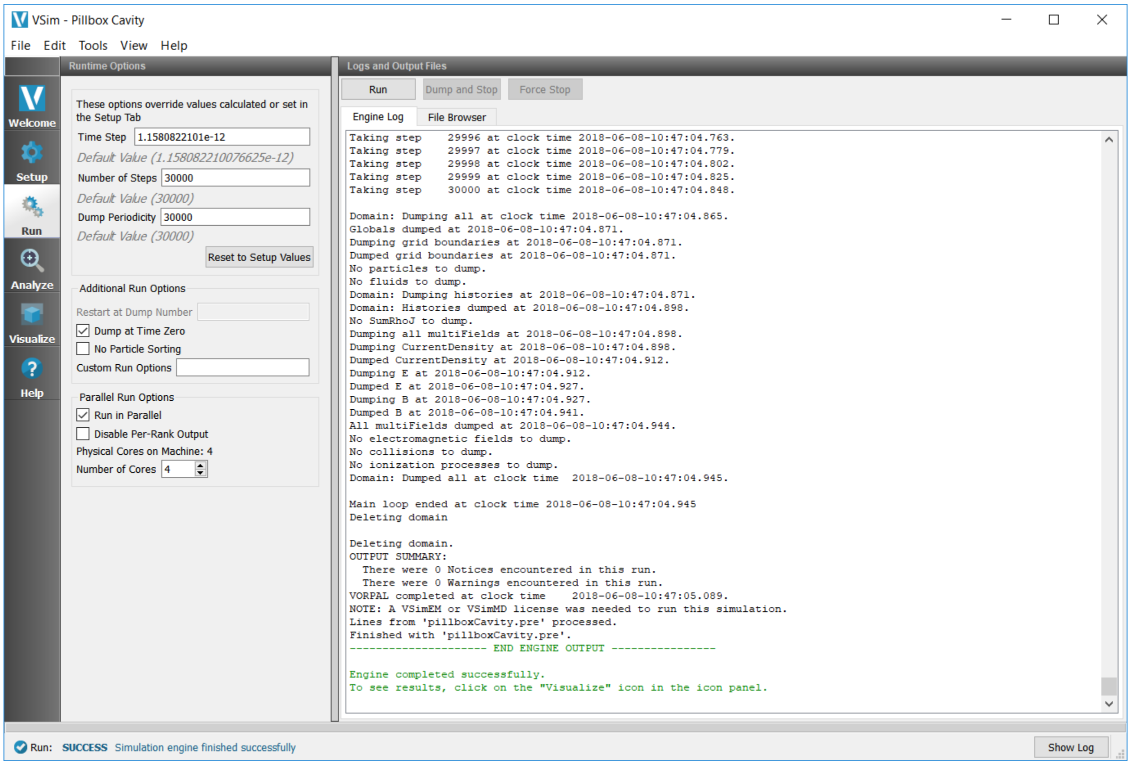
Step 3: Evolving the excited cavity

- After the first step is complete, change *Number of Steps* to 2000, change *Dump Periodicity* to 100.
- In the *Additional Run Options* Box, make sure that the *Dump at Time Zero* box is unchecked and that *Restart at Dump Number* is set to 1.
- Click run. The run has completed when you see the output, “Engine completed successfully.” A snapshot of the simulation run completion is shown in Fig. 4.14. When this run is finished, the last step should be step 32000.

Note: The simulation must be run in two steps because there must be no driving currents flowing in the simulation while dumping data used to extract the eigenmodes. So, while the drive is ringing the cavity, there is no need to dump data. We switch the dump periodicity after the driving current has shut off in order to resolve the frequency of the eignemodes of interest.

Step 4: Computing the eigenmodes

- Go to the analyzer window by selecting *Analyze* in the left column.



These options override values calculated or set in the Setup Tab

Time Step 1.1580822101e-12

Default Value (1.158082210076625e-12)

Number of Steps 30000

Default Value (30000)

Dump Periodicity 30000

Default Value (30000)

Reset to Setup Values

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

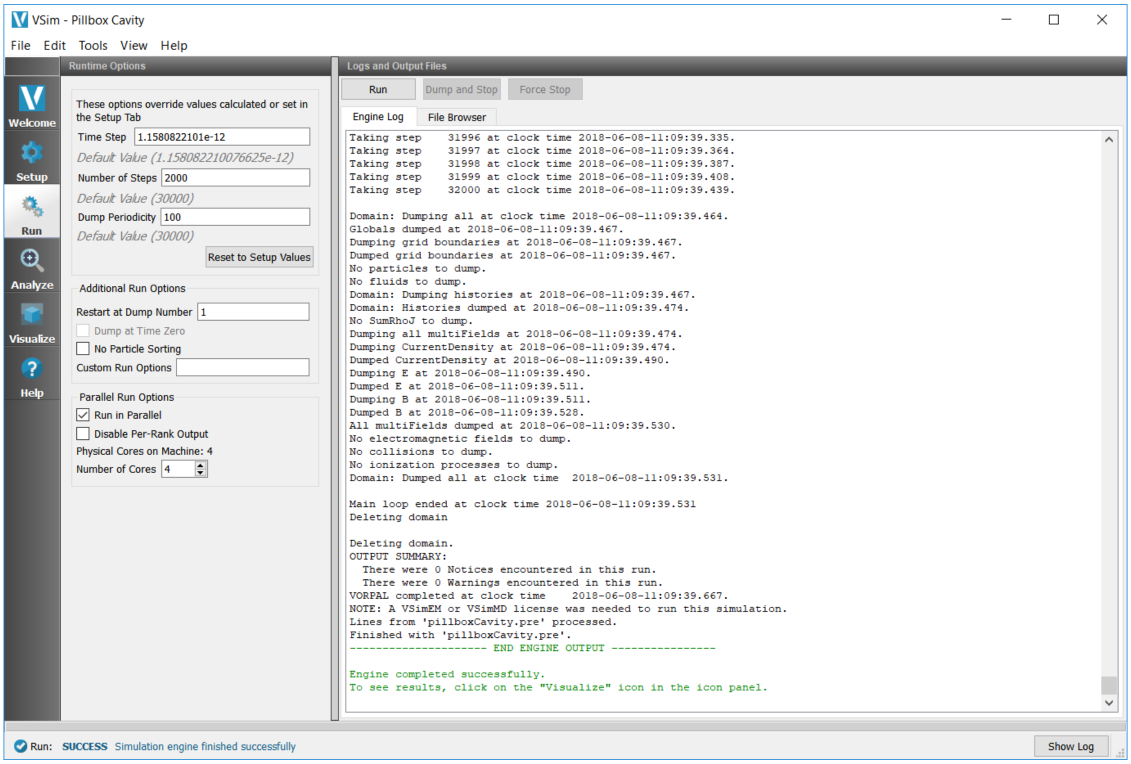
Physical Cores on Machine: 4

Number of Cores 4

```
Taking step   29996 at clock time 2018-06-08-10:47:04.763.
Taking step   29997 at clock time 2018-06-08-10:47:04.779.
Taking step   29998 at clock time 2018-06-08-10:47:04.802.
Taking step   29999 at clock time 2018-06-08-10:47:04.825.
Taking step   30000 at clock time 2018-06-08-10:47:04.848.

Domain: Dumping all at clock time 2018-06-08-10:47:04.865.
Globals dumped at 2018-06-08-10:47:04.871.
Dumping grid boundaries at 2018-06-08-10:47:04.871.
Dumped grid boundaries at 2018-06-08-10:47:04.871.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-06-08-10:47:04.871.
Domain: Histories dumped at 2018-06-08-10:47:04.898.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-08-10:47:04.898.
Dumping CurrentDensity at 2018-06-08-10:47:04.898.
Dumped CurrentDensity at 2018-06-08-10:47:04.912.
Dumping E at 2018-06-08-10:47:04.912.
Dumped E at 2018-06-08-10:47:04.927.
Dumping B at 2018-06-08-10:47:04.927.
Dumped B at 2018-06-08-10:47:04.941.
All multiFields dumped at 2018-06-08-10:47:04.944.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-08-10:47:04.945.

Main loop ended at clock time 2018-06-08-10:47:04.945
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-08-10:47:05.089.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'pillboxCavity.pre' processed.
Finished with 'pillboxCavity.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 4.13: The Run window at the end of a successful execution.



These options override values calculated or set in the Setup Tab

Time Step 1.1580822101e-12

Default Value (1.158082210076625e-12)

Number of Steps 2000

Default Value (30000)

Dump Periodicity 100

Default Value (30000)

Reset to Setup Values

Additional Run Options

Restart at Dump Number 1

☐ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

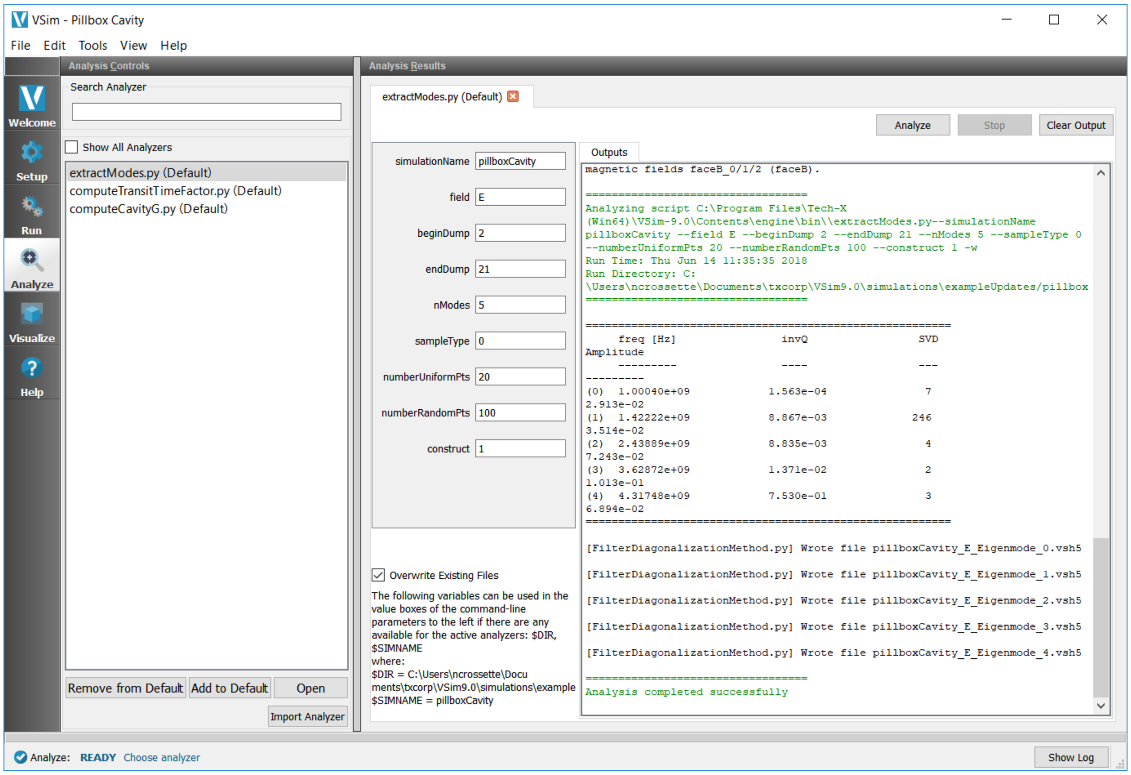
```
Taking step   31996 at clock time 2018-06-08-11:09:39.335.
Taking step   31997 at clock time 2018-06-08-11:09:39.364.
Taking step   31998 at clock time 2018-06-08-11:09:39.387.
Taking step   31999 at clock time 2018-06-08-11:09:39.408.
Taking step   32000 at clock time 2018-06-08-11:09:39.439.

Domain: Dumping all at clock time 2018-06-08-11:09:39.464.
Globals dumped at 2018-06-08-11:09:39.467.
Dumping grid boundaries at 2018-06-08-11:09:39.467.
Dumped grid boundaries at 2018-06-08-11:09:39.467.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-06-08-11:09:39.467.
Domain: Histories dumped at 2018-06-08-11:09:39.474.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-08-11:09:39.474.
Dumping CurrentDensity at 2018-06-08-11:09:39.474.
Dumped CurrentDensity at 2018-06-08-11:09:39.490.
Dumping E at 2018-06-08-11:09:39.490.
Dumped E at 2018-06-08-11:09:39.511.
Dumping B at 2018-06-08-11:09:39.511.
Dumped B at 2018-06-08-11:09:39.528.
All multiFields dumped at 2018-06-08-11:09:39.530.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-08-11:09:39.531.

Main loop ended at clock time 2018-06-08-11:09:39.531
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-08-11:09:39.667.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'pillboxCavity.pre' processed.
Finished with 'pillboxCavity.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 4.14: The Run window at the end of a successful execution.

- Select *extractModes.py* from the set of default analyzers. Then click “Open” on the bottom right of the *Analysis Controls* pane.
- Compute the electric field eigenfunctions. After the analyzer loads, ensure the following parameters are entered:
  - **simulationName**: “pillboxCavity”
  - **field**: “E”
  - **beginDump**: “2”
  - **endDump**: “21”
  - **nModes**: “5”
  - **sampleType**: “0”
  - **numberUniformPoints**: “20”
  - **numberRandomPoints**: “100”
  - **construct**: “1”

  Also, check the “Overwrite Existing Files” box. Double-check your entries against what is shown in Fig. 4.15.



simulationName: pillboxCavity
field: E
beginDump: 2
endDump: 21
nModes: 5
sampleType: 0
numberUniformPts: 20
numberRandomPts: 100
construct: 1

☑ Overwrite Existing Files

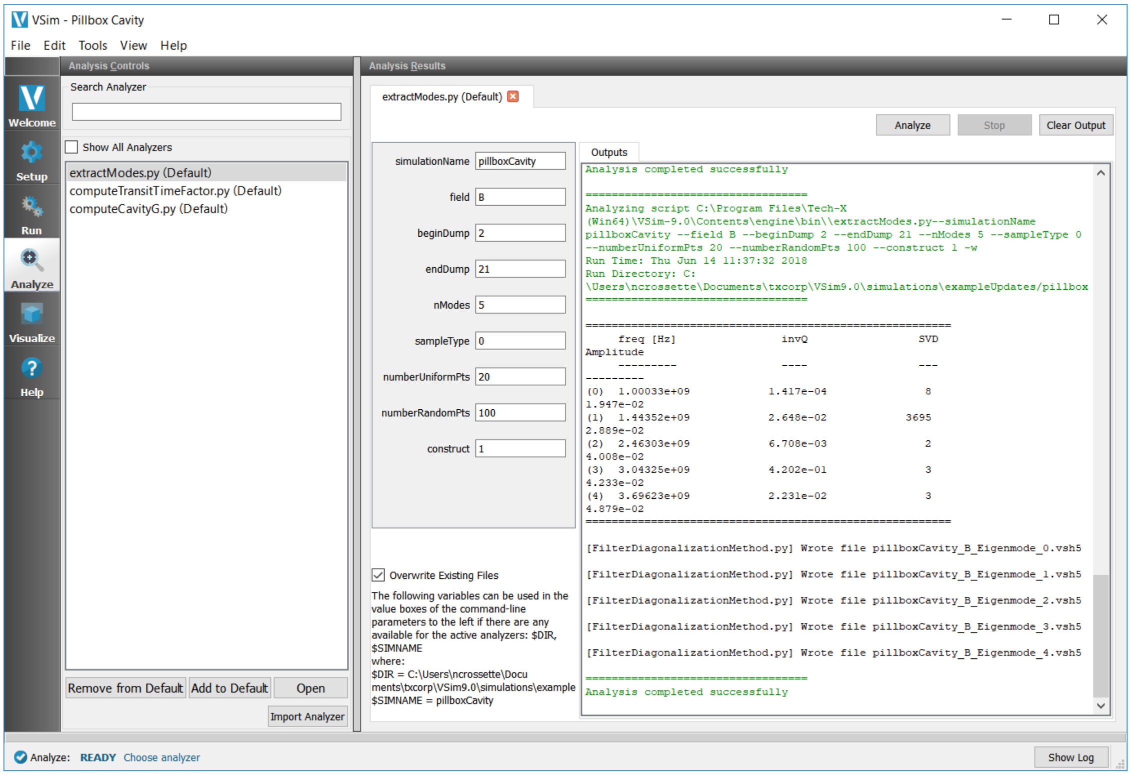
The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analyzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\ncrossette\Documents\txcorp\VSim9.0\simulations\example
$SIMNAME = pillboxCavity

```
magnetic fields faceB_0/1/2 (faceB).

==================================
Analyzing script C:\Program Files\Tech-X
(Win64)\VSim-9.0\Contents\engine\bin\\extractModes.py--simulationName
pillboxCavity --field E --beginDump 2 --endDump 21 --nModes 5 --sampleType 0
--numberUniformPts 20 --numberRandomPts 100 --construct 1 -w
Run Time: Thu Jun 14 11:35:35 2018
Run Directory: C:
\Users\ncrossette\Documents\txcorp\VSim9.0\simulations\exampleUpdates/pillbox
==================================


=======================================================
     freq [Hz]                 invQ                 SVD
Amplitude
     ---------                 ----                 ---
---------
(0)  1.00040e+09             1.563e-04               7
2.913e-02
(1)  1.42222e+09             8.867e-03             246
3.514e-02
(2)  2.43889e+09             8.835e-03               4
7.243e-02
(3)  3.62872e+09             1.371e-02               2
1.013e-01
(4)  4.31748e+09             7.530e-01               3
6.894e-02
=======================================================

[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_E_Eigenmode_0.vsh5

[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_E_Eigenmode_1.vsh5

[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_E_Eigenmode_2.vsh5

[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_E_Eigenmode_3.vsh5

[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_E_Eigenmode_4.vsh5

==================================
Analysis completed successfully
```

Analyze: READY Choose analyzer

Fig. 4.15: Computing the electric field eigenfunctions and frequencies using the extractModes.py analyzer.

- Press the *Analyze* button which is located in the upper right corner.
- Compute the magnetic field eigenfunctions. Change *field* to “B” and press *Analyze* again. After the analysis is finished, and scrolling down in the *Outputs* log pane you should see what is shown in Fig. 4.16.
- Note that *extractModes.py* outputs the frequencies of the computed modes in the *Run Output* pane. The first mode, mode 0, should have a frequency of approximately 1 GHz.



simulationName: pillboxCavity
field: B
beginDump: 2
endDump: 21
nModes: 5
sampleType: 0
numberUniformPts: 20
numberRandomPts: 100
construct: 1

☑ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analyzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\ncrossette\Documents\txcorp\VSim9.0\simulations\example
$SIMNAME = pillboxCavity

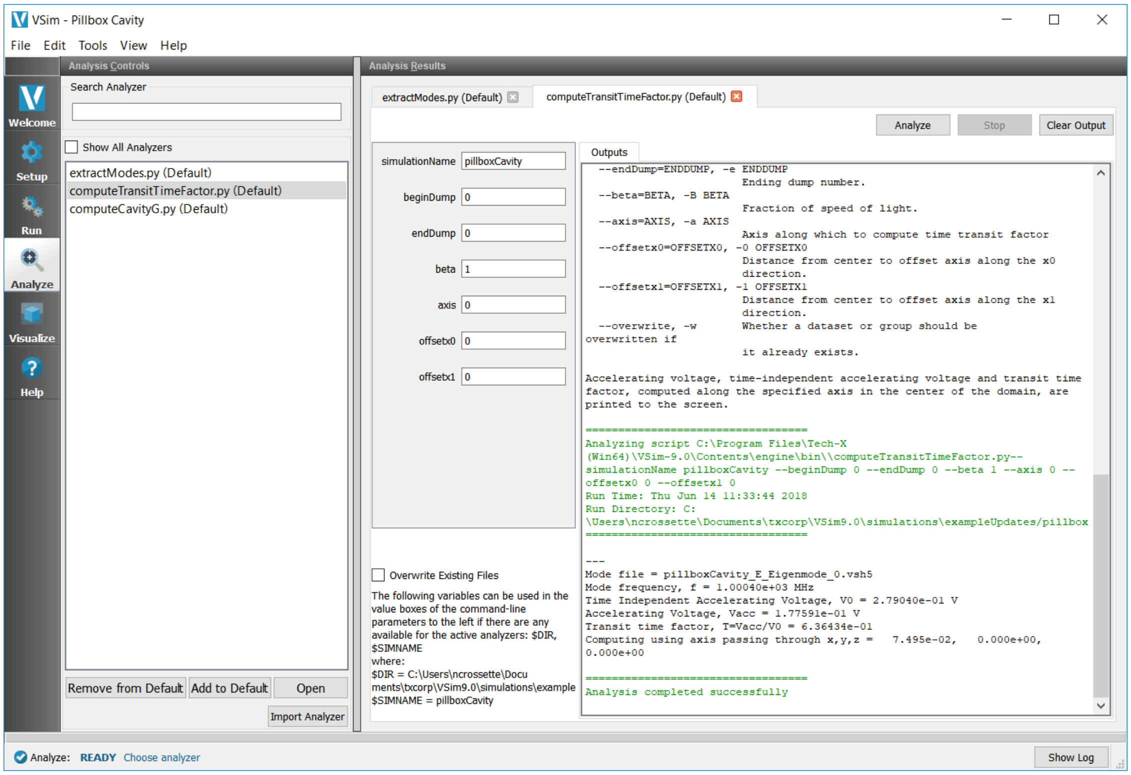
```
Analysis completed successfully

==================================
Analyzing script C:\Program Files\Tech-X
(Win64)\VSim-9.0\Contents\engine\bin\\extractModes.py--simulationName
pillboxCavity --field B --beginDump 2 --endDump 21 --nModes 5 --sampleType 0
--numberUniformPts 20 --numberRandomPts 100 --construct 1 -w
Run Time: Thu Jun 14 11:37:32 2018
Run Directory: C:
\Users\ncrossette\Documents\txcorp\VSim9.0\simulations\exampleUpdates/pillbox
==================================


========================================================
     freq [Hz]                invQ                SVD
Amplitude
     ---------                ----                ---
---------
(0)  1.00033e+09           1.417e-04                8
1.947e-02
(1)  1.44352e+09           2.648e-02             3695
2.889e-02
(2)  2.46303e+09           6.708e-03                2
4.008e-02
(3)  3.04325e+09           4.202e-01                3
4.233e-02
(4)  3.69623e+09           2.231e-02                3
4.879e-02
========================================================

[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_B_Eigenmode_0.vsh5

[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_B_Eigenmode_1.vsh5

[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_B_Eigenmode_2.vsh5

[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_B_Eigenmode_3.vsh5

[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_B_Eigenmode_4.vsh5

==================================
Analysis completed successfully
```

Fig. 4.16: The *Outputs* pane after Analyzing to determine the eigenmodes of the magentic field.

Step 5: Computing the transit time factor

- Select *computeTransitTimeFactor.py* from the default analyzers and press “Open” on the bottom right of the *Analysis Controls* pane.
- After the analyzer loads, ensure the following parameters are entered:
  - **`simulationName`**: “pillboxCavity”
  - **`beginDump`**: “0”
  - **`endDump`**: “0”
  - **`beta`**: “1”
  - **`axis`**: “0”
  - **`offsetx0`**: “0”
  - **`offsetx1`**: “0”

  And compare against what is shown in Fig. 4.17
- Press *Analyze*.
- If you have selected the closed cavity, the transit time factor (the value following “Transit time factor, T=Vacc/V0 =”) should be very close the the analytic value of $2/\pi$.



simulationName: pillboxCavity
beginDump: 0
endDump: 0
beta: 1
axis: 0
offsetx0: 0
offsetx1: 0

□ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analyzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\ncrossette\Documents\txcorp\VSim9.0\simulations\example
$SIMNAME = pillboxCavity

```
  --endDump=ENDDUMP, -e ENDDUMP
                        Ending dump number.
  --beta=BETA, -B BETA
                        Fraction of speed of light.
  --axis=AXIS, -a AXIS
                        Axis along which to compute time transit factor
  --offsetx0=OFFSETX0, -0 OFFSETX0
                        Distance from center to offset axis along the x0
                        direction.
  --offsetx1=OFFSETX1, -1 OFFSETX1
                        Distance from center to offset axis along the x1
                        direction.
  --overwrite, -w       Whether a dataset or group should be
overwritten if
                        it already exists.

Accelerating voltage, time-independent accelerating voltage and transit time
factor, computed along the specified axis in the center of the domain, are
printed to the screen.

==================================
Analyzing script C:\Program Files\Tech-X
(Win64)\VSim-9.0\Contents\engine\bin\\computeTransitTimeFactor.py--
simulationName pillboxCavity --beginDump 0 --endDump 0 --beta 1 --axis 0 --
offsetx0 0 --offsetx1 0
Run Time: Thu Jun 14 11:33:44 2018
Run Directory: C:
\Users\ncrossette\Documents\txcorp\VSim9.0\simulations\exampleUpdates/pillbox
==================================


---
Mode file = pillboxCavity_E_Eigenmode_0.vsh5
Mode frequency, f = 1.00040e+03 MHz
Time Independent Accelerating Voltage, V0 = 2.79040e-01 V
Accelerating Voltage, Vacc = 1.77591e-01 V
Transit time factor, T=Vacc/V0 = 6.36434e-01
Computing using axis passing through x,y,z =   7.495e-02,   0.000e+00,
0.000e+00

==================================
Analysis completed successfully
```

Fig. 4.17: Computing the transit time factor for the eigenmode of interest.

Step 6: Computing the geometry factor

- Select *computeCavityG.py* and click “Open”.
- If you have selected the closed cavity, then enter “pillboxCavityAnalytical” for cavityGeometryName. Otherwise, enter “pillboxCavityWithTube” for cavityGeometryName.
- Select begin dump to 0 and end dump to 2.
- If you have selected the closed cavity, the geometry factor should be very close the the analytic value of 257.

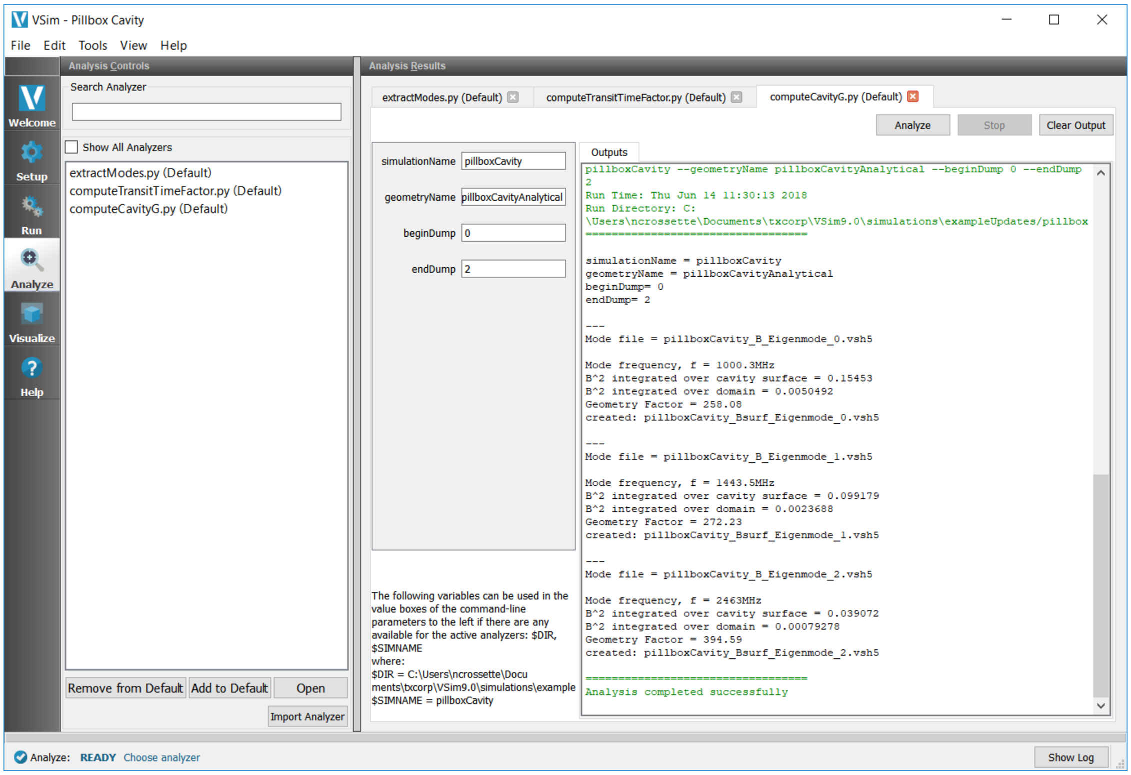
Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To see the projection of the magnetic field of the fundamental mode onto the cavity walls, do the following:

- Ensure that *Data Overview* is selected from the *Data View* drop down menu.
- Expand *Scalar Data*
- Expand *Bsurf*
- Check *BsurfMagnitude*
- Click the *Plane Controls* button at the bottom of the *Visualization Controls* pane on the left of the Composer window.


VSim - Pillbox Cavity
File Edit Tools View Help
Analysis Controls
Search Analyzer
Welcome
Setup
Run
Analyze
Visualize
Help
□ Show All Analyzers
extractModes.py (Default)
computeTransitTimeFactor.py (Default)
computeCavityG.py (Default)
Remove from Default
Add to Default
Open
Import Analyzer
Analysis Results
extractModes.py (Default)
computeTransitTimeFactor.py (Default)
computeCavityG.py (Default)
Analyze
Stop
Clear Output
Outputs
Analyze: READY Choose analyzer
Show Log

simulationName: pillboxCavity
geometryName: pillboxCavityAnalytical
beginDump: 0
endDump: 2

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analyzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\ncrossette\Documents\txcorp\VSim9.0\simulations\example
$SIMNAME = pillboxCavity

```
pillboxCavity --geometryName pillboxCavityAnalytical --beginDump 0 --endDump
2
Run Time: Thu Jun 14 11:30:13 2018
Run Directory: C:
\Users\ncrossette\Documents\txcorp\VSim9.0\simulations\exampleUpdates/pillbox
==================================

simulationName = pillboxCavity
geometryName = pillboxCavityAnalytical
beginDump= 0
endDump= 2

---
Mode file = pillboxCavity_B_Eigenmode_0.vsh5

Mode frequency, f = 1000.3MHz
B^2 integrated over cavity surface = 0.15453
B^2 integrated over domain = 0.0050492
Geometry Factor = 258.08
created: pillboxCavity_Bsurf_Eigenmode_0.vsh5

---
Mode file = pillboxCavity_B_Eigenmode_1.vsh5

Mode frequency, f = 1443.5MHz
B^2 integrated over cavity surface = 0.099179
B^2 integrated over domain = 0.0023688
Geometry Factor = 272.23
created: pillboxCavity_Bsurf_Eigenmode_1.vsh5

---
Mode file = pillboxCavity_B_Eigenmode_2.vsh5

Mode frequency, f = 2463MHz
B^2 integrated over cavity surface = 0.039072
B^2 integrated over domain = 0.00079278
Geometry Factor = 394.59
created: pillboxCavity_Bsurf_Eigenmode_2.vsh5

==================================
Analysis completed successfully
```

Fig. 4.18: Computing the geometry factor for the eigenmode of interest.

- Select *X* as the “Clip Plane Normal” and “.05” as the “Origin of Normal Vector” for “X”. Leave the “Origin of Normal Vector” for “Y” and “Z” as 0.
- Rotate the visualization by left clicking and dragging with your mouse.
- You should see a visualization of the magnitude of the magnetic field of the fundamental mode projected onto the wall of the cavity as in Fig. 4.19

To see a more quantitative visualization of the eigenmode fields, as shown in Fig. 4.20, do the following:

- Select *Field Analysis* as the *Data View*
- Select *E_x (Eigenmode)*
- Under the *Layout* drop-down menu, select *Side-by-side 2d/1d*

The Bessel function dependence of the x-component of the electric field will be clearly plotted on the right.

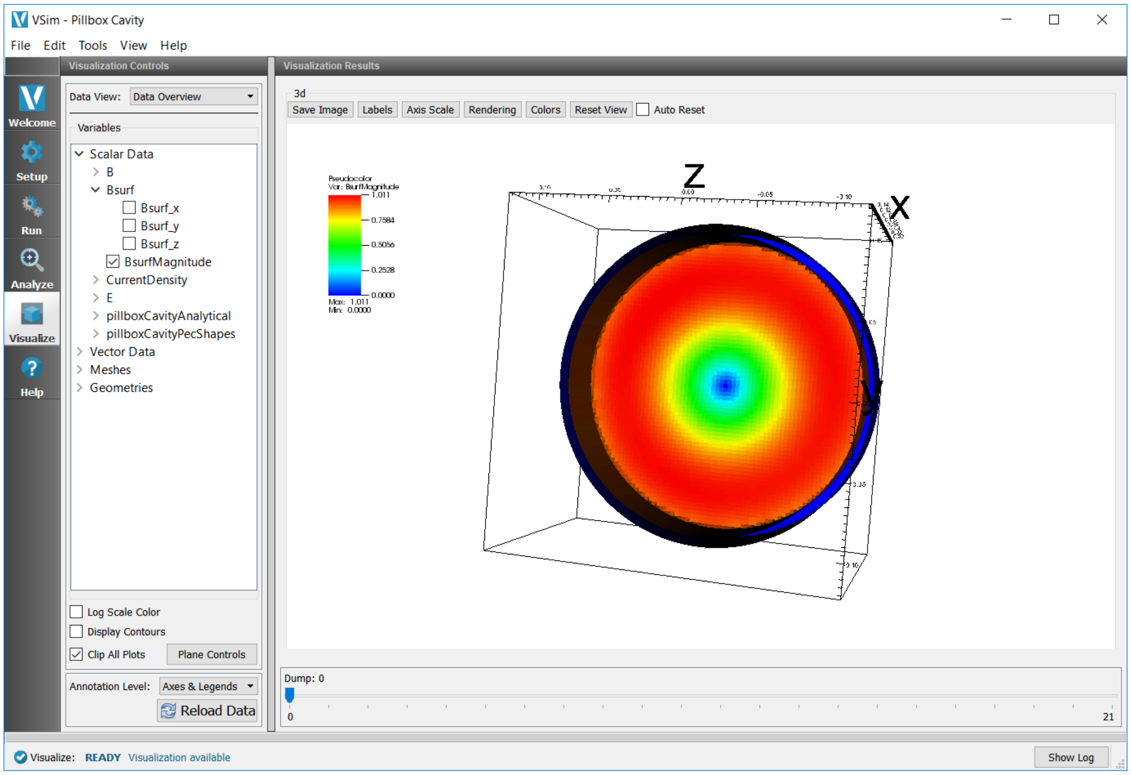
4.1.4 Rectangular Waveguide (rectangularWaveguide.sdf)

Keywords:

`Field Boundary Condition, rectangularWaveguide, Rectangular Waveguide`

Problem description

This example illustrates how to create a rectangular waveguide using the Rectangular Waveguide Field Boundary Condition and Constructive Solid Geometry.



VSim - Pillbox Cavity

Data View: Data Overview

Variables

- Scalar Data
  - B
  - Bsurf
    - □ Bsurf_x
    - □ Bsurf_y
    - □ Bsurf_z
    - ☑ BsurfMagnitude
  - CurrentDensity
  - E
  - pillboxCavityAnalytical
  - pillboxCavityPecShapes
- Vector Data
- Meshes
- Geometries

□ Log Scale Color

□ Display Contours

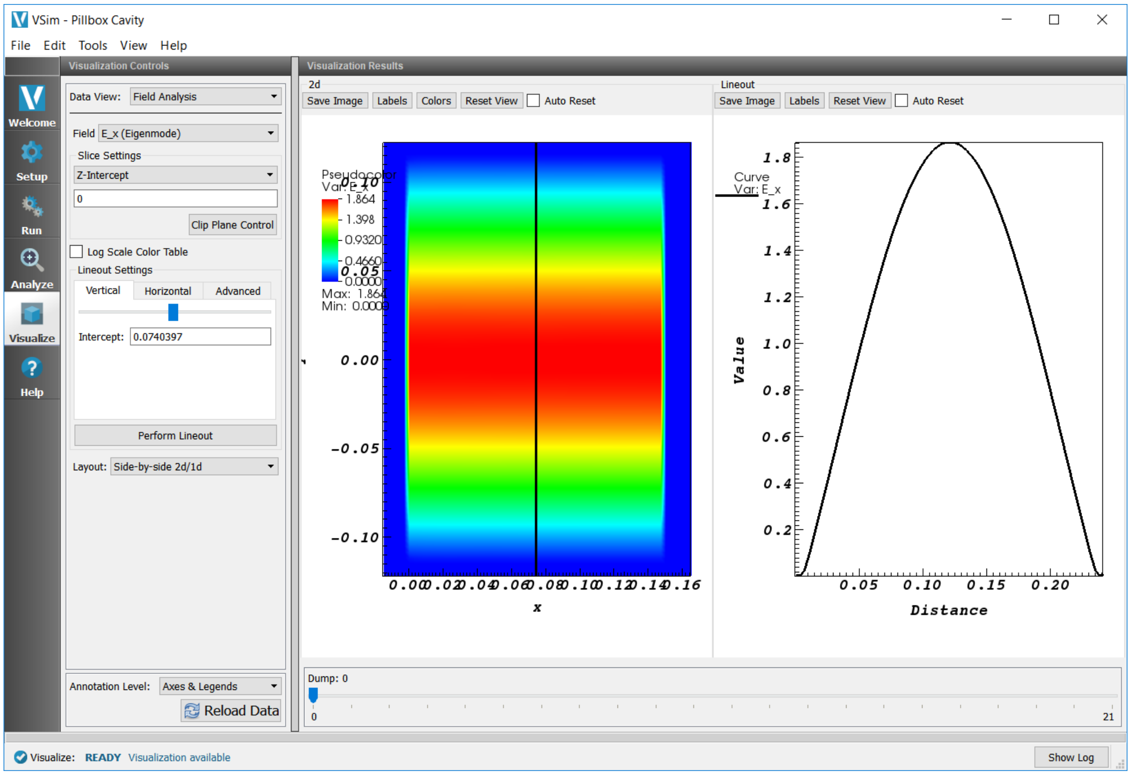
☑ Clip All Plots

Annotation Level: Axes & Legends

Dump: 0

Visualize: READY Visualization available

Fig. 4.19: The magnitude of the magnetic field on the wall of the cavity



VSim - Pillbox Cavity

Data View: Field Analysis

Field E_x (Eigenmode)

Slice Settings

Z-Intercept

0

□ Log Scale Color Table

Lineout Settings

Intercept: 0.0740397

Layout: Side-by-side 2d/1d

Annotation Level: Axes & Legends

Dump: 0

Visualize: READY Visualization available

Fig. 4.20: Axial component of the electric field in the $z = 0$ plane (left) and plot of the axial electric field along $z = 0, x = 0.07495$ (right).

Three waveguides are demonstrated in this example .

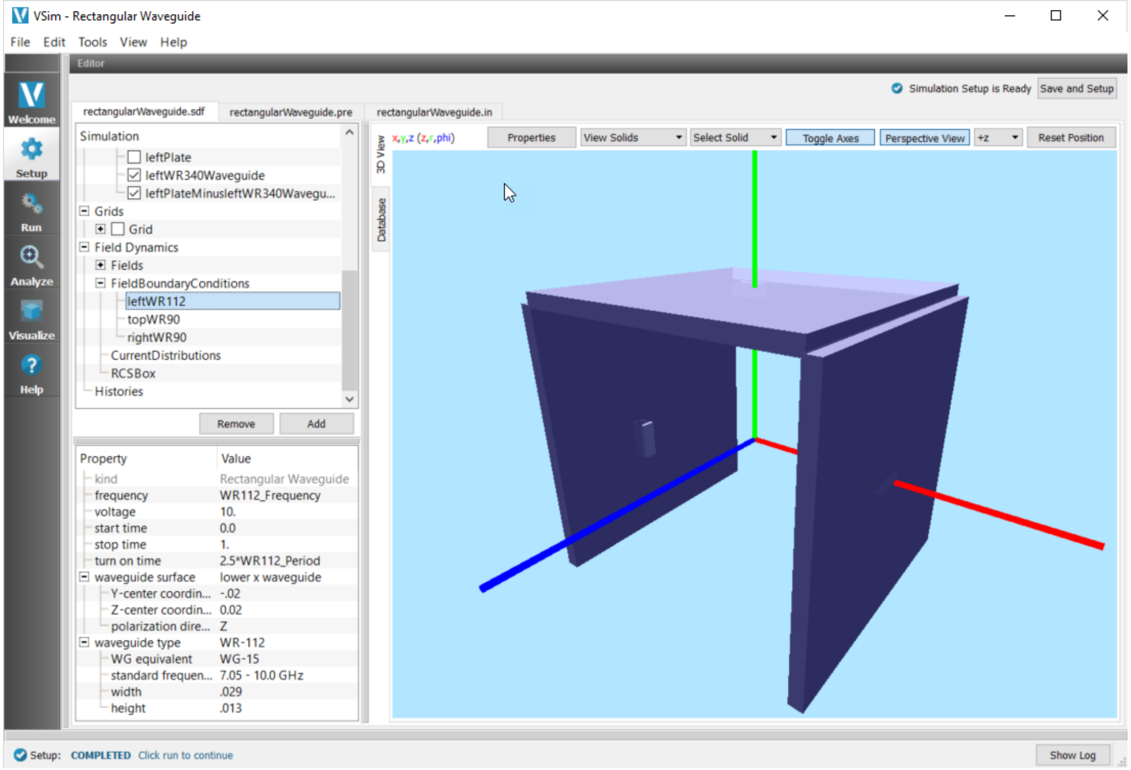
This simulation can be run with a VSimEM, VSimMD, VSimPD, or VSimPA license.

Opening the Simulation

The Rectangular Waveguide example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Cavities and Waveguides* option.
- Select *Rectangular Waveguide* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The Setup Window is shown in Fig. 4.21.



Property	Value
kind	Rectangular Waveguide
frequency	WR112_Frequency
voltage	10.
start time	0.0
stop time	1.
turn on time	2.5*WR112_Period
waveguide surface	lower x waveguide
Y-center coordin...	-.02
Z-center coordin...	0.02
polarization dire...	Z
waveguide type	WR-112
WG equivalent	WG-15
standard frequen...	7.05 - 10.0 GHz
width	.029
height	.013

Setup: COMPLETED Click run to continue

Fig. 4.21: Setup Window for the Rectangular Waveguide example.

Simulation Properties

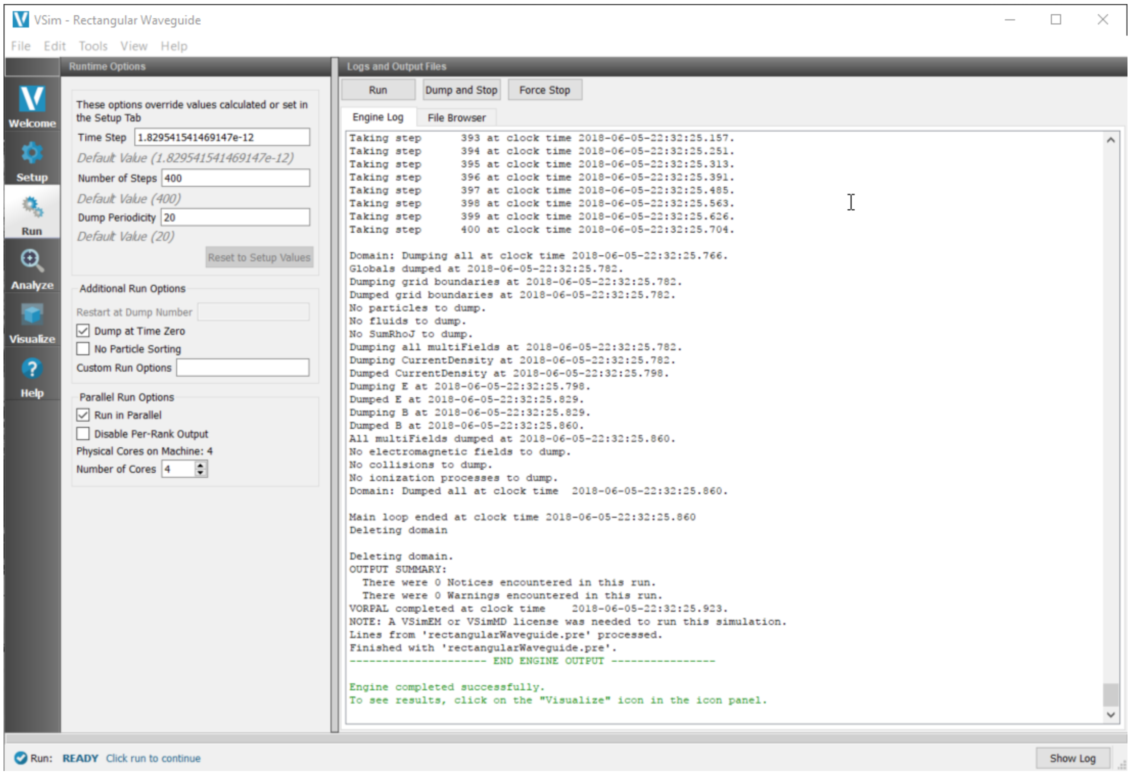
This simulation demonstrates how to create a rectangular waveguide. There are three rectangular waveguides in this simulation. Each is constructed by creating a physical waveguide on the simulation boundary, and defining the wave that is carried into the simulation. First a metal plate from a box primitive must be placed on the simulation boundary. It is important that this plate extend from at least one cell outside of the simulation boundary to at least one cell inside of the simulation. Next a box primitive corresponding to the size and orientation of the actual waveguide must be created. This is then subtracted from the previously created metal plate. It is important to note here that the polarization parameter will always be parallel to the width. The wave carried in this waveguide is then created by adding a FieldBoundaryCondtion of Rectangular Waveguide. The waveguide surface must be specified to match the intended simulation boundary and on the right location to match the physically constructed waveguide.

Several standard waveguide sizes are available, or User-Defined may be selected to specify a custom size. If no “Turn On Time” is specified, it will be set to a time of 2.5 periods of the carried signal, and a warning will be provided after running the simulation.

Running the Simulation

Once finished with the problem setup, continue as follows:

- Proceed to the Run Window by pressing the *Run* button in the left column of buttons.
- Use the default values for the stepping.
- Choose parallel computing options if desired
- To run the file, click on the *Run* button in the upper left corner of the right pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in the window below.



These options override values calculated or set in the Setup Tab

Time Step 1.829541541469147e-12
Default Value (1.829541541469147e-12)
Number of Steps 400
Default Value (400)
Dump Periodicity 20
Default Value (20)

Additional Run Options
Restart at Dump Number
☑ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options
☑ Run in Parallel
☐ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
Taking step      393 at clock time 2018-06-05-22:32:25.157.
Taking step      394 at clock time 2018-06-05-22:32:25.251.
Taking step      395 at clock time 2018-06-05-22:32:25.313.
Taking step      396 at clock time 2018-06-05-22:32:25.391.
Taking step      397 at clock time 2018-06-05-22:32:25.485.
Taking step      398 at clock time 2018-06-05-22:32:25.563.
Taking step      399 at clock time 2018-06-05-22:32:25.626.
Taking step      400 at clock time 2018-06-05-22:32:25.704.

Domain: Dumping all at clock time 2018-06-05-22:32:25.766.
Globals dumped at 2018-06-05-22:32:25.782.
Dumping grid boundaries at 2018-06-05-22:32:25.782.
Dumped grid boundaries at 2018-06-05-22:32:25.782.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-05-22:32:25.782.
Dumping CurrentDensity at 2018-06-05-22:32:25.782.
Dumped CurrentDensity at 2018-06-05-22:32:25.798.
Dumping E at 2018-06-05-22:32:25.798.
Dumped E at 2018-06-05-22:32:25.829.
Dumping B at 2018-06-05-22:32:25.829.
Dumped B at 2018-06-05-22:32:25.860.
All multiFields dumped at 2018-06-05-22:32:25.860.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-05-22:32:25.860.

Main loop ended at clock time 2018-06-05-22:32:25.860
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-05-22:32:25.923.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'rectangularWaveguide.pre' processed.
Finished with 'rectangularWaveguide.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

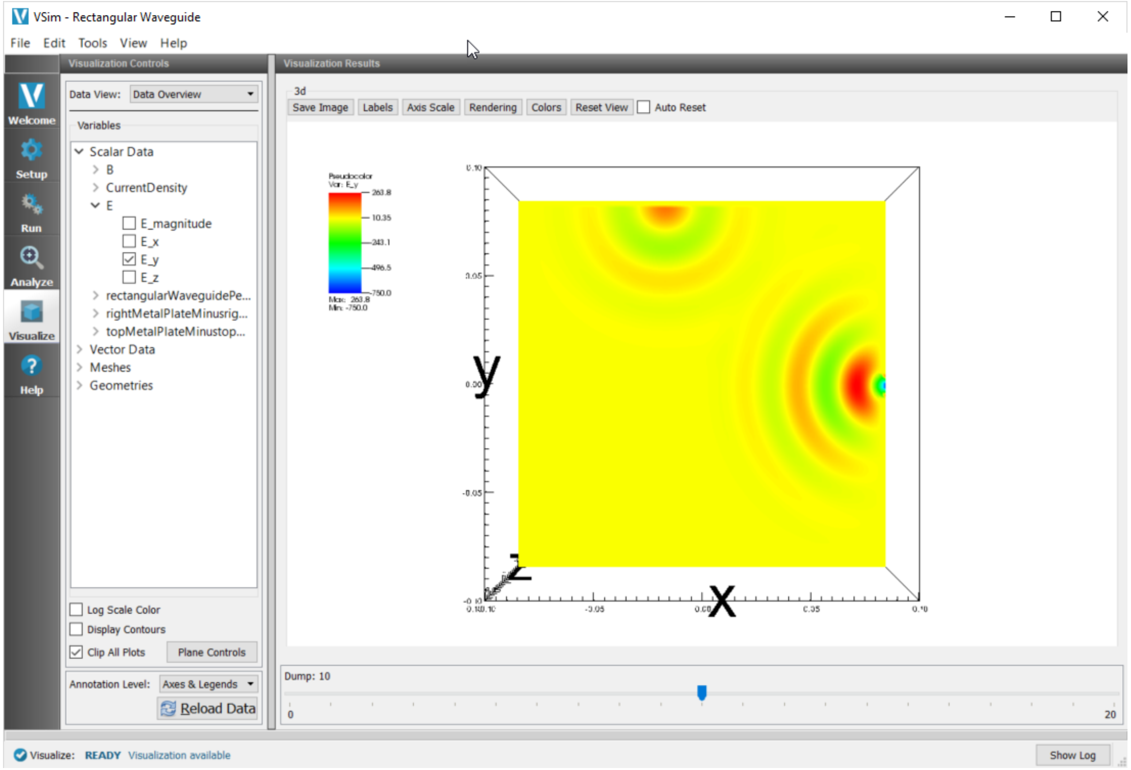
Run: READY Click run to continue

Fig. 4.22: The Run Window at the end of execution.

Visualizing the Results

After a succesful run, go to the Visualize Window by pressing *Visualize* in the left column.

Expand *Scalar Data*, *E*, and select E_y. To slice inside the field, select *Clip All Plots* in the lower left hand corner. Now step through time using the Dump slider on the bottom of the right pane. This is shown below.



Data View: Data Overview

Annotation Level: Axes & Legends

Dump: 10

Visualize: READY Visualization available

Fig. 4.23: The E_y field propagating out of the two waveguides centerd on the z axis.
The effects of the third waveguide can be viewed by adjusting the “Origin of Normal Vector” parametr under the Plane Controls button.

Further Experiments

Waveguides can be added or subtracted to this simulation.

4.1.5 S-Matrix of Box Cavity (sMatrix.sdf)

Keywords:

`electromagnetics, sMatrix`

Problem description

A common measurement made on a 2-port RF device is reflection and transmission of an RF signal, for either a single frequency, or for a range of frequencies. This measurement results in the Scattering-Matrix, or S-Matrix, whose elements S11 and S21 are the reflected and transmitted signal for unit input at Port 1. VSim provides the capability to simulate these S-Matrix parameters for arbitrarily complex devices connected to waveguides propagating TE, TM, and TEM modes. To demonstrate this capability, we show in this example how to measure S11 and S21 in a dual-mode cavity filter, connected to a WR-90 waveguide, with the narrow-band band-pass tuned to pass frequencies between 9.95 and 10.05 GHz.

The Dual Mode Cavity Filter operates by coupling the TE01 waveguide mode into the two nearly degenerate TE102 and TE201 modes of the cavity, since the length of the cavity is very close to its width. The differences in these values, along with the symmetry breaking along the waveguide axis, determine the frequency separation of the two modes. This separation is what gives the filter finite-bandwidth since frequencies between these modes are passed, and frequencies above or below the modes are rejected. A pole in the transmitted signal just below the band contributes to sharpness of the band's lower edge, but this pole moves easily to the upper frequency edge with small adjustments to the cavity dimension parameters, and the user is encouraged to experiment in finding optimal placement of this pole. Some relevant parameters are shown in Fig. 4.24.

This simulation can be performed with a VSimMD or VSimEM license.

Opening the Simulation

The Scattering Matrix example is accessed from within VSimComposer by the following actions:

- Select the *New → From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Cavities and Waveguides* option.
- Select "S-Matrix of Box Cavity" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is now shown with all the implemented physics and geometries, if applicable. See Fig. 4.25.

Simulation Properties

The simulation geometry consists of a standard WR-90 rectangular waveguide with the filter cavity (also referred to as the Device-Under-Test (DUT) in this writeup) in the center. A planar antenna in the waveguide, near the DUT, launches the incident wave while allowing reflected signals to pass through into the waveguide behind it. The waveguide ends are terminated in gradual absorbing layers with negligible reflection, and the reflected and transmitted signals are measured just in front of these absorbers.

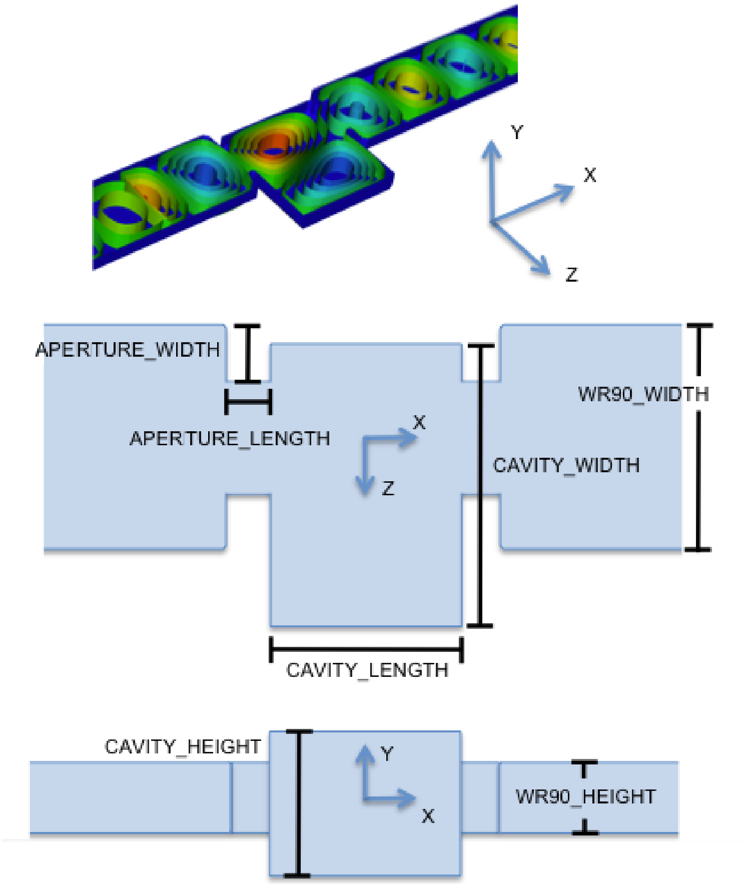
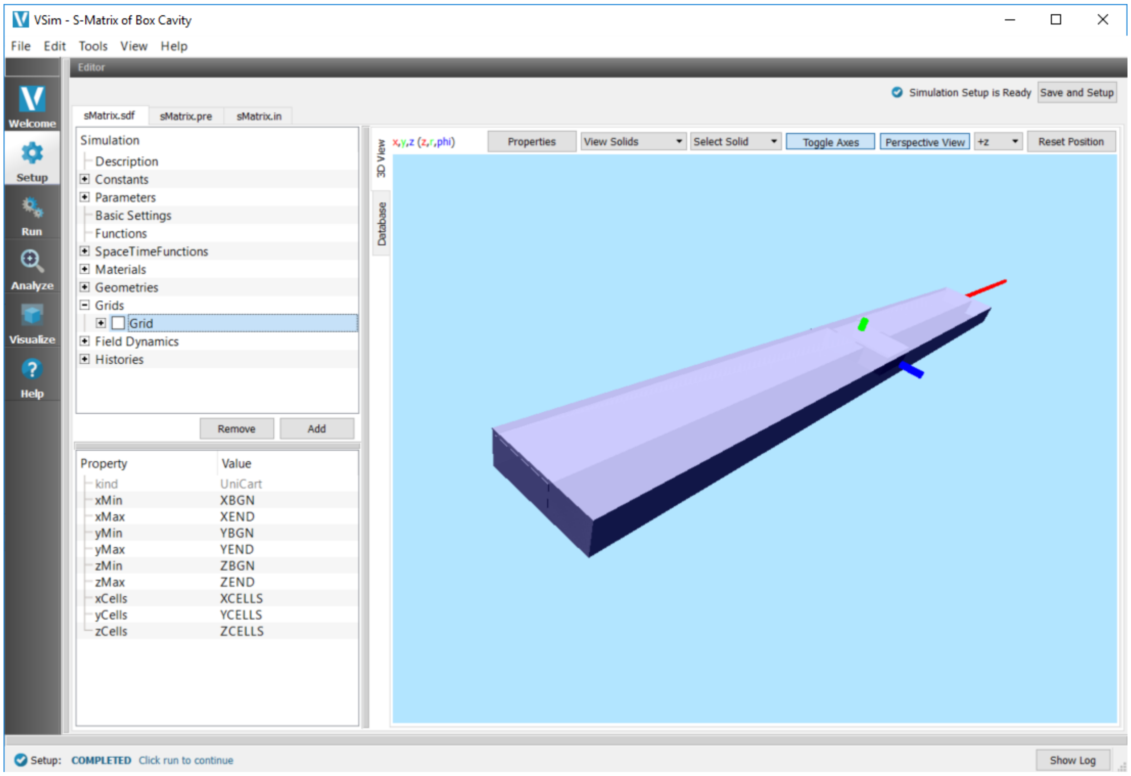


Fig. 4.24: Some relevant parameters for the S-Matrix Box Cavity example.



Simulation
- Description
- Constants
- Parameters
- Basic Settings
- Functions
- SpaceTimeFunctions
- Materials
- Geometries
- Grids
  - □ Grid
- Field Dynamics
- Histories

Property	Value
kind	UniCart
xMin	XBGN
xMax	XEND
yMin	YBGN
yMax	YEND
zMin	ZBGN
zMax	ZEND
xCells	XCELLS
yCells	YCELLS
zCells	ZCELLS

Setup: COMPLETED Click run to continue

Fig. 4.25: Setup Window for the Scattering Matrix example.

A main feature of this input file is that the waveguide description and the DUT description are short compact sections of input, that are easily substituted. Thus this example is effectively a template for an S-Matrix simulation of any device. The time histories of voltage signals used to measure S11 and S21 are also built in and automated for easy substitution. Furthermore, these signals are easily turned into S11 and S21 frequency variation curves using the standard “FFT” capabilities in VSimComposer, or if single frequency, then the S11 and S21 values are just the amplitudes of the signals.

The x axis is aligned with the length of the waveguide, and the input and output waveguide are specified with 10 variables that provide YZ bounding coordinates of the waveguide, YZ coordinates for the voltage measurement, a boolean specifying whether the excited mode is TM or not, and the waveguide cutoff frequency for the mode. Three additional analytic functions provide the interior of the waveguide’s YZ cross-section and the Y and Z components of the mode’s electric field pattern. For this simulation, these parameters all derive from WR90 dimensions, with the waveguide cross-section centered at the YZ origin. The excited mode is the standard lowest mode, TE01, and in particular note that for this mode, the Ez component of the field is zero.

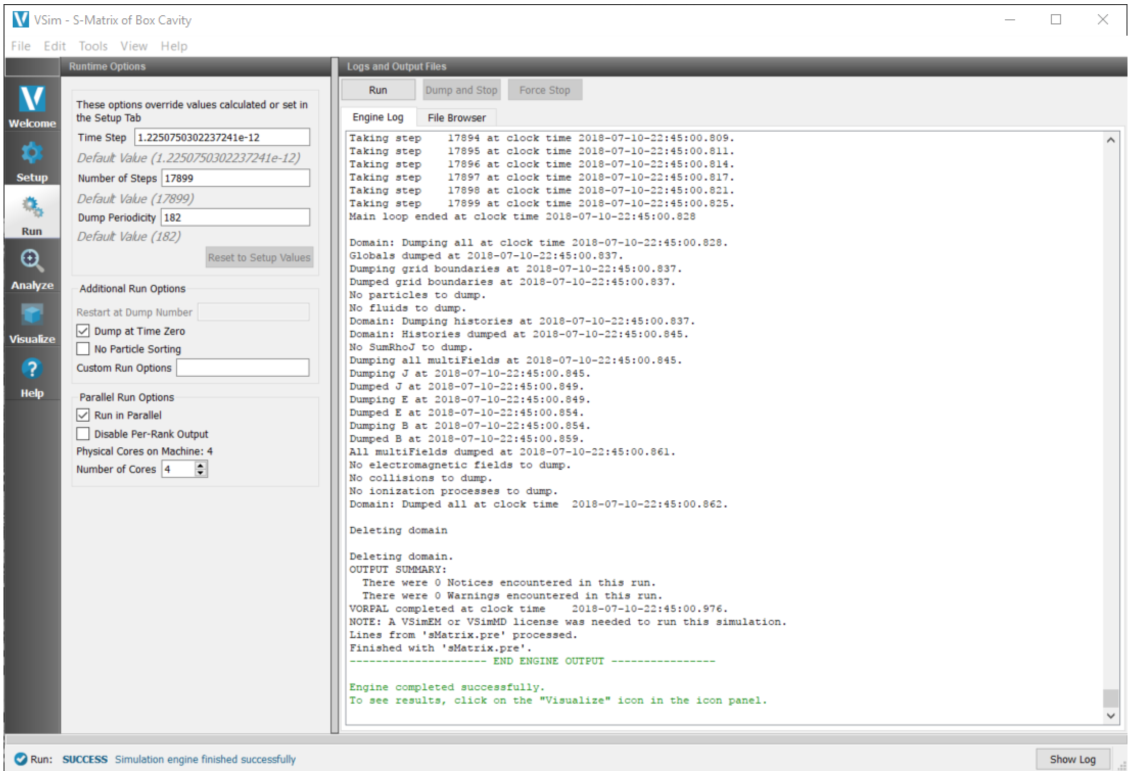
The Device-Under-Test, in this case the dual mode cavity filter, is specified by 6 variables that provide the XYZ bounding coordinates and one macro that constructs the DUT. These all derive from the dimension parameters for the dual mode cavity filter, which are height, width, and length of the rectangular cavity, and the size of the apertures connected to the waveguide.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the *Run* button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner. of the right pane. You will see the output of the

run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in the window below.



These options override values calculated or set in the Setup Tab

Time Step 1.2250750302237241e-12

Default Value (1.2250750302237241e-12)

Number of Steps 17899

Default Value (17899)

Dump Periodicity 182

Default Value (182)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Taking step    17894 at clock time 2018-07-10-22:45:00.809.
Taking step    17895 at clock time 2018-07-10-22:45:00.811.
Taking step    17896 at clock time 2018-07-10-22:45:00.814.
Taking step    17897 at clock time 2018-07-10-22:45:00.817.
Taking step    17898 at clock time 2018-07-10-22:45:00.821.
Taking step    17899 at clock time 2018-07-10-22:45:00.825.
Main loop ended at clock time 2018-07-10-22:45:00.828

Domain: Dumping all at clock time 2018-07-10-22:45:00.828.
Globals dumped at 2018-07-10-22:45:00.837.
Dumping grid boundaries at 2018-07-10-22:45:00.837.
Dumped grid boundaries at 2018-07-10-22:45:00.837.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-07-10-22:45:00.837.
Domain: Histories dumped at 2018-07-10-22:45:00.845.
No SumRhoJ to dump.
Dumping all multiFields at 2018-07-10-22:45:00.845.
Dumping J at 2018-07-10-22:45:00.845.
Dumped J at 2018-07-10-22:45:00.849.
Dumping E at 2018-07-10-22:45:00.849.
Dumped E at 2018-07-10-22:45:00.854.
Dumping B at 2018-07-10-22:45:00.854.
Dumped B at 2018-07-10-22:45:00.859.
All multiFields dumped at 2018-07-10-22:45:00.861.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-10-22:45:00.862.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-07-10-22:45:00.976.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'sMatrix.pre' processed.
Finished with 'sMatrix.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.26: The Run Window at the end of execution.

This example is more sophisticated than some of the others, in that successful determination of S-Matrix parameters is not the result of a single run, but rather a result of a procedure involving several runs. This includes at least one Calibration Run, and at least one Data Run to determine S11 and S21, and then a repetition with input switched to Port 2, in order to determine S22 and S12. All runs use the same input file and it is a set of control variables which determine the action of a particular run. Below we discuss in detail some of the features of this example.

Frequency Band vs. Single Frequency

The user may choose whether to compute a single-frequency value of the S-Matrix parameters, or to compute the variation of the parameters as a function of frequency across a user specified frequency band. The variable, FREQCENTER, specifies either the single frequency or the center frequency of the band. The variable, FREQBANDWIDTH, provides the bandwidth or is set to 0 if a single-frequency simulation is desired.

With a single frequency simulation, the variable, NUMBEROFCYCLESTODRIVE, should be large enough to ensure that the S11 and S21 histories reach a steady amplitude. The *History* data view can be used to obtain the S-Matrix value, which is just the amplitude of the signal.

With a finite frequency band, the same variable, NUMBEROFCYCLESTODRIVE, can be adjusted upward to increase the detail and sharpness of the S-Matrix variation with frequency. The variable, NUMBEROFCYCLESTOCOAST, may also need to be adjusted upward if the DUT contains internal mode oscillation of large Q (quality) factor. This variable needs to be large enough so that the signal histories have decreased to a negligible value (10^{-4}, relative to maximum) by the end of the simulation. The *FFT* button in the *History* data view is then used to give the S-Matrix variation with frequency, with the plot’s Y-axis units being dB. Be aware that it is usually necessary to zoom in significantly on this plot in order to see the frequency band of interest.

Finally, in both these cases, only the amplitude of the complex-valued S-Matrix parameters can be obtained with VSimComposer. More sophisticated post-processing (not covered in this example) is needed in order to get the phase information.

Calibration Run

The Calibration Run is done first, and the user must ensure that in the geometries, only the material of the object *metalMinuscalibrationWaveguide* is set to PEC, i.e., ensuring the material of the object *myWaveguideAndDUT* is set to empty. In the Calibration Run, the DUT is automatically omitted and replaced with a continuation of the waveguide, so that this is a near trivial simulation of a straight length of waveguide that should have effectively 100% transmission. The calibration run serves two very important purposes:

- To ensure that there is negligible (below 1% amplitude, -40 dB) reflected voltage (S11). If the reflection is too high it indicates that either the absorbing boundaries are not working well enough, or that the waveguide's "modeProfile" description is not accurate enough, and/or that there is not enough grid resolution.
- To adjust the variable DRIVENORMALIZATION, which runs in proportion to observed transmitted voltage (S21), so that the next time the calibration run is done, the transmitted voltage (S21) will be exactly unit amplitude (single frequency) or zero dB (across frequency band). For example, if the first Calibration Run shows an amplitude of 0.667 for S21, change the variable DRIVENORMALIZATION to 1.5 times its present value for the next Calibration Run, since 1/0.667 = 1.5.

Changing center frequency, or any waveguide parameter, or even the nominal cell size, will require re-calibration. If not sure, always recalibrate, when changing a parameter.

Data Run

Once the Calibration Run is successful at achieving unit transmission with negligible reflection, the Data Run is then done. The user should ensure that only the material of the object *myWaveguideAndDUT* is set to PEC, i.e., ensuring the material of the object *metalMinuscalibrationWaveguide* is set to empty.

Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The S-Matrix results are then read from the History data view in the Visualize Window. An example run, for a frequency band simulation from 8 to 12 GHz, is shown below in Fig. 4.27. The figure shows the *History* data view with the FFT's of the "S11_Voltage" history and the "S21_Voltage" history.

Further Experiments

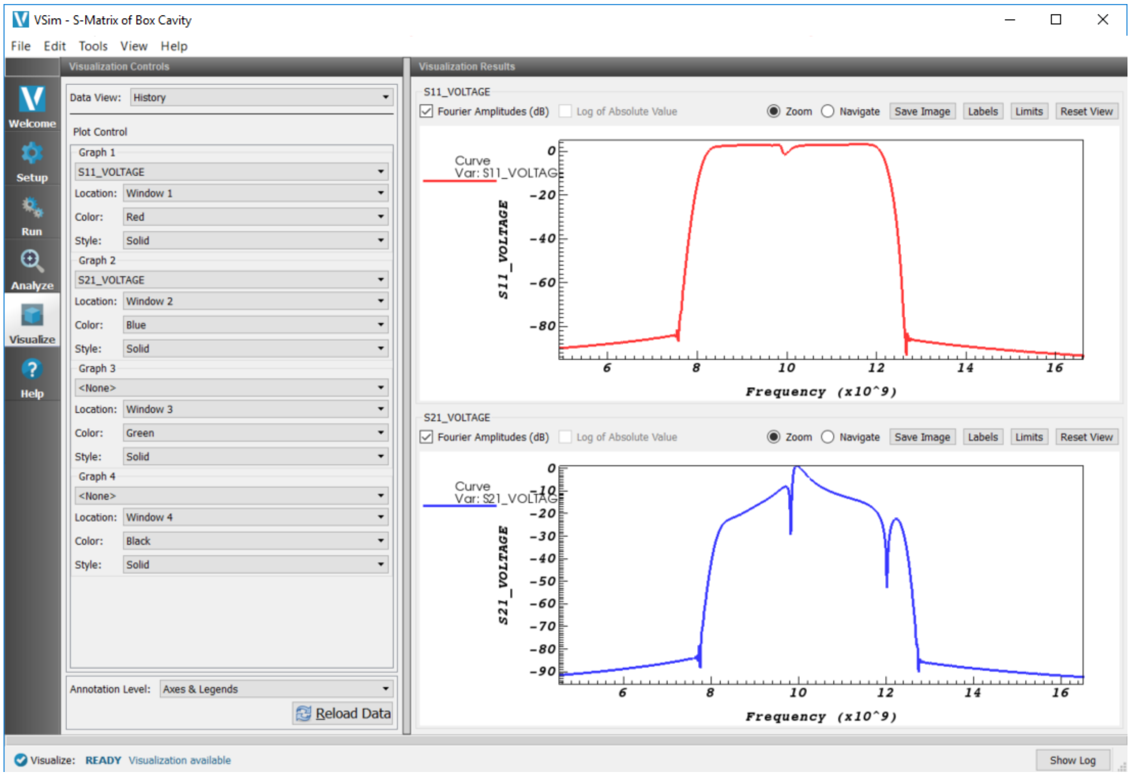
Experiment with finding optimal placement of the pole in the transmitted signal.

4.2 Cavities and Waveguides (text-based setup)

4.2.1 Coaxial Cylinder (coaxT.pre)

Keywords:

`coaxT, coaxial geometry, cylinder, current pulse, rlc circuit, step potential`



Data View: History

Plot Control

Graph 1
S11_VOLTAGE
Location: Window 1
Color: Red
Style: Solid

Graph 2
S21_VOLTAGE
Location: Window 2
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 3
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

S11_VOLTAGE

S21_VOLTAGE

Visualize: READY Visualization available

Fig. 4.27: Fourier transforms of the histories S11_Voltage and S21_Voltage as a function of frequency (in GHz).

Problem description

This example probes the electromagnetic properties of a semi-infinite coaxial cylinder. One end of the cylinder lies in the simulation space and its dimensions are large–well beyond the size of a coaxial cable. It’s outer radius is 8 cm, the inner radius is 2 cm, and the section considered is 20 cm long. The inner cylinder is shorter than the outer cylinder and there is an electron absorbing cap on the end of the outer cylinder. When the simulation initiates, a single EM pulse is launched into the open, continuous end of the geometry and propagates to the capped tip. Electrons are ejected from the tip of the inner cylinder when the pulse reaches it. The pulse has a period of ~0.7 ns and the simulation runs for 5 periods.

This computational model is equivalent to applying a step-potential at one end of the cylinder (location of the other, unseen end of the cylinder is arbitrary). The step-potential propagates at the speed of light (the medium is a vacuum) until it reaches the tip of the inner cylinder. There, an attenuating series of oscillations occurs. Gradually the tip potential stabilizes at the applied potential.

This simulation can be performed with a VSimMD license.

Opening the Simulation

The coaxT example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Introductory Examples (Text-based setup)* option.
- Press the arrow button to the left of *Introductory*.

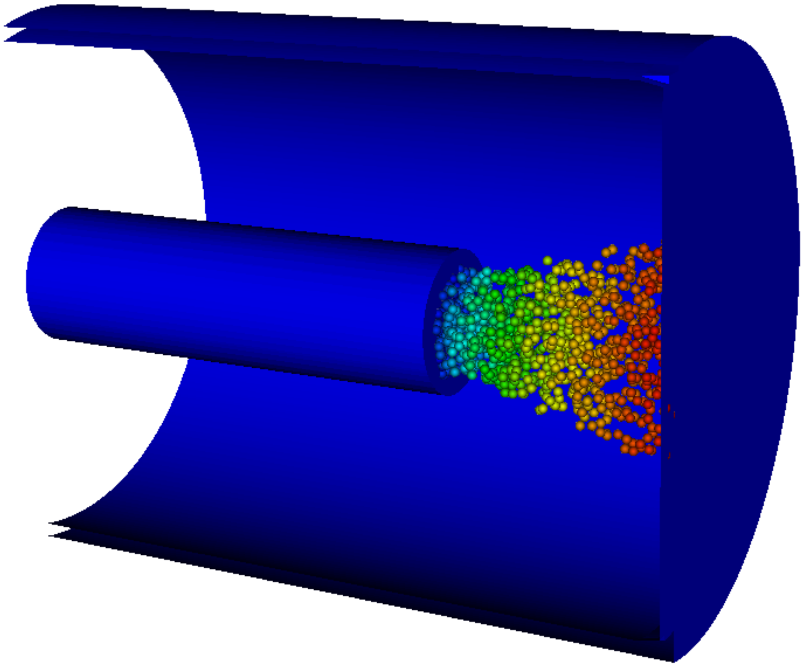


Fig. 4.28: The electrons are emitted from the tip of the inner cylinder after the field reaches it.

- Select *Coaxial Cylinder* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 4.29.

Input File Features

Primary Parameters

`EXTERNAL_RADIUS_EXTERIOR`: Radius of the coaxial cylinder's outer wall

`EXTERNAL_CYLINDER_LENGTH`: Length of the outer cylinder, determines simulation domain size

`EXTERNAL_CYLINDER_WIDTH`: Thickness of the outer cylinder wall

`NGRID`: Number of cells spanning the length of the simulation space (the axial direction)

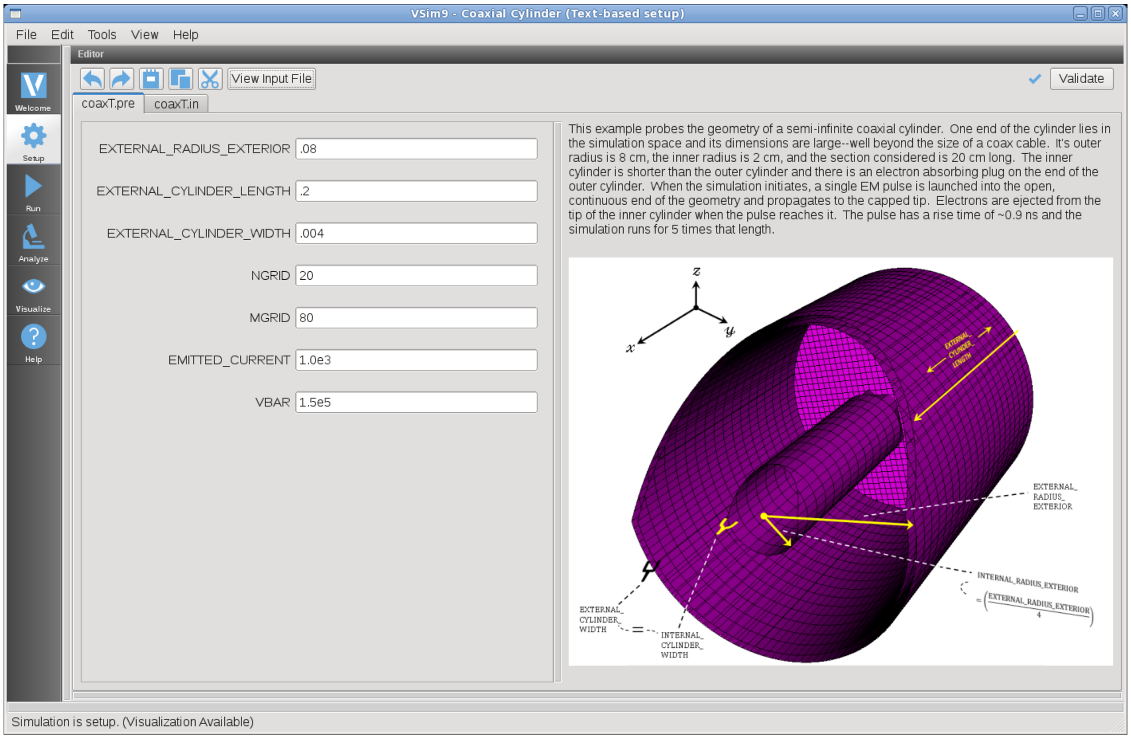
`MGRID`: Number of cells spanning the dimensions of the simulation space transverse to the cylinder axis

`EMITTED_CURRENT`: The current emitted from the tip of the inner cylinder

`VBAR`: The average velocity of electrons emitted from the inner cylinder

Secondary and Derived Variables

Can be found in the input file view.



EXTERNAL_RADIUS_EXTERIOR .08

EXTERNAL_CYLINDER_LENGTH .2

EXTERNAL_CYLINDER_WIDTH .004

NGRID 20

MGRID 80

EMITTED_CURRENT 1.0e3

VBAR 1.5e5

This example probes the geometry of a semi-infinite coaxial cylinder. One end of the cylinder lies in the simulation space and its dimensions are large--well beyond the size of a coax cable. It's outer radius is 8 cm, the inner radius is 2 cm, and the section considered is 20 cm long. The inner cylinder is shorter than the outer cylinder and there is an electron absorbing plug on the end of the outer cylinder. When the simulation initiates, a single EM pulse is launched into the open, continuous end of the geometry and propagates to the capped tip. Electrons are ejected from the tip of the inner cylinder when the pulse reaches it. The pulse has a rise time of ~0.9 ns and the simulation runs for 5 times that length.

Simulation is setup. (Visualization Available)

Fig. 4.29: Setup Window for the Coaxial Cylinder example.

INTERNAL_RADIUS_EXTERIOR: Radius of the inner cylinder's outer wall; equal to 0.25*EXTERNAL_RADIUS_EXTERIOR

INTERNAL_CYLINDER_LENGTH: Length of the inner cylinder; equal to 0.7*EXTERNAL_CYLINDER_LENGTH or EXTERNAL_CYLINDER_LENGTH - EXTERNAL_RADIUS_EXTERIOR, whichever quantity is longer.

INTERNAL_CYLINDER_WIDTH: Thickness of the inner cylinder wall

OMEGA: Equivalent frequency of incoming EM pulse. Not an actual physical quantity since the pulse is non-periodic. but it does help determine the rise time of the pulse, called PERIOD below. The rise-time depends on the cylinder radius and is equal to $2.405c$/EXTERNAL_RADIUS_EXTERIOR $= 9.01 \times 10^9$rad/sec $= 14.2$GHz.

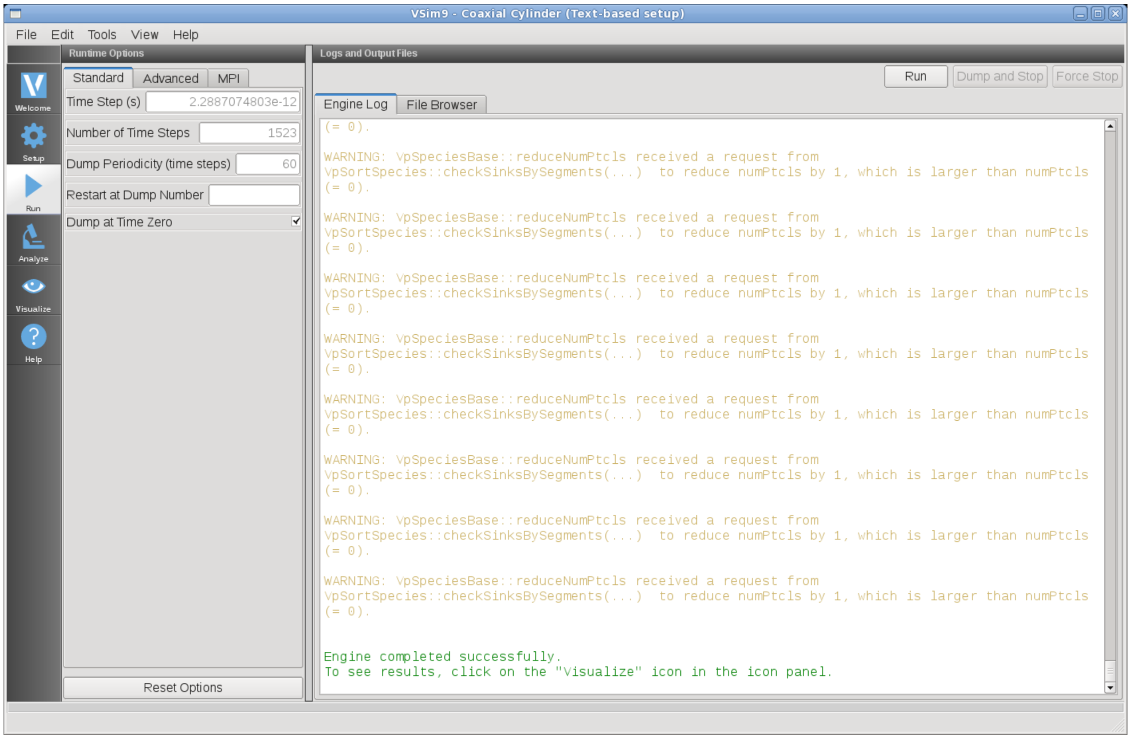
PERIOD: The rise-time of the pulse. Equal to $2\pi/$OMEGA. Not used for any physics, PERIOD determines the simulation run time.

TIMESTEPS: Determines length of simulation. By default the simulation will run for 5*PERIOD.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner. of the window. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 4.30.



Time Step (s) 2.2887074803e-12

Number of Time Steps 1523

Dump Periodicity (time steps) 60

Restart at Dump Number

Dump at Time Zero ☑

```
(= 0).

WARNING: VpSpeciesBase::reduceNumPtcls received a request from
VpSortSpecies::checkSinksBySegments(...)  to reduce numPtcls by 1, which is larger than numPtcls
(= 0).

WARNING: VpSpeciesBase::reduceNumPtcls received a request from
VpSortSpecies::checkSinksBySegments(...)  to reduce numPtcls by 1, which is larger than numPtcls
(= 0).

WARNING: VpSpeciesBase::reduceNumPtcls received a request from
VpSortSpecies::checkSinksBySegments(...)  to reduce numPtcls by 1, which is larger than numPtcls
(= 0).

WARNING: VpSpeciesBase::reduceNumPtcls received a request from
VpSortSpecies::checkSinksBySegments(...)  to reduce numPtcls by 1, which is larger than numPtcls
(= 0).

WARNING: VpSpeciesBase::reduceNumPtcls received a request from
VpSortSpecies::checkSinksBySegments(...)  to reduce numPtcls by 1, which is larger than numPtcls
(= 0).

WARNING: VpSpeciesBase::reduceNumPtcls received a request from
VpSortSpecies::checkSinksBySegments(...)  to reduce numPtcls by 1, which is larger than numPtcls
(= 0).

WARNING: VpSpeciesBase::reduceNumPtcls received a request from
VpSortSpecies::checkSinksBySegments(...)  to reduce numPtcls by 1, which is larger than numPtcls
(= 0).

WARNING: VpSpeciesBase::reduceNumPtcls received a request from
VpSortSpecies::checkSinksBySegments(...)  to reduce numPtcls by 1, which is larger than numPtcls
(= 0).


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Fig. 4.30: The Run Window at the end of execution.

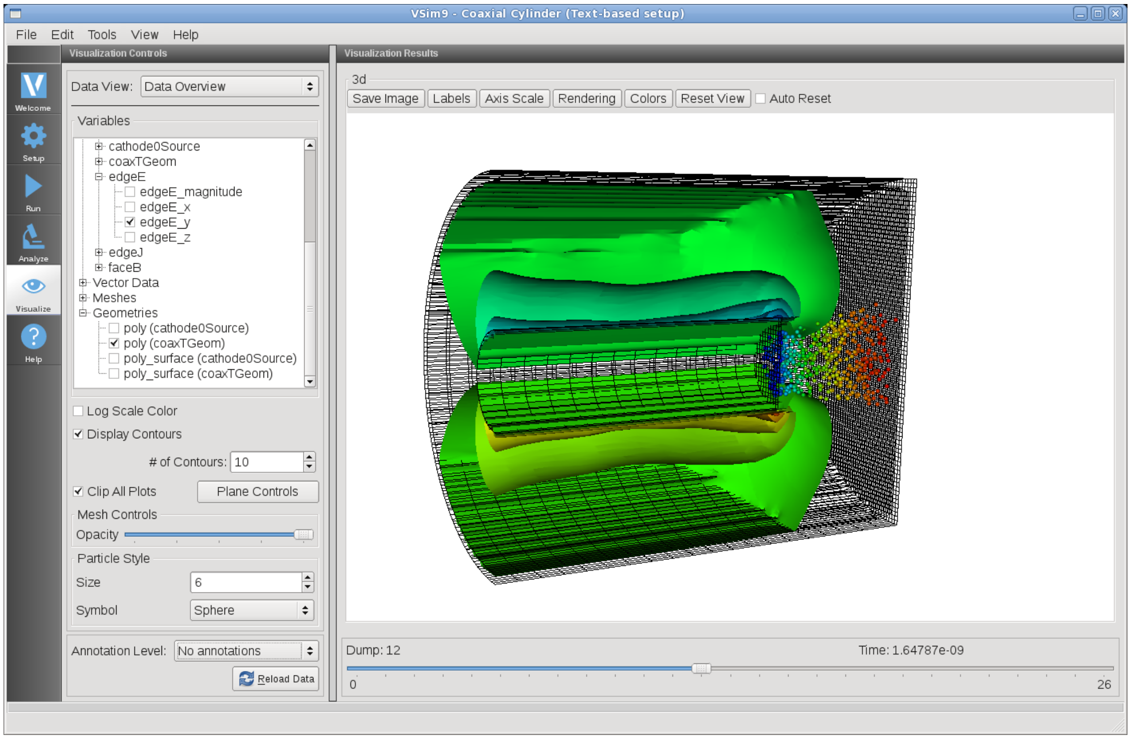
Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To create the image seen in Fig. 4.31, proceed as follows:

- Expand *Scalar Data*
- Expand *edgeE*
- Select *edgeE_y*
- Select *Clip All Plots*
- Click the *Plane Controls* button and set the normal to the Z-direction
- Select *Display Contours* and set the # of contours to 10
- Set the particle size to 6 and the symbol to *Sphere*
- Expand *Particle Data*
- Expand *electrons*
- Select *electrons_ux*
- Expand *Geometries*
- Select *poly (coaxTGeom)*
- Move the dump slider forward in time
- Click and drag to rotate



Data View: Data Overview

☑ edgeE_y

☑ poly (coaxTGeom)

□ Log Scale Color

☑ Display Contours

# of Contours: 10

☑ Clip All Plots

Size: 6

Symbol: Sphere

Annotation Level: No annotations

□ Auto Reset

Dump: 12

Time: 1.64787e-09

Fig. 4.31: Visualization of the coaxial cylinder as a color contour plot.

To obtain a clearer picture of what is happening at the cylinder tip, switch the Data View to *History*. One dimensional plots of the number of electrons (called numberOfMacroParticles), the electric potential, phi, and the current emitted and absorbed should come up automatically.

You can set Graph 2 to Location "Window 1" as in Fig. 4.32.

The potential is measured at the tip of the cylinder with the interior of the inner cylinder serving as a reference point. The potential plotted has a lot of noise on it resulting from the free electrons. It may be insightful to run the simulation once without electrons so you can see the ringing on the waveform of phi, which is not unlike the output of an oscilloscope hooked up to a coaxial cable. Electrons can be suppressed by setting the EMITTED_CURRENT parameter to 0 during setup.

The coaxial cylinder acts like an RLC circuit: the inner conductor serves as an inductor, the gap between the tip and the absorbing cap is a capacitor, and the spacing between the inner and outer cylinders constitutes a resistor. By default, the rise-time of the pulse is near the resonance period, and this makes it a good driver of the circuit.

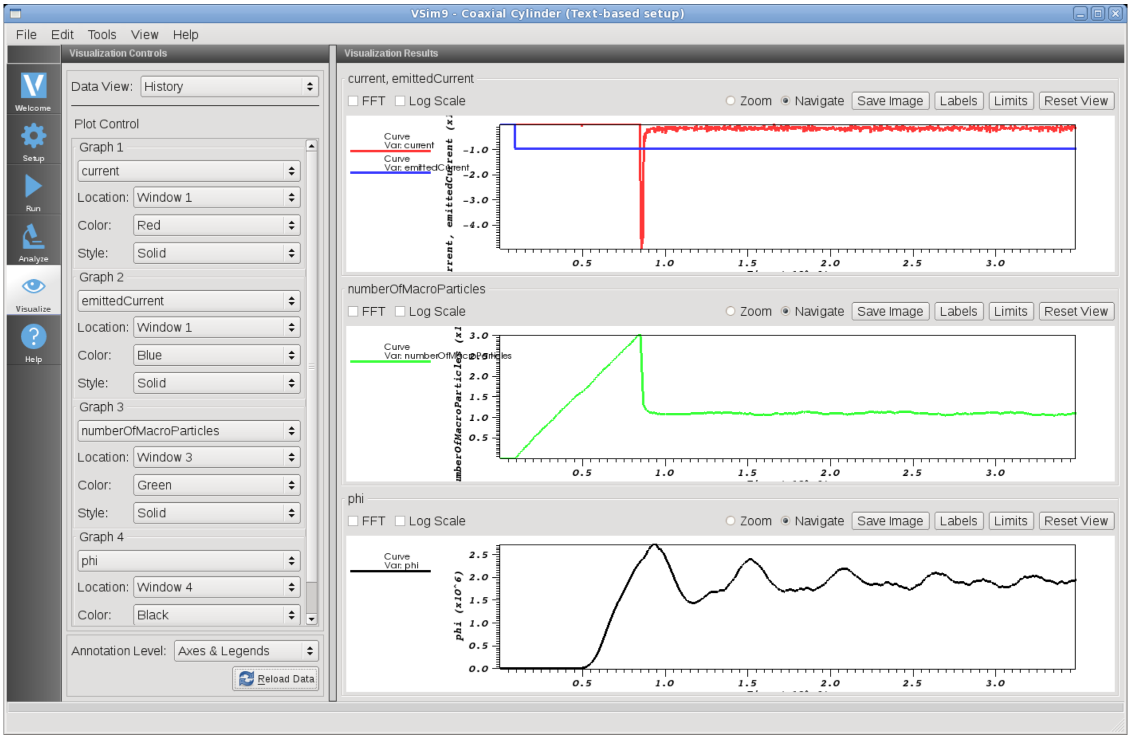
Further Experiments

Try experimenting with different geometry sizes. In particular, note the effects of radius on pulse rise-time and cylinder length on the phi History plot.

4.2.2 A15 Crab Cavity (crabCavityT.pre)

Keywords:

`electromagnetic cavities, accelerators, mode frequencies`


VSim9 - Coaxial Cylinder (Text-based setup)
File Edit Tools View Help
Visualization Controls
Visualization Results
Welcome
Setup
Run
Analyze
Visualize
Help
Plot Control
current, emittedCurrent
□ FFT □ Log Scale ○ Zoom ● Navigate Save Image Labels Limits Reset View
Curve Var: current
Curve Var: emittedCurrent
numberOfMacroParticles
Curve Var: numberOfMacroParticles
phi
Curve Var: phi
phi (x10^6)
Reload Data

Data View: History

Graph 1
current
Location: Window 1
Color: Red
Style: Solid

Graph 2
emittedCurrent
Location: Window 1
Color: Blue
Style: Solid

Graph 3
numberOfMacroParticles
Location: Window 3
Color: Green
Style: Solid

Graph 4
phi
Location: Window 4
Color: Black

Annotation Level: Axes & Legends

Fig. 4.32: The History visualization window with the electrons.

Problem Description

The Crab Cavity simulation illustrates how to extract the modes and frequencies of an accelerator cavity in a given frequency range. The range of interest here is 3.9 to 4.1 GHz. The simulation is performed by exciting the cavity with a broadly filtered pulse that excites modes in a given range. The excitation occurs through a temporally and spatially specified current source that excites the frequencies of interest. The simulation features a variable sampling frequency, allowing the cavity to first be rung up without generating excessive memory dumps. After the simulation has been rung up, sampling frequency increases, and when combined with post-processing find the modes and frequencies. The algorithm is detailed in [1].

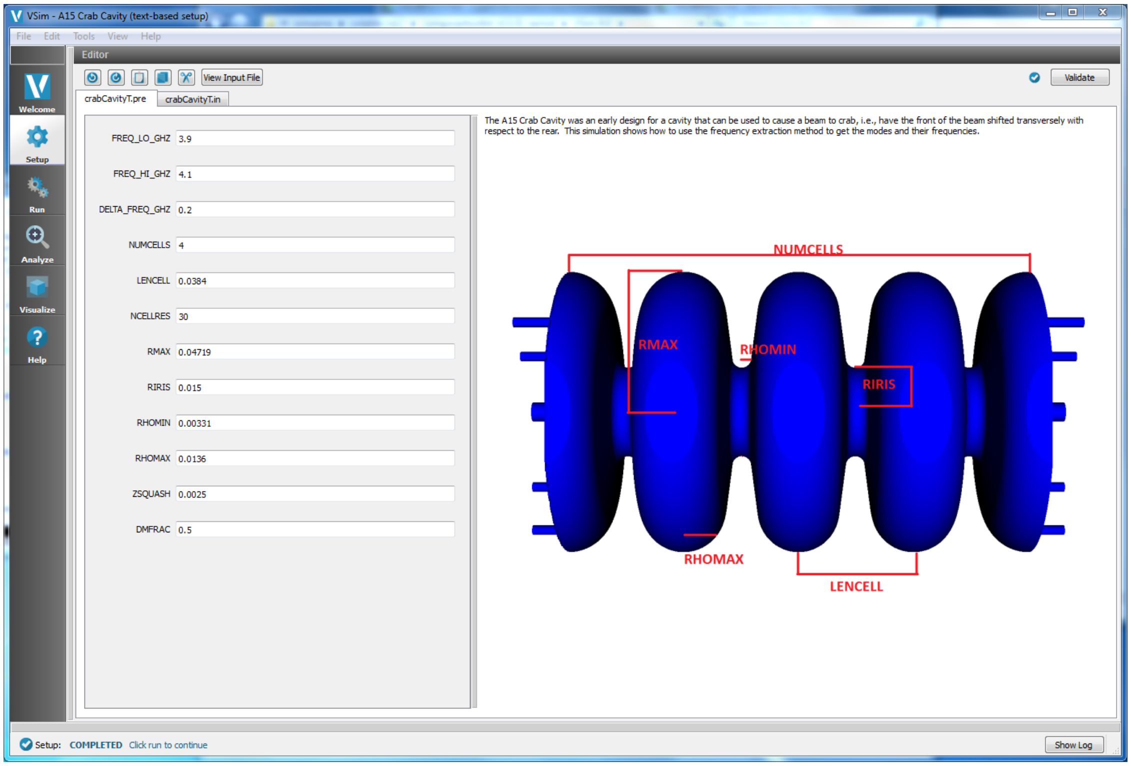
This simulation can be performed with a VSimEM, VSimMD or VSimPD license.

Opening the Simulation

The Crab Cavity example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Cavities and Waveguides (text-based setup)* option.
- Select "A15 Crab Cavity (text-based setup)" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 4.33.



The A15 Crab Cavity was an early design for a cavity that can be used to cause a beam to crab, i.e., have the front of the beam shifted transversely with respect to the rear. This simulation shows how to use the frequency extraction method to get the modes and their frequencies.

FREQ_LO_GHZ	3.9
FREQ_HI_GHZ	4.1
DELTA_FREQ_GHZ	0.2
NUMCELLS	4
LENCELL	0.0384
NCELLRES	30
RMAX	0.04719
RIRIS	0.015
RHOMIN	0.00331
RHOMAX	0.0136
ZSQUASH	0.0025
DMFRAC	0.5

Setup: COMPLETED Click run to continue

Fig. 4.33: Setup Window for the Crab Cavity example.

Input File Features

The input file allows the user to control a number of features of the Crab Cavity simulation. The FREQ_LO_GHZ and FREQ_HI_GHZ defines the range of frequencies that we are interested in extracting whereas the DELTA_FREQ_GHZ specifies the separation in frequency between the range of interest and the next nearest mode (at 4.3 GHz).

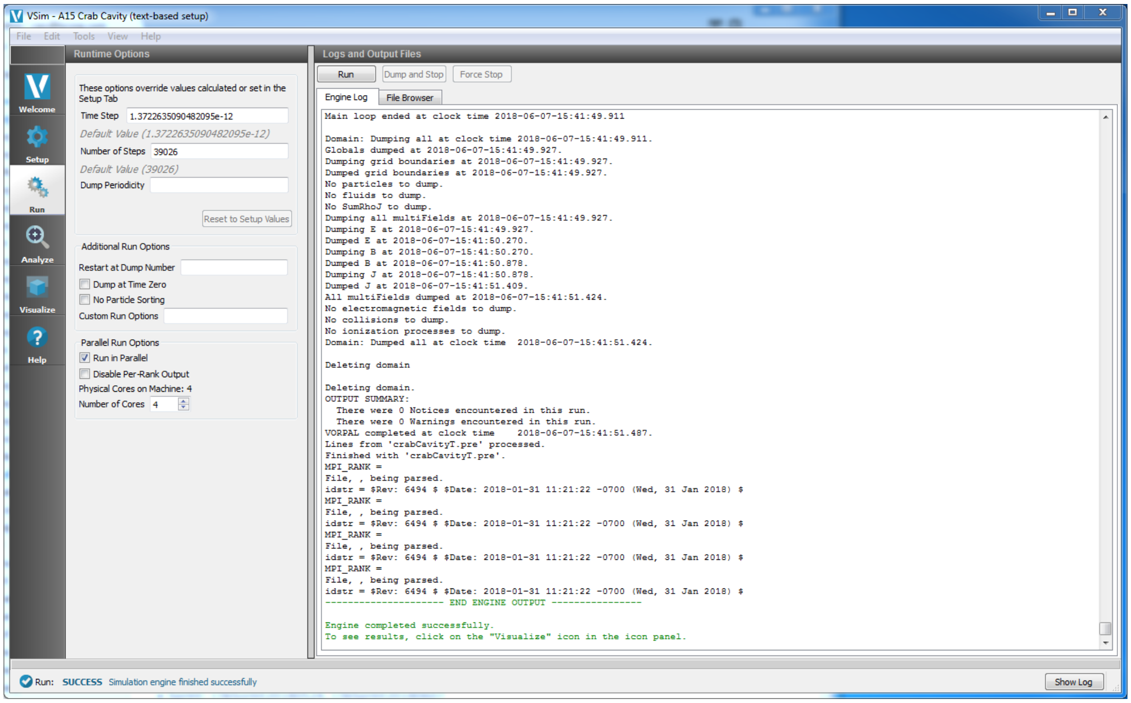
The input file is written to run for a long time, sufficient to ring up the cavity, then dump periodically during the free oscillation period. The modes and frequencies will be extracted from those dumps. This can be seen in crabCavityT.in.

The remaining key parameter values correspond to the geometry and discretization of the cells. The focus of the Crab Cavity simulation is on a four cell cavity with end holes that were originally used for measurement purposes. See [2]. The final ZSQUASH parameter is used to squeeze each cell of the cavity to eliminate the degeneracy due to cylindrical symmetry.

Running the Simulation

After examining the inputs, do the following:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- Select running in parallel as desired.
- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane on the right. You will see the output of the run in this pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.34.



These options override values calculated or set in the Setup Tab

Time Step 1.3722635090482095e-12

Default Value (1.3722635090482095e-12)

Number of Steps 39026

Default Value (39026)

Dump Periodicity

Additional Run Options

Restart at Dump Number

☐ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Main loop ended at clock time 2018-06-07-15:41:49.911

Domain: Dumping all at clock time 2018-06-07-15:41:49.911.
Globals dumped at 2018-06-07-15:41:49.927.
Dumping grid boundaries at 2018-06-07-15:41:49.927.
Dumped grid boundaries at 2018-06-07-15:41:49.927.
No particles to dump.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-07-15:41:49.927.
Dumping E at 2018-06-07-15:41:49.927.
Dumped E at 2018-06-07-15:41:50.270.
Dumping B at 2018-06-07-15:41:50.270.
Dumped B at 2018-06-07-15:41:50.878.
Dumping J at 2018-06-07-15:41:50.878.
Dumped J at 2018-06-07-15:41:51.409.
All multiFields dumped at 2018-06-07-15:41:51.424.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-07-15:41:51.424.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-07-15:41:51.487.
Lines from 'crabCavityT.pre' processed.
Finished with 'crabCavityT.pre'.
MPI_RANK =
File, , being parsed.
idstr = $Rev: 6494 $ $Date: 2018-01-31 11:21:22 -0700 (Wed, 31 Jan 2018) $
MPI_RANK =
File, , being parsed.
idstr = $Rev: 6494 $ $Date: 2018-01-31 11:21:22 -0700 (Wed, 31 Jan 2018) $
MPI_RANK =
File, , being parsed.
idstr = $Rev: 6494 $ $Date: 2018-01-31 11:21:22 -0700 (Wed, 31 Jan 2018) $
MPI_RANK =
File, , being parsed.
idstr = $Rev: 6494 $ $Date: 2018-01-31 11:21:22 -0700 (Wed, 31 Jan 2018) $
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

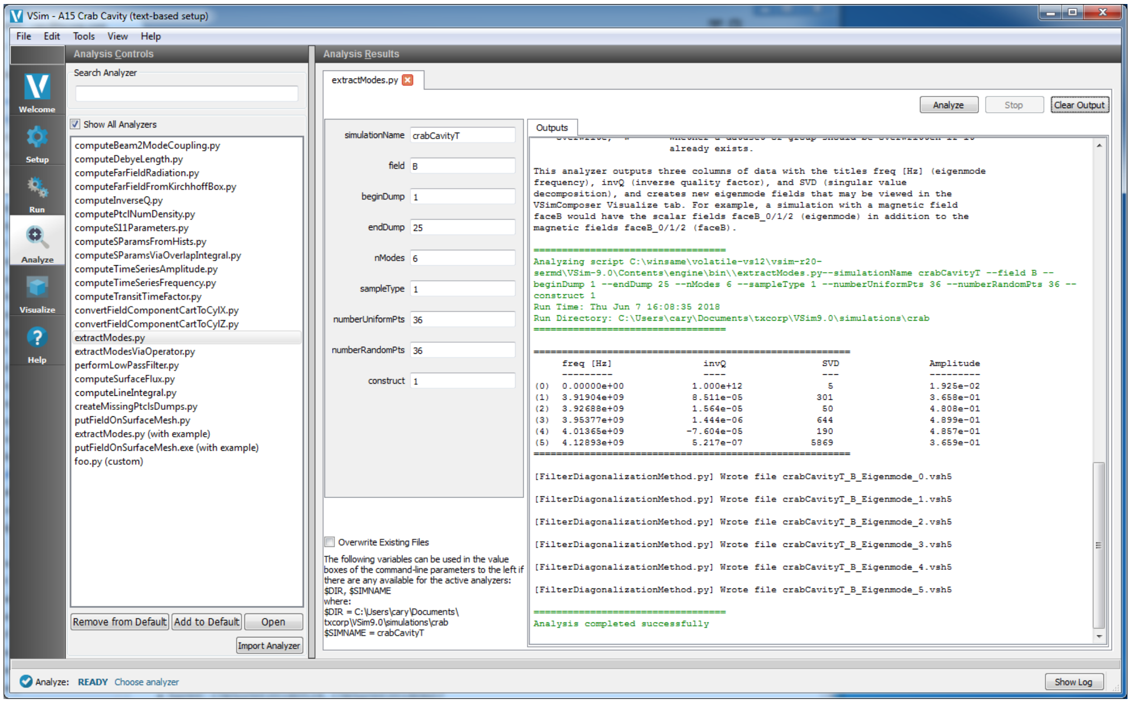
Fig. 4.34: The Run Window at the end of execution.

Analyzing the data

It is possible to extract the modes of the A15 crab cavity via post processing using the *extractModes.py - Extract Modes Analysis Script* as follows:

- Press the Analyze button in the left column of buttons.
- Click *Show All Analyzers*.
- Select *extractModes.py* and press *Open*.
- Enter the following parameters in the appropriate fields. the default simulation values are used:
  - simulationName = crabCavityT
  - field = B
  - beginDump = 1
  - endDump = 25
  - nModes = 6
  - sampleType = 1
  - numberUniformPts = 36
  - numberRandomPts = 36
  - construct = 1
- Click the *Analyze* button in the upper right corner of the window.

As shown in Fig. 4.35 below, three columns of data with the titles “freq [Hz]” (eigenmode frequency), “invQ” (inverse quality factor), and “SVD” (singular value decomposition) will be output in the right pane. The analysis has completed when you see the output “Analysis completed successfully.” One can see 6 modes, but the first one is not real as one can see from its unrealistic value of invQ, which should in fact be zero for this ideal (non-lossy) cavity, and the fact that it has zero frequency.


VSim - A15 Crab Cavity (text-based setup)
File Edit Tools View Help
Welcome
Setup
Run
Analyze
Visualize
Help
Analysis Controls
Search Analyzer
☑ Show All Analyzers
computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
computeSurfaceFlux.py
computeLineIntegral.py
createMissingPtclsDumps.py
putFieldOnSurfaceMesh.py
extractModes.py (with example)
putFieldOnSurfaceMesh.exe (with example)
foo.py (custom)
Remove from Default
Add to Default
Open
Import Analyzer
Analysis Results
extractModes.py
Analyze
Stop
Clear Output
Outputs
Analyze: READY Choose analyzer
Show Log

simulationName crabCavityT
field B
beginDump 1
endDump 25
nModes 6
sampleType 1
numberUniformPts 36
numberRandomPts 36
construct 1

☐ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analyzers:
$DIR, $SIMNAME
where:
$DIR = C:\Users\cary\Documents\txcorp\VSim9.0\simulations\crab
$SIMNAME = crabCavityT

```
                        already exists.

This analyzer outputs three columns of data with the titles freq [Hz] (eigenmode
frequency), invQ (inverse quality factor), and SVD (singular value
decomposition), and creates new eigenmode fields that may be viewed in the
VSimComposer Visualize tab. For example, a simulation with a magnetic field
faceB would have the scalar fields faceB_0/1/2 (eigenmode) in addition to the
magnetic fields faceB_0/1/2 (faceB).

==================================
Analyzing script C:\winsame\volatile-vs12\vsim-r20-
sermd\VSim-9.0\Contents\engine\bin\\extractModes.py--simulationName crabCavityT --field B --
beginDump 1 --endDump 25 --nModes 6 --sampleType 1 --numberUniformPts 36 --numberRandomPts 36 --
construct 1
Run Time: Thu Jun 7 16:08:35 2018
Run Directory: C:\Users\cary\Documents\txcorp\VSim9.0\simulations\crab
==================================

========================================================
     freq [Hz]               invQ              SVD               Amplitude
     ---------               ----              ---               ---------
(0)  0.00000e+00          1.000e+12              5               1.925e-02
(1)  3.91904e+09          8.511e-05            301               3.658e-01
(2)  3.92688e+09          1.564e-05             50               4.808e-01
(3)  3.95377e+09          1.444e-06            644               4.899e-01
(4)  4.01365e+09         -7.604e-05            190               4.857e-01
(5)  4.12893e+09          5.217e-07           5869               3.659e-01
========================================================

[FilterDiagonalizationMethod.py] Wrote file crabCavityT_B_Eigenmode_0.vsh5

[FilterDiagonalizationMethod.py] Wrote file crabCavityT_B_Eigenmode_1.vsh5

[FilterDiagonalizationMethod.py] Wrote file crabCavityT_B_Eigenmode_2.vsh5

[FilterDiagonalizationMethod.py] Wrote file crabCavityT_B_Eigenmode_3.vsh5

[FilterDiagonalizationMethod.py] Wrote file crabCavityT_B_Eigenmode_4.vsh5

[FilterDiagonalizationMethod.py] Wrote file crabCavityT_B_Eigenmode_5.vsh5

==================================
Analysis completed successfully
```

Fig. 4.35: The Analysis window at the end of execution of the extractModes.py script.

The magnetic fields at each of the eigenmode frequencies will be available to view in the Visualize Window under the *B* Field.

Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the electric field:

- Expand *Scalar Data*
- Expand *E*
- Select *E_z*
- Expand *Geometries*
- Select *poly*
- Check *Clip All Plots* and click on *Plane Controls*
- Fix the coloring by clicking on *Colors* and set *Fix Minimum* to -0.0005 and *Fix Maximum* to 0.0005.
- Move the slider at the bottom of the right pane to see the electric field at different times.

One can instead view the eigenmodes, which are so labeled under *B*. E.g., Unclick *E_z* click *B_y (Eigenmode)*. For these plots a better choice of minimum and maximum are +- 0.005. The slider can be moved to see the eigenmodes. Slider positions past the last eigenmode will display only the last eigenmode.

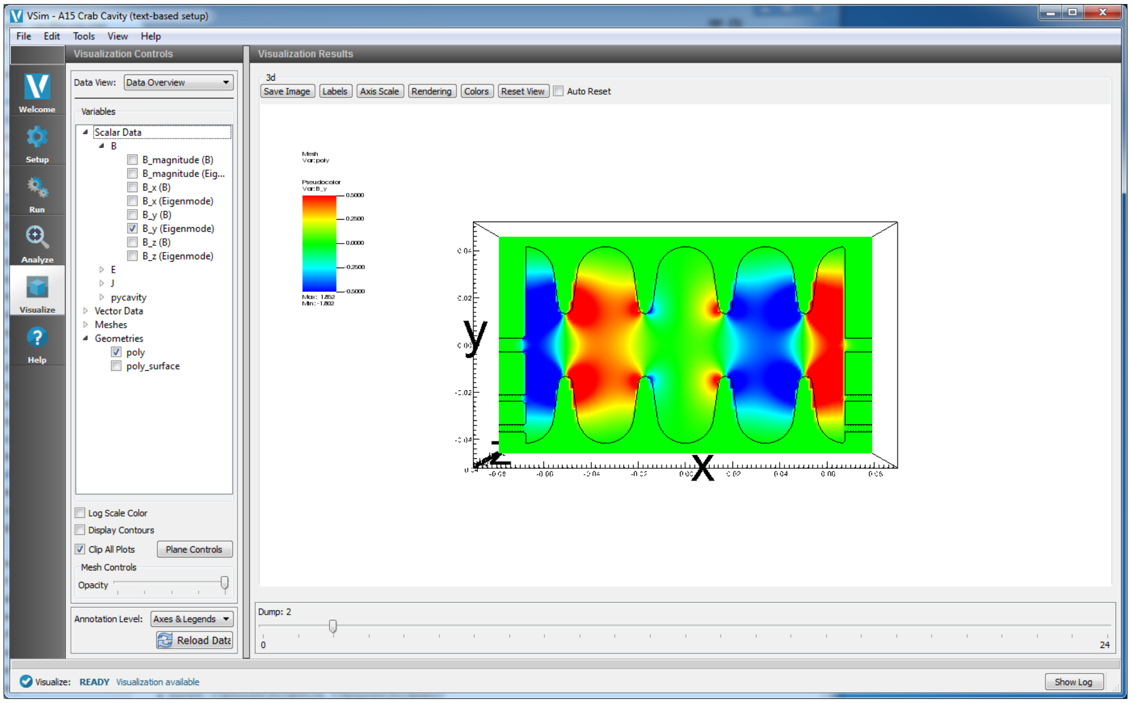


Fig. 4.36: Visualization of the electric field as a color contour plot.

Further Experiments

Additional experiments worth investigating are:

- Use Histories to record the power flow, to compute the coupling efficiency.
- Simulate one period of the waveguide with periodic boundary conditions and a user-defined phase shift, and use the frequency extraction feature to compute the waveguide modes and dispersion curves.

References

[1] G. R. Werner and J.R. Cary, “Extracting modes and frequencies from time-domain simulations with filter-diagonalization”, J. Comp. Phys., 227 (10), 5200-5214, 2008.

[2] T. M. Austin et al., “Validation of frequency extraction calculations from time-domain simulations of accelerator cavities”, Comput. Sci. Disc., 4, 015004, 2011.

4.2.3 Stairstep Cavity in coordinateGrid (emCavityCoordProdT.pre)

Keywords:

`stairstep boundary, coordinateGrid, Klystron cavity`

Problem description

This example demonstrates how to set up a complex geometry structure in VSim that uses the *coordinateGrid* system for a varying mesh size. There are two benefits of constructing a grid of kind `coordProdGrid` via coordinateGrid blocks. The first is a flexible choice of either Cartesian (x, y, z) or cylindrical (z, r, ϕ) coordinate systems to construct the grid. The second is that it enables one to vary the cell size along each axis of the grid. For example, a fine grid resolution can be used in a region of the domain consisting of complicated geometry, while a coarse resolution can be used in a different region of the domain where the geometry is simple. This method reduces the memory requirement for large multiscale simulations. The gridBoundary block (implemented in this example through the geometry macro) is used by VSim to represent complex geometrical surfaces with boundary conditions.

This example simulates a klystron cavity using a non-uniform Cartesian mesh generated by VSim's *coordinateGrid* system. Klystron cavities have wide applications as RF power sources by amplifying an RF input with electron beams. The simulated cavity is defined by a set of VSim geometry macros. Grid cell size is varied in the longitudinal direction so that a fine mesh exists at the round nose surface connecting the center drifting tube and outer ring cavity. Larger cell sizes are used at both ends of the drifting tube. The fundamental transverse magnetic (TM) mode is excited by a Gaussian current pulse.

This simulation can be performed with either a VSimEM, VSimMD, or VSimPD license.

Opening the Simulation

The emCavityCoordProdT example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Cavities and Waveguides (text-based setup)* option.
- Select "Stairstep Cavity in Coordinate Grid (text-based setup)" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the Setup Window, as shown in Fig. 4.37.

Input File Features

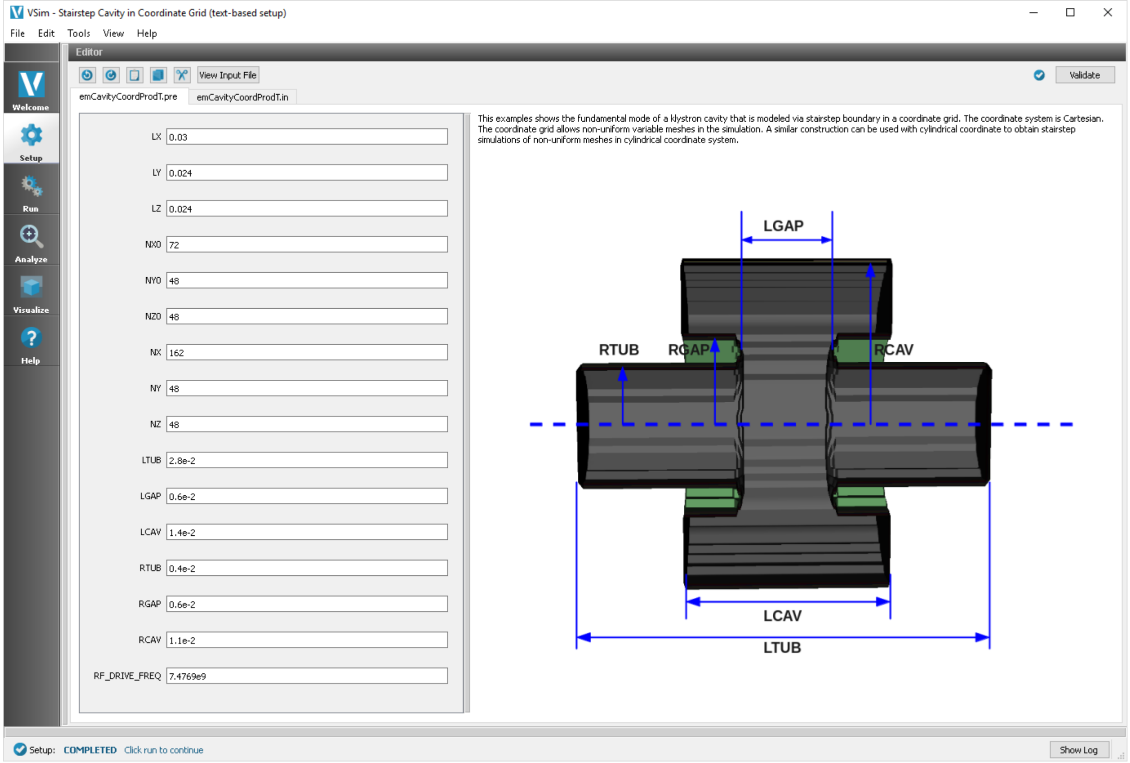
- At the top of the Editor pane, click *View Input File*.

The first important feature of this input file is the setup of the non-uniform simulation grid. Scroll to the Grid block on line 262 (Fig. 4.38). The variable spacing in x in the non-uniform grid is specified in the definitions of sectionBreaks and deltaAtBreaks in the coordinateGrid dir0 block. The deltaAtBreaks field specifies the grid cell spacing at each of the sectionBreaks positions, and the grid is generated such that the cell spacing transitions gradually between these positions. In this specific example, on lines 266 and 267, from $x =$ XBGN to $x =$ CAV_START the cell spacing transitions from $\Delta x =$ DX to $\Delta x =$ DX/3.0, then from $x =$ CAV_START to $x =$ CAV_END the cell spacing stays at a constant value of $\Delta x =$ DX/3.0, and finally from $x =$ CAV_END to $x =$ XEND the cell spacing transitions from $\Delta x =$ DX/3.0 back to $\Delta x =$ DX.

The second important feature of this input file is the electromagnetic solver for this type of grid. Both the Faraday and Ampere updaters are set to kind curlUpdaterCoordProd. By setting interiorness = cellcenter in both updaters, the curl operation is performed with a stair-stepped gridBoundary.

Running the simulation

Because the cells are not uniformly spaced, the number of cells in the simulation is unknown until calculated by VSim's Vorpal engine. However, the number of cells in each dimension is required for VSim to preprocess the input file. To correctly set the number of cells NX in the input file, take the following steps:



VSim - Stairstep Cavity in Coordinate Grid (text-based setup)

LX 0.03

LY 0.024

LZ 0.024

NX0 72

NY0 48

NZ0 48

NX 162

NY 48

NZ 48

LTUB 2.8e-2

LGAP 0.6e-2

LCAV 1.4e-2

RTUB 0.4e-2

RGAP 0.6e-2

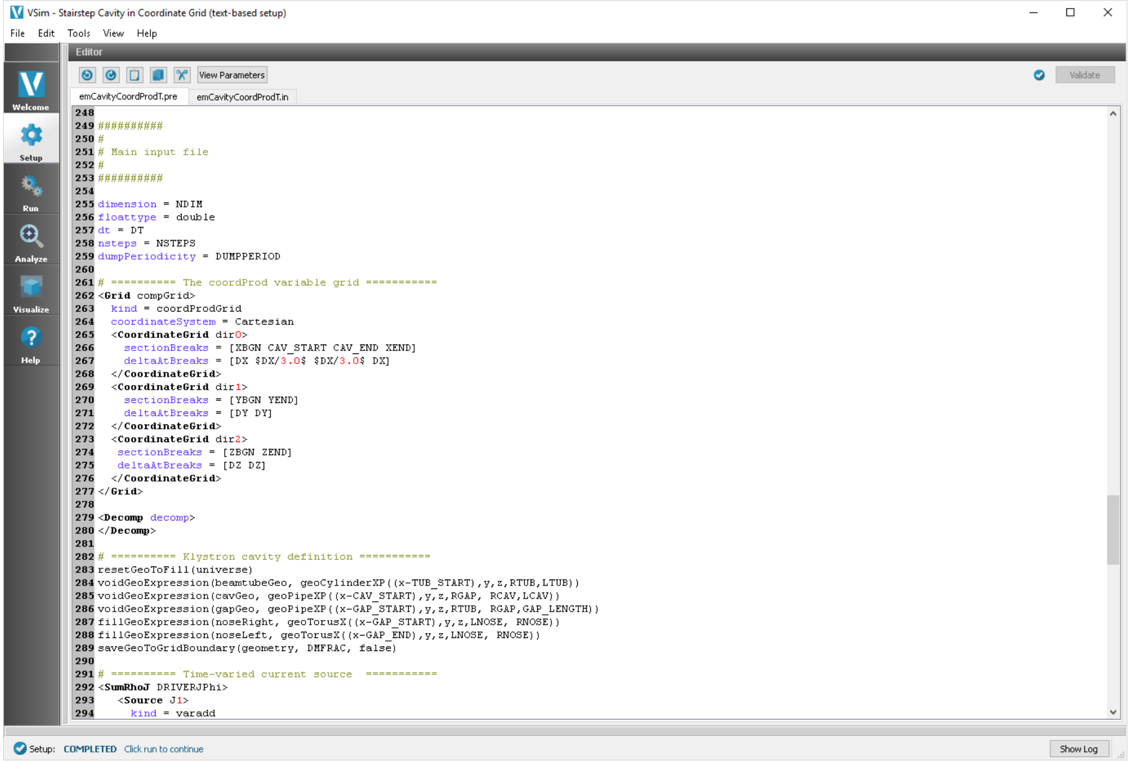
RCAV 1.1e-2

RF_DRIVE_FREQ 7.4769e9

This examples shows the fundamental mode of a klystron cavity that is modeled via stairstep boundary in a coordinate grid. The coordinate system is Cartesian. The coordinate grid allows non-uniform variable meshes in the simulation. A similar construction can be used with cylindrical coordinate to obtain stairstep simulations of non-uniform meshes in cylindrical coordinate system.

Setup: COMPLETED Click run to continue

Fig. 4.37: Setup Window for the emCavityCoordProdT example.



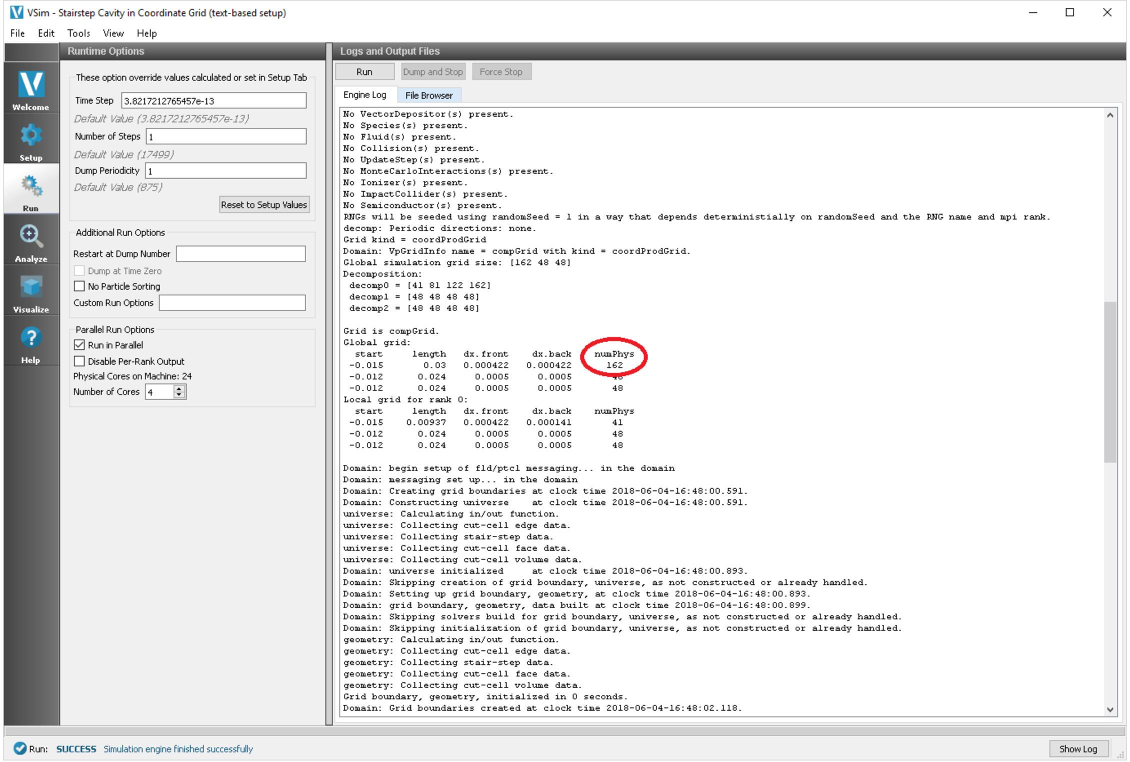
VSim - Stairstep Cavity in Coordinate Grid (text-based setup)

```

##########
#
# Main input file
#
##########

dimension = NDIM
floattype = double
dt = DT
nsteps = NSTEPS
dumpPeriodicity = DUMPPERIOD

# ========== The coordProd variable grid ===========
<Grid compGrid>
  kind = coordProdGrid
  coordinateSystem = Cartesian
  <CoordinateGrid dir0>
    sectionBreaks = [XBGN CAV_START CAV_END XEND]
    deltaAtBreaks = [DX $DX/3.0$ $DX/3.0$ DX]
  </CoordinateGrid>
  <CoordinateGrid dir1>
    sectionBreaks = [YBGN YEND]
    deltaAtBreaks = [DY DY]
  </CoordinateGrid>
  <CoordinateGrid dir2>
   sectionBreaks = [ZBGN ZEND]
   deltaAtBreaks = [DZ DZ]
  </CoordinateGrid>
</Grid>

<Decomp decomp>
</Decomp>

# ========== Klystron cavity definition ===========
resetGeoToFill(universe)
voidGeoExpression(beamtubeGeo, geoCylinderXP((x-TUB_START),y,z,RTUB,LTUB))
voidGeoExpression(cavGeo, geoPipeXP((x-CAV_START),y,z,RGAP, RCAV,LCAV))
voidGeoExpression(gapGeo, geoPipeXP((x-GAP_START),y,z,RTUB, RGAP,GAP_LENGTH))
fillGeoExpression(noseRight, geoTorusX((x-GAP_START),y,z,LNOSE, RNOSE))
fillGeoExpression(noseLeft, geoTorusX((x-GAP_END),y,z,LNOSE, RNOSE))
saveGeoToGridBoundary(geometry, DMFRAC, false)

# ========== Time-varied current source  ===========
<SumRhoJ DRIVERJPhi>
   <Source J1>
     kind = varadd
```

Setup: COMPLETED Click run to continue

Fig. 4.38: Input file for the emCavityCoordProdT example, showing the setup of the *coordinateGrid* system.

- Set the parameter `NX0` (the default is 72). This specifies the grid spacing (`DX`) at the ends of the simulation domain as `NX0/LX`.
- Run the simulation for one time-step by clicking the **Run** button in the left column of buttons, and entering "1" in both the *Number of Steps* and *Dump Periodicity* fields.
- After the simulation completes, scroll through the log file to find the value of `numPhys` in the first row of `Global grid`, as circled in Fig. 4.39.
- Go back to the Setup Window by clicking **Setup** in the left column of buttons, and enter this value into the field for `NX`. For the default values of this example, this number should be 162.



These option override values calculated or set in Setup Tab

Time Step 3.8217212765457e-13
Default Value (3.8217212765457e-13)
Number of Steps 1
Default Value (17499)
Dump Periodicity 1
Default Value (875)

Additional Run Options

Restart at Dump Number
□ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 24
Number of Cores 4

```
No VectorDepositor(s) present.
No Species(s) present.
No Fluid(s) present.
No Collision(s) present.
No UpdateStep(s) present.
No MonteCarloInteractions(s) present.
No Ionizer(s) present.
No ImpactCollider(s) present.
No Semiconductor(s) present.
RNGs will be seeded using randomSeed = 1 in a way that depends deterministially on randomSeed and the RNG name and mpi rank.
decomp: Periodic directions: none.
Grid kind = coordProdGrid
Domain: VpGridInfo name = compGrid with kind = coordProdGrid.
Global simulation grid size: [162 48 48]
Decomposition:
 decomp0 = [41 81 122 162]
 decomp1 = [48 48 48 48]
 decomp2 = [48 48 48 48]

Grid is compGrid.
Global grid:
  start     length   dx.front    dx.back    numPhys
 -0.015       0.03   0.000422   0.000422       162
 -0.012      0.024     0.0005     0.0005        48
 -0.012      0.024     0.0005     0.0005        48
Local grid for rank 0:
  start     length   dx.front    dx.back    numPhys
 -0.015    0.00937   0.000422   0.000141        41
 -0.012      0.024     0.0005     0.0005        48
 -0.012      0.024     0.0005     0.0005        48

Domain: begin setup of fld/ptcl messaging... in the domain
Domain: messaging set up... in the domain
Domain: Creating grid boundaries at clock time 2018-06-04-16:48:00.591.
Domain: Constructing universe    at clock time 2018-06-04-16:48:00.591.
universe: Calculating in/out function.
universe: Collecting cut-cell edge data.
universe: Collecting stair-step data.
universe: Collecting cut-cell face data.
universe: Collecting cut-cell volume data.
Domain: universe initialized     at clock time 2018-06-04-16:48:00.893.
Domain: Skipping creation of grid boundary, universe, as not constructed or already handled.
Domain: Setting up grid boundary, geometry, at clock time 2018-06-04-16:48:00.893.
Domain: grid boundary, geometry, data built at clock time 2018-06-04-16:48:00.899.
Domain: Skipping solvers build for grid boundary, universe, as not constructed or already handled.
Domain: Skipping initialization of grid boundary, universe, as not constructed or already handled.
geometry: Calculating in/out function.
geometry: Collecting cut-cell edge data.
geometry: Collecting stair-step data.
geometry: Collecting cut-cell face data.
geometry: Collecting cut-cell volume data.
Grid boundary, geometry, initialized in 0 seconds.
Domain: Grid boundaries created at clock time 2018-06-04-16:48:02.118.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.39: Location of the `numPhys` output to be entered into the `NX` field by the user.

The simulation is now ready to run. Return to the Run Window, enter the desired values for *Number of Steps* and *Dump Periodicity* and click *Run* once again. The run has completed when you see the output, "Engine completed successfully" as shown in Fig. 4.40. This will require approximately two hours of computation time when run in parallel on four processors on a modern CPU.

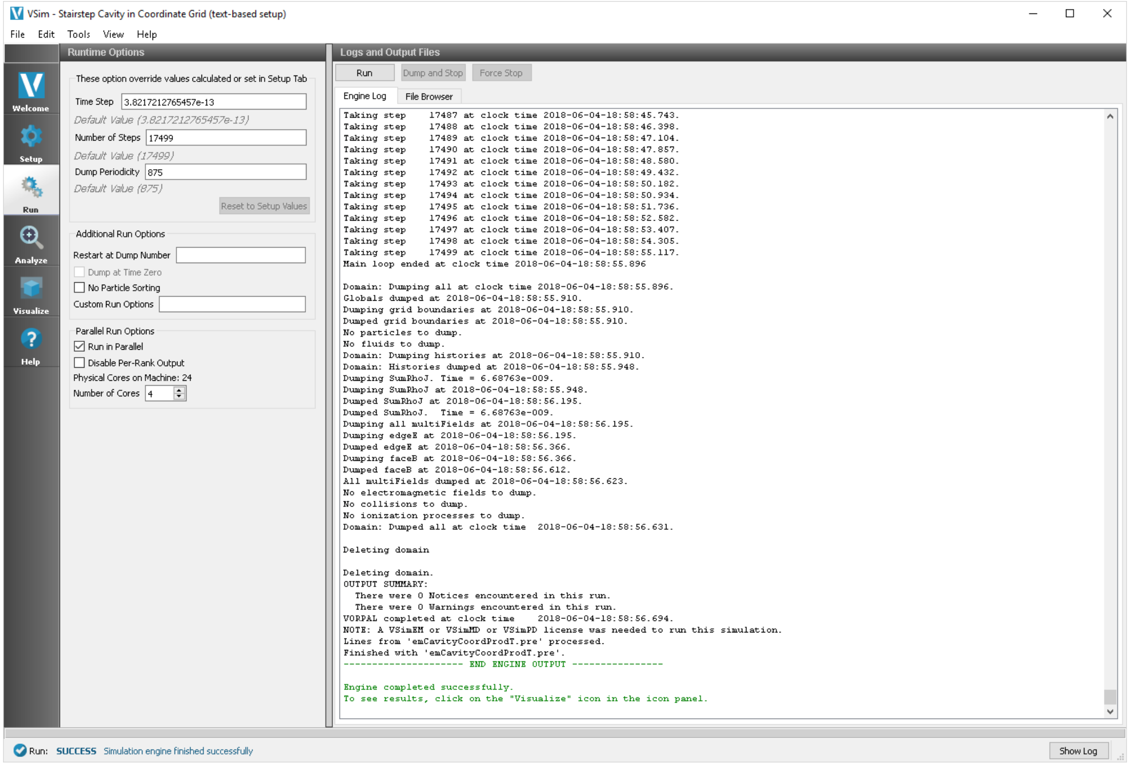
Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by clicking **Visualize** in the left column of buttons.

To create the plot as shown in Fig. 4.41:

- In the Variables section of the Visualization Controls pane, Expand *Geometries*
- Select *poly_surface*
- Expand *Meshes*



These option override values calculated or set in Setup Tab

Time Step 3.8217212765457e-13
Default Value (3.8217212765457e-13)
Number of Steps 17499
Default Value (17499)
Dump Periodicity 875
Default Value (875)

Additional Run Options

Restart at Dump Number
□ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 24
Number of Cores 4

```
Taking step    17487 at clock time 2018-06-04-18:58:45.743.
Taking step    17488 at clock time 2018-06-04-18:58:46.398.
Taking step    17489 at clock time 2018-06-04-18:58:47.104.
Taking step    17490 at clock time 2018-06-04-18:58:47.857.
Taking step    17491 at clock time 2018-06-04-18:58:48.580.
Taking step    17492 at clock time 2018-06-04-18:58:49.432.
Taking step    17493 at clock time 2018-06-04-18:58:50.182.
Taking step    17494 at clock time 2018-06-04-18:58:50.934.
Taking step    17495 at clock time 2018-06-04-18:58:51.736.
Taking step    17496 at clock time 2018-06-04-18:58:52.582.
Taking step    17497 at clock time 2018-06-04-18:58:53.407.
Taking step    17498 at clock time 2018-06-04-18:58:54.305.
Taking step    17499 at clock time 2018-06-04-18:58:55.117.
Main loop ended at clock time 2018-06-04-18:58:55.896

Domain: Dumping all at clock time 2018-06-04-18:58:55.896.
Globals dumped at 2018-06-04-18:58:55.910.
Dumping grid boundaries at 2018-06-04-18:58:55.910.
Dumped grid boundaries at 2018-06-04-18:58:55.910.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-06-04-18:58:55.910.
Domain: Histories dumped at 2018-06-04-18:58:55.948.
Dumping SumRhoJ. Time = 6.68763e-009.
Dumping SumRhoJ at 2018-06-04-18:58:55.948.
Dumped SumRhoJ at 2018-06-04-18:58:56.195.
Dumped SumRhoJ.  Time = 6.68763e-009.
Dumping all multiFields at 2018-06-04-18:58:56.195.
Dumping edgeE at 2018-06-04-18:58:56.195.
Dumped edgeE at 2018-06-04-18:58:56.366.
Dumping faceB at 2018-06-04-18:58:56.366.
Dumped faceB at 2018-06-04-18:58:56.612.
All multiFields dumped at 2018-06-04-18:58:56.623.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-04-18:58:56.631.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-04-18:58:56.694.
NOTE: A VSimEM or VSimMD or VSimPD license was needed to run this simulation.
Lines from 'emCavityCoordProdT.pre' processed.
Finished with 'emCavityCoordProdT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

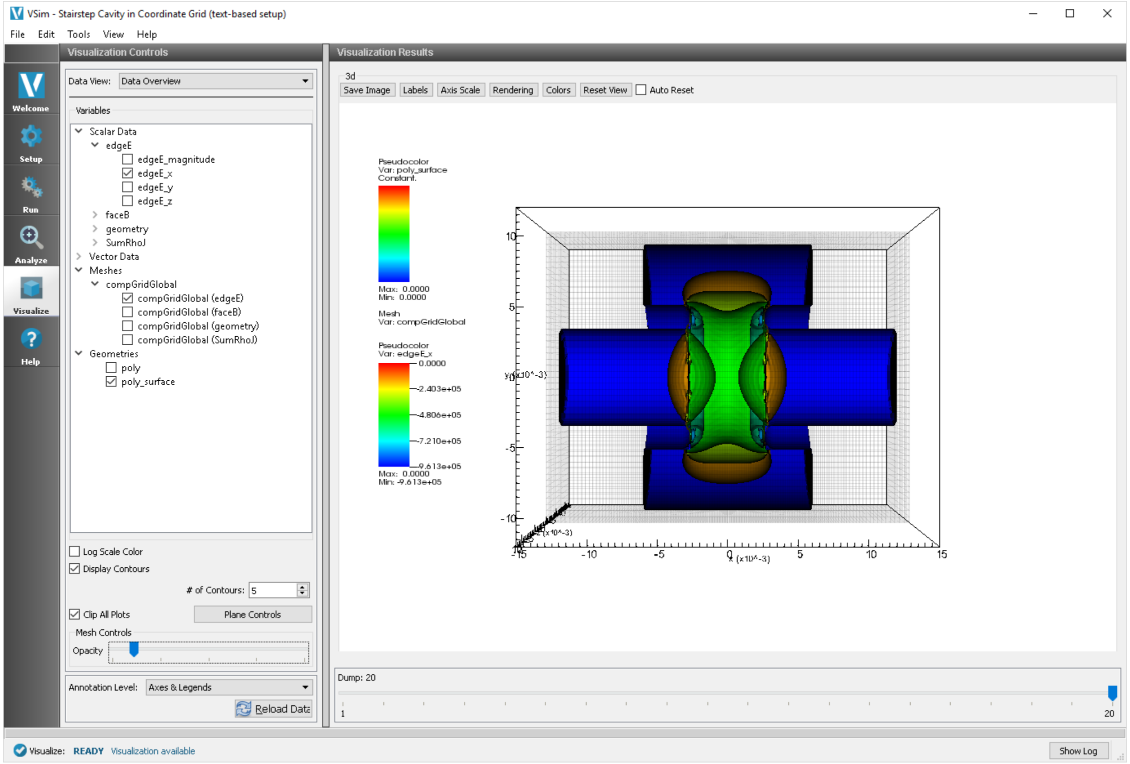
Run: **SUCCESS** Simulation engine finished successfully

Fig. 4.40: The Run Window during the execution.

- Expand *compGridGlobal*
- Select any of the grid choices, for they are all identical for this simulation
- Expand *Scalar Data*
- Expand *edgeE*
- Select *edgeE_x*
- Near the bottom of the Visualization Controls pane, select *Clip All Plots*
- Click the *Plane Controls* button and set the normal in the Z-direction (which should already selected by default)
- Select *Display Contours*
- Decrease the Opacity by moving the Opacity slider the left to better see the fields through the grid
- At the bottom of the Visualization Results pane, move dump slider forward in time

Further Experiments

The *coordinateGrid* system is also capable of creating non-uniform grids in cylindrical coordinates, by setting `coordinateSystem = Cylindrical` in the `Grid` block. The `curlUpdaterCoordProd` updater also works with the same settings in cylindrical coordinates.



Data View: Data Overview

Variables
- Scalar Data
  - edgeE
    - ☐ edgeE_magnitude
    - ☑ edgeE_x
    - ☐ edgeE_y
    - ☐ edgeE_z
  - faceB
  - geometry
  - SumRhoJ
- Vector Data
- Meshes
  - compGridGlobal
    - ☑ compGridGlobal (edgeE)
    - ☐ compGridGlobal (faceB)
    - ☐ compGridGlobal (geometry)
    - ☐ compGridGlobal (SumRhoJ)
- Geometries
  - ☐ poly
  - ☑ poly_surface

☐ Log Scale Color

☑ Display Contours

# of Contours: 5

☑ Clip All Plots

Annotation Level: Axes & Legends

Dump: 20

Visualize: READY Visualization available

Fig. 4.41: Visualization of the x-component of the electric field in the simulation of a klystron cavity with a non-uniform mesh.

4.3 Radiation Generation

4.3.1 A6 Magnetron 1: Modes (a6Magnetron1Modes.sdf)

Keywords:

`magnetron, cavity modes, A6`

Problem Description

This VSimMD example simulates MIT's cylindrical A6 magnetron cavity with no outlets in three dimensions. The structure is generated using shape primitives within the VSim composer. The cavity is excited by a sinc pulse ping using a distributed current source within one of the resonant cavities. The spectrum of the cavity is used to find the modes, and FDM is used to extract the exact mode profile of the cold cavity.

This simulation can be performed with a VSimMD license.

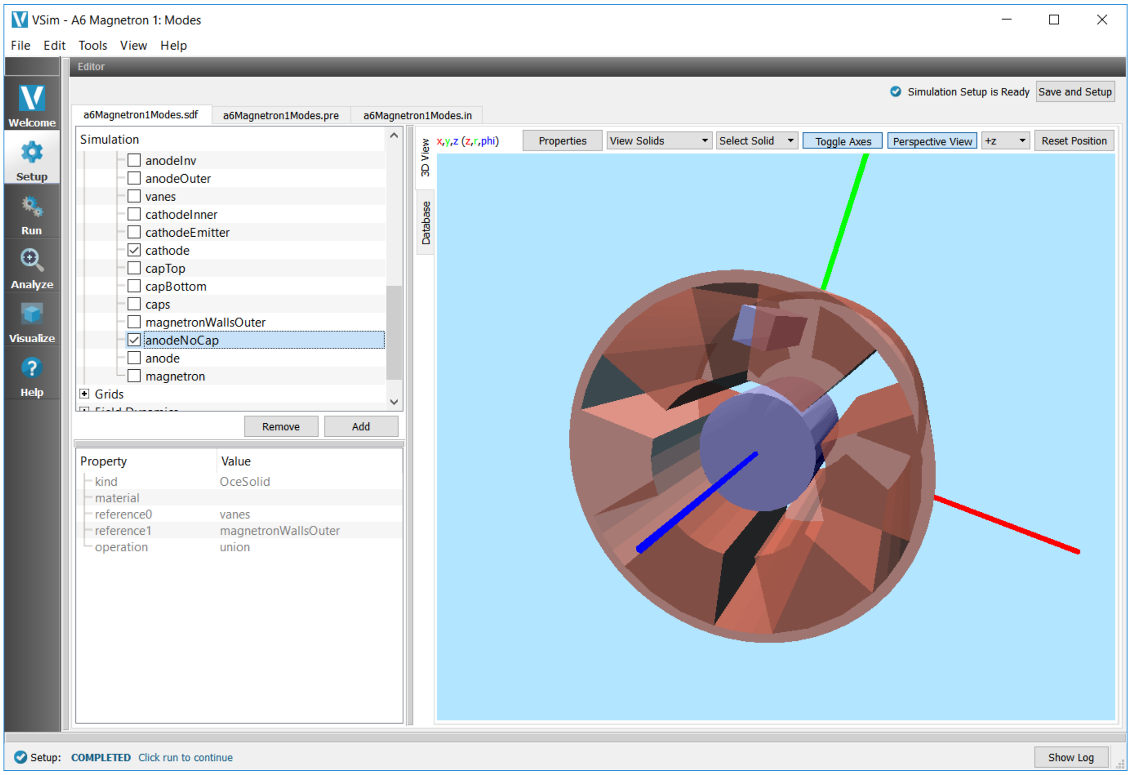
Opening the Simulation

The A6 Magnetron example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Radiation Generation* option.

- Select “A6 Magnetron 1:Modes” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the setup window as shown in Fig. 4.42. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



Simulation

- □ anodeInv
- □ anodeOuter
- □ vanes
- □ cathodeInner
- □ cathodeEmitter
- ☑ cathode
- □ capTop
- □ capBottom
- □ caps
- □ magnetronWallsOuter
- ☑ anodeNoCap
- □ anode
- □ magnetron

⊞ Grids

Property	Value
kind	OceSolid
material	
reference0	vanes
reference1	magnetronWallsOuter
operation	union

Setup: COMPLETED Click run to continue

Fig. 4.42: Setup window for the A6 Magnetron example showing cathode and anode vanes only.

Simulation Properties

The A6 Magnetron example includes several constants for easy adjustment of simulation properties. User changeable parameters include:

- **RADIUS_ANODE** → Inner radius of anode.
- **RADIUS_ANODE_OUTER** → Outer radius of anode.
- **ANGLE_CAVITY** → Angle of resonant cavity openings, in degrees.
- **THICKNESS_WALL_OUTER** → Thickness of all walls.
- **WIDTH_VANES** → Total width of anode vanes in z-direction.
- **RADIUS_CATHODE** → Radius of the emitting section of the cathode.
- **RADIUS_CATHODE_INNER** → Radius of the inner section of the cathode.
- **WIDTH_CATHODE** → Width (in z-direction) of the emitting section of the cathode.
- **FREQ_LOW** → Lower frequency of excitation source range.

- **FREQ_HIGH** → Upper frequency of excitation source range.
- **(X,Y,Z)POS_CURR** → Position of excitation distributed current source.
- **(XY,Z)SIZE_CURR** → Size of excitation distributed current source region.
- **(X,Y,Z)POS_HIST** → Position of electric field history.

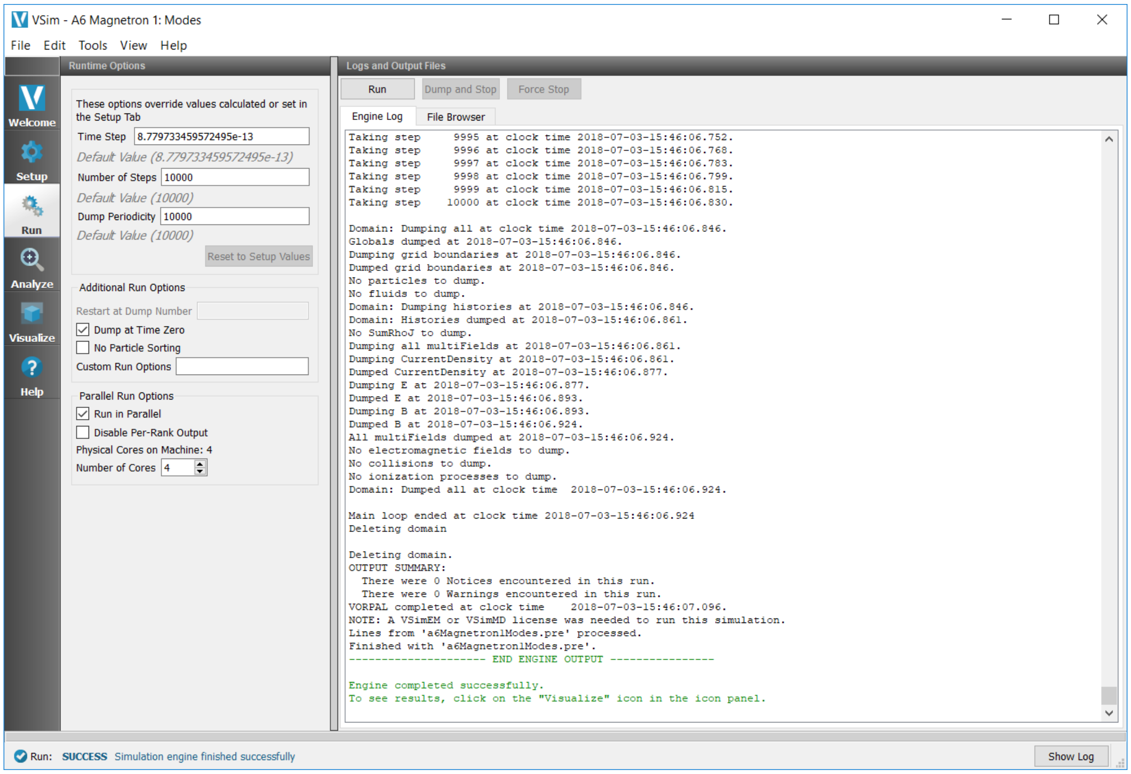
The axis of the cavity coincides with the z-axis and the center is at z = 0. The emitting cathode region is 4.0 cm long. All surfaces are perfect electric conductors. Histories of the electric and magnetic fields are taken at the inside of one of the cavities to find the modes. The FFT of the history shows the mode frequencies, and the exact value and profile is found using *extractModes.py - Extract Modes Analysis Scripts*.

The excitation frequency range can be set using the constants FREQ_LOW and FREQ_HIGH. The total excitation time is calculated in TIME_EXCITE. The simulation should be run for longer than TIME_EXCITE to allow the excitation source to complete.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 4.43.



These options override values calculated or set in the Setup Tab

Time Step 8.779733459572495e-13

Default Value (8.779733459572495e-13)

Number of Steps 10000

Default Value (10000)

Dump Periodicity 10000

Default Value (10000)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Taking step     9995 at clock time 2018-07-03-15:46:06.752.
Taking step     9996 at clock time 2018-07-03-15:46:06.768.
Taking step     9997 at clock time 2018-07-03-15:46:06.783.
Taking step     9998 at clock time 2018-07-03-15:46:06.799.
Taking step     9999 at clock time 2018-07-03-15:46:06.815.
Taking step    10000 at clock time 2018-07-03-15:46:06.830.

Domain: Dumping all at clock time 2018-07-03-15:46:06.846.
Globals dumped at 2018-07-03-15:46:06.846.
Dumping grid boundaries at 2018-07-03-15:46:06.846.
Dumped grid boundaries at 2018-07-03-15:46:06.846.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-07-03-15:46:06.846.
Domain: Histories dumped at 2018-07-03-15:46:06.861.
No SumRhoJ to dump.
Dumping all multiFields at 2018-07-03-15:46:06.861.
Dumping CurrentDensity at 2018-07-03-15:46:06.861.
Dumped CurrentDensity at 2018-07-03-15:46:06.877.
Dumping E at 2018-07-03-15:46:06.877.
Dumped E at 2018-07-03-15:46:06.893.
Dumping B at 2018-07-03-15:46:06.893.
Dumped B at 2018-07-03-15:46:06.924.
All multiFields dumped at 2018-07-03-15:46:06.924.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-03-15:46:06.924.

Main loop ended at clock time 2018-07-03-15:46:06.924
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-07-03-15:46:07.096.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'a6Magnetron1Modes.pre' processed.
Finished with 'a6Magnetron1Modes.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.43: Run window for the A6 Magentron mode extraction example after the initial run..

The simulation is best run in two steps: The first with course dump periodicity to excite the modes, and the second with fine dump periodicity to observe the modes.

- Initially, the simulation is run for 10,000 time steps, writing one dump file at the end, to allow the current source excitation to finish.
- After the initial run, change the Number of Time Steps to 2,000, the Dump Periodicity to 50, and enter 1 into Restart at Dump Number. This will record details of the excited field after the source has finished.

Visualizing the results

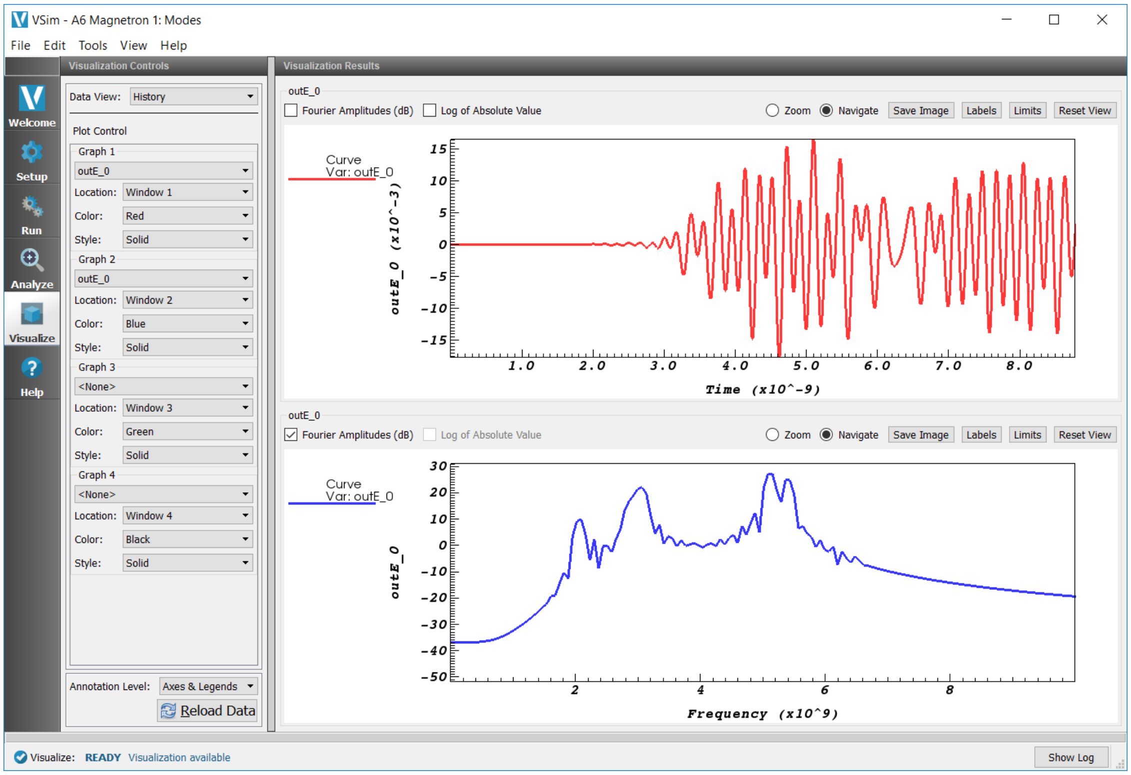
After performing the above actions, continue as follows:

- Proceed to the Visualize window by pressing the Visualize button in the left column of buttons.

To visualize a run to determine the resonant frequency, select *History* from the *Data View* pull-down menu. Select outE_0 for *Graph 1* and *Graph 2*, and click *Fourier Amplitudes (dB)* to the left of one of the outE_0 plot in the *Visualization Results* pane. .

- Proceed to the Visualize window by pressing the Visualize icon in the left panel.
- Select History under Data View.
- For Graph 3 and Graph 4, select <None> to be plotted.
- Then for Graph 2 select the *Fourier Amplitudes (dB)* checkbox * In the upper right corner of each plot, select Limits and set X-Axis max to 1e10.
- The result should be that shown in Fig. 4.44.

Note that running the simulation longer will more sharply resolve the mode frequencies in the FFT.



Data View: History

Graph 1
outE_0
Location: Window 1
Color: Red
Style: Solid

Graph 2
outE_0
Location: Window 2
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 3
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

outE_0
□ Fourier Amplitudes (dB) □ Log of Absolute Value

outE_0
☑ Fourier Amplitudes (dB) □ Log of Absolute Value

Visualize: READY Visualization available

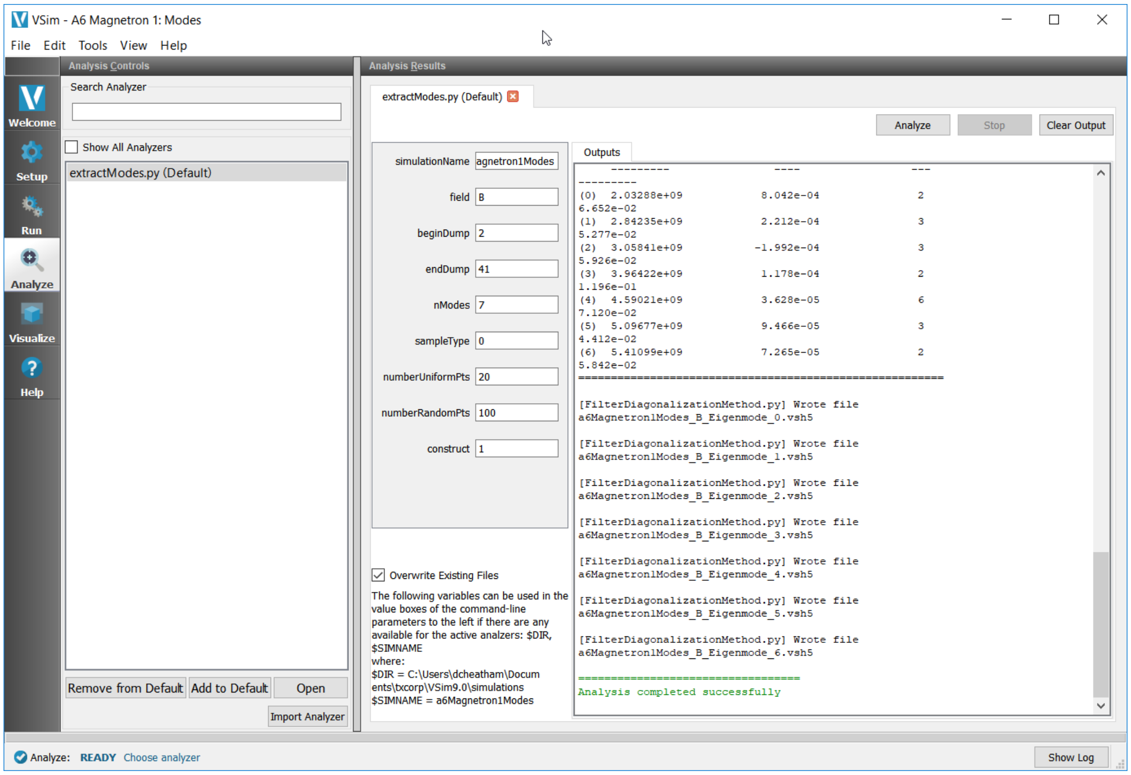
Fig. 4.44: Fourier transform of outE_0 versus time (in Hertz).

Analyzing the spectrum

It is possible to extract the modes of the A6 magnetron cavity via post processing using the *extractModes.py - Extract Modes Analysis Script* as follows:

- Press the Analyze button in the left column of buttons.
- Select *extractModes.py(default)*.
- Enter the following parameters in the appropriate fields:
  - simulationName = magnetronA6Modes
  - field = B
  - beginDump = 2
  - endDump = 41
  - nModes = 7
  - sampleType = 0
  - numberUniformPts = 20
  - numberRandomPts = 100
  - construct = 1
- Click the *Analyze* button in the upper right corner of the window.

Three columns of data with the titles “freq [Hz]” (Eigenmode frequency), “invQ” (inverse quality factor), and “SVD” (singular value decomposition) will be output in the right pane. The analysis has completed when you see the output “Analysis completed successfully.” One can see 7 modes in Fig. 4.3.1.



simulationName: agnetron1Modes

field: B

beginDump: 2

endDump: 41

nModes: 7

sampleType: 0

numberUniformPts: 20

numberRandomPts: 100

construct: 1

☑ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\dcheatham\Documents\txcorp\VSim9.0\simulations
$SIMNAME = a6Magnetron1Modes

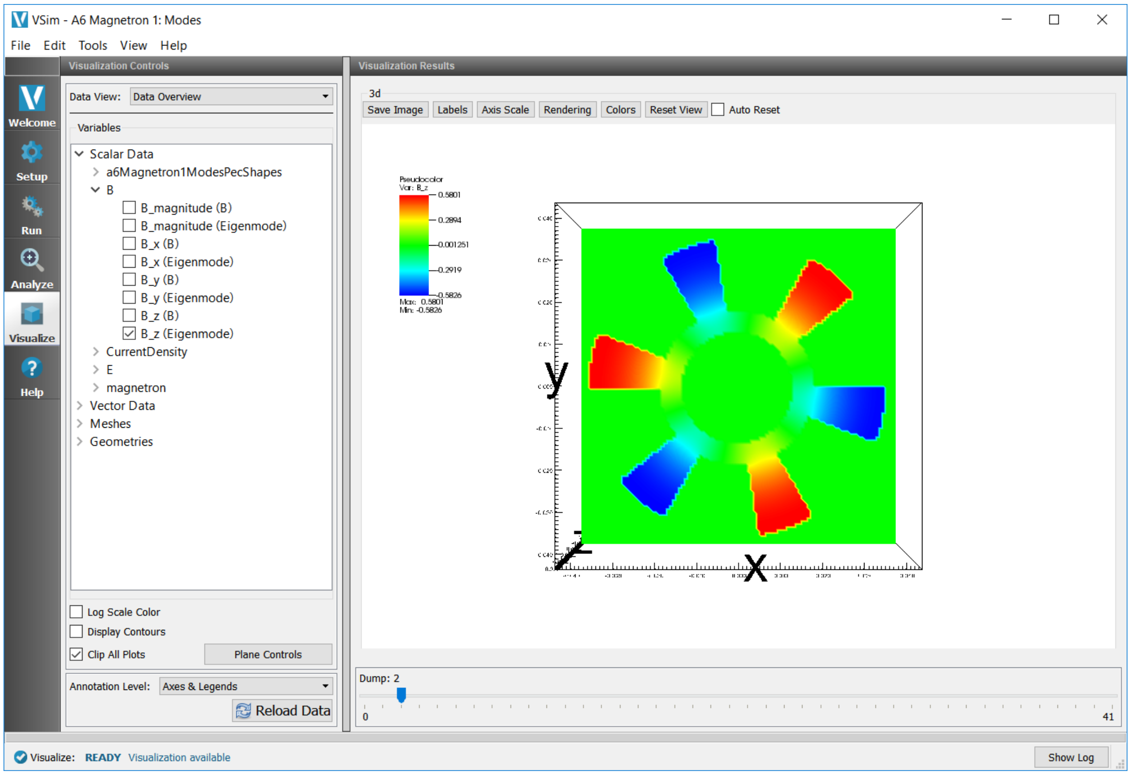
```
     ---------            ----              ---
---------
(0)  2.03288e+09          8.042e-04           2
6.652e-02
(1)  2.84235e+09          2.212e-04           3
5.277e-02
(2)  3.05841e+09         -1.992e-04           3
5.926e-02
(3)  3.96422e+09          1.178e-04           2
1.196e-01
(4)  4.59021e+09          3.628e-05           6
7.120e-02
(5)  5.09677e+09          9.466e-05           3
4.412e-02
(6)  5.41099e+09          7.265e-05           2
5.842e-02
=========================================================

[FilterDiagonalizationMethod.py] Wrote file
a6Magnetron1Modes_B_Eigenmode_0.vsh5

[FilterDiagonalizationMethod.py] Wrote file
a6Magnetron1Modes_B_Eigenmode_1.vsh5

[FilterDiagonalizationMethod.py] Wrote file
a6Magnetron1Modes_B_Eigenmode_2.vsh5

[FilterDiagonalizationMethod.py] Wrote file
a6Magnetron1Modes_B_Eigenmode_3.vsh5

[FilterDiagonalizationMethod.py] Wrote file
a6Magnetron1Modes_B_Eigenmode_4.vsh5

[FilterDiagonalizationMethod.py] Wrote file
a6Magnetron1Modes_B_Eigenmode_5.vsh5

[FilterDiagonalizationMethod.py] Wrote file
a6Magnetron1Modes_B_Eigenmode_6.vsh5

===================================
Analysis completed successfully
```

Analyze: READY Choose analyzer

- Proceed to the Visualize window by pressing the Visualize icon in the left panel.

- Select *Data Overview* under *Data View*.
- Expand *Scalar Data*
- Expand *B*
- Select *B_z (Eigenmode)*.
- Select *Clip All Plots* in the bottom left corner.
- Move the dump slider first to dump 2 and then to dump 5.

The axial magnetic field inside the cavity during the π and 2π mode operations are shown in Fig. 4.45 and Fig. 4.46, respectively:



Data View: Data Overview

- ☐ B_magnitude (B)
- ☐ B_magnitude (Eigenmode)
- ☐ B_x (B)
- ☐ B_x (Eigenmode)
- ☐ B_y (B)
- ☐ B_y (Eigenmode)
- ☐ B_z (B)
- ☑ B_z (Eigenmode)

☐ Log Scale Color
☐ Display Contours
☑ Clip All Plots

Annotation Level: Axes & Legends

☐ Auto Reset

Max: 0.5801
Min: -0.5826

Dump: 2

Visualize: READY Visualization available

Fig. 4.45: Visualization of the axial B-field in the π Eigenmode.

Further Experiments

The values of FREQ_LOW and FREQ_HIGH can be adjusted to find additional modes, or to focus in on a specific mode. Narrowing the excitation range to fewer modes will produce a mode accurate frequency and Q-factor extraction for the mode.

4.3.2 A6 Magnetron 2: Power (a6Magnetron2Power.sdf)

Keywords:

magnetron, bunching, space charge, A6

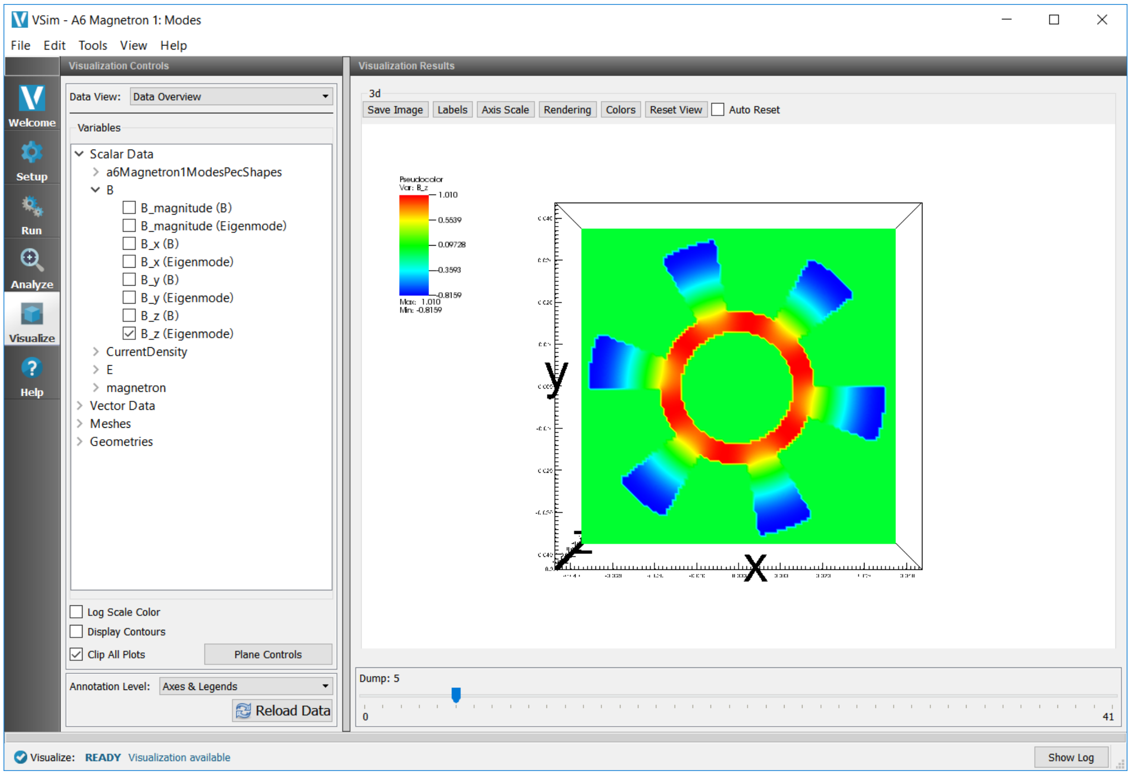


Fig. 4.46: Visualization of the axial B-field in the 2π Eigenmode.

Problem Description

This VSimMD example simulates MIT's cylindrical A6 magnetron cavity with a slot outlet in three dimensions. The geometry was defined by using VSimComposer's constructive solid geometry (CSG) capabilities. The cathode-anode voltage is ramped up from zero to around 360 kV by a current distribution source. Electrons are emitted from the emitter section of the cathode, and undergo $E \times B$ drift. Bunching of the space-charge distribution occurs and kinetic energy from the electrons is transfered to the electromagnetic modes of the cavity. If the simulation is run long enough, it will be seen that the 2π mode dominates.

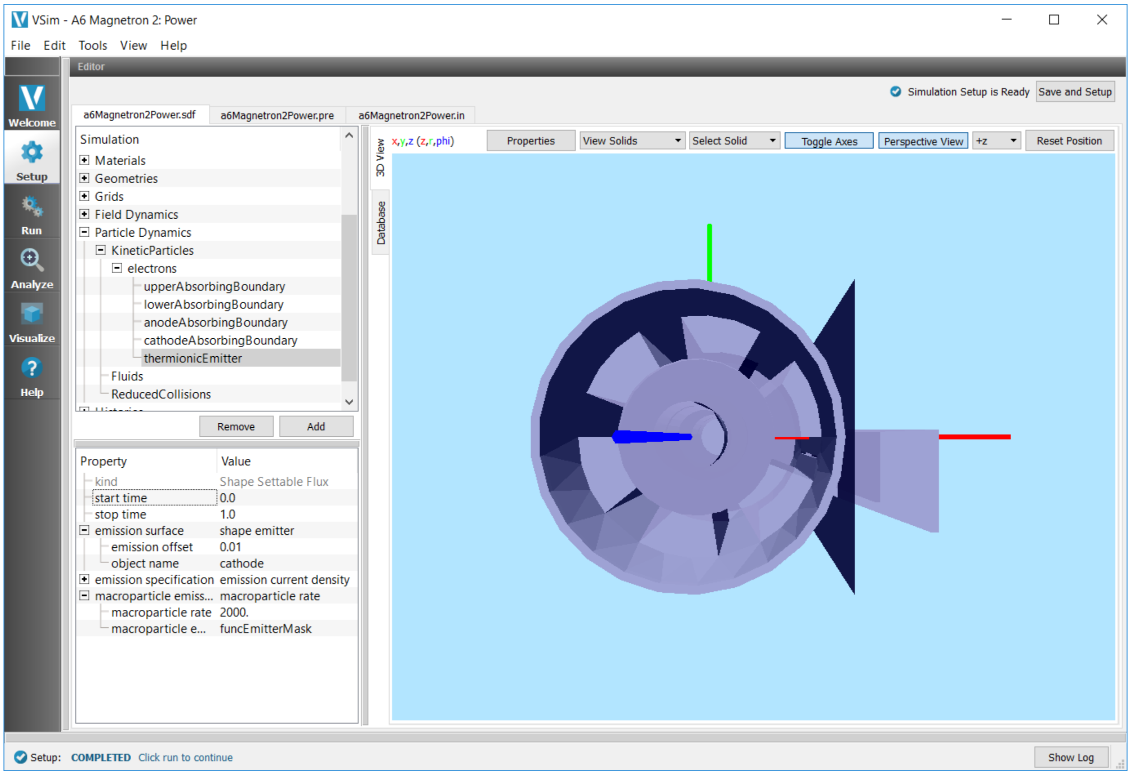
This simulation can be performed with a VSimMD license.

Opening the Simulation

The A6 Magnetron example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Radiation Generation* option.
- Select *A6 Magnetron 2: Power* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 4.47. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or de-select the box next to `Grid`.



Property	Value
kind	Shape Settable Flux
start time	0.0
stop time	1.0
emission surface	shape emitter
emission offset	0.01
object name	cathode
emission specification	emission current density
macroparticle emiss...	macroparticle rate
macroparticle rate	2000.
macroparticle e...	funcEmitterMask

Setup: COMPLETED Click run to continue

Fig. 4.47: Setup Window for the A6 Magnetron example.

Simulation Properties

The A6 Magnetron example includes several constants for easy adjustment of simulation properties. User changable parameters include:

- **RADIUS_ANODE** → Inner radius of anode.
- **RADIUS_ANODE_OUTER** → Outer radius of anode.
- **ANGLE_CAVITY** → Angle of resonant cavity openings, in degrees.
- **THICKNESS_WALL_OUTER** → Thickness of all walls.
- **WIDTH_MAGNETRON** → Total width of magnetron in z-direction.
- **RADIUS_OUTLET** → Radius of outlet horn in x-direction.
- **WIDTH_IRIS** → Width of outlet slit opening.
- **WIDTH_VANES** → Total width of anode vanes in z-direction.
- **RADIUS_CATHODE** → Radius of the emitting section of the cathode.
- **RADIUS_CATHODE_INNER** → Radius of the inner section of the cathode.
- **WIDTH_CATHODE** → Width (in z-direction) of the emitting section of the cathode.
- **(X,Y,Z)POS_HIST** → Position of electric field history.

The axial magnetic field is uniform with a constant value of $B_z = 0.6$ T. There is an opening at the back of one of the cavities that allows microwave energy to leave the magnetron through a horn antenna and into a matched absorbing layer (MAL) boundary. A Poynting Flux history records the power output through the antenna. The emitting region of the cathode has a slightly larger radius than the rest of the cathode. There are also end caps, on either side of the vanes, in electrical contact with the cathode.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- Here you can set run parameters, including how many cores to run with (under the MPI tab).
- When you are finished setting run parameters, click on the *Run* button in the upper right corner. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.”

This simulation is setup so that the cathode emission current and anode-cathode (AK) voltage ramp up relatively slowly (over many RF periods). Once the AK voltage is high enough, the bunching of the electrons will occur. If the simulation is run for long enough, around 100000 time steps, the 2π mode will eventually dominate as has been seen experimentally for this magnetron.

Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

After the simulation has been run for a sufficient amount of time, spokes will form in the electron distribution (bunching). Since the A6 is a six-cavity magnetron, operation in the 2π mode will correspond to six spokes. To visualize the spokes:

- Select *Phase Space* from the *Data View* pull-down menu
- Select *electrons_x* for the *X-axis*
- Select *electrons_y* for the *Y-axis*
- Use the *Dump* bar at the bottom of the screen to advance through the solution and visualize each dump file

Eventually, a steady state will be reached in which the electron distribution has six spokes similar to Fig. 4.48.

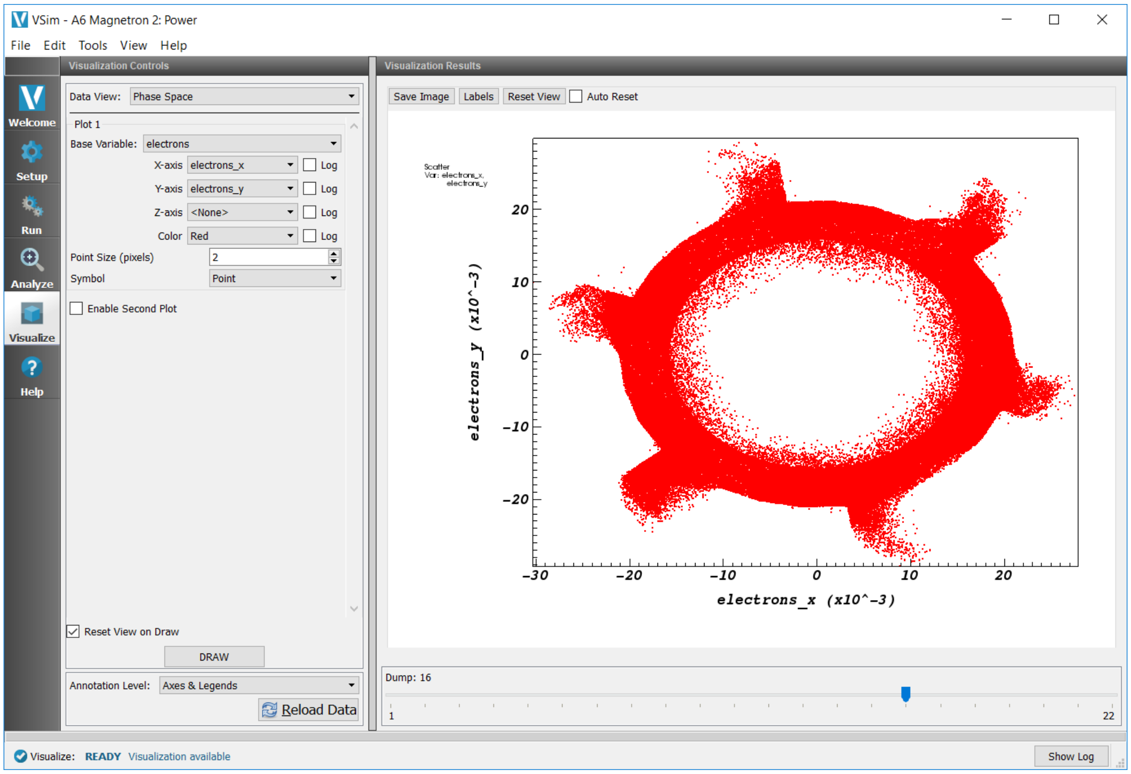
To visualize the axial magnetic field during the 2π mode operation, proceed as follows:

- Select *Data Overview* from the *Data View* pull-down menu
- Expand *B*
- Select *B_z*
- Expand *Geometries*
- Select *poly (a6Magnetron2PowerPecShapes)*
- Select *Clip All Plots*
- Select *Colors*
- Check *Fix Minimum* and *Fix Maximum* and set to -0.4 and 0.4 respectively
- Move the dump slider forward in time to see the evolution.

Image Fig. 4.49 was taken at Dump 16.

To determine the operating frequency:

- Select *History* from the *Data View* pull-down menu



VSim - A6 Magnetron 2: Power

Data View: Phase Space

Plot 1

Base Variable: electrons

X-axis: electrons_x □ Log

Y-axis: electrons_y □ Log

Z-axis: <None> □ Log

Color: Red □ Log

Point Size (pixels): 2

Symbol: Point

□ Enable Second Plot

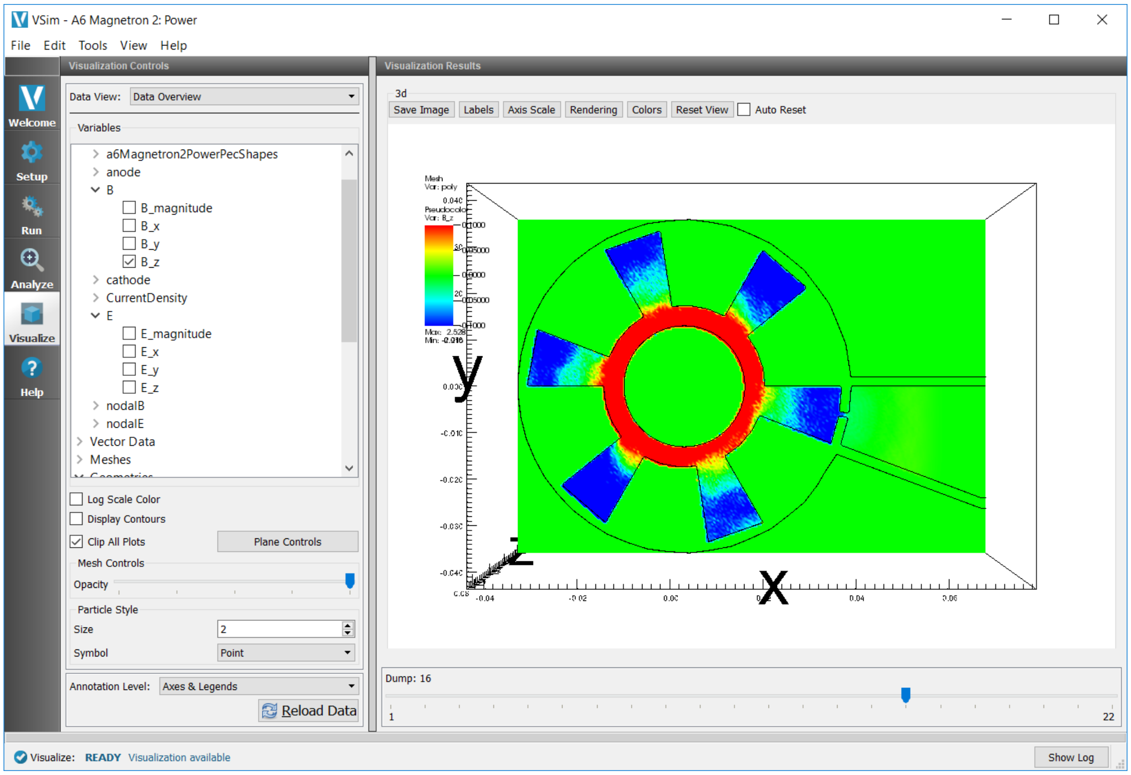
☑ Reset View on Draw

Annotation Level: Axes & Legends

Dump: 16

Visualize: READY Visualization available

Fig. 4.48: Phase-space plot of the electron distribution showing the formation of six spokes corresponding to operation in the 2π mode.



VSim - A6 Magnetron 2: Power

Data View: Data Overview

Variables

- a6Magnetron2PowerPecShapes
- anode
- B
  - □ B_magnitude
  - □ B_x
  - □ B_y
  - ☑ B_z
- cathode
- CurrentDensity
- E
  - □ E_magnitude
  - □ E_x
  - □ E_y
  - □ E_z
- nodalB
- nodalE
- Vector Data
- Meshes

□ Log Scale Color

□ Display Contours

☑ Clip All Plots

Mesh Controls

Opacity

Particle Style

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

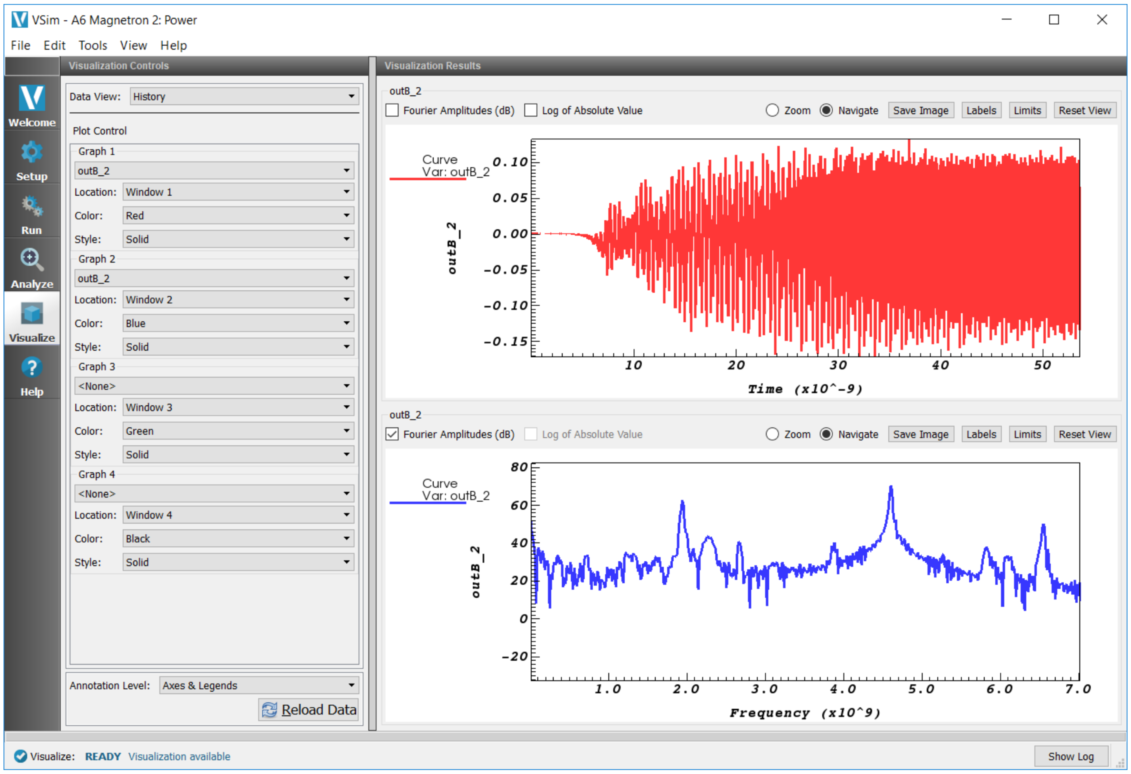
Dump: 16

Visualize: READY Visualization available

Fig. 4.49: Axial magnetic field of the magnetron operating in the 2π mode.

- Select *outB_2* for *Graph 1* and *Graph 2*
- Click *Fourier Amplitudes (dB)* to the left of one of the plots in the *Visualization Results* pane
- Zoom in on the maximum of this plot to determine the approximate resonance mode frequencies

The resulting plot will resemble Fig. 4.50. If the simulation has been run for long enough, the peak at 4.6 GHz, which corresponds to the 2π mode, should be the most prominent.


VSim - A6 Magnetron 2: Power

Data View: History

Plot Control

Graph 1
outB_2
Location: Window 1
Color: Red
Style: Solid

Graph 2
outB_2
Location: Window 2
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 3
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

outB_2
□ Fourier Amplitudes (dB) □ Log of Absolute Value ○ Zoom ● Navigate

outB_2
☑ Fourier Amplitudes (dB) □ Log of Absolute Value ○ Zoom ● Navigate

Visualize: READY Visualization available

Fig. 4.50: Plot of outB_2 vs. time and vs. frequency (Fourier transform).

Further Experiments

The power output and operating mode is affected by the E/B ratio and the geometry. The user could try adjusting the strength of the magnetic field and the geometry of the cathode, including the radius of the emitting region and the configuration of the end-caps, to see how this affects magnetron operation.

4.3.3 Gyrotron Mode (gyrotronMode.sdf)

Keywords:

gyrotron

Problem description

This VSimMD example illustrates a very high order mode, TE-22-6, propagating in a cylindrical waveguide, very near to the cutoff frequency, which is a common situation in a gyrotron. The example is intended to allow investigation of the axial phase and group velocity of such a mode, as a function of frequency, and to highlight the intricacies of simulating a mode that is propagating within a percent or two of its cutoff frequency.

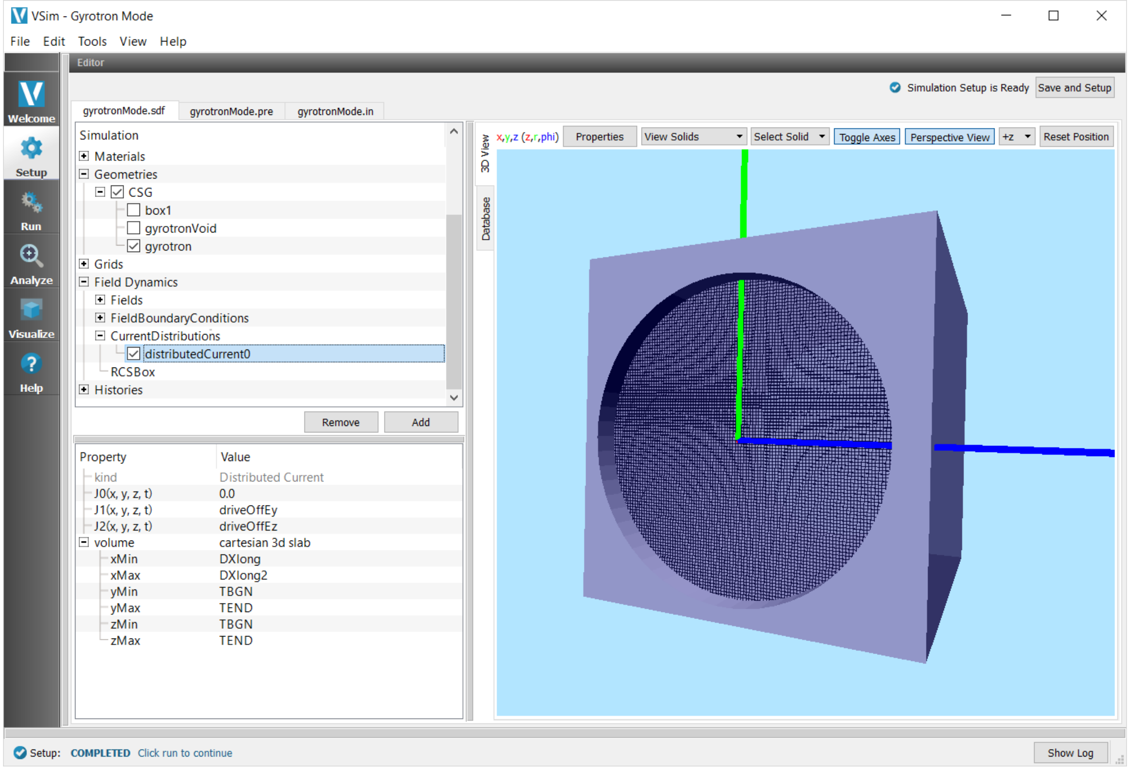
This simulation can be performed with a VSimMD license.

Opening the Simulation

The Gyrotron Mode example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Radiation Generation* option.
- Select "Gyrotron Mode" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is now shown with all the implemented physics and geometries, if applicable. See Fig. 4.51.



Property	Value
kind	Distributed Current
J0(x, y, z, t)	0.0
J1(x, y, z, t)	driveOffEy
J2(x, y, z, t)	driveOffEz
volume	cartesian 3d slab
xMin	DXlong
xMax	DXlong2
yMin	TBGN
yMax	TEND
zMin	TBGN
zMax	TEND

Setup: COMPLETED Click run to continue

Fig. 4.51: Setup Window for the Gyrotron Mode example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 4.51.

Simulation Properties

There are only two geometrical input parameters; the waveguide radius and length. The user may also control the excitation frequency, the duration of the simulation, and the nature of the excitation, specifically whether it is pulsed or continuous-wave. Additional exposed parameters include the grid sizes, and the tuning of the exiting wave boundary condition, which allows for more in-depth study with this example.

The excitation may be pulsed or continuous-wave, depending on the parameter, KEEP_DRIVE_ON.

Pulsed Simulation (KEEP_DRIVE_ON=0)

In this case, the wave is driven for half of the simulation duration, with a smooth turn-on / turn-off time window. Then, for the remaining half of the periods, the excitation propagates freely. The axial profile of the pulse will be very short, typically just one or two axial wavelengths. It will propagate slowly down the waveguide, as expected from the group velocity which is very small near cutoff. In the center of the pulse the TE-22-6 mode is preserved, but because this is a pulse, nearby modes in frequency are also present. One can observe a rich set of other mode patterns just a few grid planes away from the center of the pulse.

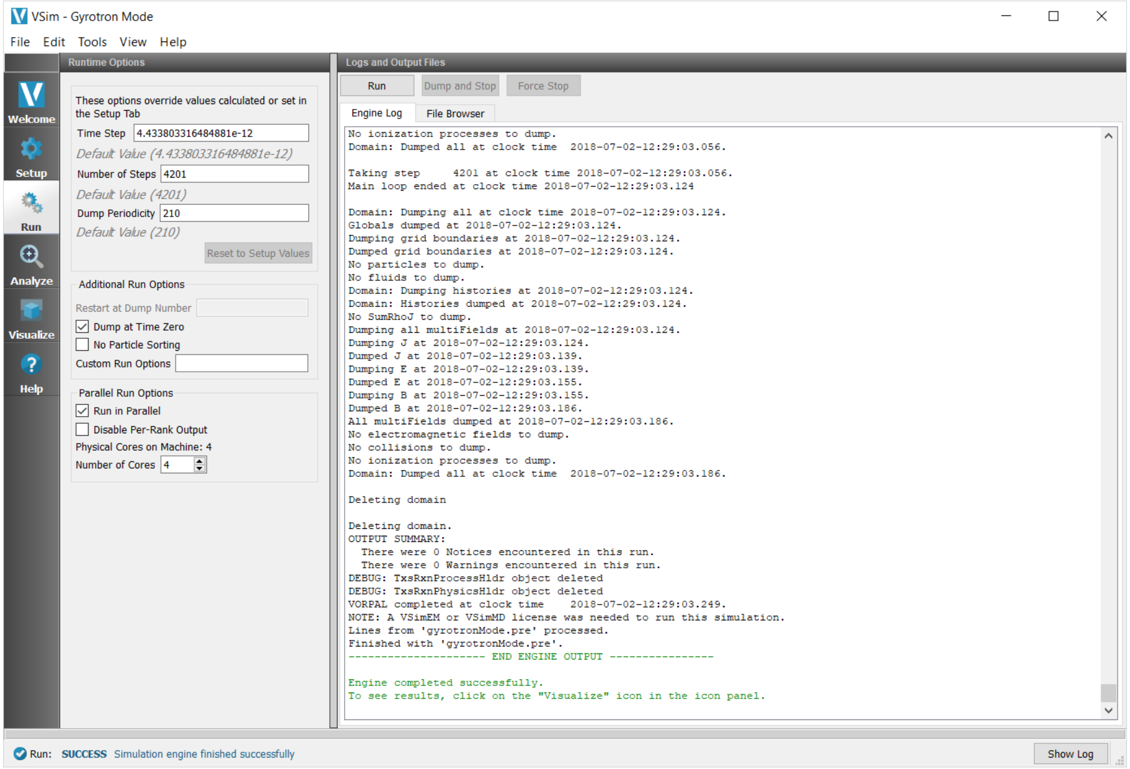
Continuous-Wave Simulation (KEEP_DRIVE_ON=1)

The drive may be kept on, instead of having it turn off halfway through the simulation. After an initial transient, this sets up a single TE-22-6 traveling wave mode pattern throughout the waveguide. This allows for accurate measurement of the axial wavenumber, beta, for the mode.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 4.52.



These options override values calculated or set in the Setup Tab

Time Step 4.433803316484881e-12

Default Value (4.433803316484881e-12)

Number of Steps 4201

Default Value (4201)

Dump Periodicity 210

Default Value (210)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-02-12:29:03.056.

Taking step     4201 at clock time 2018-07-02-12:29:03.056.
Main loop ended at clock time 2018-07-02-12:29:03.124

Domain: Dumping all at clock time 2018-07-02-12:29:03.124.
Globals dumped at 2018-07-02-12:29:03.124.
Dumping grid boundaries at 2018-07-02-12:29:03.124.
Dumped grid boundaries at 2018-07-02-12:29:03.124.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-07-02-12:29:03.124.
Domain: Histories dumped at 2018-07-02-12:29:03.124.
No SumRhoJ to dump.
Dumping all multiFields at 2018-07-02-12:29:03.124.
Dumping J at 2018-07-02-12:29:03.124.
Dumped J at 2018-07-02-12:29:03.139.
Dumping E at 2018-07-02-12:29:03.139.
Dumped E at 2018-07-02-12:29:03.155.
Dumping B at 2018-07-02-12:29:03.155.
Dumped B at 2018-07-02-12:29:03.186.
All multiFields dumped at 2018-07-02-12:29:03.186.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-02-12:29:03.186.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
DEBUG: TxsRxnProcessHldr object deleted
DEBUG: TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-07-02-12:29:03.249.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'gyrotronMode.pre' processed.
Finished with 'gyrotronMode.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

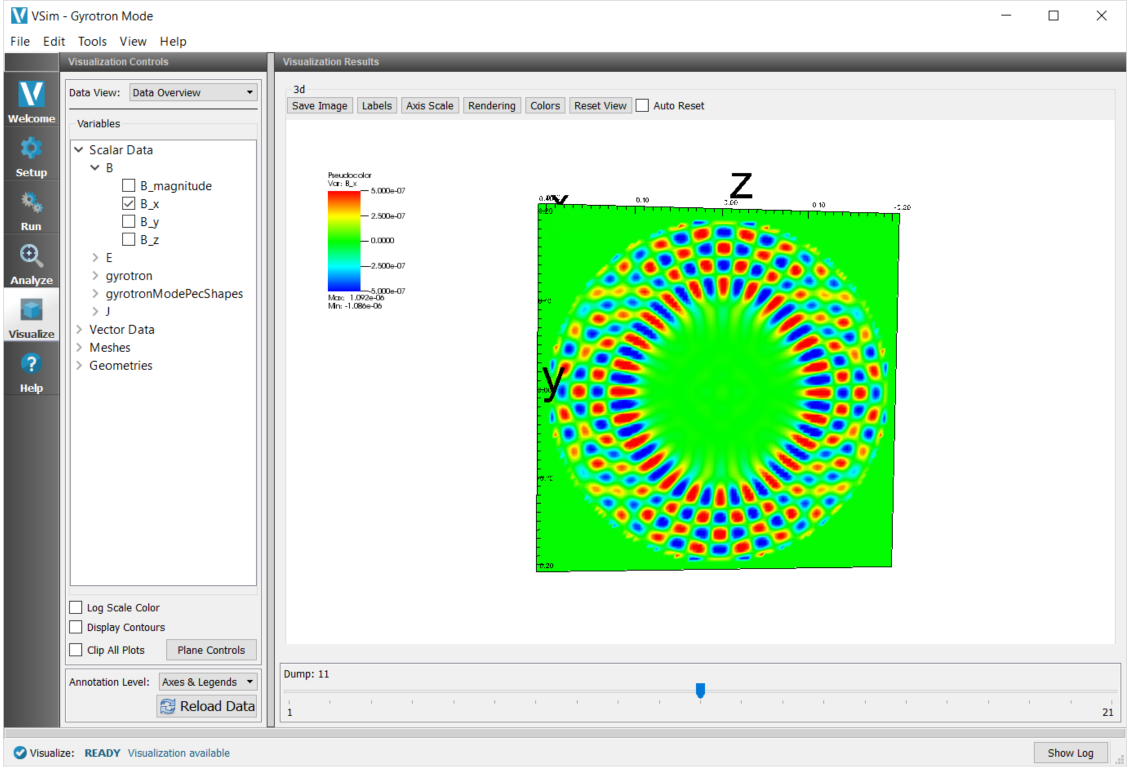
Run: **SUCCESS** Simulation engine finished successfully

Fig. 4.52: The Run Window at the end of execution.

Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.



VSim - Gyrotron Mode

Data View: Data Overview

☐ Log Scale Color
☐ Display Contours
☐ Clip All Plots

Annotation Level: Axes & Legends

☐ Auto Reset

Dump: 11

Visualize: READY Visualization available

Fig. 4.53: Illustration of the mode pattern, and propagation of the mode down the length of the tube.

The B_x field is the best component for looking at in this simulation, as shown in Fig. 4.53.

- Expand *Scalar Data*.
- Expand *B*
- Select *B_x*
- Using the cursor, grab the image and rotate it from right to left by 90 degrees.
- Move the dump slider to dump 11.

The initial parameters are selected so that the excitation frequency is just barely above cutoff. While the axial phase velocity is high in this case, the group velocity is quite low, and the simulation shows a narrow wavepacket slowly moving down the length of the tube, while remarkably still maintaining the very high order TE-22-6 pattern. Contamination of the pattern increases as the duration of the excitation is reduced, since more frequencies are brought into the transient. The user is encouraged to look at the mode pattern and contamination properties as frequency and duration are varied.

The TE-22-6 mode's cutoff frequency, for the suggested initial radius of 20 cm, is known analytically to be 10.8845 GHz, which derives from the value of the 6th root of the J_{22} bessel function, which is 45.624312. However, the user will note that the suggested initial drive frequency is below this, at 10.74 GHz, and yet the wave appears to propagate! This illustrates an important property of finite-difference dispersion, that in fact the speed of light is ever-so-slightly slower in the finite-difference-time-domain simulation than in reality. In most cases, this is hardly noticed, however, when operating this close to the cutoff frequency of a waveguide, this difference can be readily seen, as this example illustrates. The discrepancy between the discrete FDTD cutoff frequency and the analytic cutoff frequency, depends on the grid resolution of the wave, and in general decreases as δx^2, where δx is the grid size.

A very useful piece of information is the FDTD cutoff frequency. This may be found with a series of simulations, each at different drive frequencies, ω. The KEEP_DRIVE_ON parameter should be set to 1, so that the axial wavelength, β, can be measured from the field plots. A plot of ω^2 vs. β^2 should be essentially linear, with the intercept on the ω^2 axis being the FDTD cutoff frequency, ω^2_{cutoff} ($\omega^2 = \omega^2_{cutoff} + c^2\beta^2$), and with slope being the FDTD speed-of-light-squared. A spreadsheet showing this exercise for the suggested initial values of the example is shown below. The result of this study is that the FDTD cutoff frequency is actually 10.675 GHz, or 2% below the known analytical result, for the initial suggested grid resolution.

Further Experiments

The user is encouraged to repeat the simulations discussed in the previous section with a finer resolution, to see how the FDTD cutoff frequency approaches the analytic result as resolution improves.

The detailed TE-22-6 mode pattern is very carefully crafted using polynomial fitting functions, and is introduced into the axial magnetic field, B_x, at the left side of the simulation. There is no direct option to use a different mode, although the user may attempt to edit the detail of the input to do so.

Finally, a boundary condition tuning parameter, VPHASE_PORT, is offered to allow the user to experiment with tuning of the outgoing wave boundary condition in this near cutoff scenario. In this circumstance, the optimal phase velocity may be 5 to 10 times the speed of light.

An additional exposed user parameter, FREQ_CUTOFF, is offered, and may be used to store the value derived from the simulations discussed in the previous section. By default, this parameter is not used. However the user may look into the detail of input file, and notice a comment line that indicates how this parameter might be used to set the value of VPHASE_PORT more accurately.

4.3.4 Helix Traveling Wave Tube 1: Dispersion (helixTwt1Dispersion.sdf)

Keywords:

`Helix TWT Dispersion Analysis Run`

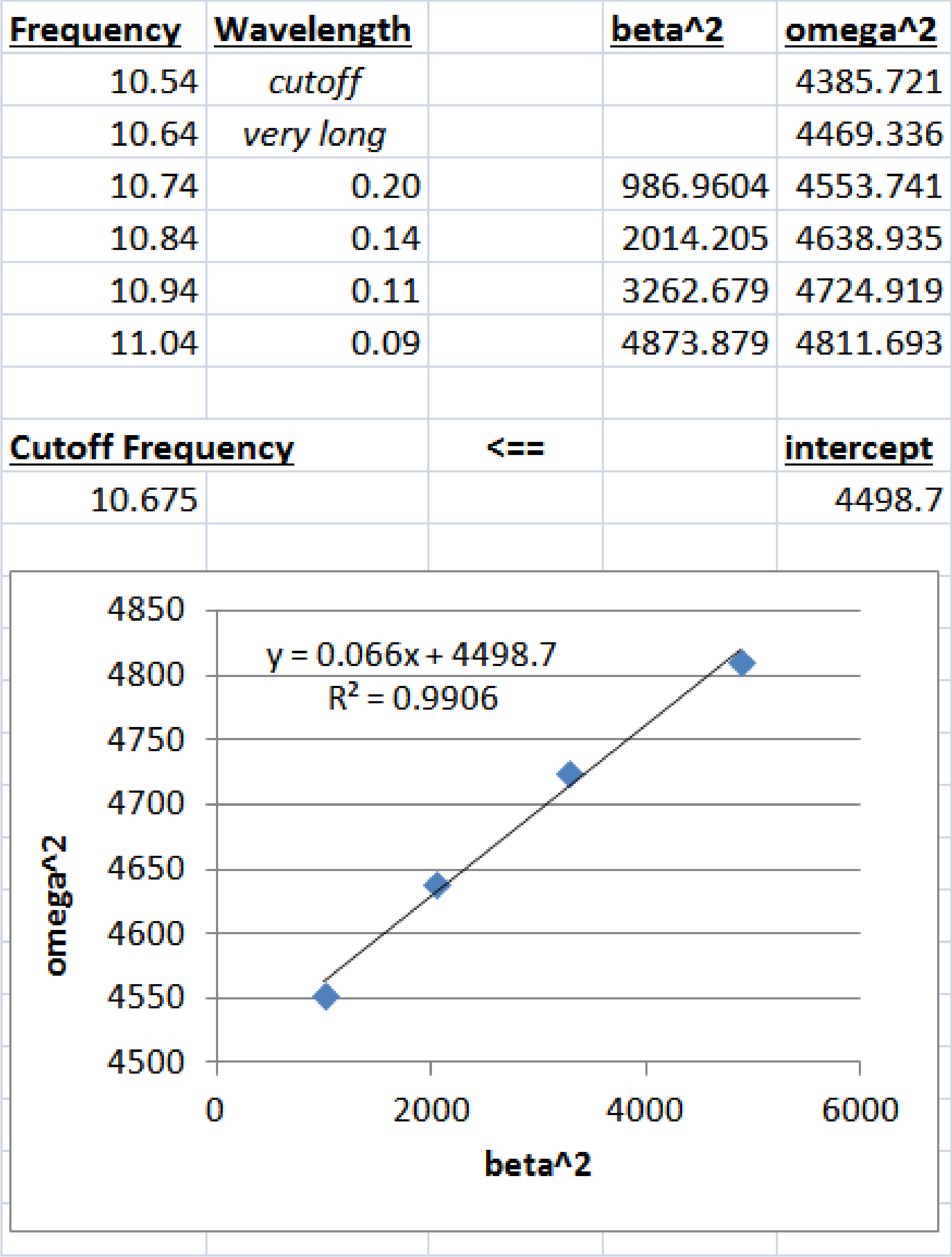
Problem description

This VSimMD example is one of a set of simulations showing different calculations to aid the design of a helix traveling wave tube (TWT) in three dimensions. The 100-turn helix with end feeds is imported from a CAD file, but all other geometrical parts are created with the Constructive Solid Geometry (CSG) capabilities within VSimComposer. The dependence of the geometries on the constants and parameters will be discussed in *Helix Traveling Wave Tube 2: Impedance and Attenuation (helixTwt2ImpedAtten.sdf)*.

This simulation addresses the dispersion analysis of the tube and as such runs with a grid covering a reduced number of helix turns. An impulse signal is excited between the helix and the body tube (which would be the vacuum interface in a real device) and periodic boundary conditions are enforced at the two ends. The tube is allowed to run for sufficiently long to observe the behaviour at relatively low frequencies. We are able to recover the phase velocity of the wave on the helix (and so the structure/RF curve on an omega beta diagram) from this simulation. It differs from the other simulations of helix TWT in that the simulated region contains no coaxial coupler. As well the attenuator is outside the simulated region. For other studies of the TWT, see

- *Helix Traveling Wave Tube 2: Impedance and Attenuation (helixTwt2ImpedAtten.sdf)*
- *Helix Traveling Wave Tube 3: Power Run (helixTwt3PowerRun.sdf)*

This simulation can be performed with a VSimMD or VSimPD license.



Frequency	Wavelength		beta^2	omega^2
10.54	*cutoff*			4385.721
10.64	*very long*			4469.336
10.74	0.20		986.9604	4553.741
10.84	0.14		2014.205	4638.935
10.94	0.11		3262.679	4724.919
11.04	0.09		4873.879	4811.693
Cutoff Frequency		<==		intercept
10.675				4498.7

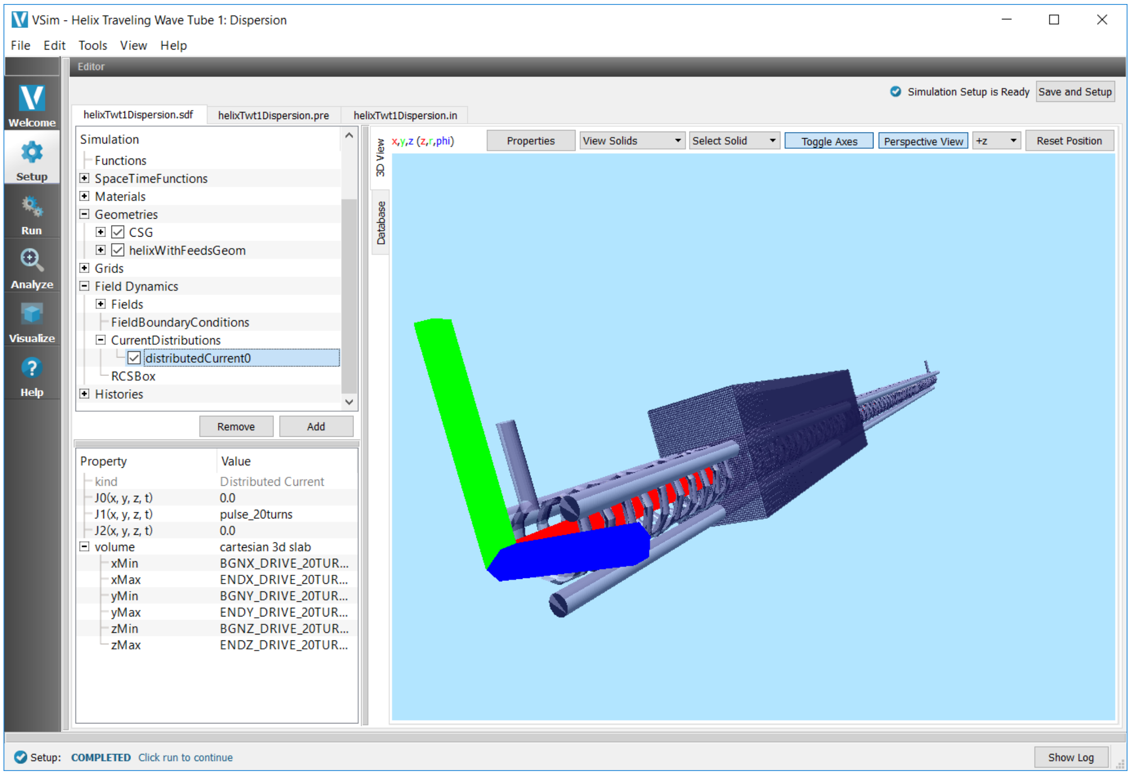
Fig. 4.54: Computing the FDTD cutoff frequency of the TE-22-6 mode.

Opening the Simulation

The Helix TWT example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Radiation Generation* option.
- Select "Helix Traveling Wave Tube 1: Dispersion" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 4.55. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



VSim - Helix Traveling Wave Tube 1: Dispersion

Simulation Setup is Ready

helixTwt1Dispersion.sdf | helixTwt1Dispersion.pre | helixTwt1Dispersion.in

- Simulation
  - Functions
  - ⊞ SpaceTimeFunctions
  - ⊞ Materials
  - ⊟ Geometries
    - ⊞ ☑ CSG
    - ⊞ ☑ helixWithFeedsGeom
  - ⊞ Grids
  - ⊟ Field Dynamics
    - ⊞ Fields
    - FieldBoundaryConditions
    - ⊟ CurrentDistributions
      - ☑ distributedCurrent0
    - RCSBox
  - ⊞ Histories

Property	Value
kind	Distributed Current
J0(x, y, z, t)	0.0
J1(x, y, z, t)	pulse_20turns
J2(x, y, z, t)	0.0
⊟ volume	cartesian 3d slab
xMin	BGNX_DRIVE_20TUR...
xMax	ENDX_DRIVE_20TUR...
yMin	BGNY_DRIVE_20TUR...
yMax	ENDY_DRIVE_20TUR...
zMin	BGNZ_DRIVE_20TUR...
zMax	ENDZ_DRIVE_20TUR...

Setup: COMPLETED Click run to continue

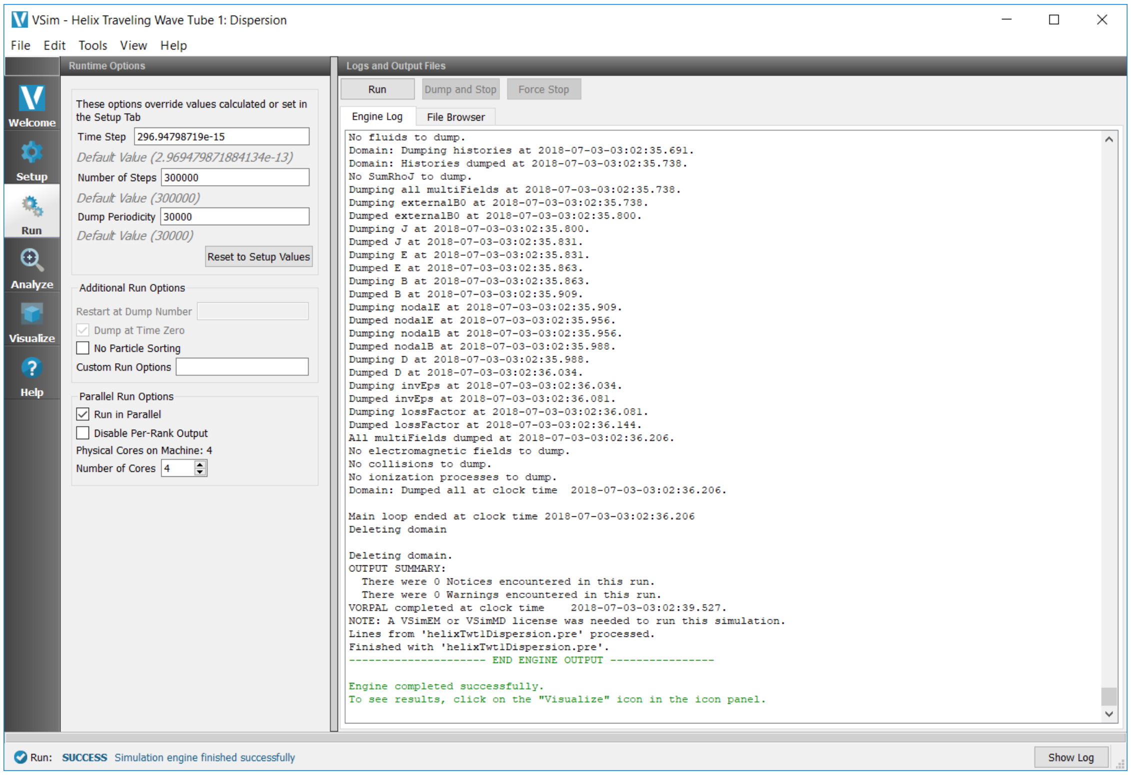
Fig. 4.55: Setup Window for the Helix TWT example.

The geometry of the helix can be made more visible by changing the default "opacity" of the surfaces in the Setup Window, as shown in figure Fig. 4.55. This setting is made available by clicking on the "Properties" button.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 4.56.



These options override values calculated or set in the Setup Tab

Time Step 296.94798719e-15

Default Value (2.969479871884134e-13)

Number of Steps 300000

Default Value (300000)

Dump Periodicity 30000

Default Value (30000)

Reset to Setup Values

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
No fluids to dump.
Domain: Dumping histories at 2018-07-03-03:02:35.691.
Domain: Histories dumped at 2018-07-03-03:02:35.738.
No SumRhoJ to dump.
Dumping all multiFields at 2018-07-03-03:02:35.738.
Dumping externalB0 at 2018-07-03-03:02:35.738.
Dumped externalB0 at 2018-07-03-03:02:35.800.
Dumping J at 2018-07-03-03:02:35.800.
Dumped J at 2018-07-03-03:02:35.831.
Dumping E at 2018-07-03-03:02:35.831.
Dumped E at 2018-07-03-03:02:35.863.
Dumping B at 2018-07-03-03:02:35.863.
Dumped B at 2018-07-03-03:02:35.909.
Dumping nodalE at 2018-07-03-03:02:35.909.
Dumped nodalE at 2018-07-03-03:02:35.956.
Dumping nodalB at 2018-07-03-03:02:35.956.
Dumped nodalB at 2018-07-03-03:02:35.988.
Dumping D at 2018-07-03-03:02:35.988.
Dumped D at 2018-07-03-03:02:36.034.
Dumping invEps at 2018-07-03-03:02:36.034.
Dumped invEps at 2018-07-03-03:02:36.081.
Dumping lossFactor at 2018-07-03-03:02:36.081.
Dumped lossFactor at 2018-07-03-03:02:36.144.
All multiFields dumped at 2018-07-03-03:02:36.206.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-03-03:02:36.206.

Main loop ended at clock time 2018-07-03-03:02:36.206
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-07-03-03:02:39.527.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'helixTwt1Dispersion.pre' processed.
Finished with 'helixTwt1Dispersion.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.56: The Run Window at the end of execution.

The simulation allows a frequency domain analysis to be performed. In order to resolve low frequency signals, a large number of steps is required. Increasing the number of steps further may help to improve the frequency domain resolution, especially at the low frequency end of the spectrum. The Dump Periodicity may be increased to reduce the number of data dumps and save space at the expense of being able to view up to date simulation data while it runs.

Visualizing the results

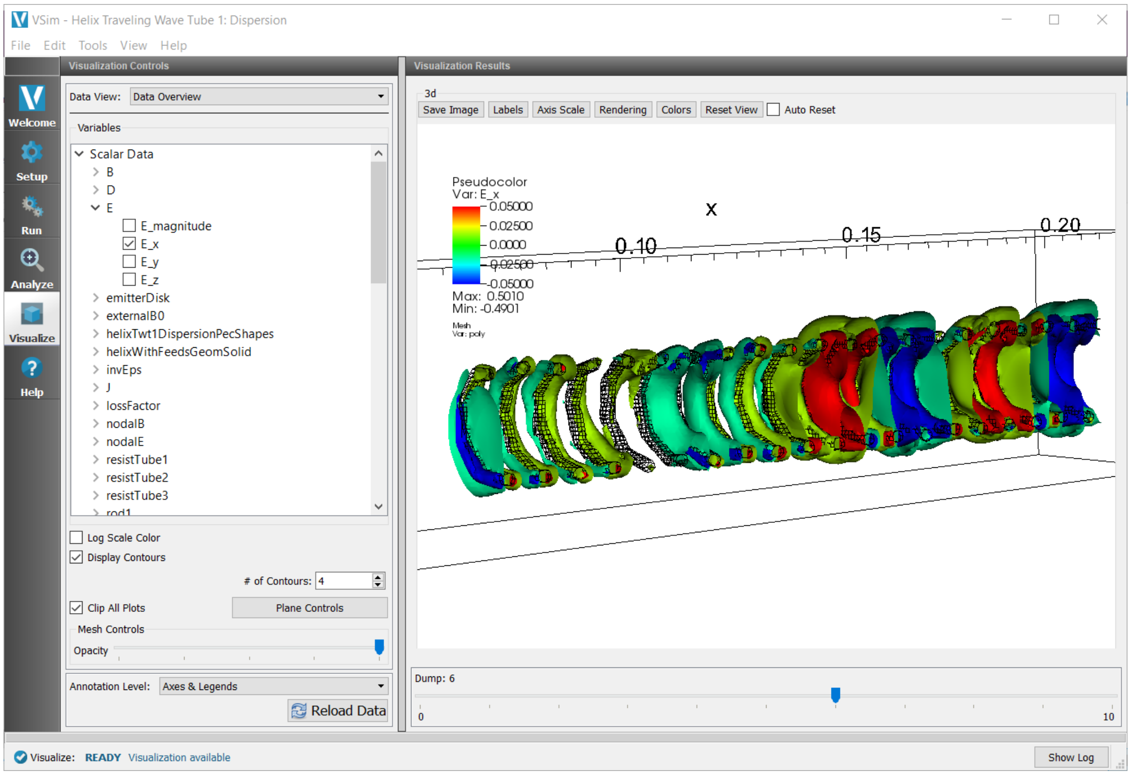
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To see the fields inside the tube as shown in Fig. 4.57, do the following:

- Expand *E*
- Select *E_x*
- Expand *Geometries*
- Select *poly (helixTwtWithFeeds)*
- Select *Clip All Plots*
- Select *Colors*
- Check *Fix Minimum* and *Fix Maximum* and set to -.05 and 0.05 respectively
- Move the dump slider forward in time to see the evolution
- Click and drag to rotate the image
- Select *Display Contours* and set # of contours to 4.

The wave travels along the helix, and the strongest fields occur between turns. The individual modes are not separated out in this case.



Data View: Data Overview

☐ Log Scale Color

☑ Display Contours

# of Contours: 4

☑ Clip All Plots

Annotation Level: Axes & Legends

Dump: 6

Visualize: READY Visualization available

Fig. 4.57: Longitudinal field, E_x, for the dispersion run at time step 120,000.

The *History* records can be used to calculate the dispersion curve:

- Set EonAxisA_0 under Graph 1
- Set EonAxisA_1 under Graph 2
- Under Graph 3 and Graph 4 change the dataset to <*None*>
- Check "Fourier Amplitudes (dB)" button at top left of plot
- Select Zoom radio button
- Using the *Limits* button above the plot set both to have limits of 0 to 2e9.

The result is shown in Fig. 4.58 (to which we have added vertical measuring lines using an external image editing software). A series of peaks can be seen. The mode frequencies correspond to the maxima of this plot. Having more than one history is important as due to mode variation in space, one history may pick up modes another misses and vice-versa.

Record the frequencies at which these peaks occur, eg in a spreadsheet. With the view mode switched from *Zoom* to *Navigate*, a wheel mouse can be scrolled up and down to zoom in and out. This may expedite the process of collecting the data. Or, as we have done, one can add vertical measuring lines after opening the image in some external software.

The first seven frequencies from the 300000 step simulation are listed in the table below. To determine the phase velocity v_n for each mode frequency, first note that the wave number for the n-th mode is given by

$$k_n = \frac{2\pi}{L} n$$

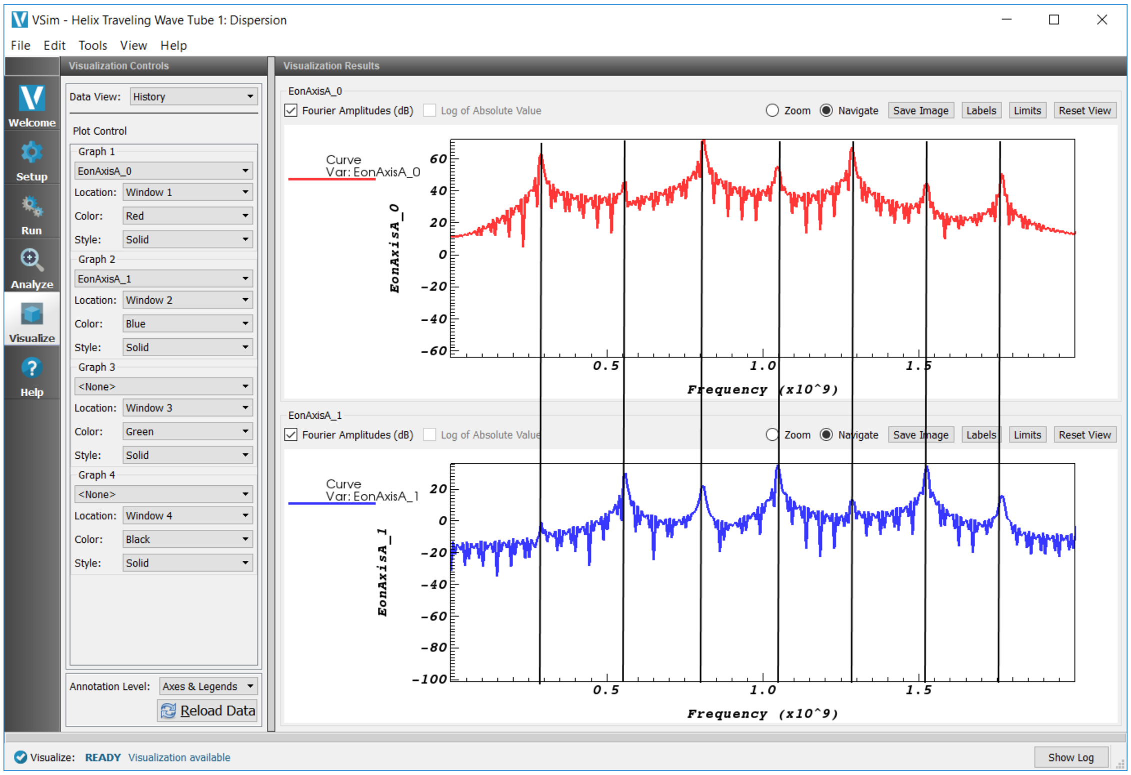
where L is the length of the helix TWT. The phase velocity for the n-th mode is then

$$\frac{v_n}{c} = \frac{2\pi f_n}{ck_n} = \frac{Lf_n}{cn}$$

where f_n is the frequency of the n-th mode and c is the speed of light. The first 10 mode phase velocities are listed in the table below for L = 15 cm. For large frequencies, we should expect the phase velocity to approach the ratio of the helix pitch (0.75 cm) to the circumference (6.28 cm), or 0.119.

n	frequency (GHz)	phase velocity (c)
1	0.285	0.1425
2	0.555	0.1387
3	0.810	0.1350
4	1.048	0.1310
5	1.285	0.1285
6	1.520	0.1267
7	1.770	0.1264

Higher resolution and longer duration simulation could be used to better measure the frequencies and, hence, determine the phase velocity. Even more precise frequencies could be obtained by the Filter Diagonalization Method.



VSim - Helix Traveling Wave Tube 1: Dispersion

Data View: History

Plot Control

Graph 1: EonAxisA_0, Location: Window 1, Color: Red, Style: Solid

Graph 2: EonAxisA_1, Location: Window 2, Color: Blue, Style: Solid

Graph 3: <None>, Location: Window 3, Color: Green, Style: Solid

Graph 4: <None>, Location: Window 4, Color: Black, Style: Solid

Annotation Level: Axes & Legends

EonAxisA_0: ☑ Fourier Amplitudes (dB) □ Log of Absolute Value ○ Zoom ● Navigate

EonAxisA_1: ☑ Fourier Amplitudes (dB) □ Log of Absolute Value ○ Zoom ● Navigate

Visualize: READY Visualization available

Fig. 4.58: Fourier transform of various histories after 300000 timesteps.

Further tests

The axial phase velocity is chosen to be synchronous with the beam. Adjust the helix parameters (in an external CAD editor) and observe the changes to the phase velocity.

Restarting after the default 300000 steps allows more accurate definition of the frequencies.

Use the Filter Diagonalization Method to get the frequencies more precisely.

4.3.5 Helix Traveling Wave Tube 2: Impedance and Attenuation (helixTwt2ImpedAtten.sdf)

Keywords:

`Helix TWT Impedance and Attenuation Run`

Problem description

This VSimMD example is one of a set of simulations showing different calculations to aid the design of a helix traveling wave tube (TWT) in three dimensions. The 100-turn helix with end feeds is imported from a CAD file, but all other geometrical parts are created with the Constructive Solid Geometry (CSG) capabilities within VSimComposer.

An input signal is sent into a short section of coaxial input waveguide and a similar section of coaxial waveguide at the opposite end of the tube provides an output power port. The geometry includes three dielectric support rods, each clad by a section of resistive tubing for attenuation. The interaction impedance run enables the user to calculate the transverse impedance and Pierce interaction impedance of the helix TWT. The transverse impedance is relevant for impedance matching at the input and output coaxial ports, and the Pierce interaction impedance is related to the interaction of the particles with the signal, and thus the signal gain.

The user may wish to run this simulation type multiple times, varying parameters such as the coax radius and dielectric permittivity, in order to result in a design with the correct impedance parameters.

Related simulations:

- *Helix Traveling Wave Tube 1: Dispersion (helixTwt1Dispersion.sdf)*
- *Helix Traveling Wave Tube 3: Power Run (helixTwt3PowerRun.sdf)*

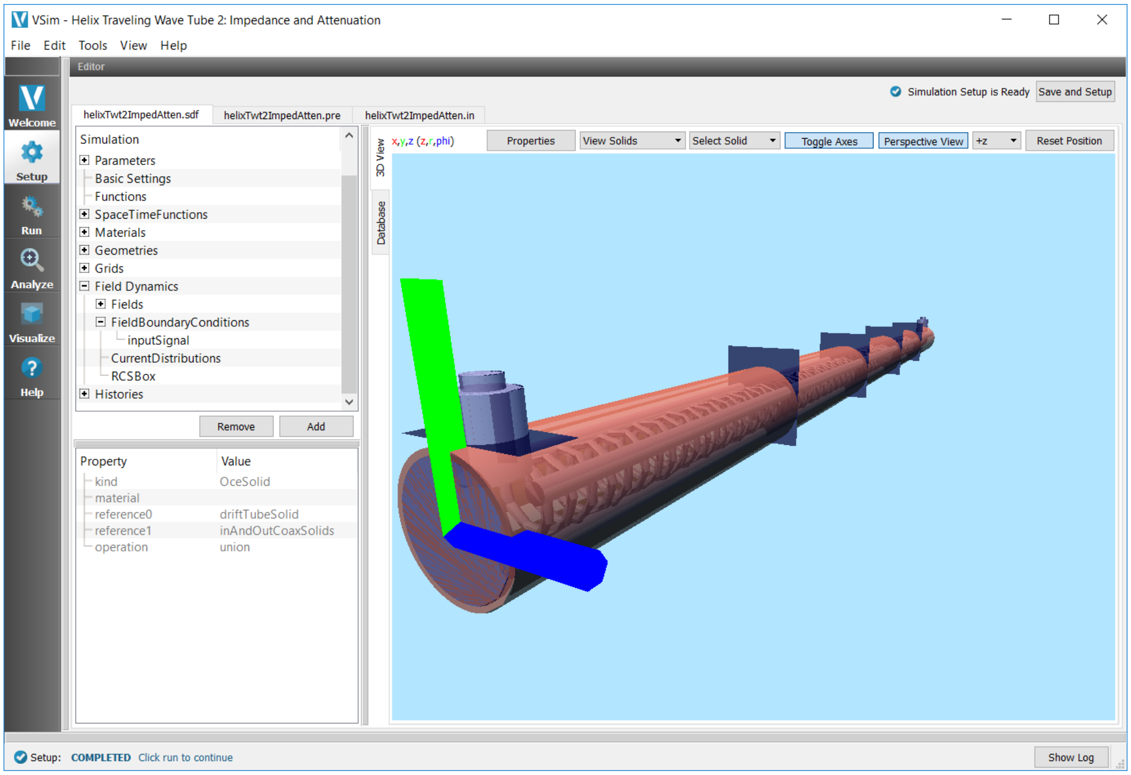
This simulation can be performed with a VSimMD or VSimPD license.

Opening the Simulation

The Helix TWT Impedance and Attenuation example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Radiation Generation* option.
- Select “Helix Traveling Wave Tube 2: Impedance and Attenuation” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 4.59, with the *Elements Tree* in the upper center, and the *Property Editor* in the lower center. The right pane shows a 3D view of the geometry as well as the grid, if its visibility has been activated (which it was not when this image was captured).



Property	Value
kind	OceSolid
material	
reference0	driftTubeSolid
reference1	inAndOutCoaxSolids
operation	union

Setup: COMPLETED Click run to continue

Fig. 4.59: Setup Window for the Helix TWT impedance and attenuation example.

Geometry details

The various geometrical objects can all be seen in the tree by pulling down the bar separating the Elements Tree from the Property Editor and then expanding *Geometries*, *CSG*, and *Grids*. Make sure *tube* and *Grid* are unclicked, *helixWithFeedsGeom* is clicked, and that the *Toggle Axes* button is set to remove the axes from the view. This allows one to see the interior geometrical objects, including the incoming feed, the dielectric support rods, the resistive tubes in the center, the particle emission disk at the left, and various planes where measurements are taken. The geometry is shown in Fig. 4.60.

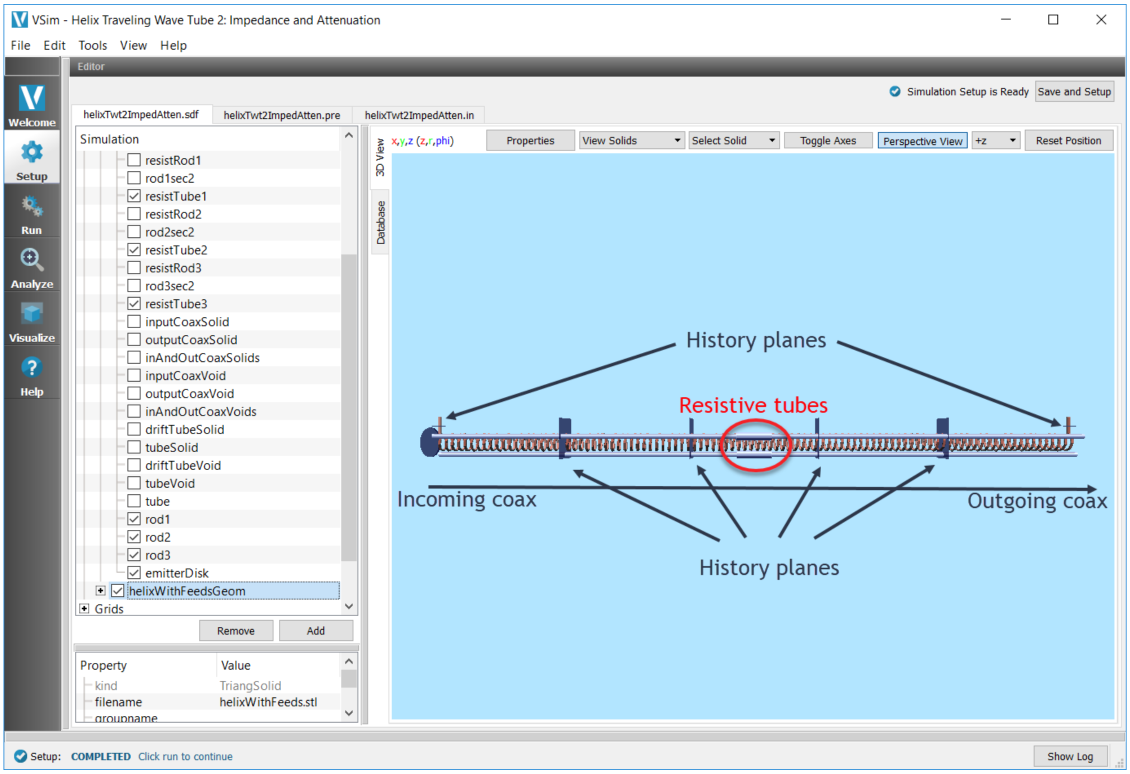
Constants and Parameters

Pulling the separator bar between the Elements Tree and the Property Editor and opening the Constants part of the tree gives the view shown in Fig. 4.61.

There are three types of constants. The first set of constants, from `PI` through `ELECMASSEV` are not changeable by the user. These are the various mathematical and physical constants that the simulation will use. The second set of constants are those with `HELIX` in the name. These must correspond to the helix geometry, the beginning, mid-radius, wire radius, pitch, and number of turns of the helix. These cannot be set arbitrarily, as the helix was imported as an STL file. Instead they must be set to match the imported helix parameters. The remaining constants define fundamental geometry quantities, such as where the tube begins and its radius, other physical simulation parameters, such as the wave frequency, and numerical parameters, such as the number of cells in each direction.

Moving the scroll bar and opening the Parameters part of the tree shows the Parameters, values that come from arithmetically combining constants and other parameters. This is shown in Fig. 4.62.

As an example, `LENGTH_HELIX` is shown. The expression shows that this is the number of helix turns times the helix pitch. It also shows the value. Of course, the expression is editable, while the value is not.



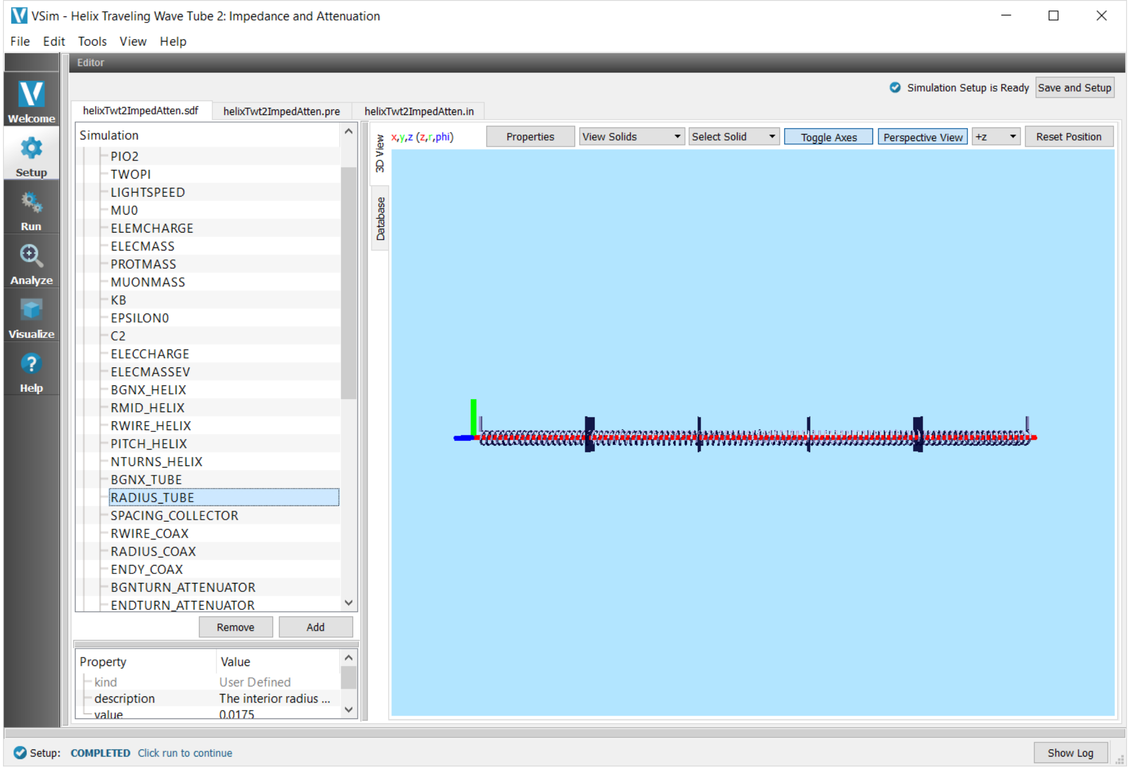
Simulation
- □ resistRod1
- □ rod1sec2
- ☑ resistTube1
- □ resistRod2
- □ rod2sec2
- ☑ resistTube2
- □ resistRod3
- □ rod3sec2
- ☑ resistTube3
- □ inputCoaxSolid
- □ outputCoaxSolid
- □ inAndOutCoaxSolids
- □ inputCoaxVoid
- □ outputCoaxVoid
- □ inAndOutCoaxVoids
- □ driftTubeSolid
- □ tubeSolid
- □ driftTubeVoid
- □ tubeVoid
- □ tube
- ☑ rod1
- ☑ rod2
- ☑ rod3
- ☑ emitterDisk
- ☑ helixWithFeedsGeom

Grids

Property	Value
kind	TriangSolid
filename	helixWithFeeds.stl
groupname	

Setup: COMPLETED Click run to continue

Fig. 4.60: Interior geometry for the Helix TWT impedance and attenuation example.

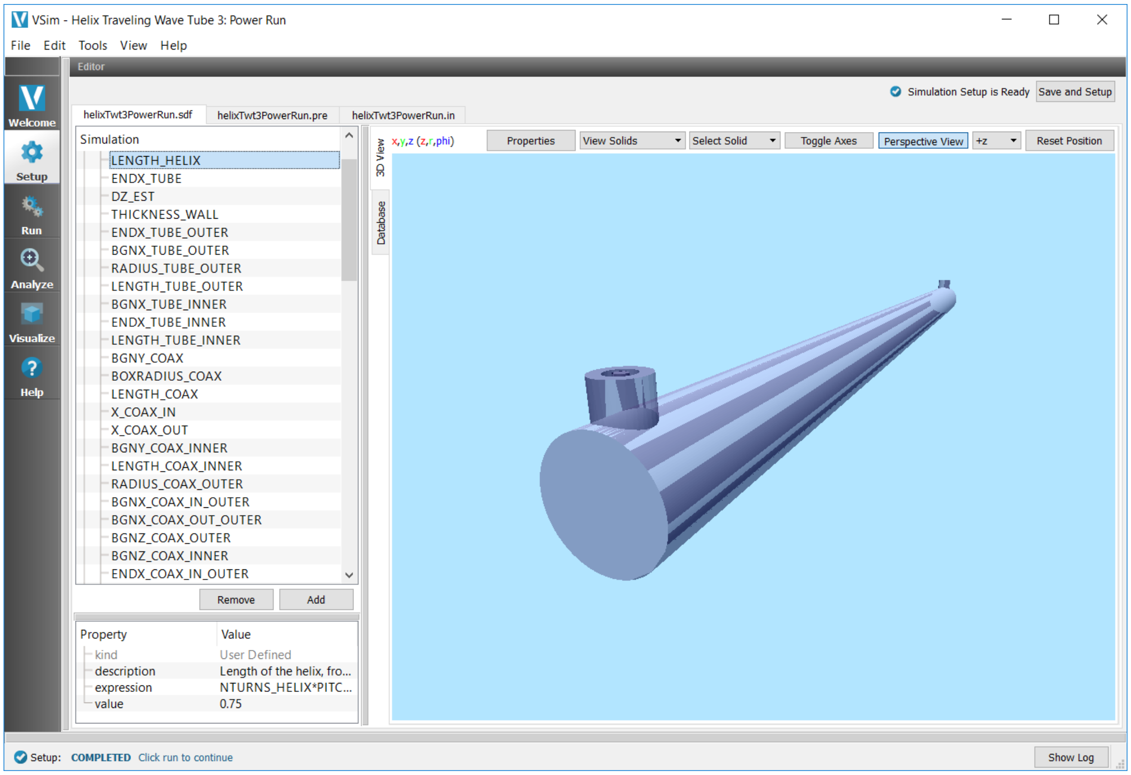


Simulation
- PI02
- TWOPI
- LIGHTSPEED
- MU0
- ELEMCHARGE
- ELECMASS
- PROTMASS
- MUONMASS
- KB
- EPSILON0
- C2
- ELECCHARGE
- ELECMASSEV
- BGNX_HELIX
- RMID_HELIX
- RWIRE_HELIX
- PITCH_HELIX
- NTURNS_HELIX
- BGNX_TUBE
- RADIUS_TUBE
- SPACING_COLLECTOR
- RWIRE_COAX
- RADIUS_COAX
- ENDY_COAX
- BGNTURN_ATTENUATOR
- ENDTURN_ATTENUATOR

Property	Value
kind	User Defined
description	The interior radius ...
value	0.0175

Setup: COMPLETED Click run to continue

Fig. 4.61: Constants for the Helix TWT impedance and attenuation example.



Simulation
- LENGTH_HELIX
- ENDX_TUBE
- DZ_EST
- THICKNESS_WALL
- ENDX_TUBE_OUTER
- BGNX_TUBE_OUTER
- RADIUS_TUBE_OUTER
- LENGTH_TUBE_OUTER
- BGNX_TUBE_INNER
- ENDX_TUBE_INNER
- LENGTH_TUBE_INNER
- BGNY_COAX
- BOXRADIUS_COAX
- LENGTH_COAX
- X_COAX_IN
- X_COAX_OUT
- BGNY_COAX_INNER
- LENGTH_COAX_INNER
- RADIUS_COAX_OUTER
- BGNX_COAX_IN_OUTER
- BGNX_COAX_OUT_OUTER
- BGNZ_COAX_OUTER
- BGNZ_COAX_INNER
- ENDX_COAX_IN_OUTER

Property	Value
kind	User Defined
description	Length of the helix, fro...
expression	NTURNS_HELIX*PITC...
value	0.75

Setup: COMPLETED Click run to continue

Fig. 4.62: Parameters for the Helix TWT impedance and attenuation example.

Both constants and parameters have a *description* field that allows the user to document the quantity.

Materials

To bring materials in the simulation, in the right pane, select the *Database* tab, select one or more materials (with ctrl-click) and hit the button *Add To Simulation*. The materials will then appear under *Materials* in the tree view. At this point one can change the properties of the materials including the name. In this example we imported *resistive damper*, changed its name to *LossyMaterial*, and changed its value of conductivity to *0.55*. This is shown in Fig. 4.63.

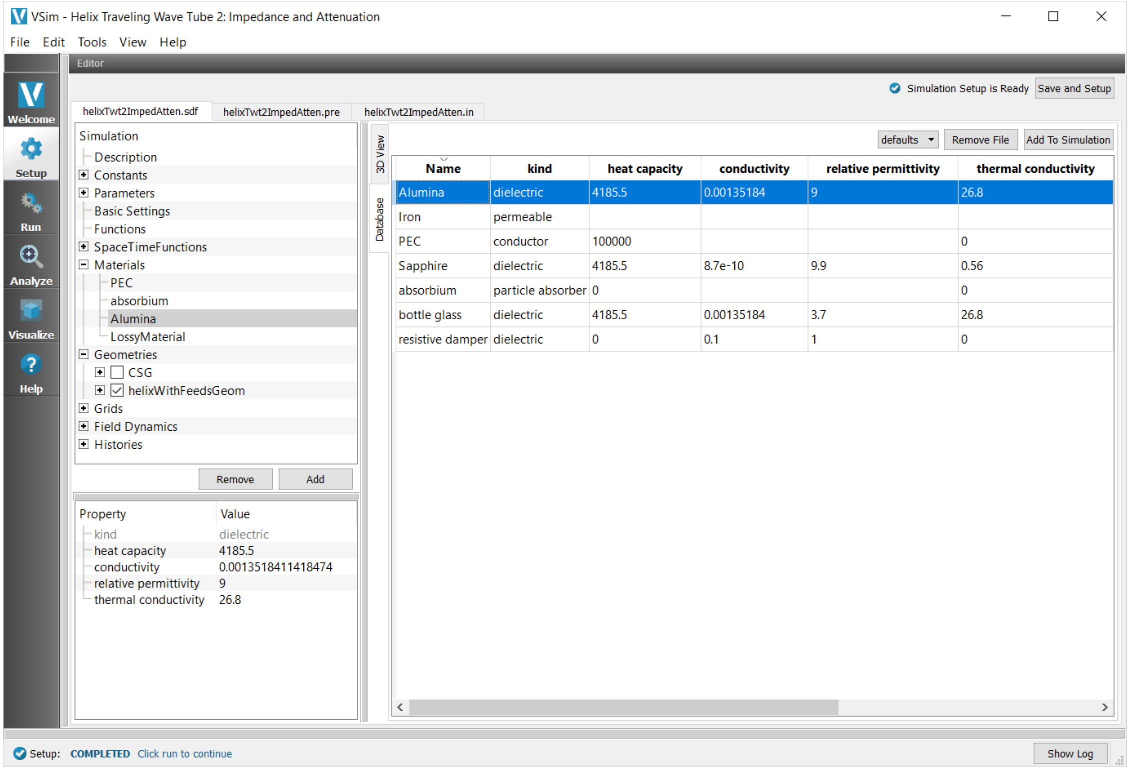
Once one has materials in the simulation, one can set the materials of any of the geometries. Click on the geometry, then in the Property Editor pane, double click on the material value. A context menu will allow you to set the material of the geometry to any material in the simulation.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.64.

The simulation to determine the impedance should run long for any mismatch at the outgoing boundary to stabilize. That is, the simulation must be run long enough for the electromagnetic wave to reach the far end of the tube and for any reflections to return some distance to the last history in the tube. This will take about 100,000 steps. On a 4-core machine, we have observed 0.23s/step, so this simulation will take about 7 hours. This simulation parallelizes well up



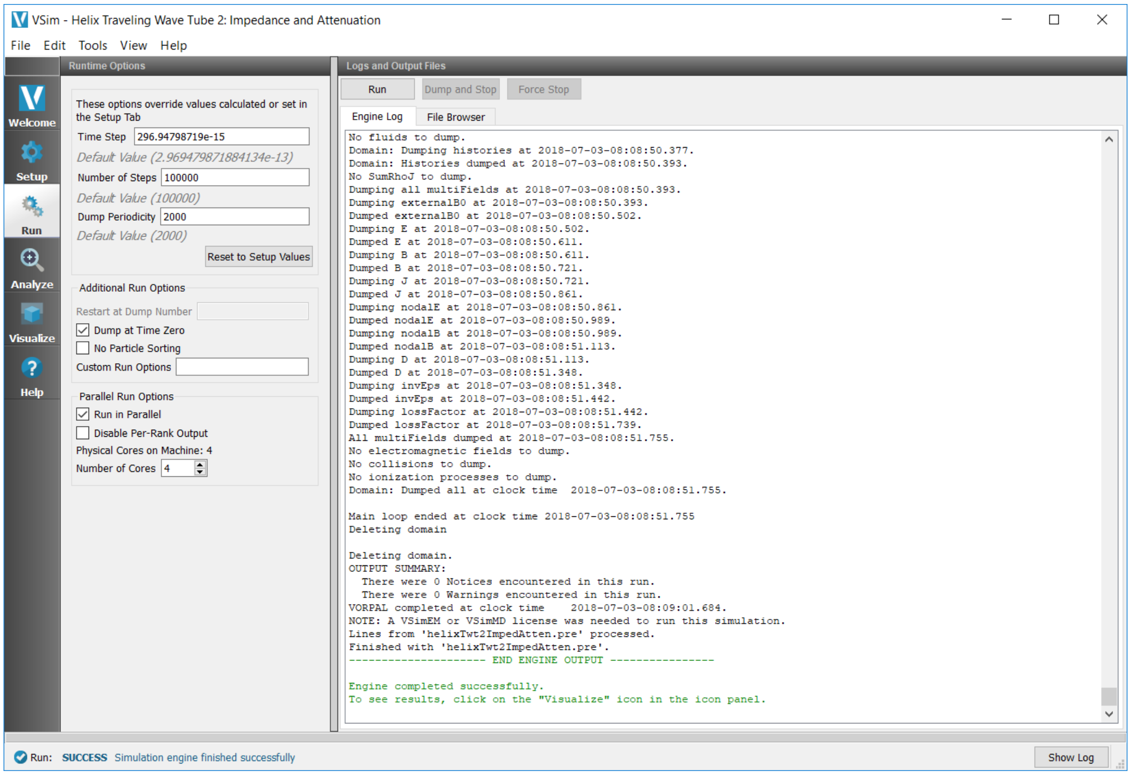
Simulation Setup is Ready

Name	kind	heat capacity	conductivity	relative permittivity	thermal conductivity
Alumina	dielectric	4185.5	0.00135184	9	26.8
Iron	permeable				
PEC	conductor	100000			0
Sapphire	dielectric	4185.5	8.7e-10	9.9	0.56
absorbium	particle absorber	0			0
bottle glass	dielectric	4185.5	0.00135184	3.7	26.8
resistive damper	dielectric	0	0.1	1	0

Property	Value
kind	dielectric
heat capacity	4185.5
conductivity	0.0013518411418474
relative permittivity	9
thermal conductivity	26.8

Setup: COMPLETED Click run to continue

Fig. 4.63: Materials for the Helix TWT impedance and attenuation example.



These options override values calculated or set in the Setup Tab

Time Step 296.94798719e-15

Default Value (2.969479871884134e-13)

Number of Steps 100000

Default Value (100000)

Dump Periodicity 2000

Default Value (2000)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
No fluids to dump.
Domain: Dumping histories at 2018-07-03-08:08:50.377.
Domain: Histories dumped at 2018-07-03-08:08:50.393.
No SumRhoJ to dump.
Dumping all multiFields at 2018-07-03-08:08:50.393.
Dumping externalB0 at 2018-07-03-08:08:50.393.
Dumped externalB0 at 2018-07-03-08:08:50.502.
Dumping E at 2018-07-03-08:08:50.502.
Dumped E at 2018-07-03-08:08:50.611.
Dumping B at 2018-07-03-08:08:50.611.
Dumped B at 2018-07-03-08:08:50.721.
Dumping J at 2018-07-03-08:08:50.721.
Dumped J at 2018-07-03-08:08:50.861.
Dumping nodalE at 2018-07-03-08:08:50.861.
Dumped nodalE at 2018-07-03-08:08:50.989.
Dumping nodalB at 2018-07-03-08:08:50.989.
Dumped nodalB at 2018-07-03-08:08:51.113.
Dumping D at 2018-07-03-08:08:51.113.
Dumped D at 2018-07-03-08:08:51.348.
Dumping invEps at 2018-07-03-08:08:51.348.
Dumped invEps at 2018-07-03-08:08:51.442.
Dumping lossFactor at 2018-07-03-08:08:51.442.
Dumped lossFactor at 2018-07-03-08:08:51.739.
All multiFields dumped at 2018-07-03-08:08:51.755.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-03-08:08:51.755.

Main loop ended at clock time 2018-07-03-08:08:51.755
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-07-03-08:09:01.684.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'helixTwt2ImpedAtten.pre' processed.
Finished with 'helixTwt2ImpedAtten.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.64: The Run Window at the end of execution.

to 16 cores, so with a sufficient license and workstation or cluster, one can bring this simulation time down to about 2 hours.

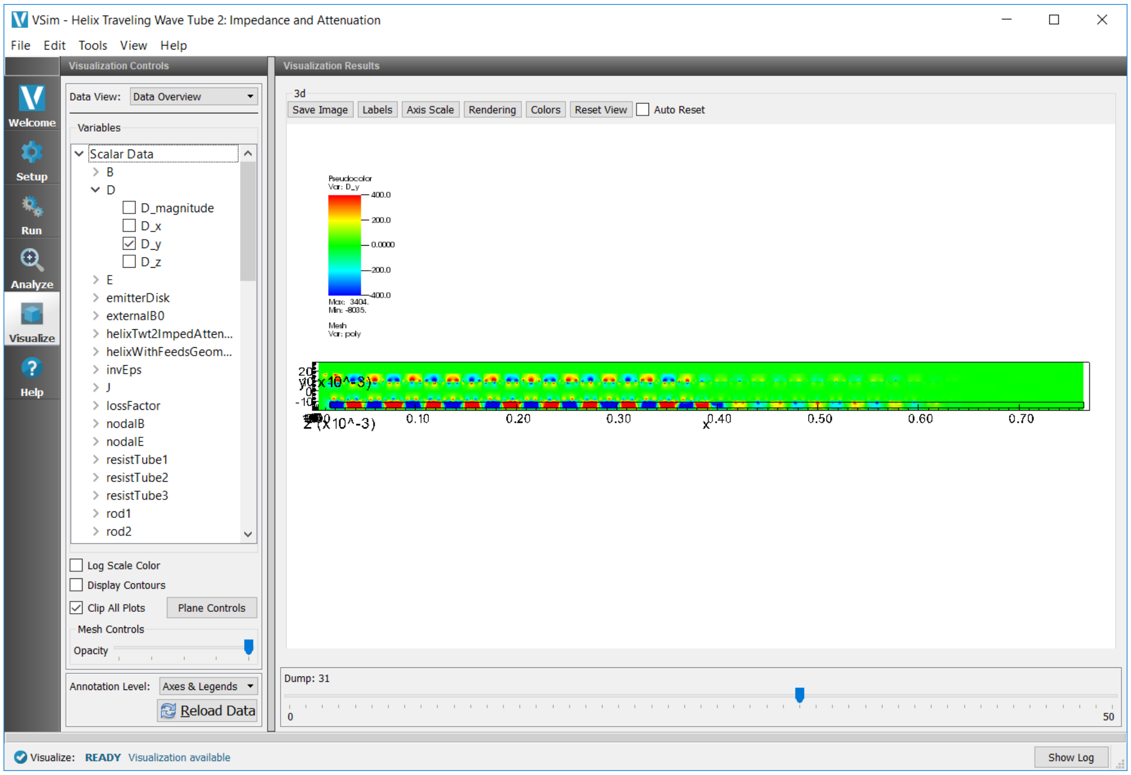
Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To see the fields inside the tube as shown in Fig. 4.65, continue as follows:

- Select Data View: *Data Overview*
- Expand *Scalar Data* then *D*, then select field *D_y*.
- Expand *Geometries* then select *poly (rod1)*.
- Check *Clip All Plots*, which cuts through the data at the $z = 0$ plane.
- Select the *Colors* option from the upper menu bar in the *Visualuization Results* pane.
- In the *Color Options* window: check *Fix Minimum* and set it to -400, then check *Fix Maximum*, set it to 400, and click *OK*.
- Move the dump slider forward in time to see the evolution.



VSim - Helix Traveling Wave Tube 2: Impedance and Attenuation

Data View: Data Overview

Variables

Scalar Data
- B
- D
  - □ D_magnitude
  - □ D_x
  - ☑ D_y
  - □ D_z
- E
- emitterDisk
- externalB0
- helixTwt2ImpedAtten...
- helixWithFeedsGeom...
- invEps
- J
- lossFactor
- nodalB
- nodalE
- resistTube1
- resistTube2
- resistTube3
- rod1
- rod2

□ Log Scale Color
□ Display Contours
☑ Clip All Plots

Annotation Level: Axes & Legends

3d

□ Auto Reset

Dump: 31

Visualize: READY Visualization available

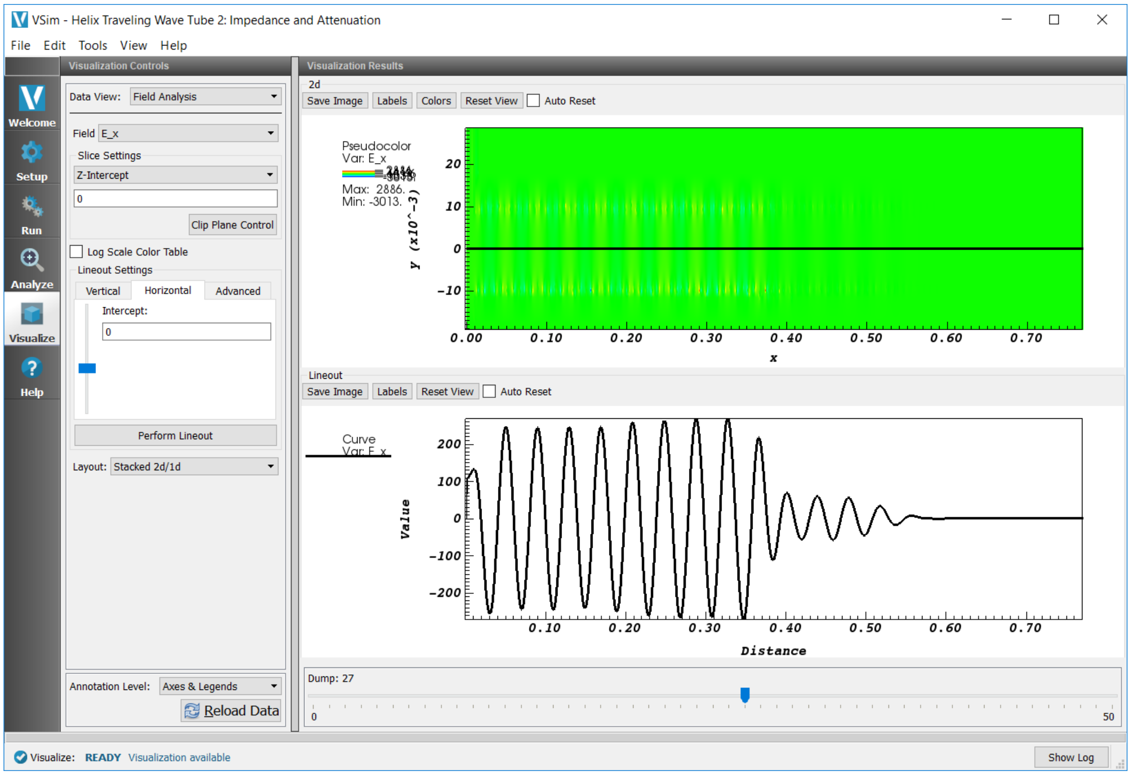
Fig. 4.65: The transverse displacement field, Dy, on the central x-y plane at dump 31.

This plot shows the transverse displacement field. Once can see that it is confined inside the tube (sanity check), and that it is most intense inside the dielectric rod at the bottom. The field is larger at the left, as it is just entering and propagating down the tube.

At any time one can leave this visualization pane to move back to the run pane to see how the simulation is progressing.

The longitudinal field inside the tube is shown in Fig. 4.66, which can be obtained by the steps:

- Select Data View: *Field Analysis*
- Select Field *E_x*.
- Select *Horizontal* under *Lineout Settings*, set the Intercept to `0`, and click *Perform Lineout*.
- Under *Layout* select *Stacked 2d/1d*
- Move the dump slider forward in time to see the evolution.



Data View: Field Analysis

Field: E_x

Slice Settings: Z-Intercept, 0

Lineout Settings: Horizontal, Intercept: 0

Layout: Stacked 2d/1d

Annotation Level: Axes & Legends

Dump: 27

Visualize: READY Visualization available

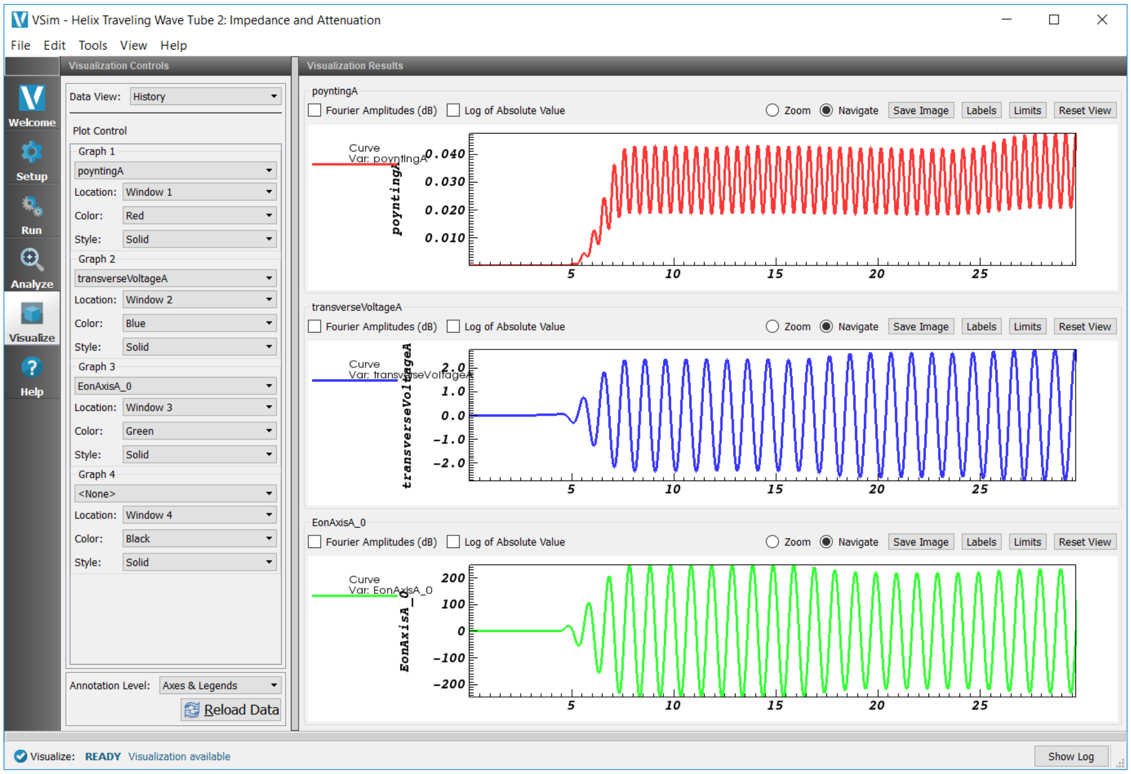
Fig. 4.66: The longitudinal field, Ex, on axis at dump 27, which is time step 54,000.

As seen in Fig. 4.66, the longitudinal field has dropped from about `240 V/m` to about `68 V/m` in the center of the simulation where the resistive tubes are. This corresponds to about `14 dB` of attenuation. The purpose of this attenuation is to have sufficient damping so that reflections coming back from the end to the beginning and then reflection again do not grow, as that would change the device into an oscillator, with energy growth that could destroy the system. If the *Helix Traveling Wave Tube 3: Power Run (helixTwt3PowerRun.sdf)* shows that this is happening, one can return to this run and increase the conductivity of the LossyMaterial or the length of the resistive tube (through `BGNTURN_ATTENUATOR` and `ENDTURN_ATTENUATOR`) to provide more attenuation.

Histories contain the time evolution of quantities defined in the input file. These can be seen by selecting the *Data View*, *History*. To determine various impedances we want particular histories obtained by the process:

- Select Data View: *History*
- Under Graph 1 select *poyntingA*
- Under Graph 2 select *transverseVoltageA*
- Under Graph 3 select *EonAxisA_0*
- Under Graph 4 select *<None>*

The result is shown in Fig. 4.67. The power through the plane was defined such that incoming is negative.



VSim - Helix Traveling Wave Tube 2: Impedance and Attenuation

Data View: History

Plot Control

Graph 1: poyntingA; Location: Window 1; Color: Red; Style: Solid

Graph 2: transverseVoltageA; Location: Window 2; Color: Blue; Style: Solid

Graph 3: EonAxisA_0; Location: Window 3; Color: Green; Style: Solid

Graph 4: <None>; Location: Window 4; Color: Black; Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

File Edit Tools View Help
Visualization Controls
Visualization Results
Welcome
Setup
Run
Analyze
Visualize
Help
Reload Data
Show Log
poyntingA
☐ Fourier Amplitudes (dB) ☐ Log of Absolute Value
○ Zoom ● Navigate Save Image Labels Limits Reset View
Curve
Var: poyntingA
poyntingA
0.040
0.030
0.020
0.010
5
10
15
20
25
transverseVoltageA
☐ Fourier Amplitudes (dB) ☐ Log of Absolute Value
○ Zoom ● Navigate Save Image Labels Limits Reset View
Curve
Var: transverseVoltageA
transverseVoltageA
2.0
1.0
0.0
−1.0
−2.0
5
10
15
20
25
EonAxisA_0
☐ Fourier Amplitudes (dB) ☐ Log of Absolute Value
○ Zoom ● Navigate Save Image Labels Limits Reset View
Curve
Var: EonAxisA_0
EonAxisA_0
200
100
0
−100
−200
5
10
15
20
25

Fig. 4.67: Poynting power (W), transverse voltage (V), and electric field (V/m), at Plane A, along the helix TWT as a function of time (s).

Impedance parameters of interest are the transverse impedance

$$Z_t = \frac{V_t^2}{2P}$$

and the Pierce interaction impedance

$$Z_p = \frac{E_x^2 \lambda^2}{8\pi^2 P}$$

where P is the poynting power (recorded in the history poyntingA), V_t is the transverse Voltage amplitude (recorded in the histories transverseVoltage), E_x is the electric field amplitude (recorded in the history EonAxis), and λ is the wavelength of the field along the helix TWT axis.

The histories show the graphs of these quantities. To get precise values for any of these, one can press the *Limits* button, which will pop up a window withe the precise values. First, the *X-Axis* limits show that the units are ns. Secondly, one needs to choose a consistent time period, where all amplitudes are roughly constants. The period $28ns < t < 32ns$ is chosen. One can now adjust the limits until the peaks line up with the limits. We want average poynting power. We find $P_{min} = 18.\ mW$ and $P_{max} = 44.\ mW$. Hence, $P_{av} = 31.\ mW$. During that same time interval we find $V_t = 2.4\ V$ and $E_x = 240\ V/m$.

Fig. 4.66 can be used to obtain the wavelength. One can see four wavelengths between $0.20m$ and $0.357m$. Therefore the wavelength is $(.357m - .20m)/4 = 0.039m$

We now compute

$$Z_t = \frac{2.4 * 2.4}{2 * .031} = 92.9\ \Omega$$

and the Pierce interaction impedance is

$$Z_p = \frac{240^2 \cdot 0.039^2}{8\pi 0.031} = 35.8\ \Omega.$$

Further Experiments

As noted above, one can change the attenuation by varying the conductivity of the resistive tubs or their length. For any given length, there is a maximum attainable attenuation, as there is no attenuation at zero conductivity (infinite resistance, i.e., vacuum) and none as well at infinite conductivity (metallic shielding). So if more than 14 dB attenuation is needed one can vary the conductivity, but a maximum will be observed, and if that is insufficient one will have to vary the rod length.

With additional computing resources, one could increase the grid resolution so that the resistive tube could be made thinner. As it is made thinner, one can go to greater conductivity without having the skin depth less than the resistive tube thickness.

The coupling is determined by the transverse impedance of the structure, which in turn depends on the capacitance provided by the rods. Varying the relative permittivity changes the transverse impedance.

4.3.6 Helix Traveling Wave Tube 3: Power Run (helixTwt3PowerRun.sdf)

Keywords:

`Helix TWT Power Run`

Problem description

This VSimMD example is the last of a set of simulations showing different calculations to aid the design of a helix traveling wave tube (TWT) in three dimensions. The 100-turn helix with end feeds is imported from a CAD file, but all other geometrical parts are created with the Constructive Solid Geometry (CSG) capabilities within VSimComposer.

An input signal is sent into a short section of coaxial input waveguide and a similar section of coaxial waveguide provides an output power port. The geometries in and constant parameters of this simulation are described in more detail in *Helix Traveling Wave Tube 2: Impedance and Attenuation (helixTwt2ImpedAtten.sdf)*. Electrons are injected at the left end of the tube. Gain can be computed from the ratio of voltages in the input and output waveguides.

The user may wish to run this simulation type multiple times, varying parameters such as the input signal power and the electron energy, in order to result in a design with maximum output power.

Related simulations:

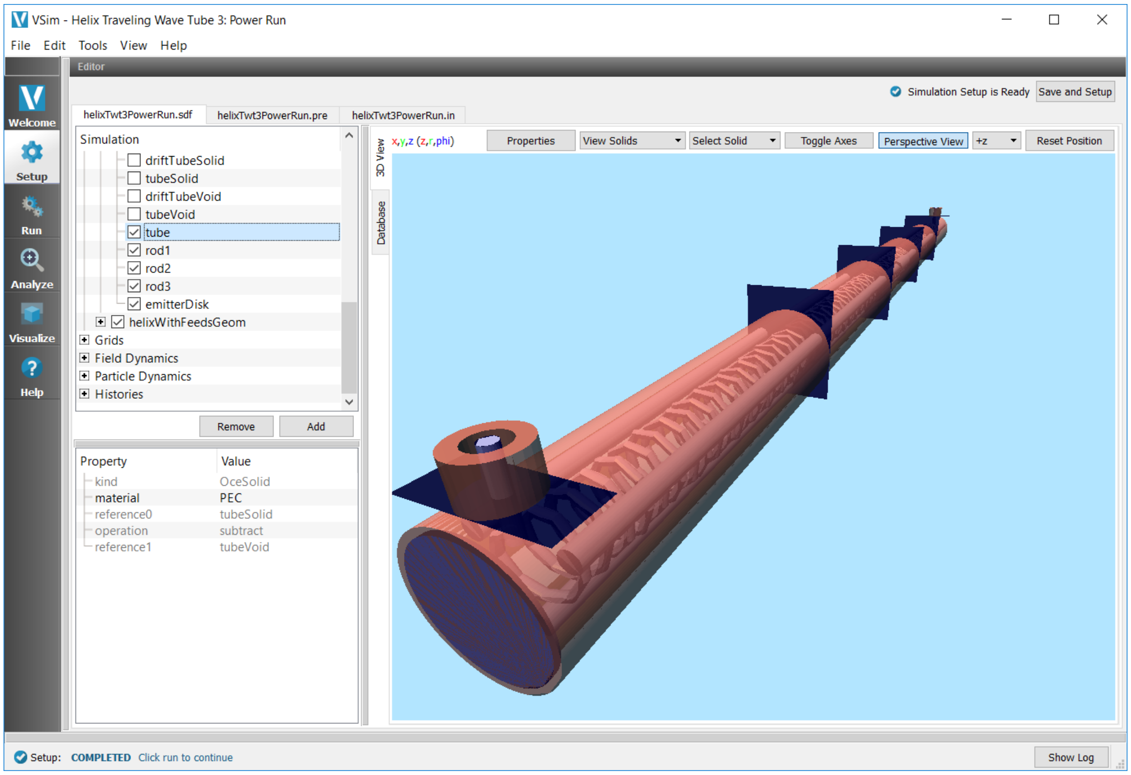
This simulation can be performed with a VSimMD or VSimPD license.

Opening the Simulation

The Helix TWT Power Run example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Radiation Generation* option.
- Select “Helix Travelling Wave Tube 3: Power Run” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 4.68. You can expand the tree elements and navigate through the various properties. Some of these changes will affect the geometry, and so one should review the look of the geometry in the right pane as one changed geometrical variables. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`. One can, e.g., hide the tube to see inside it.



Simulation
- ☐ driftTubeSolid
- ☐ tubeSolid
- ☐ driftTubeVoid
- ☐ tubeVoid
- ☑ tube
- ☑ rod1
- ☑ rod2
- ☑ rod3
- ☑ emitterDisk
- ⊞ ☑ helixWithFeedsGeom
- ⊞ Grids
- ⊞ Field Dynamics
- ⊞ Particle Dynamics
- ⊞ Histories

Property	Value
kind	OceSolid
material	PEC
reference0	tubeSolid
operation	subtract
reference1	tubeVoid

Setup: COMPLETED Click run to continue

Fig. 4.68: Setup Window for the Helix TWT example.

The geometry of the helix can be made more visible by changing the default "opacity" of the surfaces in the Setup Window, as shown in figure Fig. 4.68. This setting is made available by clicking on the "Properties" button.

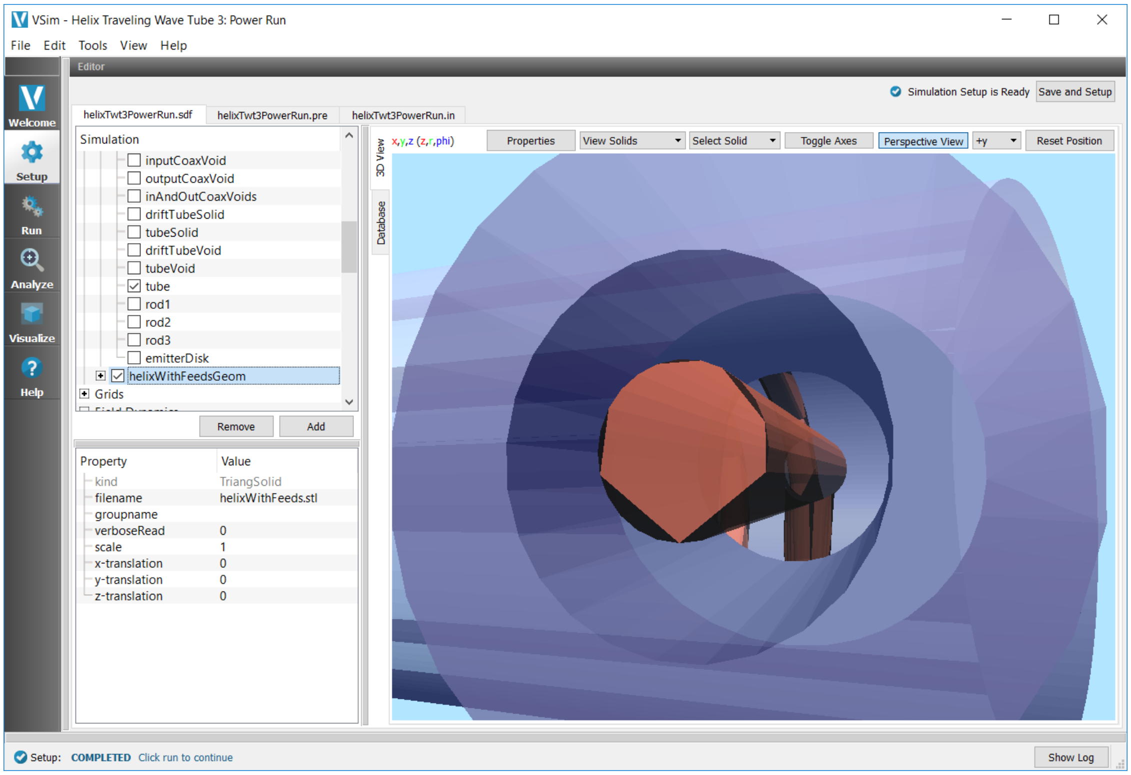
Additional detail of the geometry is shown in figure Fig. 4.69. The top figure shows the dielectric rods and the bottom figure shows how the coaxial waveguide connects to the helical wire.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 4.70.

The default number of time steps will run the simulation long enough to verify that the electron beam is traveling down the tube, that the input signal has entering the simulation and propagated down the tube, that the amplified signal is leaving the system, and that the amplification has reached a steady state. However, the simulation has not been run long enough to ensure that there are no deleterious, backward wave oscillations. To determine that, one should run the simulation twice as long (ensuring a backward and forward traversal) or more, depending on the growth rate of the oscillation.



Simulation

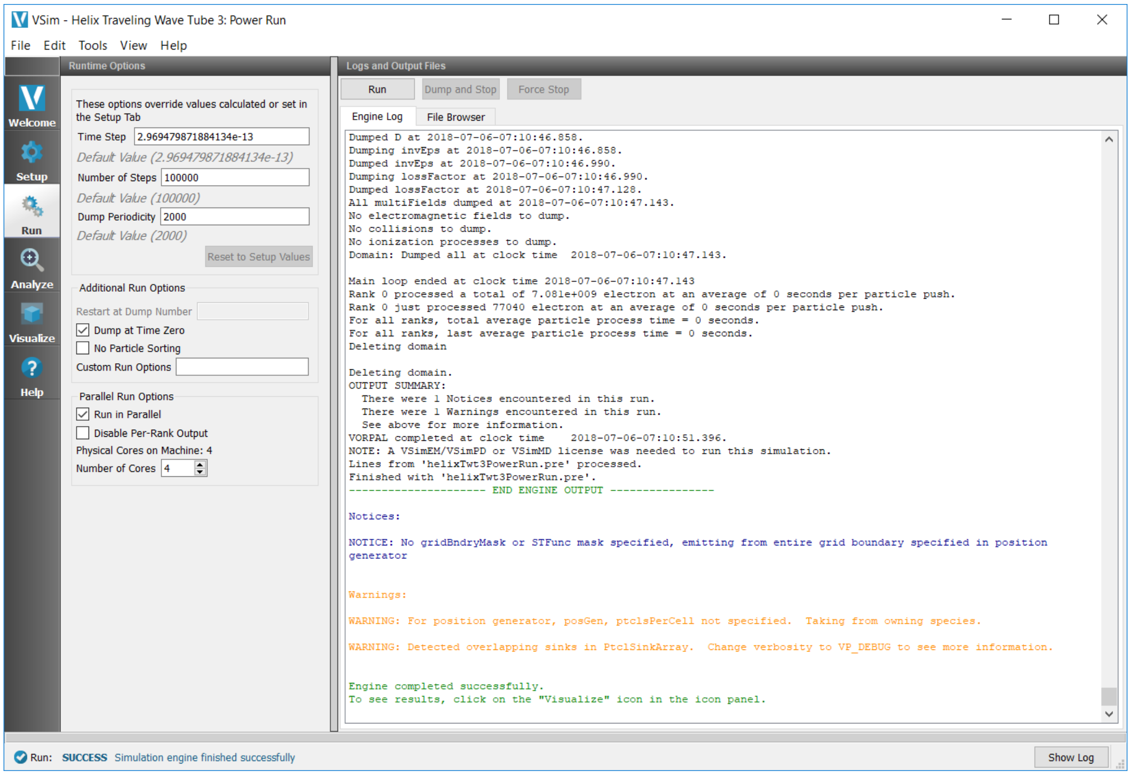
- □ inputCoaxVoid
- □ outputCoaxVoid
- □ inAndOutCoaxVoids
- □ driftTubeSolid
- □ tubeSolid
- □ driftTubeVoid
- □ tubeVoid
- ☑ tube
- □ rod1
- □ rod2
- □ rod3
- □ emitterDisk
- ☑ helixWithFeedsGeom

Grids

Property	Value
kind	TriangSolid
filename	helixWithFeeds.stl
groupname	
verboseRead	0
scale	1
x-translation	0
y-translation	0
z-translation	0

Setup: COMPLETED Click run to continue

Fig. 4.69: A view in through the input coax.



These options override values calculated or set in the Setup Tab

Time Step 2.969479871884134e-13

Default Value (2.969479871884134e-13)

Number of Steps 100000

Default Value (100000)

Dump Periodicity 2000

Default Value (2000)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Dumped D at 2018-07-06-07:10:46.858.
Dumping invEps at 2018-07-06-07:10:46.858.
Dumped invEps at 2018-07-06-07:10:46.990.
Dumping lossFactor at 2018-07-06-07:10:46.990.
Dumped lossFactor at 2018-07-06-07:10:47.128.
All multiFields dumped at 2018-07-06-07:10:47.143.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-06-07:10:47.143.

Main loop ended at clock time 2018-07-06-07:10:47.143
Rank 0 processed a total of 7.081e+009 electron at an average of 0 seconds per particle push.
Rank 0 just processed 77040 electron at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 1 Notices encountered in this run.
  There were 1 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-07-06-07:10:51.396.
NOTE: A VSimEM/VSimPD or VSimMD license was needed to run this simulation.
Lines from 'helixTwt3PowerRun.pre' processed.
Finished with 'helixTwt3PowerRun.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: No gridBndryMask or STFunc mask specified, emitting from entire grid boundary specified in position
generator


Warnings:

WARNING: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.70: The Run Window at the end of execution.

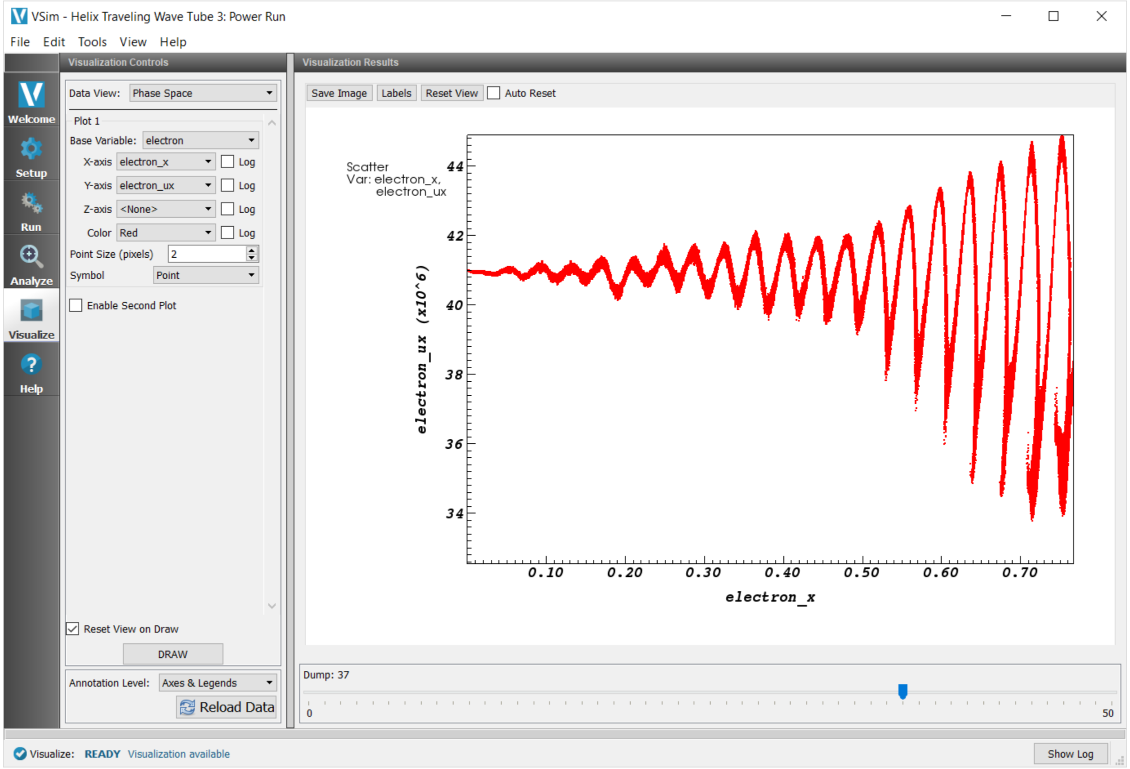
Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the *Visualize* button in the left column of buttons.

The particle phase space, Fig. 4.71, shows how the energy is being extracted from electron beam. To generate this image:

- For *Data View* select *Phase Space*.
- Set *X-axis* to electron_x.
- Set *Y-axis* to electron_ux.
- Press *Draw*.
- Move the dump slider forward in time to see the evolution
- The image is at dump 37.



VSim - Helix Traveling Wave Tube 3: Power Run

Data View: Phase Space

Plot 1

Base Variable: electron

X-axis electron_x □ Log

Y-axis electron_ux □ Log

Z-axis <None> □ Log

Color Red □ Log

Point Size (pixels) 2

Symbol Point

□ Enable Second Plot

☑ Reset View on Draw

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 37

Visualize: READY Visualization available

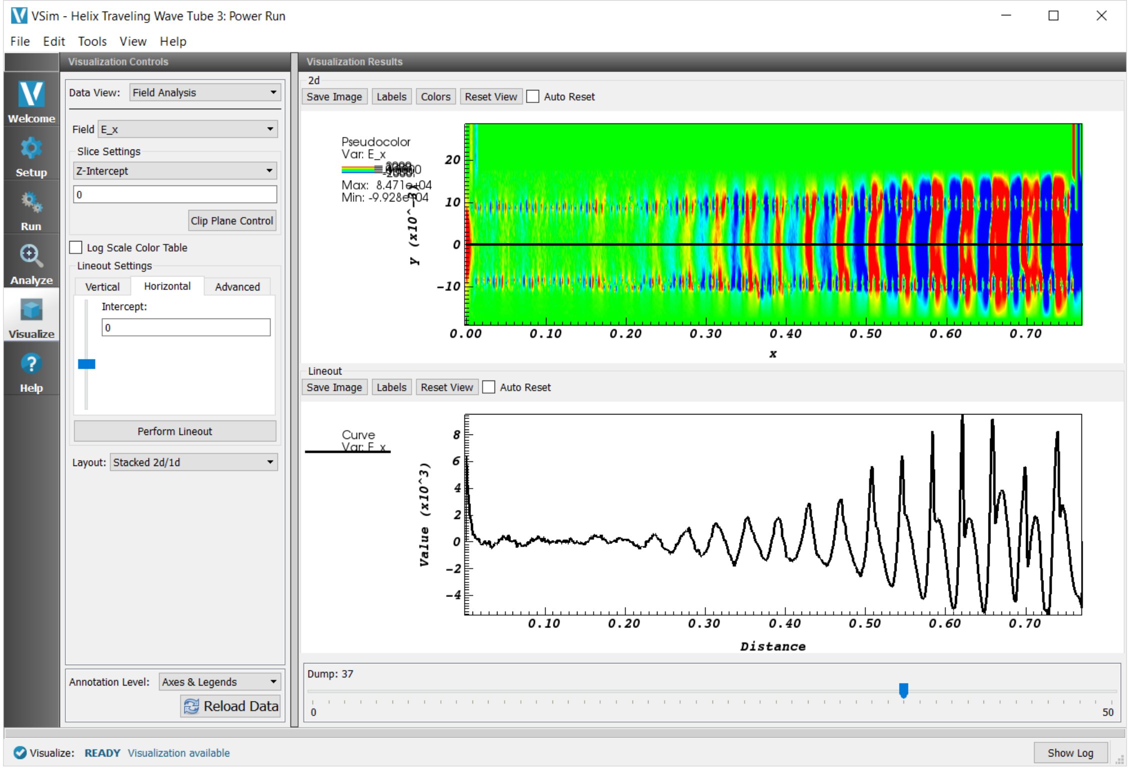
Fig. 4.71: Longitudinal phase space of the electron beam.

One can see in Fig. 4.71 that the beam has been overdriven, such that trapping is beginning to occur. Hence, one must either reduce the input power or one must reduce the gain. In the middle of the tube one can see that the beam oscillation for one cycle decays a bit before taking off again. This is where the attenuator is located.

The effect of overdriving the tube can also be seen in the longitudinal field, Fig. 4.72. This image is obtained by

- For *Data View* select *Field Analysis*.
- Set *Field* to E_x.
- For *Lineout Settings*, choose *Horizontal* with Intercept of 0.

- For *Layout* select Stacked 2d/1d.
- Press *Perform Lineout*.
- Set the *Colors* minima and maxima to $/pm2e3$.
- Move the dump slider forward in time to see the evolution
- The image is at dump 37.


VSim - Helix Traveling Wave Tube 3: Power Run

Data View: Field Analysis

Field: E_x

Slice Settings: Z-Intercept

0

☐ Log Scale Color Table

Lineout Settings: Horizontal

Intercept: 0

Layout: Stacked 2d/1d

Annotation Level: Axes & Legends

Dump: 37

Visualize: READY Visualization available

Fig. 4.72: Longitudinal electric field in the center of the tube.

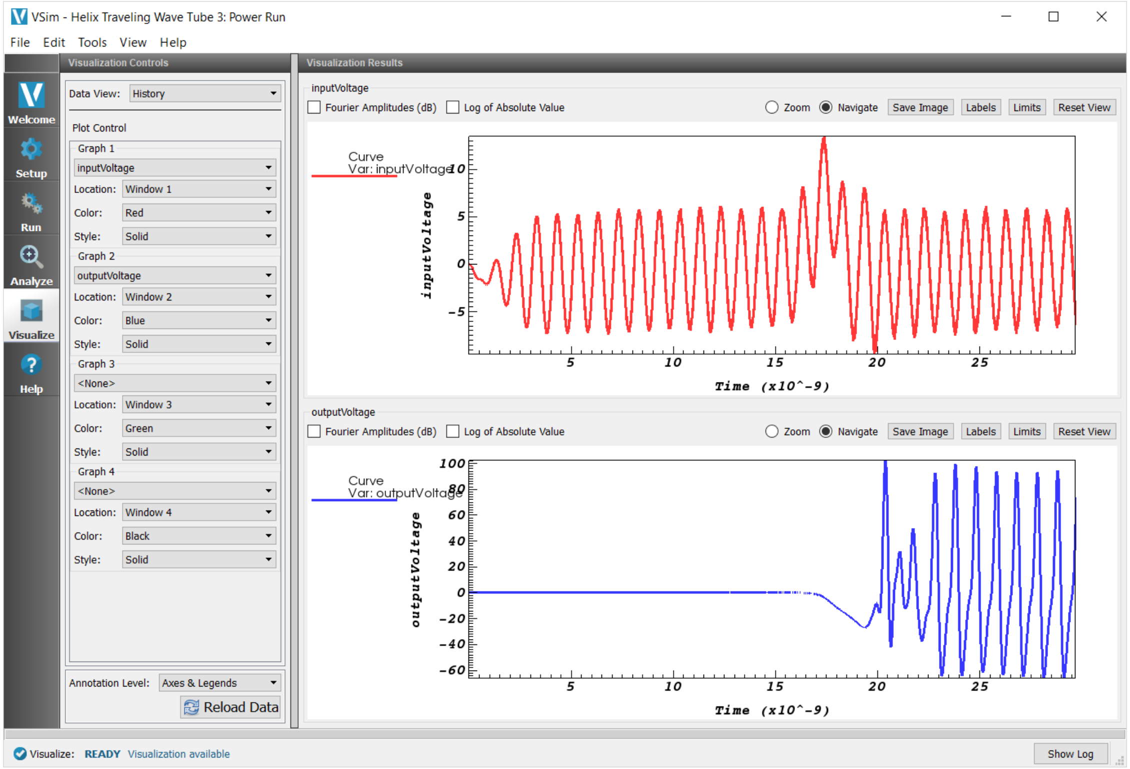
As expected, the longitudinal field is largely confined within the helix. Again, at around x=0.4, one sees the field being damped out by the attenuator. Because the tube has been overdriven, harmonics are appearing in the field at the right. This image shows that the harmonics occur at about 1/5 of the current output power, indicating the amount by which one should decrease the input power or the gain to obtain linear operation.

The gain can be seen in the History records, which are available under the *History* Data View. A sample of these is shown in Fig. 4.73.

To obtain this history image:

- For *Data View* select *History*.
- For *Graph 1* select inputVoltage.
- For *Graph 2* select outputVoltage.
- For *Graph 3* and *Graph 4* select <None>

This image shows that the voltage gains is about a factor of 10 or 20 dB. The voltage history also shows the harmonics in the output that come from overdriving the tube.



Data View: History

Plot Control

Graph 1
inputVoltage
Location: Window 1
Color: Red
Style: Solid

Graph 2
outputVoltage
Location: Window 2
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 3
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 4.73: Input and output voltage histories.

Further Experiments

As noted at the beginning, this run could be run for many more time steps to determine whether backward oscillations are present. Additionally, one can experiment with the beam energy to determine what energy gives the most gain. Varying the input power can determine the maximum output power, which happens when the beam begins trapping at the end of the tube, or the input power at which one obtains large gain while remaining in the linear regime.

4.3.7 Klystron (klystron.sdf)

Keywords:

klystron

Problem description

This VSimMD example simulates a two cavity klystron in three dimensions. The first cavity is driven at its lowest resonant frequency. The resultant cavity voltage creates a velocity modulation in the electron beam which translates to charge modulation as the beam travels in the tube between cavities. The charge modulation then drives the second cavity. The cavities are loaded to give them finite Q.

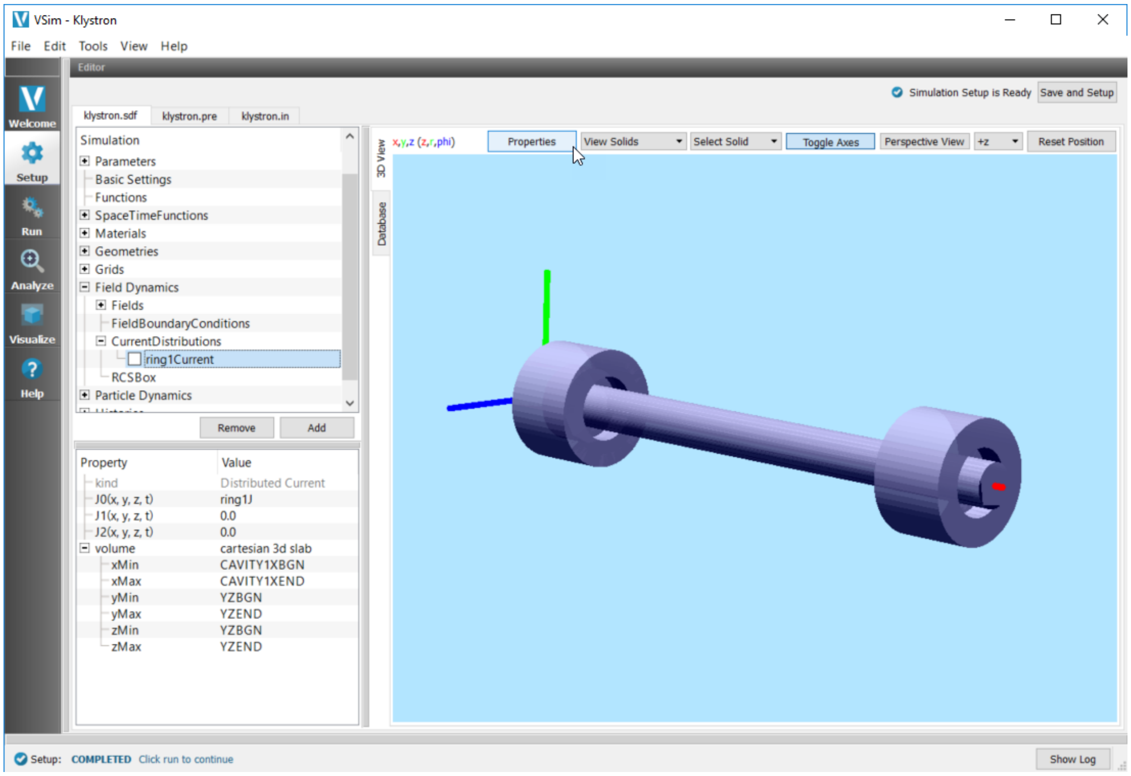
This simulation can be performed with a VSimMD or VSimPD license.

Opening the Simulation

The Klystron example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example...* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Radiation Generation* option.
- Select *Klystron* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the setup window as shown in Fig. 4.74. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to *Grid*.



Property	Value
kind	Distributed Current
J0(x, y, z, t)	ring1J
J1(x, y, z, t)	0.0
J2(x, y, z, t)	0.0
volume	cartesian 3d slab
xMin	CAVITY1XBGN
xMax	CAVITY1XEND
yMin	YZBGN
yMax	YZEND
zMin	YZBGN
zMax	YZEND

Setup: COMPLETED Click run to continue

Fig. 4.74: Setup Window for the Klystron example.

This example illustrates two methods for generating geometries. Under *Geometries* in the elements tree there are two paths, *CSG* and *load1Geom*. The CSG components are constructed from primitives within VSim. The *load1Geom* was imported as an STL file. Highlighting any geometry under the CSG group shows how it was created, either as a primitive with parameters or by operations on other shapes.

Simulation Properties

This simulation is set up to do a Power Run with full capabilities. After completing the Power Run and visualizing the results, you may wish to refine the performance of the klystron by adjusting the setup. Some useful tuning procedures are described in the **Further Experiments** section. These include the **Resonant Frequency Run** and the **Attenuation Calibration Run**.

Some constants that you may wish to modify include:

FREQUENCY:The frequency (in Hz) at which the signal across cavity is driven. This can be tuned to the resonant frequency once determined.

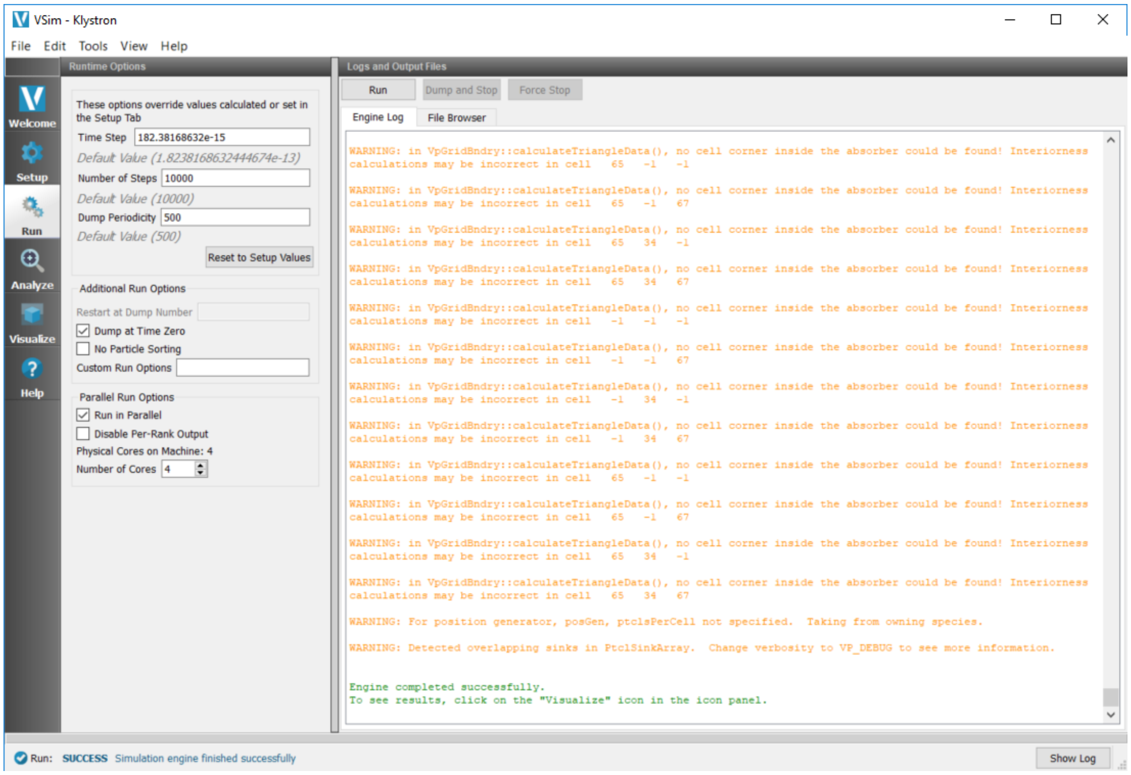
BEAMRADIUS: The radius of the emitted beam of electrons into the klystron.

BEAMCURRENT: The current of the electron beam.

Running the Simulation

With the default setup, complete the Power Run with the following steps:

- Proceed to the Run window by pressing the *Run* button in the left column.
- To run the file, click on the *Run* button in the upper left corner of the right pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.75 below.



These options override values calculated or set in the Setup Tab

Time Step 182.38168632e-15

Default Value (1.8238168632444674e-13)

Number of Steps 10000

Default Value (10000)

Dump Periodicity 500

Default Value (500)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness
calculations may be incorrect in cell   65   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness
calculations may be incorrect in cell   65   -1   67

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness
calculations may be incorrect in cell   65   34   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness
calculations may be incorrect in cell   65   34   67

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness
calculations may be incorrect in cell   -1   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness
calculations may be incorrect in cell   -1   -1   67

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness
calculations may be incorrect in cell   -1   34   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness
calculations may be incorrect in cell   -1   34   67

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness
calculations may be incorrect in cell   65   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness
calculations may be incorrect in cell   65   -1   67

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness
calculations may be incorrect in cell   65   34   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness
calculations may be incorrect in cell   65   34   67

WARNING: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.75: The Run Window at the end of execution.

Visualizing the Results

After the the simulation run has completed successfully, you may proceed to the Visualize Window by pressing the *Visualize* button in the left column. To reproduce Fig. 4.76 follow these steps:

- Select *Data Overview* from the *Data View* pull-down menu.
- Expand *Particle Data*
- Expand *electrons0*
- Select *electrons0_ux*

- Expand *Geometries*
- Select *poly_surface (klystronPecShapes)*
- Select *Clip All Plots*
- Step through time using the dump slider on the bottom of the right pane.

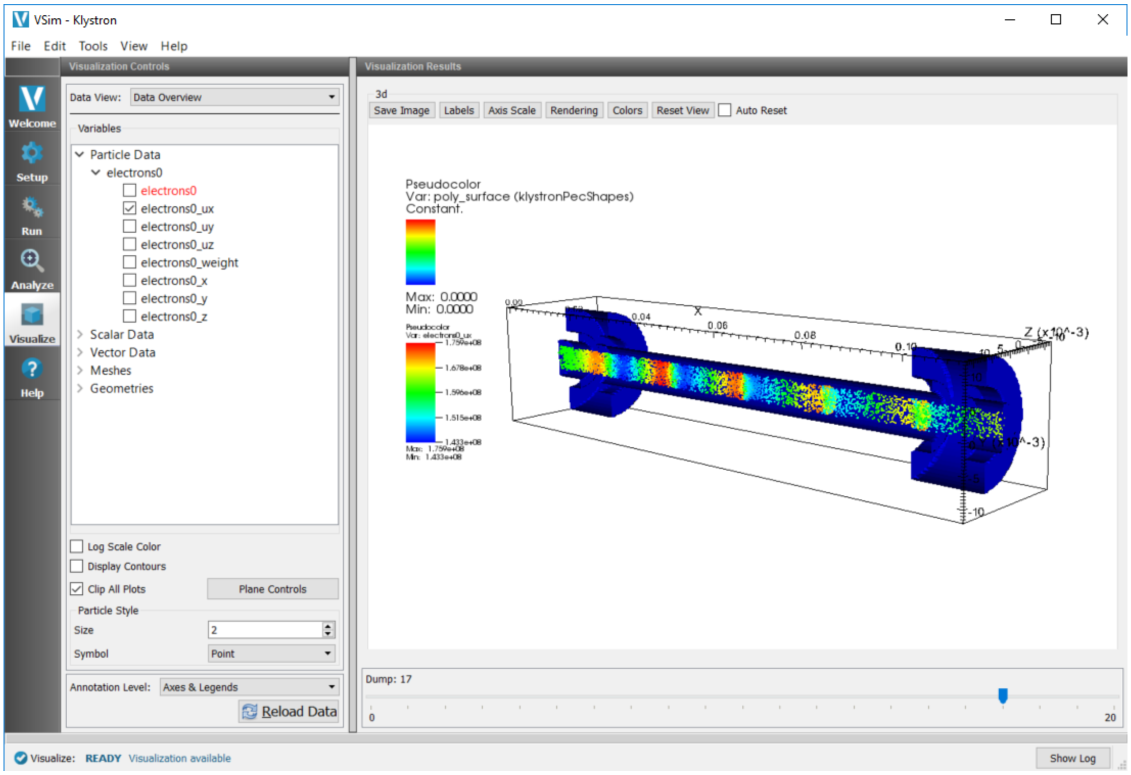


Fig. 4.76: A power run with an electron beam.

Further Experiments

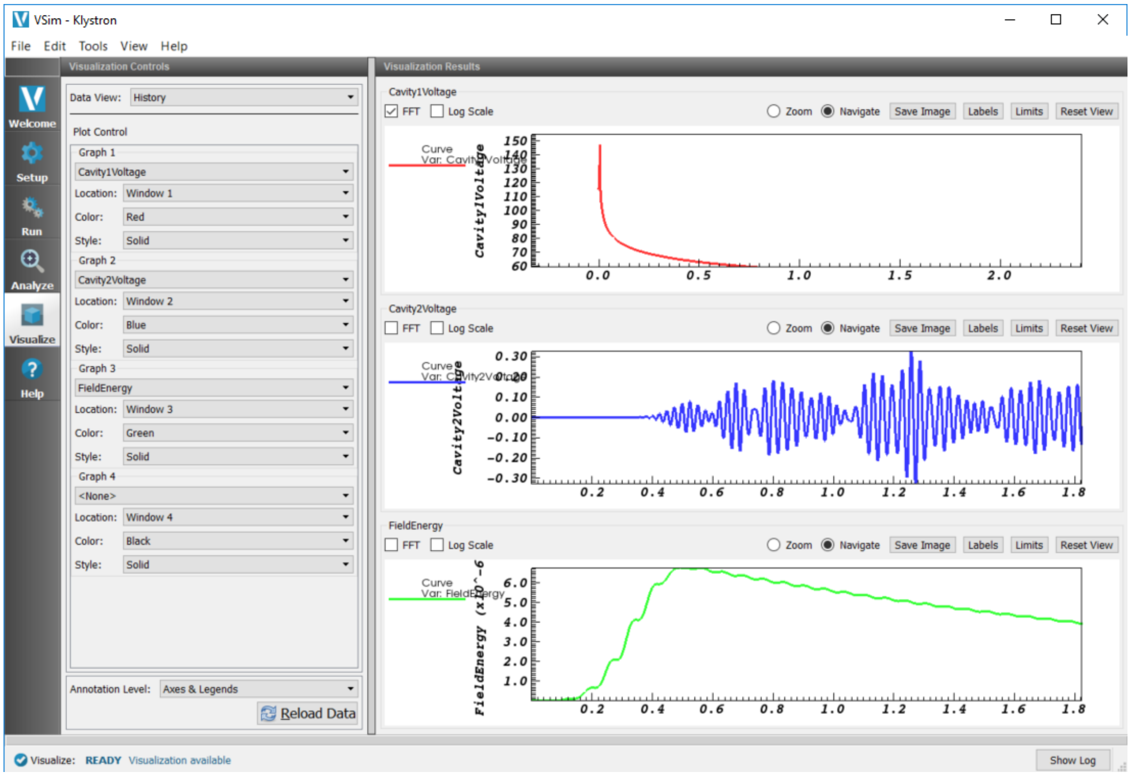
The **Attenuation Calibration Run** and **Resonant Frequency Run** are outlined below. These experiments will allow you to tune the klystron. You may want to iterate through these experiments to get the desired performance. Once the cavity performance is satisfactory you can repeat the Power Run to see the effects on the electrons. To see the full behavior of the device, increase the number of steps to (5 x Default). This will allow you to see the saturation of the second cavity.

Resonant Frequency Run

To determine the resonant frequency of the first cavity we will analyze the fourier transform of its voltage. In the Setup window, under *Basic Settings*, set *particles* to *no particles*. Then, under *SpaceTimeFunctions*, in *ring1J* change “rampUp” to “rampUpAndDown”. This will ping Cavity 1 and allow us to observe the ringing signal. Run the simulation with this setup.

To determine the resonant frequency proceed to the Visualize window. Select *History* from the *Data View* pull-down menu. Click FFT to the left of the Cavity1Voltage plot in the *Visualization Results* pane. The resulting plot will resemble Fig. 4.77. Zoom in on the maximum of this plot to determine the resonant frequency. You can now update the FREQUENCY under *Contants* in the Setup window with this new value and use it to drive future simulations.

Attenuation Calibration Run


VSim - Klystron

File Edit Tools View Help

Visualization Controls

Data View: History

Plot Control

Graph 1
Cavity1Voltage
Location: Window 1
Color: Red
Style: Solid

Graph 2
Cavity2Voltage
Location: Window 2
Color: Blue
Style: Solid

Graph 3
FieldEnergy
Location: Window 3
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Reload Data

Visualize: READY Visualization available

Fig. 4.77: Fourier transform of Cavity1_Voltage versus time (in GHz).

The user can integrate this run type in order to calibrate the observed attenuation to the desired loss. The attenuation can be tuned by modifying the *conductivity* of the material, *resistive damper*. The Q of the cavity can be computed using a feature of the Analysis Tab, as described below.

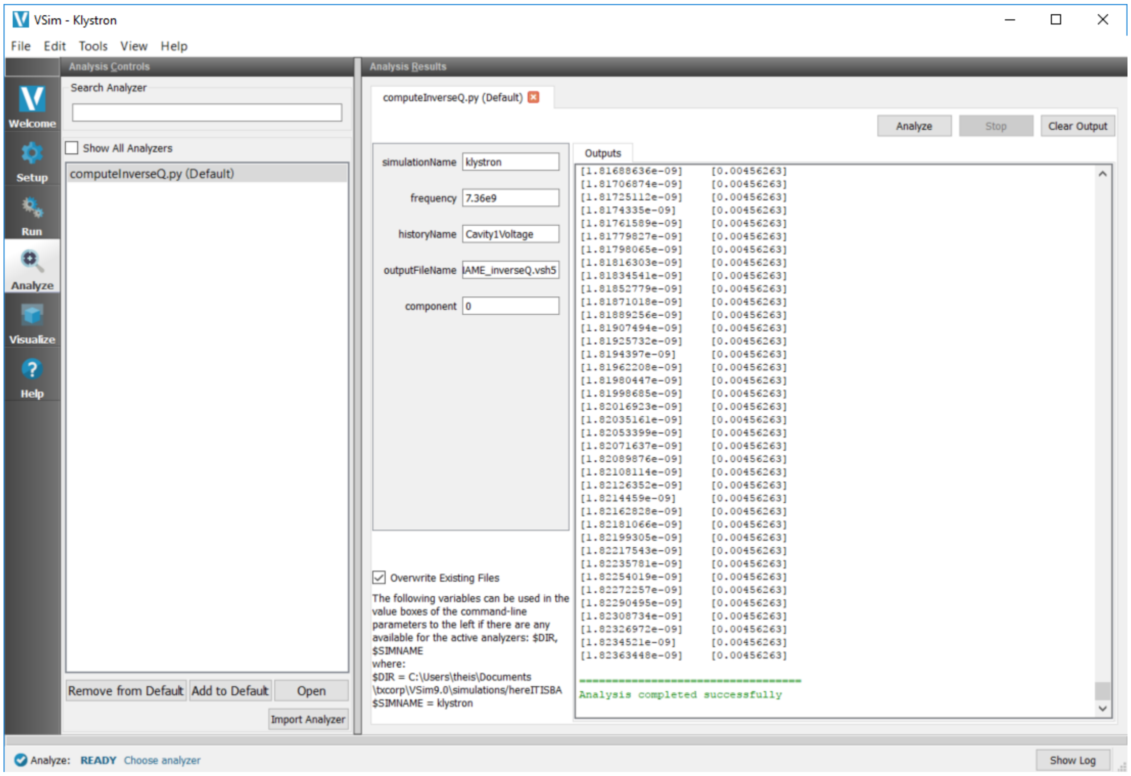
For the **Attenuation Calibration Run**, use the same Setup as the **Resonant Frequency Run**. After running the simulation, the quality factors, Q_1 and Q_2, for cavities 1 and 2 can be calculated using the *computeInverseQ.py* script in the Analyze window.

- Press the Analyze button in the left column of buttons.
- Select the *computeInverseQ.py* analyzer, then *Open*.
- Enter Cavity1Voltage or Cavity2Voltage in the *historyName* field to designate the history to analyze.
- Enter the value of the FREQUENCY constant as defined in the Setup window in the *frequency* field to designate the frequency at which the history will be analyzed.
- Update the *outputFileName* field if desired
- Click the *Analyze* button in the top right corner of the window. As shown in Fig. 4.78 below. Two columns of data with the titles "Time (s)" and "Inverse Q" will be output in the right pane. The analysis has completed when you see the output "Analysis completed successfully."

Scrolling through or plotting the output data in the Visualize window enables the user to understand the Klystron's performance. The user may iterate this run type to achieve the desired attenuation.

4.3.8 2D Magnetron (magnetron2D.sdf)

Keywords:



simulationName: klystron

frequency: 7.36e9

historyName: Cavity1Voltage

outputFileName: IAME_inverseQ.vsh5

component: 0

☑ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analyzers: $DIR, $SIMNAME
where:
$DIR = C:\Users\theis\Documents\txcorp\VSim9.0\simulations/hereITISBA
$SIMNAME = klystron

```
[1.81688636e-09]    [0.00456263]
[1.81706874e-09]    [0.00456263]
[1.81725112e-09]    [0.00456263]
[1.8174335e-09]     [0.00456263]
[1.81761589e-09]    [0.00456263]
[1.81779827e-09]    [0.00456263]
[1.81798065e-09]    [0.00456263]
[1.81816303e-09]    [0.00456263]
[1.81834541e-09]    [0.00456263]
[1.81852779e-09]    [0.00456263]
[1.81871018e-09]    [0.00456263]
[1.81889256e-09]    [0.00456263]
[1.81907494e-09]    [0.00456263]
[1.81925732e-09]    [0.00456263]
[1.8194397e-09]     [0.00456263]
[1.81962208e-09]    [0.00456263]
[1.81980447e-09]    [0.00456263]
[1.81998685e-09]    [0.00456263]
[1.82016923e-09]    [0.00456263]
[1.82035161e-09]    [0.00456263]
[1.82053399e-09]    [0.00456263]
[1.82071637e-09]    [0.00456263]
[1.82089876e-09]    [0.00456263]
[1.82108114e-09]    [0.00456263]
[1.82126352e-09]    [0.00456263]
[1.8214459e-09]     [0.00456263]
[1.82162828e-09]    [0.00456263]
[1.82181066e-09]    [0.00456263]
[1.82199305e-09]    [0.00456263]
[1.82217543e-09]    [0.00456263]
[1.82235781e-09]    [0.00456263]
[1.82254019e-09]    [0.00456263]
[1.82272257e-09]    [0.00456263]
[1.82290495e-09]    [0.00456263]
[1.82308734e-09]    [0.00456263]
[1.82326972e-09]    [0.00456263]
[1.8234521e-09]     [0.00456263]
[1.82363448e-09]    [0.00456263]

=====================================
Analysis completed successfully
```

Analyze: READY Choose analyzer

Fig. 4.78: The Analysis window at the end of execution of the computeInverseQ.py script.

magnetron

Problem description

This VSimMD example simulates a rising sun magnetron in two dimensions. A load is added to one cavity, representing a coupler to the magnetron through the quality factor, Q. Upon configuring an electrostatic voltage across the anode and cathode, particles are introduced to the simulation, exhibiting a five spoke pi-mode.

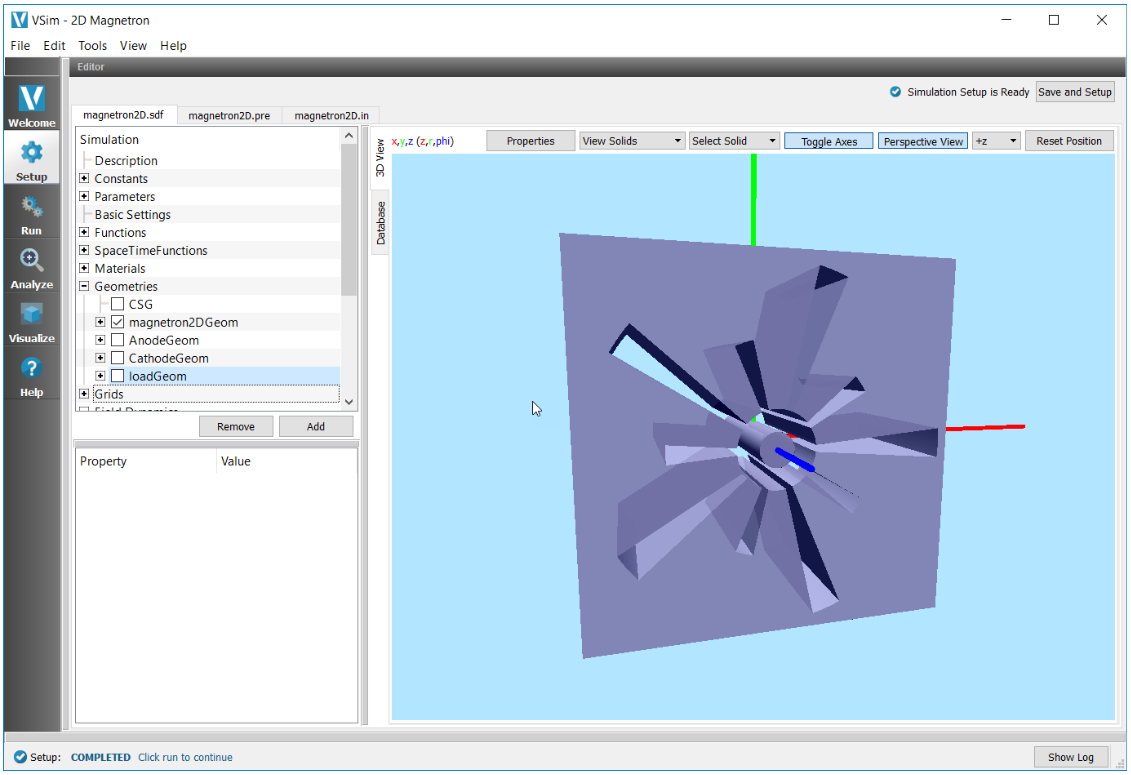
This simulation can be performed with a VSimMD or VSimPD license.

Opening the Simulation

The 2D Magnetron example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Radiation Generation* option.
- Select “2D Magnetron” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is now shown with all the implemented physics and geometries. See Fig. 4.79.



Simulation Setup is Ready

Setup: COMPLETED Click run to continue

Fig. 4.79: Setup Window for the 2D Magnetron example.

Simulation Properties

As seen in Fig. 4.80 of the rising sun magnetron, the radius of the cathode is RCATHODE and the radius of the anode is RANODE. Long cavities have radius RCAVITY1 and opening angle ANGLECAVITY1. Short cavities have radius RCAVITY2 and opening angle ANGLECAVITY2. These dimensions control the spectrum and thus the operating frequency, which for the default parameters is approximately 960 MHz.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.81 below.

Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column.
- Expand *Particle Data*
- Expand *electrons*
- Select *electrons*

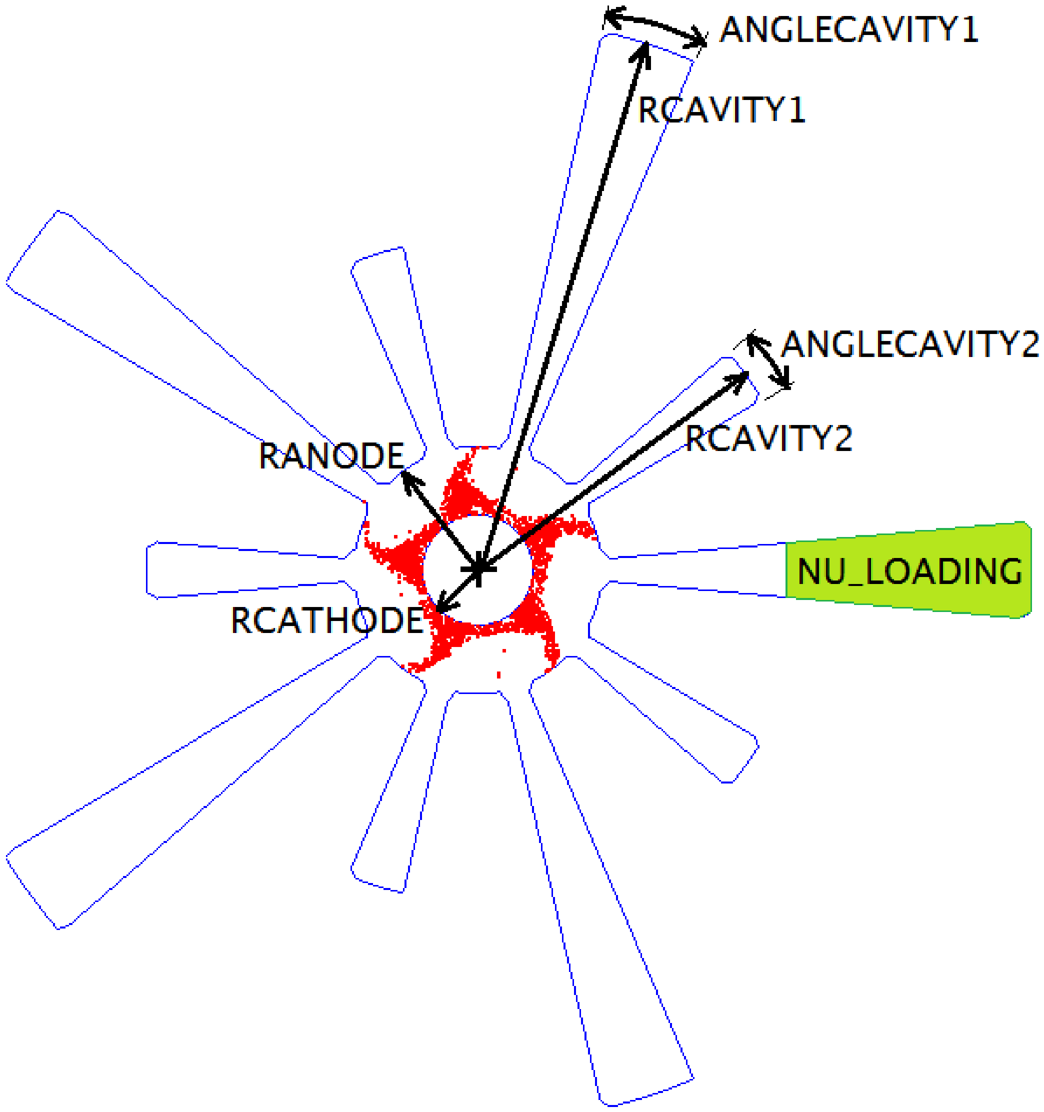
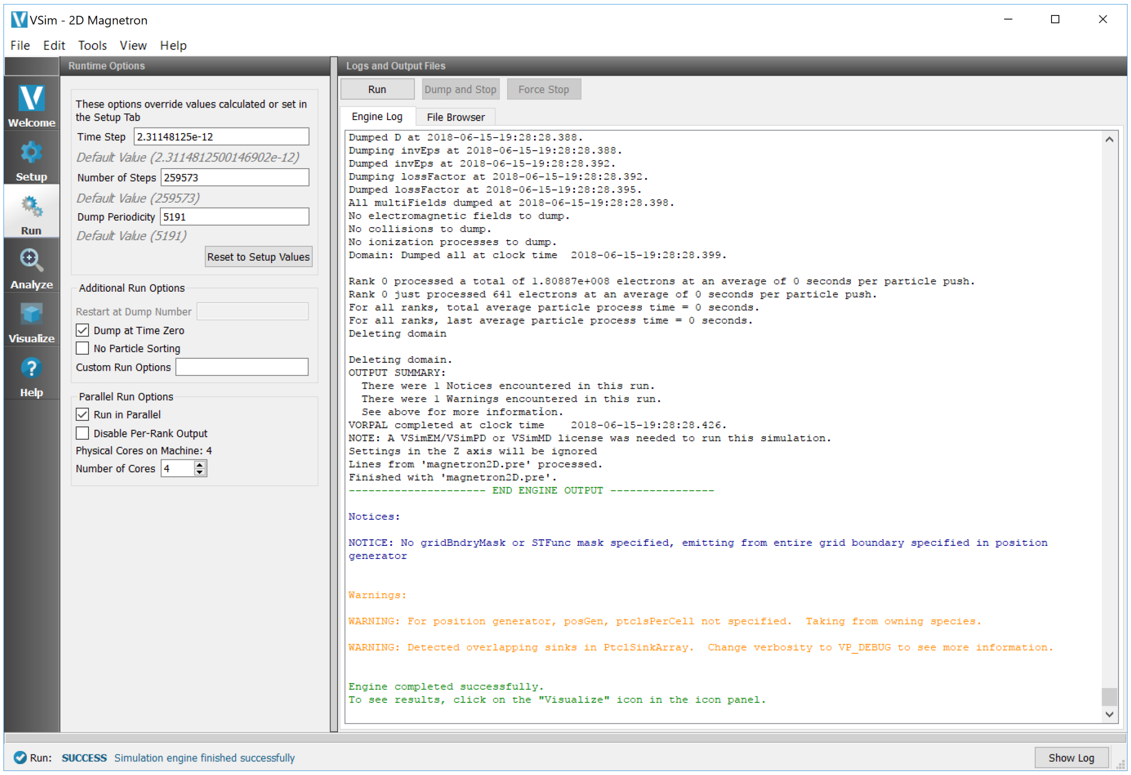


Fig. 4.80: Some exposed variables of the 2D Magnetron example.



These options override values calculated or set in the Setup Tab

Time Step 2.31148125e-12

Default Value (2.3114812500146902e-12)

Number of Steps 259573

Default Value (259573)

Dump Periodicity 5191

Default Value (5191)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Dumped D at 2018-06-15-19:28:28.388.
Dumping invEps at 2018-06-15-19:28:28.388.
Dumped invEps at 2018-06-15-19:28:28.392.
Dumping lossFactor at 2018-06-15-19:28:28.392.
Dumped lossFactor at 2018-06-15-19:28:28.395.
All multiFields dumped at 2018-06-15-19:28:28.398.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-15-19:28:28.399.

Rank 0 processed a total of 1.80887e+008 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 641 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 1 Notices encountered in this run.
  There were 1 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-06-15-19:28:28.426.
NOTE: A VSimEM/VSimPD or VSimMD license was needed to run this simulation.
Settings in the Z axis will be ignored
Lines from 'magnetron2D.pre' processed.
Finished with 'magnetron2D.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: No gridBndryMask or STFunc mask specified, emitting from entire grid boundary specified in position
generator


Warnings:

WARNING: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.81: The Run Window at the end of execution.

- Expand *Geometries*
- Select *poly (magnetron2DGeomSolid)*

The electron modes can be viewed in the right pane. Use the dump slider on the bottom of the right pane to step through time. When electrons are emitted from the cathode, the four spoke, 650 MHz is present during startup. At approximately 250 ns, the five spoke begins to dominate and eventually appears as seen in Fig. 4.82.

4.4 Radiation Generation (text-based setup)

4.4.1 Gyrotron Mode (gyrotronModeT.pre)

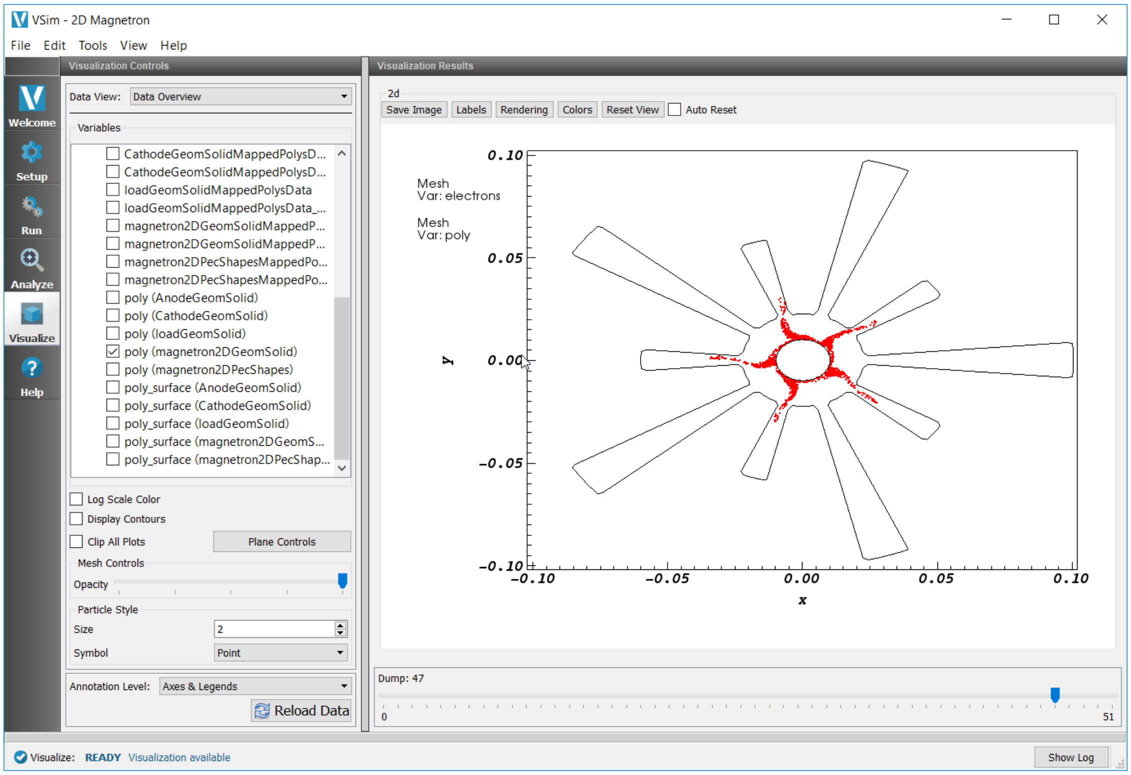
Keywords:

gyrotron

Problem description

This VSimMD example illustrates a very high order mode, TE-22-6, propagating in a cylindrical waveguide, very near to the cutoff frequency, which is a common situation in a gyrotron. The example is intended to allow investigation of the axial phase and group velocity of such a mode, as a function of frequency, and to highlight the intricacies of simulating a mode that is propagating within a percent or two of its cutoff frequency.

This simulation can be performed with a VSimMD license.



VSim - 2D Magnetron

Data View: Data Overview

Variables

- ☐ CathodeGeomSolidMappedPolysD...
- ☐ CathodeGeomSolidMappedPolysD...
- ☐ loadGeomSolidMappedPolysData
- ☐ loadGeomSolidMappedPolysData_...
- ☐ magnetron2DGeomSolidMappedP...
- ☐ magnetron2DGeomSolidMappedP...
- ☐ magnetron2DPecShapesMappedPo...
- ☐ magnetron2DPecShapesMappedPo...
- ☐ poly (AnodeGeomSolid)
- ☐ poly (CathodeGeomSolid)
- ☐ poly (loadGeomSolid)
- ☑ poly (magnetron2DGeomSolid)
- ☐ poly (magnetron2DPecShapes)
- ☐ poly_surface (AnodeGeomSolid)
- ☐ poly_surface (CathodeGeomSolid)
- ☐ poly_surface (loadGeomSolid)
- ☐ poly_surface (magnetron2DGeomS...
- ☐ poly_surface (magnetron2DPecShap...

☐ Log Scale Color

☐ Display Contours

☐ Clip All Plots

Particle Style

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

Dump: 47

Visualize: READY Visualization available

Fig. 4.82: The five spoke pi-mode at 600 ns.

Opening the Simulation

The Gyrotron Mode example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Radiation Generation (text-based setup)* option.
- Select "Gyrotron Mode (text-based setup)" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is now shown with all the implemented physics and geometries, if applicable. See Fig. 4.83.

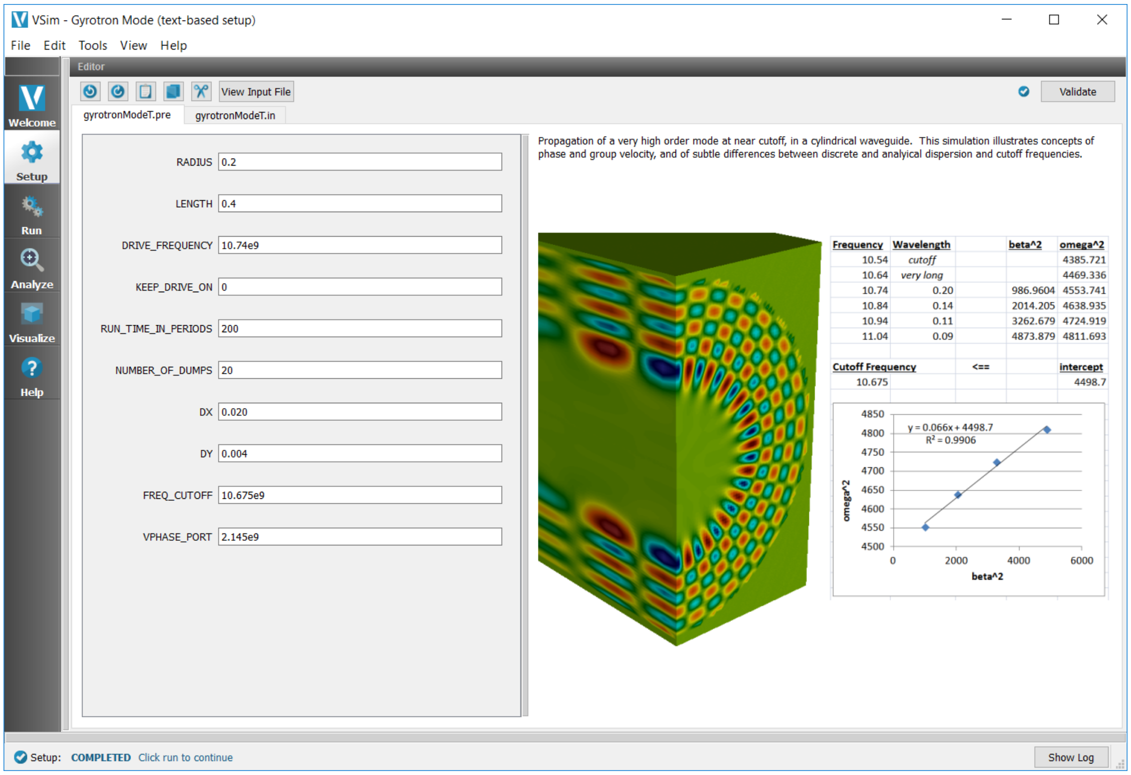
The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 4.83.

Input File Features

There are only two geometrical input parameters; the waveguide radius and length. The user may also control the excitation frequency, the duration of the simulation, and the nature of the excitation, specifically whether it is pulsed or continuous-wave. Additional exposed parameters include the grid sizes, and the tuning of the exiting wave boundary condition, which allows for more in-depth study with this example.

The excitation may be pulsed or continuous-wave, depending on the parameter, KEEP_DRIVE_ON.

Pulsed Simulation (KEEP_DRIVE_ON=0)



RADIUS 0.2

LENGTH 0.4

DRIVE_FREQUENCY 10.74e9

KEEP_DRIVE_ON 0

RUN_TIME_IN_PERIODS 200

NUMBER_OF_DUMPS 20

DX 0.020

DY 0.004

FREQ_CUTOFF 10.675e9

VPHASE_PORT 2.145e9

Propagation of a very high order mode at near cutoff, in a cylindrical waveguide. This simulation illustrates concepts of phase and group velocity, and of subtle differences between discrete and analyical dispersion and cutoff frequencies.

Frequency	Wavelength		beta^2	omega^2
10.54	*cutoff*			4385.721
10.64	*very long*			4469.336
10.74	0.20		986.9604	4553.741
10.84	0.14		2014.205	4638.935
10.94	0.11		3262.679	4724.919
11.04	0.09		4873.879	4811.693
Cutoff Frequency		<==		intercept
10.675				4498.7

Setup: COMPLETED Click run to continue

Fig. 4.83: Setup Window for the Gyrotron Mode example.

In this case, the wave is driven for half of the simulation duration, with a smooth turn-on / turn-off time window. Then, for the remaining half of the periods, the excitation propagates freely. The axial profile of the pulse will be very short, typically just one or two axial wavelengths. It will propagate slowly down the waveguide, as expected from the group velocity which is very small near cutoff. In the center of the pulse the TE-22-6 mode is preserved, but because this is a pulse, nearby modes in frequency are also present. One can observe a rich set of other mode patterns just a few grid planes away from the center of the pulse.

Continuous-Wave Simulation (KEEP_DRIVE_ON=1)

The drive may be kept on, instead of having it turn off halfway through the simulation. After an initial transient, this sets up a single TE-22-6 traveling wave mode pattern throughout the waveguide. This allows for accurate measurement of the axial wavenumber, beta, for the mode.

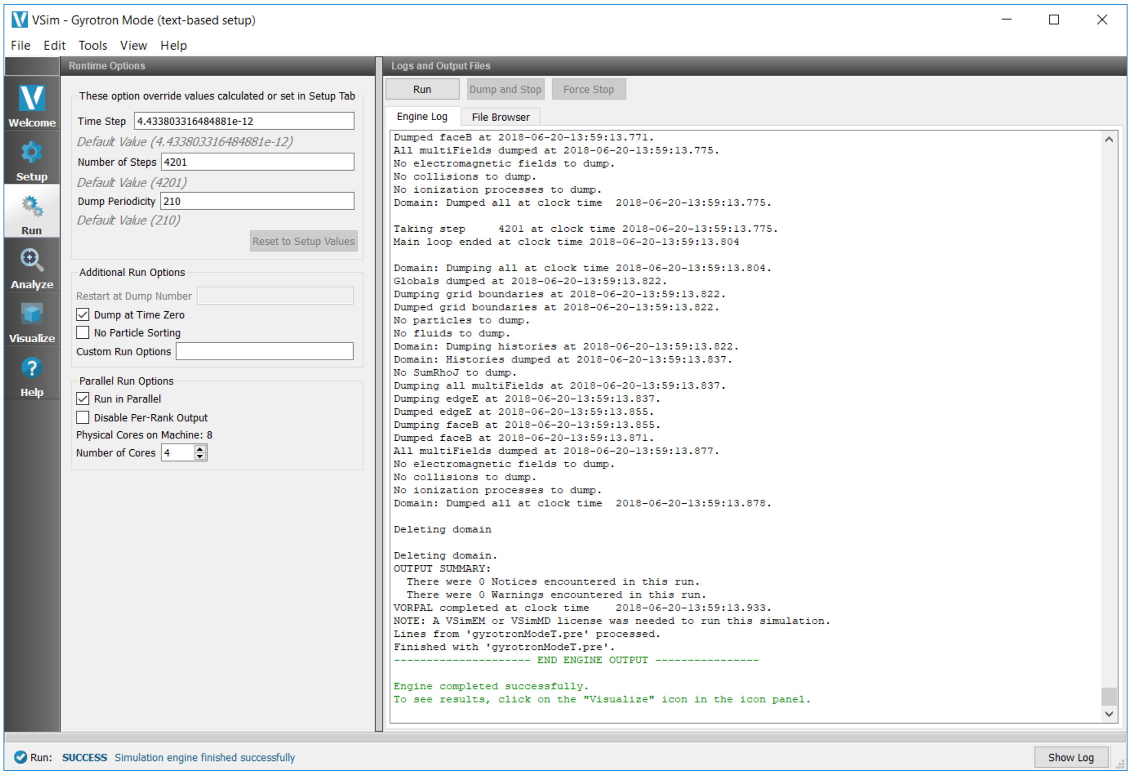
Running the simulation

After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 4.84 below.

Visualizing the results

After performing the above actions, continue as follows:



These option override values calculated or set in Setup Tab

Time Step 4.433803316484881e-12
Default Value (4.433803316484881e-12)
Number of Steps 4201
Default Value (4201)
Dump Periodicity 210
Default Value (210)

Additional Run Options

Restart at Dump Number
☑ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
☐ Disable Per-Rank Output
Physical Cores on Machine: 8
Number of Cores 4

```
Dumped faceB at 2018-06-20-13:59:13.771.
All multiFields dumped at 2018-06-20-13:59:13.775.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-20-13:59:13.775.

Taking step     4201 at clock time 2018-06-20-13:59:13.775.
Main loop ended at clock time 2018-06-20-13:59:13.804

Domain: Dumping all at clock time 2018-06-20-13:59:13.804.
Globals dumped at 2018-06-20-13:59:13.822.
Dumping grid boundaries at 2018-06-20-13:59:13.822.
Dumped grid boundaries at 2018-06-20-13:59:13.822.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-06-20-13:59:13.822.
Domain: Histories dumped at 2018-06-20-13:59:13.837.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-20-13:59:13.837.
Dumping edgeE at 2018-06-20-13:59:13.837.
Dumped edgeE at 2018-06-20-13:59:13.855.
Dumping faceB at 2018-06-20-13:59:13.855.
Dumped faceB at 2018-06-20-13:59:13.871.
All multiFields dumped at 2018-06-20-13:59:13.877.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-20-13:59:13.878.

Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-20-13:59:13.933.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'gyrotronModeT.pre' processed.
Finished with 'gyrotronModeT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.84: The Run Window at the end of execution.

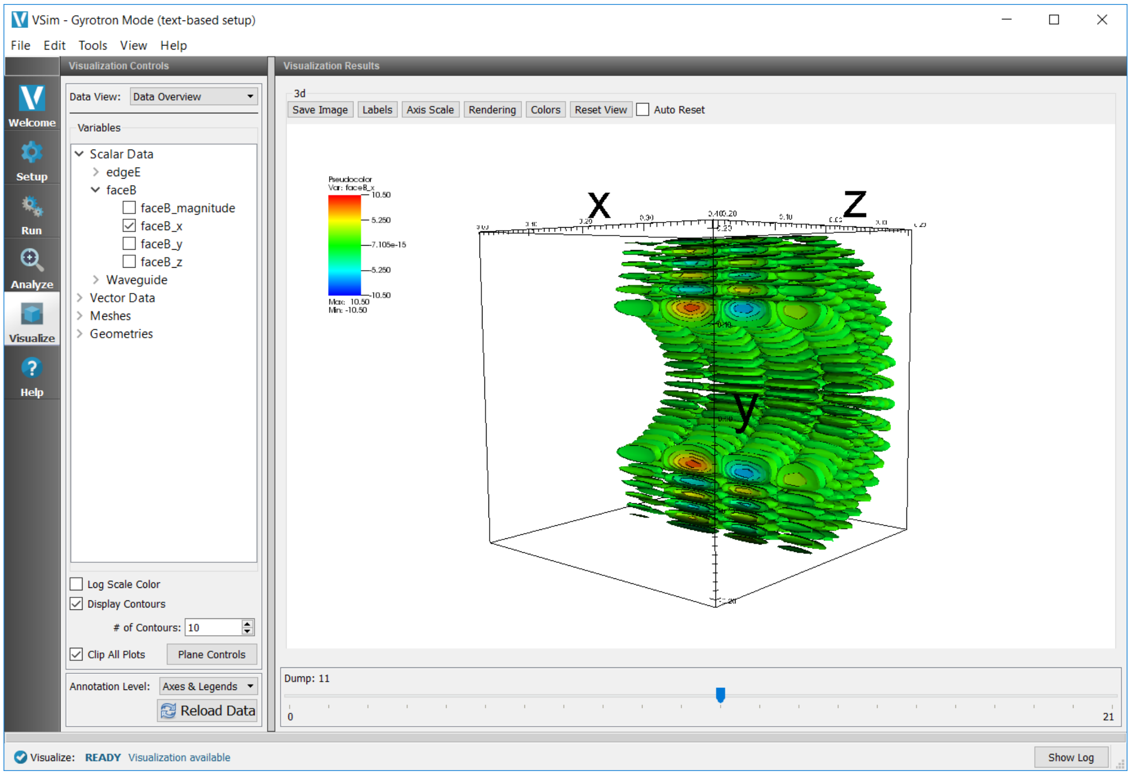
- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The B_x field is the best component for looking at in this simulation, as shown in Fig. 4.85. To create this plot:

- Expand *Scalar Data*.
- Expand *faceB*.
- Select *faceB_x*.
- Select *Display Contours* and set *# of Coutours* to 10.
- Select *Clip All Plots*.
- Move the dump slider to dump 11.

The initial parameters are selected so that the excitation frequency is just barely above cutoff. While the axial phase velocity is high in this case, the group velocity is quite low, and the simulation shows a narrow wavepacket slowly moving down the length of the tube, while remarkably still maintaining the very high order TE-22-6 pattern. Contamination of the pattern increases as the duration of the excitation is reduced, since more frequencies are brought into the transient. The user is encouraged to look at the mode pattern and contamination properties as frequency and duration are varied.

The TE-22-6 mode's cutoff frequency, for the suggested initial radius of 20 cm, is known analytically to be 10.8845 GHz, which derives from the value of the 6th root of the J_{22} bessel function, which is 45.624312. However, the user will note that the suggested initial drive frequency is below this, at 10.74 GHz, and yet the wave appears to propagate! This illustrates an important property of finite-difference dispersion, that in fact the speed of light is ever-so-slightly slower in the finite-difference-time-domain simulation than in reality. In most cases, this is hardly noticed, however, when operating this close to the cutoff frequency of a waveguide, this difference can be readily seen, as this example illustrates. The discrepancy between the discrete FDTD cutoff frequency and the analytic cutoff frequency, depends on the grid resolution of the wave, and in general decreases as δx^2, where δx is the grid size.



VSim - Gyrotron Mode (text-based setup)

Data View: Data Overview

- Scalar Data
  - edgeE
  - faceB
    - □ faceB_magnitude
    - ☑ faceB_x
    - □ faceB_y
    - □ faceB_z
  - Waveguide
- Vector Data
- Meshes
- Geometries

□ Log Scale Color

☑ Display Contours

# of Contours: 10

☑ Clip All Plots

Annotation Level: Axes & Legends

□ Auto Reset

Pseudocolor
Var: faceB_x
Max: 10.50
Min: -10.50

Dump: 11

Visualize: READY Visualization available

Fig. 4.85: Illustration of the mode pattern, and propagation of the mode down the length of the tube.

A very useful piece of information is the FDTD cutoff frequency. This may be found with a series of simulations, each at different drive frequencies, ω. The KEEP_DRIVE_ON parameter should be set to 1, so that the axial wavelength, β, can be measured from the field plots. A plot of ω^2 vs. β^2 should be essentially linear, with the intercept on the ω^2 axis being the FDTD cutoff frequency, ω^2_{cutoff} ($\omega^2 = \omega^2_{cutoff} + c^2\beta^2$), and with slope being the FDTD speed-of-light-squared. A spreadsheet showing this exercise for the suggested initial values of the example is shown below. The result of this study is that the FDTD cutoff frequency is actually 10.675 GHz, or 2% below the known analytical result, for the initial suggested grid resolution.

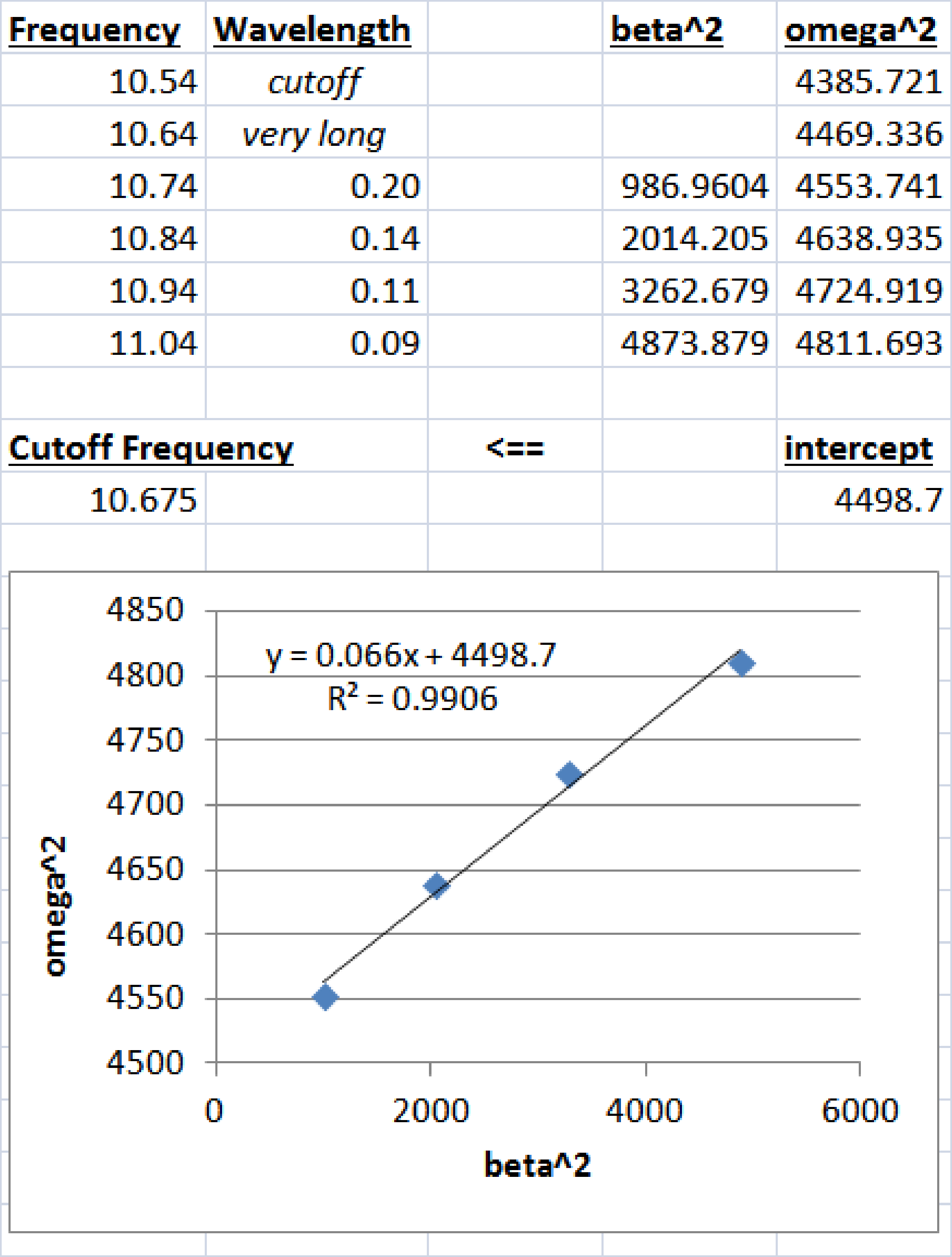
Further Experiments

The user is encouraged to repeat the simulations discussed in the previous section with a finer resolution, to see how the FDTD cutoff frequency approaches the analytic result as resolution improves.

The detailed TE-22-6 mode pattern is very carefully crafted using polynomial fitting functions, and is introduced into the axial magnetic field, B_x, at the left side of the simulation. There is no direct option to use a different mode, although the user may attempt to edit the detail of the input to do so.

Finally, a boundary condition tuning parameter, VPHASE_PORT, is offered to allow the user to experiment with tuning of the outgoing wave boundary condition in this near cutoff scenario. In this circumstance, the optimal phase velocity may be 5 to 10 times the speed of light.

An additional exposed user parameter, FREQ_CUTOFF, is offered, and may be used to store the value derived from the simulations discussed in the previous section. By default, this parameter is not used. However the user may look into the detail of input file, and notice a comment line that indicates how this parameter might be used to set the value of VPHASE_PORT more accurately.



Frequency	Wavelength		beta^2	omega^2
10.54	*cutoff*			4385.721
10.64	*very long*			4469.336
10.74	0.20		986.9604	4553.741
10.84	0.14		2014.205	4638.935
10.94	0.11		3262.679	4724.919
11.04	0.09		4873.879	4811.693
Cutoff Frequency		<==		intercept
10.675				4498.7

Fig. 4.86: Computing the FDTD cutoff frequency of the TE-22-6 mode.

4.4.2 Klystron (klystronT.pre)

Keywords:

klystronT

Problem description

This VSimMD example simulates a two cavity klystron in three dimensions. First, one cavity is pinged, and the resulting spectrum yields the resonant frequency used for later simulations. The two cavities are then loaded to simulate couplers to the cavities and the signal gain is demonstrated in a power run with an electron beam.

This simulation can be performed with a VSimMD, VSimEM or VSimPD license.

Opening the Simulation

The Klystron example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Radiation Generation (text-based setup)* option.
- Select “Klystron (text-based setup)” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 4.87 .

Input File Features

As seen in Fig. 4.88, a cylindrical tube of length TUBE_LENGTH and radius REND_TUBE is connected by a gap to two cavities. The cavities have length CAVITY_LENGTH and beginning and ending radii RBGN_CAVITY and REND_CAVITY, respectively. The centers of the cavities are a distance CAVITY_CENTER from the ends of the tube, and the gaps connecting the tube and cavities have length GAP_LENGTH. When electrons are included using INCLUDE_PARTICLES = 1, they are emitted from the end of the tube near cavity 1 in a beam with radius BEAM_RADIUS.

The following are three run types accommodated in the klystronT.pre file:

Resonant Frequency Run

The purpose of the resonant frequency run type is to identify the frequency at which to drive the klystron in later simulations. Cavity 1 is pinged and an analysis of the fourier transform of the generated gap voltage yields the resonant frequency.

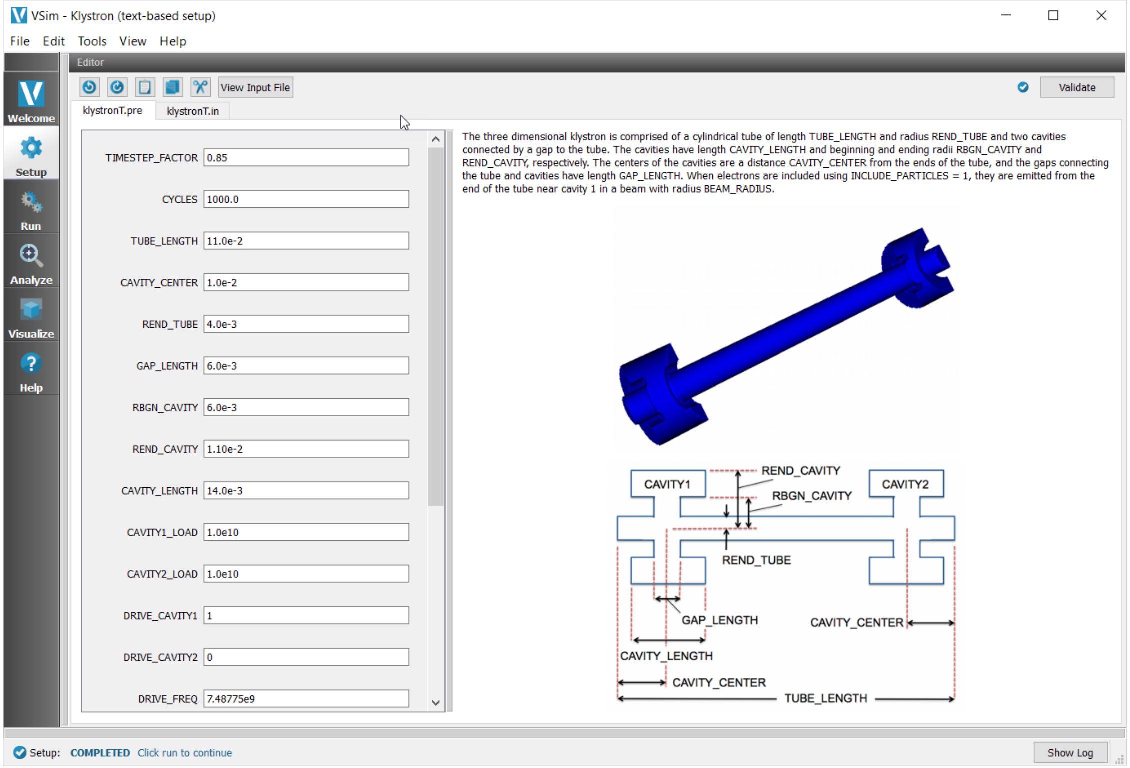
Attenuation Calibration Run

Both klystron cavities are loaded in order to simulate couplers to the cavities. The user can integrate this run type in order to calibrate the observed attenuation to the desired loss.

Power Run

Finally, an electron beam is emitted inside the klystron from the end of the tube near cavity 1. The previous runs can be iterated to ensure that the output power gain is as desired.

Designate the run type as follows:



TIMESTEP_FACTOR	0.85
CYCLES	1000.0
TUBE_LENGTH	11.0e-2
CAVITY_CENTER	1.0e-2
REND_TUBE	4.0e-3
GAP_LENGTH	6.0e-3
RBGN_CAVITY	6.0e-3
REND_CAVITY	1.10e-2
CAVITY_LENGTH	14.0e-3
CAVITY1_LOAD	1.0e10
CAVITY2_LOAD	1.0e10
DRIVE_CAVITY1	1
DRIVE_CAVITY2	0
DRIVE_FREQ	7.48775e9

The three dimensional klystron is comprised of a cylindrical tube of length TUBE_LENGTH and radius REND_TUBE and two cavities connected by a gap to the tube. The cavities have length CAVITY_LENGTH and beginning and ending radii RBGN_CAVITY and REND_CAVITY, respectively. The centers of the cavities are a distance CAVITY_CENTER from the ends of the tube, and the gaps connecting the tube and cavities have length GAP_LENGTH. When electrons are included using INCLUDE_PARTICLES = 1, they are emitted from the end of the tube near cavity 1 in a beam with radius BEAM_RADIUS.

Setup: COMPLETED Click run to continue

Fig. 4.87: Setup Window for the Klystron example.

Resonant Frequency Run

Set TURN_DRIVE_OFF = 1, DRIVE_CAVITY1 = 1, DRIVE_CAVITY2 = 0, CAVITY1_LOAD = 0.0, CAVITY2_LOAD = 0.0, and INCLUDE_PARTICLES = 0.

Attenuation Calibration Run

Set TURN_DRIVE_OFF = 1, CAVITY1_LOAD and CAVITY2_LOAD to the desired values, and INCLUDE_PARTICLES = 0. To calibrate cavity 1, set DRIVE_CAVITY1 = 1 and DRIVE_CAVITY2 = 0. To calibrate cavity 2, set DRIVE_CAVITY1 = 0 and DRIVE_CAVITY2 = 1.

Power Run

Set TURN_DRIVE_OFF = 0, DRIVE_CAVITY1 = 1, DRIVE_CAVITY2 = 0, CAVITY1_LOAD and CAVITY2_LOAD to the desired values, and INCLUDE_PARTICLES = 1.

Running the simulation

After setting the desired run type, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.89 below.

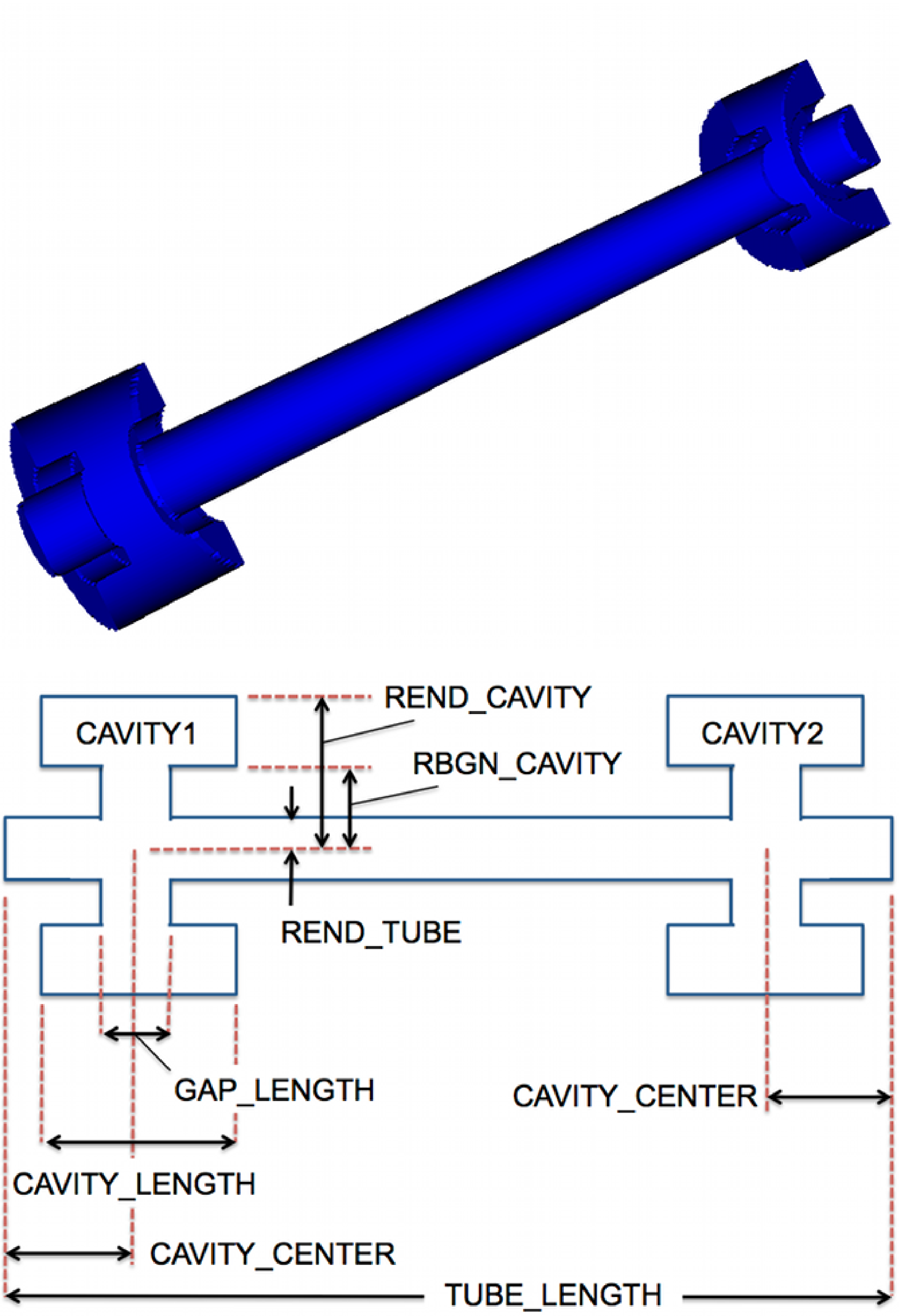
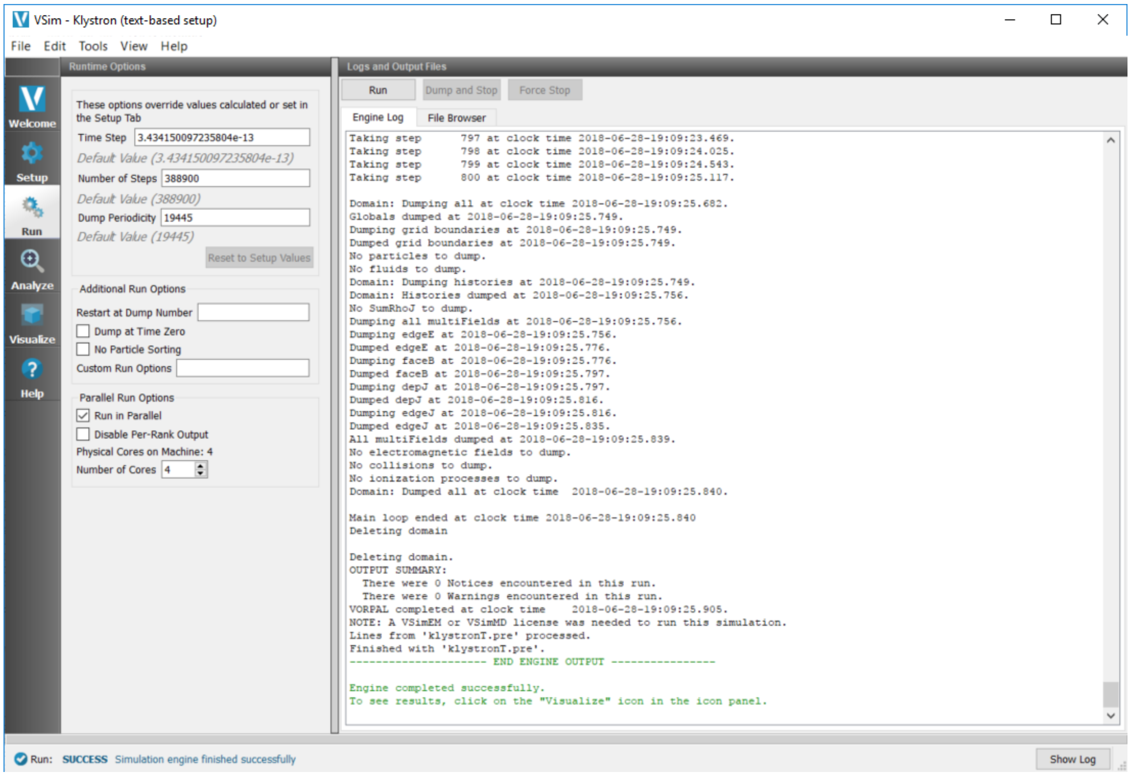


Fig. 4.88: Some exposed variables of the Klystron example.



These options override values calculated or set in the Setup Tab

Time Step 3.434150097235804e-13

Default Value (3.434150097235804e-13)

Number of Steps 388900

Default Value (388900)

Dump Periodicity 19445

Default Value (19445)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Taking step      797 at clock time 2018-06-28-19:09:23.469.
Taking step      798 at clock time 2018-06-28-19:09:24.025.
Taking step      799 at clock time 2018-06-28-19:09:24.543.
Taking step      800 at clock time 2018-06-28-19:09:25.117.

Domain: Dumping all at clock time 2018-06-28-19:09:25.682.
Globals dumped at 2018-06-28-19:09:25.749.
Dumping grid boundaries at 2018-06-28-19:09:25.749.
Dumped grid boundaries at 2018-06-28-19:09:25.749.
No particles to dump.
No fluids to dump.
Domain: Dumping histories at 2018-06-28-19:09:25.749.
Domain: Histories dumped at 2018-06-28-19:09:25.756.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-28-19:09:25.756.
Dumping edgeE at 2018-06-28-19:09:25.756.
Dumped edgeE at 2018-06-28-19:09:25.776.
Dumping faceB at 2018-06-28-19:09:25.776.
Dumped faceB at 2018-06-28-19:09:25.797.
Dumping depJ at 2018-06-28-19:09:25.797.
Dumped depJ at 2018-06-28-19:09:25.816.
Dumping edgeJ at 2018-06-28-19:09:25.816.
Dumped edgeJ at 2018-06-28-19:09:25.835.
All multiFields dumped at 2018-06-28-19:09:25.839.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-28-19:09:25.840.

Main loop ended at clock time 2018-06-28-19:09:25.840
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-28-19:09:25.905.
NOTE: A VSimEM or VSimMD license was needed to run this simulation.
Lines from 'klystronT.pre' processed.
Finished with 'klystronT.pre'.
-------------------- END ENGINE OUTPUT ---------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 4.89: The Run Window at the end of execution.

Visualizing the results

After running the desired run type, continue as follows:

- Proceed to the Visualize Window by pressing the *Visualize* button in the left column of buttons.

Resonant Frequency Run

To visualize a run to determine the resonant frequency, select *History* from the *Data View* pull-down menu at the top of the *CONTROLS* pane. Select Cavity1_Voltage in the *CONTROLS* pane, and click FFT to the left of the Cavity1_Voltage plot in the *VISUALIZATION* pane. The resulting plot will resemble Fig. 4.90. Zoom in on the maximum of this plot to determine the resonant frequency.

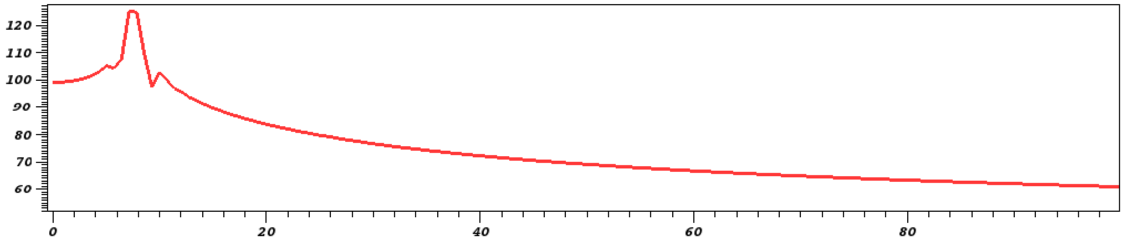
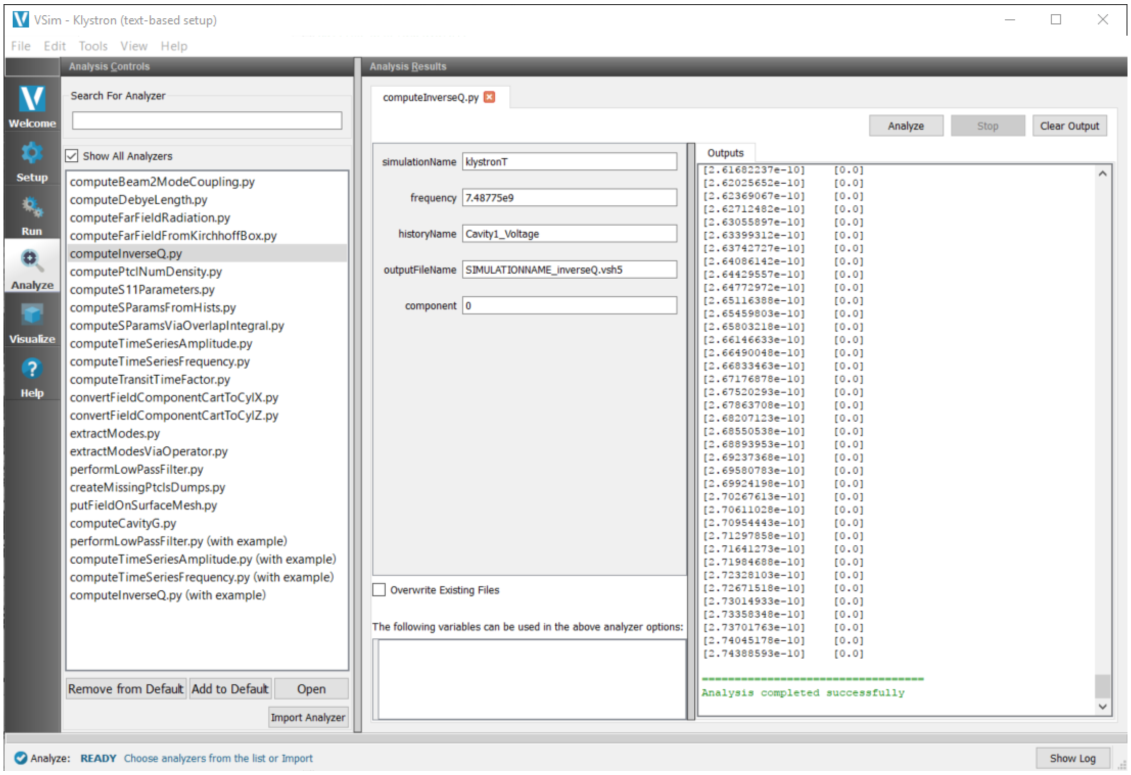


Fig. 4.90: Fourier transform of Cavity1_Voltage versus time (in GHz).

Attenuation Calibration Run

For the **Attenuation Calibration Run**, the quality factors Q_1 and Q_2 for cavities 1 and 2 can be calculated using the *computeInverseQ - Compute Inverse Q Analysis* as follows:

- Press the Analyze button in the left column of buttons.
- Select *computeInverseQ.py* from the pull down menu in the *Control* section of the window.
- Enter Cavity1_Voltage or Cavity2_Voltage in the *history* field to designate the history to analyze.
- Enter the value of the input parameter DRIVE_FREQ in the *frequency* field to designate the frequency at which the history will be analyzed.
- Click the *Analyze* button in the top right corner of the window. As shown in Fig. 4.91, two columns of data with the titles "Time (s)" and "Inverse Q" will be output in the right pane. The analysis has completed when you see the output "Analysis completed successfully."


VSim - Klystron (text-based setup)
File Edit Tools View Help
Analysis Controls
Welcome
Setup
Run
Analyze
Visualize
Help
Search For Analyzer
☑ Show All Analyzers
computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
createMissingPtclsDumps.py
putFieldOnSurfaceMesh.py
computeCavityG.py
performLowPassFilter.py (with example)
computeTimeSeriesAmplitude.py (with example)
computeTimeSeriesFrequency.py (with example)
computeInverseQ.py (with example)
Remove from Default
Add to Default
Open
Import Analyzer
Analysis Results
computeInverseQ.py
Analyze
Stop
Clear Output
Outputs
☐ Overwrite Existing Files
The following variables can be used in the above analyzer options:
Analyze: READY Choose analyzers from the list or Import
Show Log

simulationName klystronT

frequency 7.48775e9

historyName Cavity1_Voltage

outputFileName SIMULATIONNAME_inverseQ.vsh5

component 0

```
[2.61682237e-10]    [0.0]
[2.62025652e-10]    [0.0]
[2.62369067e-10]    [0.0]
[2.62712482e-10]    [0.0]
[2.63055897e-10]    [0.0]
[2.63399312e-10]    [0.0]
[2.63742727e-10]    [0.0]
[2.64086142e-10]    [0.0]
[2.64429557e-10]    [0.0]
[2.64772972e-10]    [0.0]
[2.65116388e-10]    [0.0]
[2.65459803e-10]    [0.0]
[2.65803218e-10]    [0.0]
[2.66146633e-10]    [0.0]
[2.66490048e-10]    [0.0]
[2.66833463e-10]    [0.0]
[2.67176878e-10]    [0.0]
[2.67520293e-10]    [0.0]
[2.67863708e-10]    [0.0]
[2.68207123e-10]    [0.0]
[2.68550538e-10]    [0.0]
[2.68893953e-10]    [0.0]
[2.69237368e-10]    [0.0]
[2.69580783e-10]    [0.0]
[2.69924198e-10]    [0.0]
[2.70267613e-10]    [0.0]
[2.70611028e-10]    [0.0]
[2.70954443e-10]    [0.0]
[2.71297858e-10]    [0.0]
[2.71641273e-10]    [0.0]
[2.71984688e-10]    [0.0]
[2.72328103e-10]    [0.0]
[2.72671518e-10]    [0.0]
[2.73014933e-10]    [0.0]
[2.73358348e-10]    [0.0]
[2.73701763e-10]    [0.0]
[2.74045178e-10]    [0.0]
[2.74388593e-10]    [0.0]

=====================================
Analysis completed successfully
```

Fig. 4.91: The Analysis window at the end of execution of the computeInverseQ.py script.

Scrolling through or plotting the output data enables the user to calculate the values of $1/Q_1$ and $1/Q_2$, and thus Q_1 and Q_2. The user may iterate this run type to calibrate the quality factors Q_1 and Q_2 by varying the values of the input parameters CAVITY1_LOAD and CAVITY2_LOAD. Note that Q_1 and Q_2 are inversely proportional to CAVITY1_LOAD and CAVITY2_LOAD, respectively.

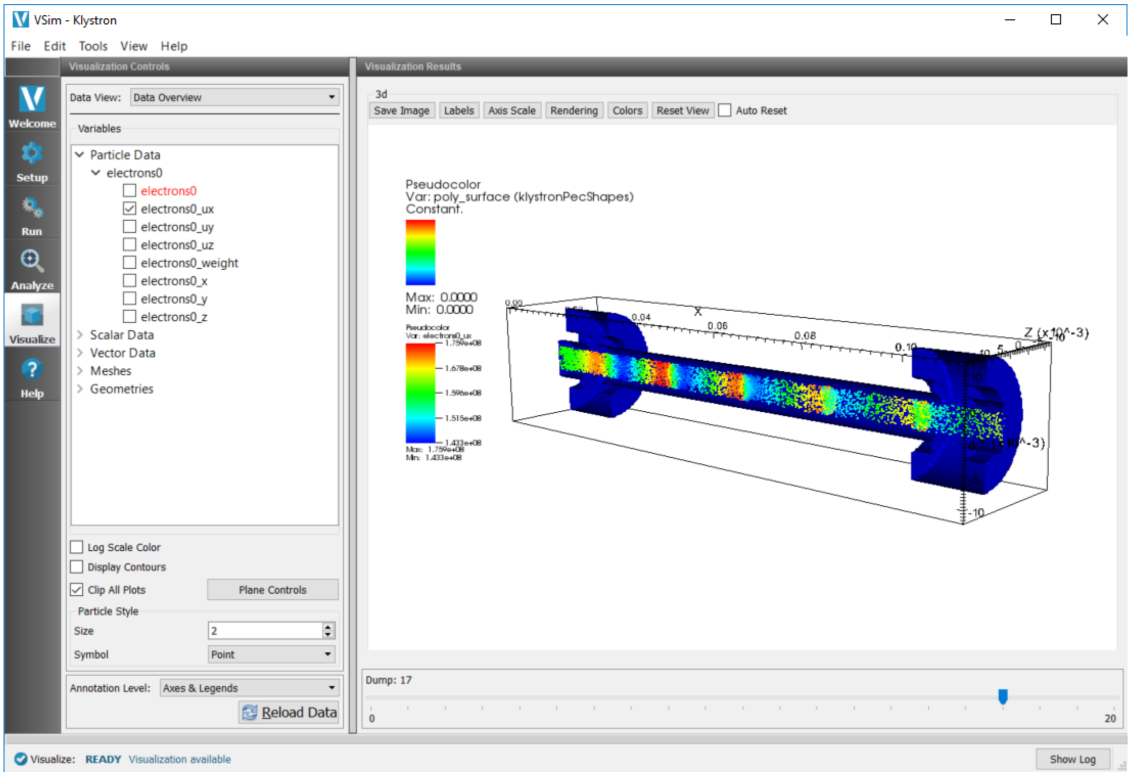
Power Run

In the power run, we introduce an electron beam to the simulation as seen in Fig. 4.92. You can reproduce this image by doing the following:

- Press the *Visualize* button in the left column of buttons.
- Expand *Particle Data*
- Expand *electrons0*
- Select *electrons0_ux*
- Expand *Geometries*

- Select *poly_surface (klystronPecShapes)*
- Select *Display Contours*
- Select *Clip All Plots*

You can change the color table to something that better shows the physics, and in this image the color table is set to “hot_desaturated”.



Data View: Data Overview

Variables

- Particle Data
  - electrons0
    - □ electrons0
    - ☑ electrons0_ux
    - □ electrons0_uy
    - □ electrons0_uz
    - □ electrons0_weight
    - □ electrons0_x
    - □ electrons0_y
    - □ electrons0_z
- Scalar Data
- Vector Data
- Meshes
- Geometries

□ Log Scale Color

□ Display Contours

☑ Clip All Plots

Particle Style

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 17

Visualize: READY Visualization available

Fig. 4.92: A power run with an electron beam.

Further Experiments

Try varying the parameter TUBE_LENGTH in order to maximize the gain.

4.4.3 2D Magnetron (magnetron2DT.pre)

Keywords:

magnetron, electromagnetic cavities, mode frequencies

Problem description

This VSimMD example simulates a rising sun magnetron in two dimensions. The spectrum of the magnetron is first tuned through geometric properties, yielding the operating frequency. A load is then added to one cavity, representing a coupler to the magnetron through the quality factor, Q. Upon configuring an electrostatic voltage across the anode and cathode, particles are introduced to the simulation, exhibiting a five spoke pi-mode.

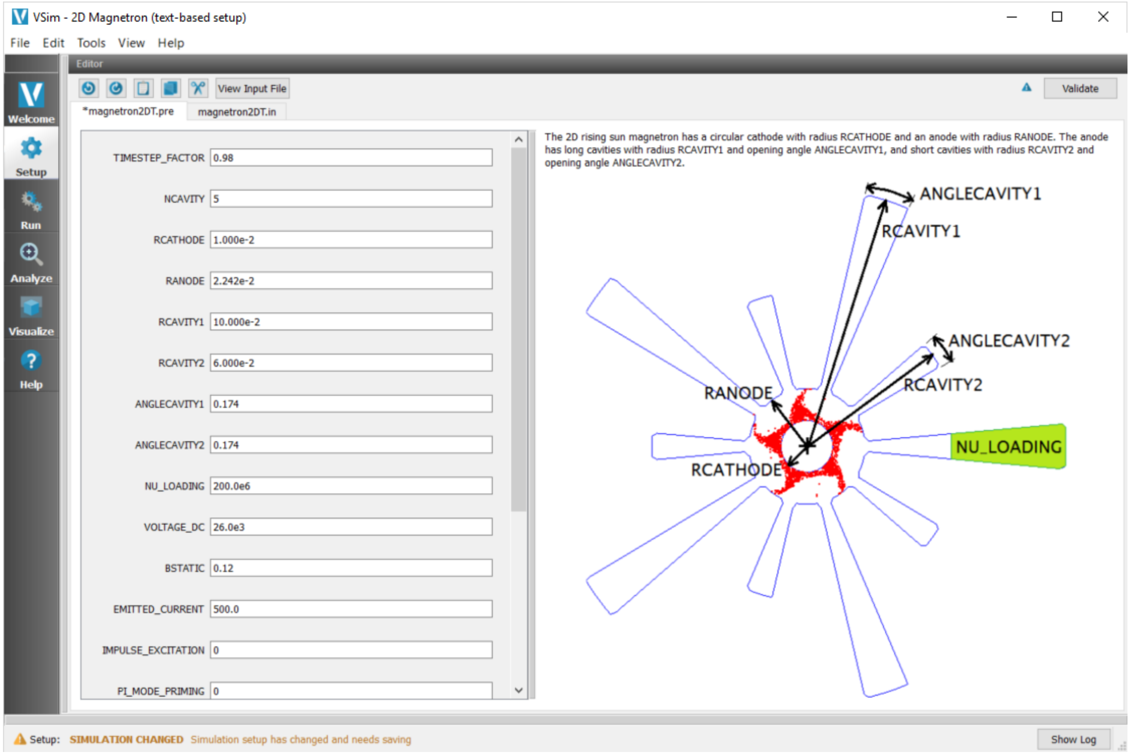
This simulation can be performed with a VSimMD or VSimPD license.

Opening the Simulation

The 2D Magnetron example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Radiation Generation (text-based setup)* option.
- Select “2D Magnetron (text-based setup)” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 4.93.



TIMESTEP_FACTOR 0.98

NCAVITY 5

RCATHODE 1.000e-2

RANODE 2.242e-2

RCAVITY1 10.000e-2

RCAVITY2 6.000e-2

ANGLECAVITY1 0.174

ANGLECAVITY2 0.174

NU_LOADING 200.0e6

VOLTAGE_DC 26.0e3

BSTATIC 0.12

EMITTED_CURRENT 500.0

IMPULSE_EXCITATION 0

PI_MODE_PRIMING 0

The 2D rising sun magnetron has a circular cathode with radius RCATHODE and an anode with radius RANODE. The anode has long cavities with radius RCAVITY1 and opening angle ANGLECAVITY1, and short cavities with radius RCAVITY2 and opening angle ANGLECAVITY2.

Setup: SIMULATION CHANGED Simulation setup has changed and needs saving

Fig. 4.93: Setup Window for the 2D Magnetron example.

Input File Features

As seen in Fig. 4.94 of the rising sun magnetron, the radius of the cathode is RCATHODE and the radius of the anode is RANODE. Long cavities have radius RCAVITY1 and opening angle ANGLECAVITY1. Short cavities have radius RCAVITY2 and opening angle ANGLECAVITY2. These geometrical properties control the spectrum and thus the operating frequency, which for the default parameters is approximately 960 MHz.

One long cavity is loaded with damping parameter NU_LOADING, which may be used to tune the quality factor Q of the magnetron, which is inversely proportional to NU_LOADING. Once the design of the magnetron is specified, electrons may be emitted from the cathode by setting INCLUDE_PARTICLES = 1.

The following are four run types accommodated in the magnetron2DT.pre file:

The Mode Spectrum

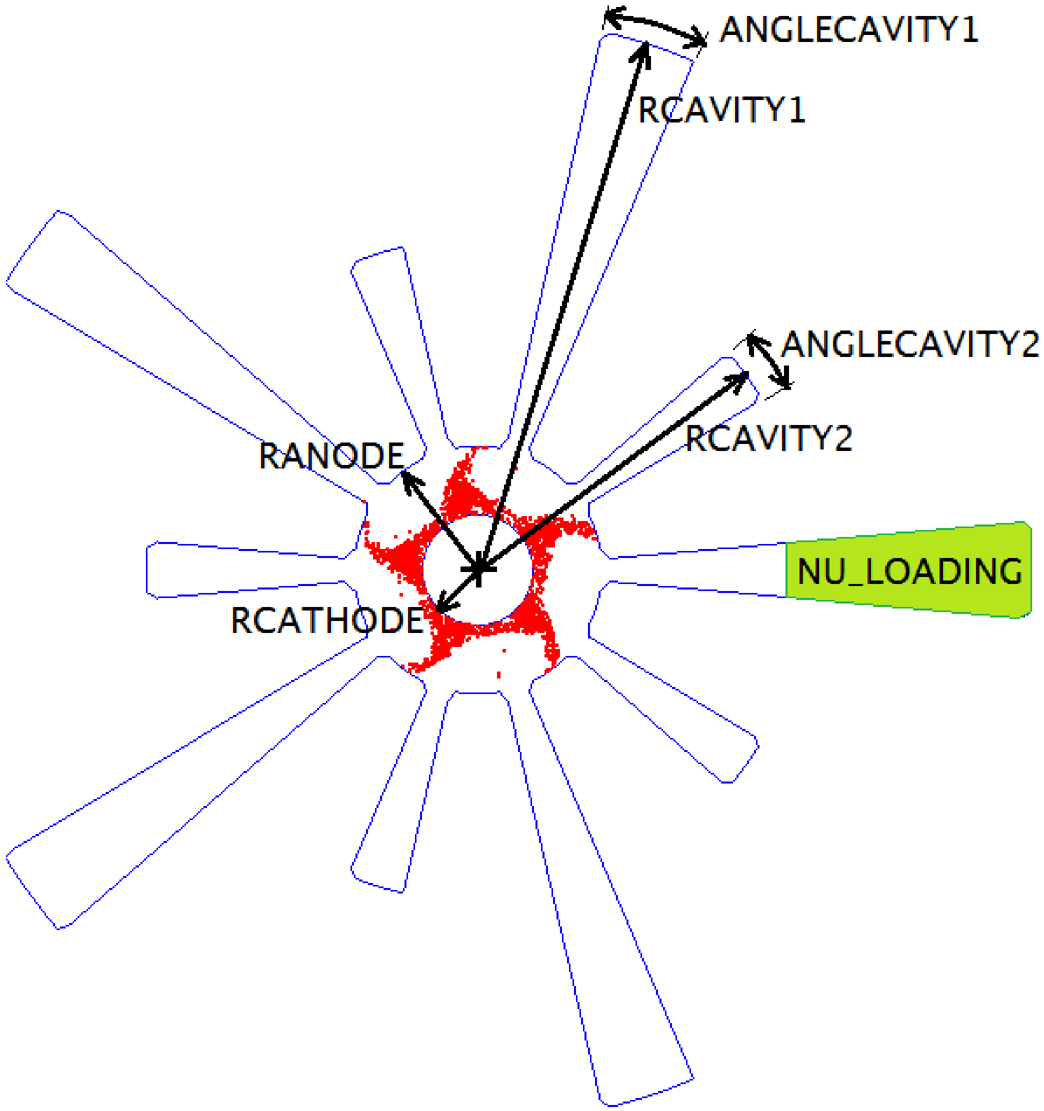


Fig. 4.94: Some exposed variables of the 2D Magnetron example.

The magnetron is rung up with a magnetic field perpendicular to the plane of the magnetron, confined to one of the long cavities. The generated voltage spectrum can then be analyzed to anticipate the magnetron operating frequency.

Calibrating the Pi-Mode and Quality Factor

The magnetron is again rung up, but with a magnetic field profile with null lines part way up a long cavity arm at the approximate location required for a pi-mode. At this point, the magnetron geometry can be modified to yield the desired operating frequency. The load can also be tuned at this point to result in the appropriate cavity quality factor.

Calibrating the DC Voltage

A specific DC voltage between the anode and cathode is required to support a given operating mode for particles in the magnetron. Thus, in this run type, an electrostatic field generated between the cathode and anode is tuned using a combined source with feedback and drain, and a finite-difference divergence-free 1/r current profile.

Run with Particles

Finally, electrons are emitted into the magnetron from the cathode. The previous runs can be iterated to ensure that the electrons exhibit the desired mode.

The four basic run types may be configured as follows:

The Mode Spectrum

Set PI_MODE_PRIMING = 0, INCLUDE_DC_VOLTAGE = 0, INCLUDE_PARTICLES = 0, and IMPULSE_EXCITATION = 1.

Calibrating the Pi-Mode and Quality Factor

Set IMPULSE_EXCITATION = 0, INCLUDE_DC_VOLTAGE = 0, INCLUDE_PARTICLES = 0, and PI_MODE_PRIMING = 1.

Calibrating the DC Voltage

Set IMPULSE_EXCITATION = 0, PI_MODE_PRIMING = 0, INCLUDE_PARTICLES = 0, and INCLUDE_DC_VOLTAGE = 1.

Run with Particles

Set IMPULSE_EXCITATION = 0, PI_MODE_PRIMING = 0, INCLUDE_DC_VOLTAGE = 1, and INCLUDE_PARTICLES = 1.

Running the simulation

For each of the setups described above, continue as follows:

- Proceed to the Run Window by pressing the *Run* button in the left column.
- To run the file, click on the *Run* button in the upper left corner of the right pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.95.

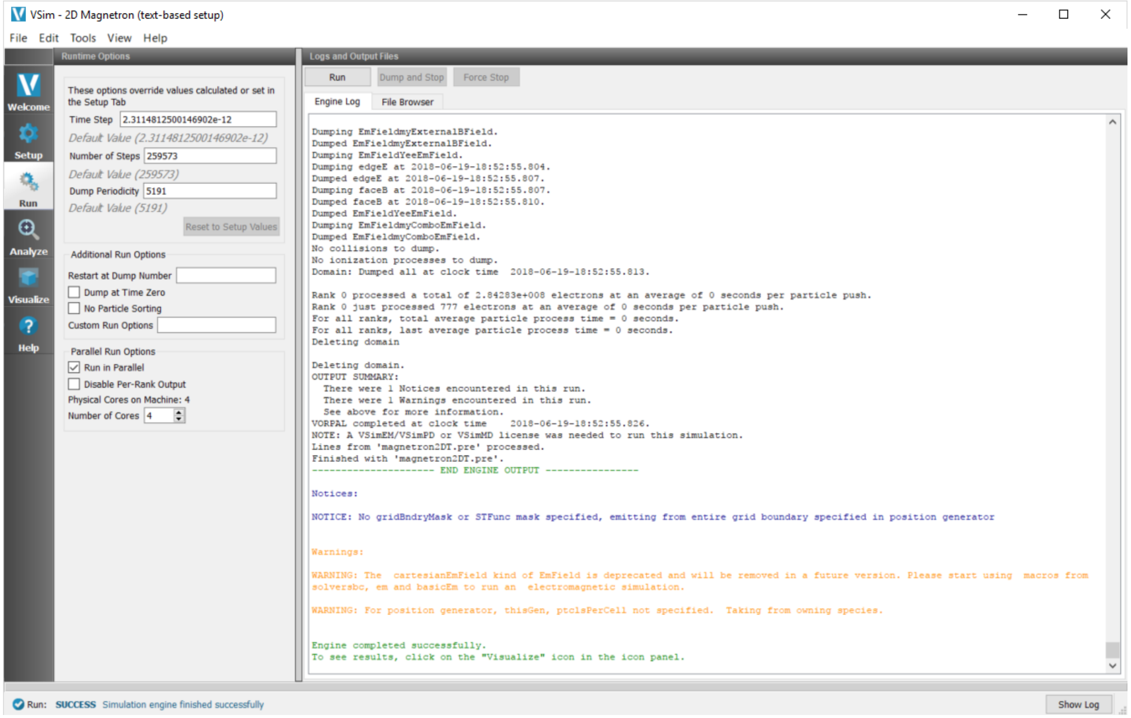
Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column.

Visualization of the four basic run types is described below.

The Mode Spectrum



These options override values calculated or set in the Setup Tab

Time Step 2.3114812500146902e-12
Default Value (2.3114812500146902e-12)
Number of Steps 259573
Default Value (259573)
Dump Periodicity 5191
Default Value (5191)

Additional Run Options
Restart at Dump Number
□ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options
☑ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
Dumping EmFieldmyExternalBField.
Dumped EmFieldmyExternalBField.
Dumping EmFieldYeeEmField.
Dumping edgeE at 2018-06-19-18:52:55.804.
Dumped edgeE at 2018-06-19-18:52:55.807.
Dumping faceB at 2018-06-19-18:52:55.807.
Dumped faceB at 2018-06-19-18:52:55.810.
Dumped EmFieldYeeEmField.
Dumping EmFieldmyComboEmField.
Dumped EmFieldmyComboEmField.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-19-18:52:55.813.

Rank 0 processed a total of 2.84283e+008 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 777 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 1 Notices encountered in this run.
  There were 1 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-06-19-18:52:55.826.
NOTE: A VSimEM/VSimPD or VSimMD license was needed to run this simulation.
Lines from 'magnetron2DT.pre' processed.
Finished with 'magnetron2DT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: No gridBndryMask or STFunc mask specified, emitting from entire grid boundary specified in position generator


Warnings:

WARNING: The  cartesianEmField kind of EmField is deprecated and will be removed in a future version. Please start using  macros from
solversbc, em and basicEm to run an  electromagnetic simulation.

WARNING: For position generator, thisGen, ptclsPerCell not specified.  Taking from owning species.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.95: The Run Window at the end of execution.

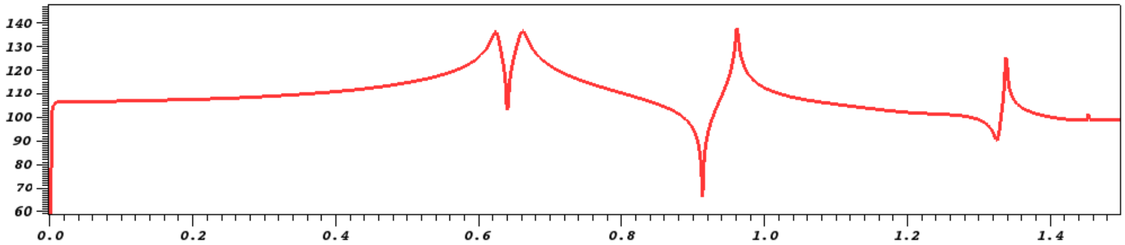


Fig. 4.96: Fourier transform of voltage across rung up cavity as a function of frequency (in GHz).

In Fig. 4.96, the fourier transform of the voltage across the rung up cavity is plotted versus frequency (in GHz). This indicates that we should expect modes near 650 MHz and 960 MHz. To generate this plot, select *History* from the *Data View* pull-down menu at the top of the *Visualization Controls* pane. Choose the *cavity1Voltage* plot in the. Then in the *Visualization Results* pane, click FFT to the left of the cavity1Voltage plot, and zoom in to the relevant area.

Calibrating the Pi-Mode and Quality Factor

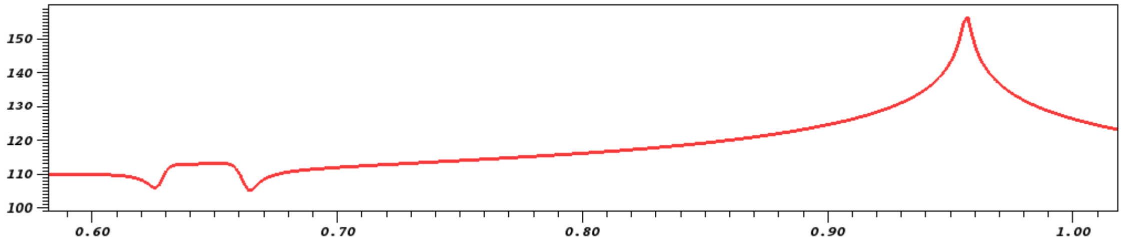


Fig. 4.97: Fourier transform of voltage across rung up cavity as a function of frequency (in GHz).

In Fig. 4.97, the fourier transform of the voltage across the rung up cavity is plotted versus frequency (in GHz). The 960 MHz mode is more pronounced than in the previous run, but the 650 MHz mode remains. If we instead seek another operating frequency, we could change the geometry of the magnetron to iteratively tune this spectrum.

Another quantity we may wish to tune is the magnetron quality factor, Q. We expect Q to be inversely proportional to NU_LOADING, but it is possible to measure Q using VSim. Using the decay of the cavity1Voltage history, Q may be calculated as

$$Q = \pi \ f \ \left(t_2 - t_1\right) \ \ln^{-1}\left[\frac{V\left(t_1\right)}{V\left(t_2\right)}\right]$$

where f is the operating frequency, $V\left(t\right)$ is measured as in Fig. 4.98. We calculate Q to be 400 with f = 960 MHz.

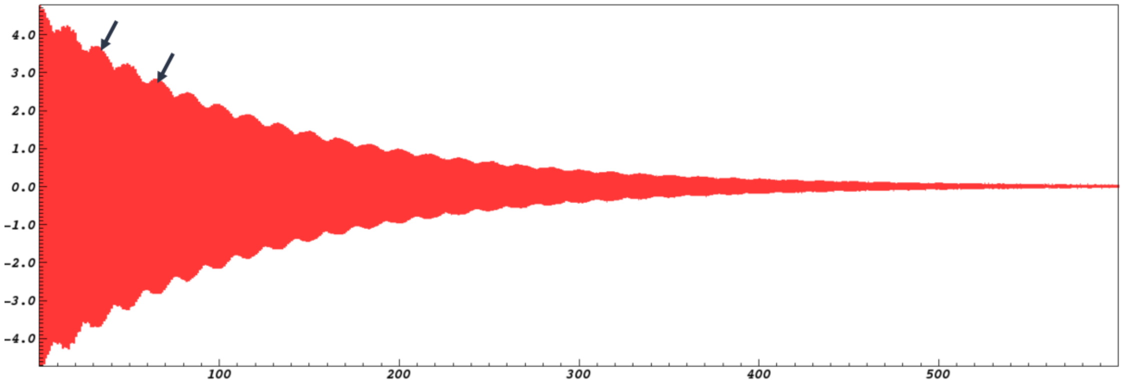


Fig. 4.98: Decay of voltage across cavity 1 (in kV) as a function of time (in ns). The two indicated points could be used to calculate the quality factor.

Calibrating the DC Voltage

An electrostatic field is generated between the cathode and anode using a combined source with feedback and drain, and a finite-difference divergence-free $1/r$ current profile. In Fig. 4.99, the resulting voltage between the anode and cathode, the history cathodeAnodeVoltage (in kV), is plotted versus time (in ns). The user controls the average value of this voltage through the variable VOLTAGE_DC.

Run with Particles

When electrons are emitted from the cathode, the four spoke, 650 MHz is present during startup. At approximately 250 ns, the five spoke begins to dominate and eventually appears as seen in Fig. 4.100.

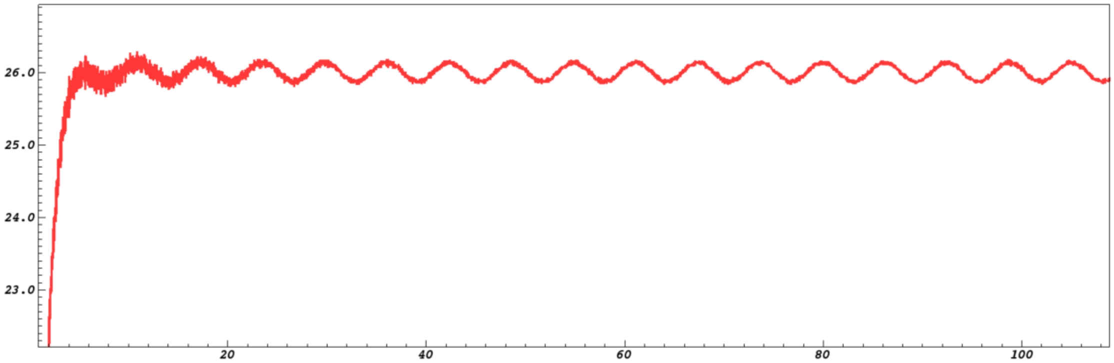
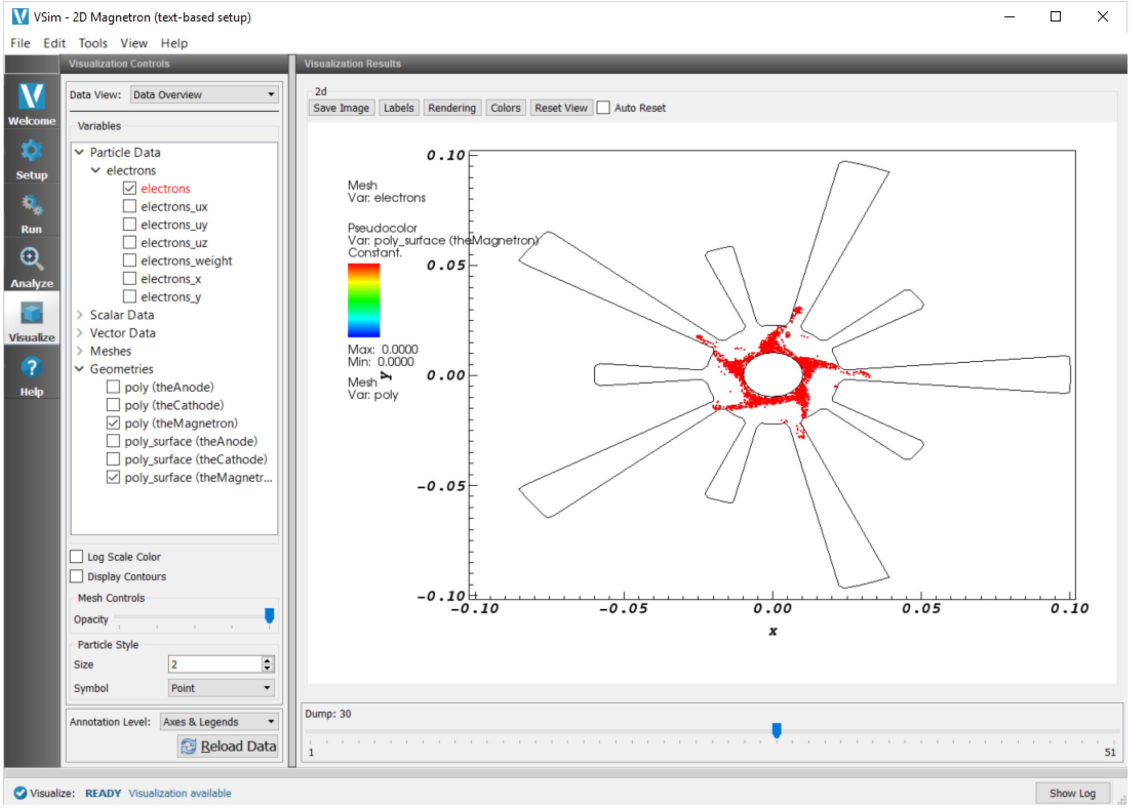


Fig. 4.99: Voltage between the anode and cathode (in kV) as a function of time (in ns), recorded through the history cathodeAnodeVoltage.



Data View: Data Overview

Variables

- Particle Data
  - electrons
    - ☑ electrons
    - ☐ electrons_ux
    - ☐ electrons_uy
    - ☐ electrons_uz
    - ☐ electrons_weight
    - ☐ electrons_x
    - ☐ electrons_y
- Scalar Data
- Vector Data
- Meshes
- Geometries
  - ☐ poly (theAnode)
  - ☐ poly (theCathode)
  - ☑ poly (theMagnetron)
  - ☐ poly_surface (theAnode)
  - ☐ poly_surface (theCathode)
  - ☑ poly_surface (theMagnetr...

☐ Log Scale Color

☐ Display Contours

Mesh Controls

Opacity

Particle Style

Size 2

Symbol Point

Annotation Level: Axes & Legends

☐ Auto Reset

Dump: 30

Visualize: READY Visualization available

Fig. 4.100: The five spoke pi-mode.

Further Experiments

Try varying RCATHODE and observing the effect on the spoke formation.

4.5 Multipacting

4.5.1 Multipacting Growth in Waveguide (multipactingGrowth.sdf)

Keywords:

`multipacting`

Problem description

Multipacting, which is the resonant build up of secondary electrons, is often a concern in microwave devices. Anytime there is an oscillating electromagnetic field across a gap between two surfaces there exists the possibility that for the right voltage across the gap a resonance condition will exist allowing the exponential build up of secondary electrons. A coaxial waveguide is such a type of structure where these conditions can exist.

This simulation can be performed with the VSimMD or VSimPD license.

Opening the Simulation

The Multipacting Growth example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Multipacting* option.
- Select “Multipacting Growth in Waveguide” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 4.101. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.

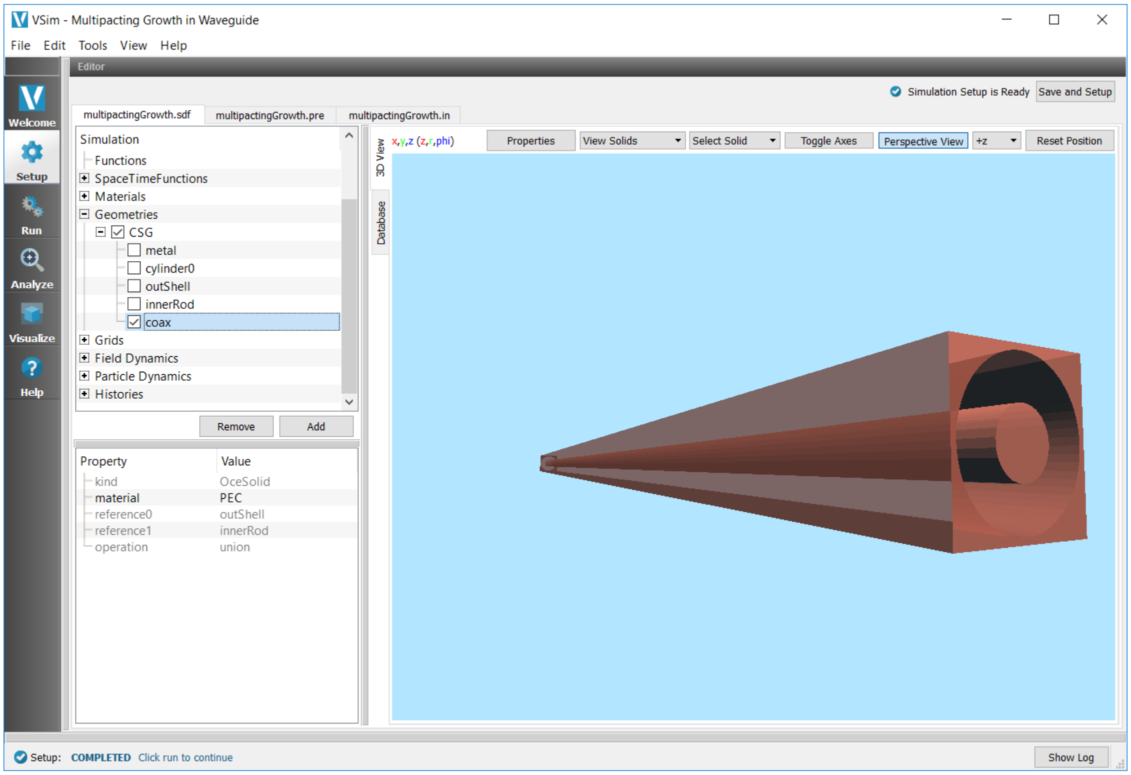
Simulation Properties

This example contains a number of *Parameters* to allow for easy manipulation of the device. Those include:

- R_O: The outer coax radius
- R_I: The inner coax radius
- FREQUENCY: The wave launcher frequency

SpaceTimeFunctions are used to create expressions defining the drive frequency and amplitude of the applied field.

CSG is used to create the coax structure by combining cylinders and cubes.



Simulation Setup is Ready

Property	Value
kind	OceSolid
material	PEC
reference0	outShell
reference1	innerRod
operation	union

Setup: COMPLETED Click run to continue

Fig. 4.101: Setup Window for the Multipacting Growth example.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane. You will see the output of the run in that pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.102

Visualizing the results

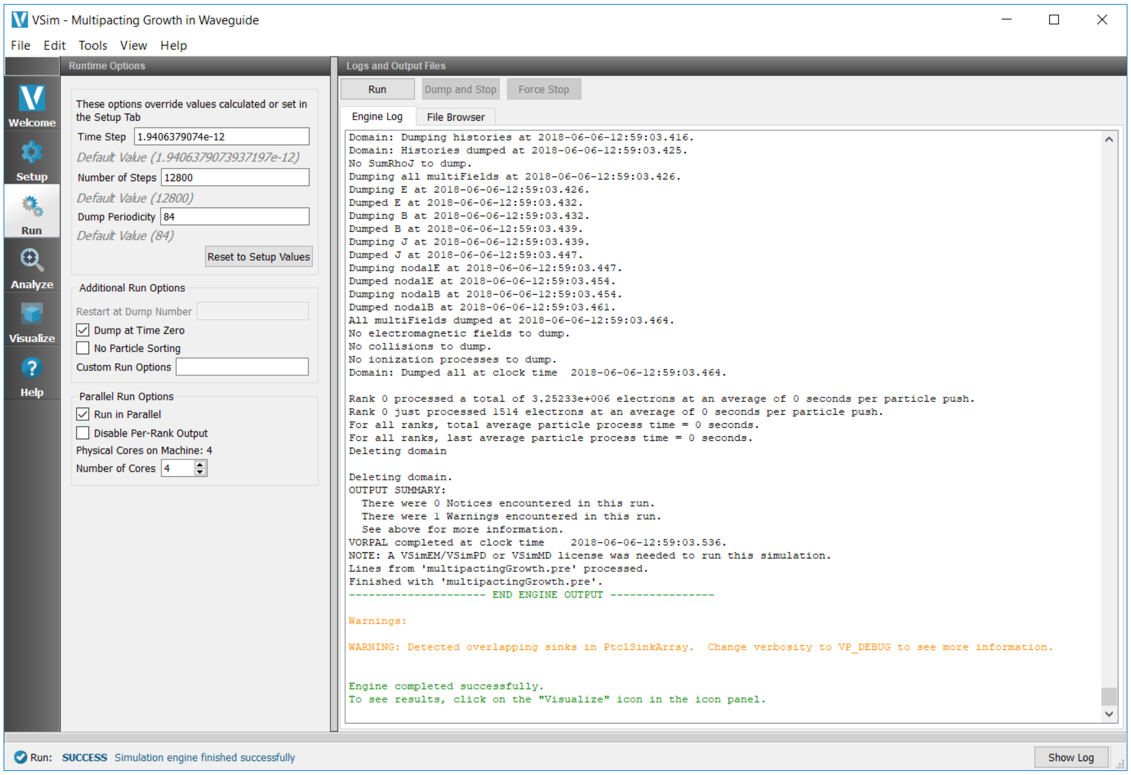
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view growth in the number of electrons, as shown in Fig. 4.103, do the following:

- Select *History* from the *Data View* pull down menu
- Set Graphs 1&2 to “None”
- Graph 3 should already be set to *numElectrons* (if not, set it)

The overall trend in the number of electrons is an exponential growth with an oscillatory signal that corresponds to the frequency of the traveling wave.



These options override values calculated or set in the Setup Tab

Time Step 1.9406379074e-12
Default Value (1.9406379073937197e-12)
Number of Steps 12800
Default Value (12800)
Dump Periodicity 84
Default Value (84)

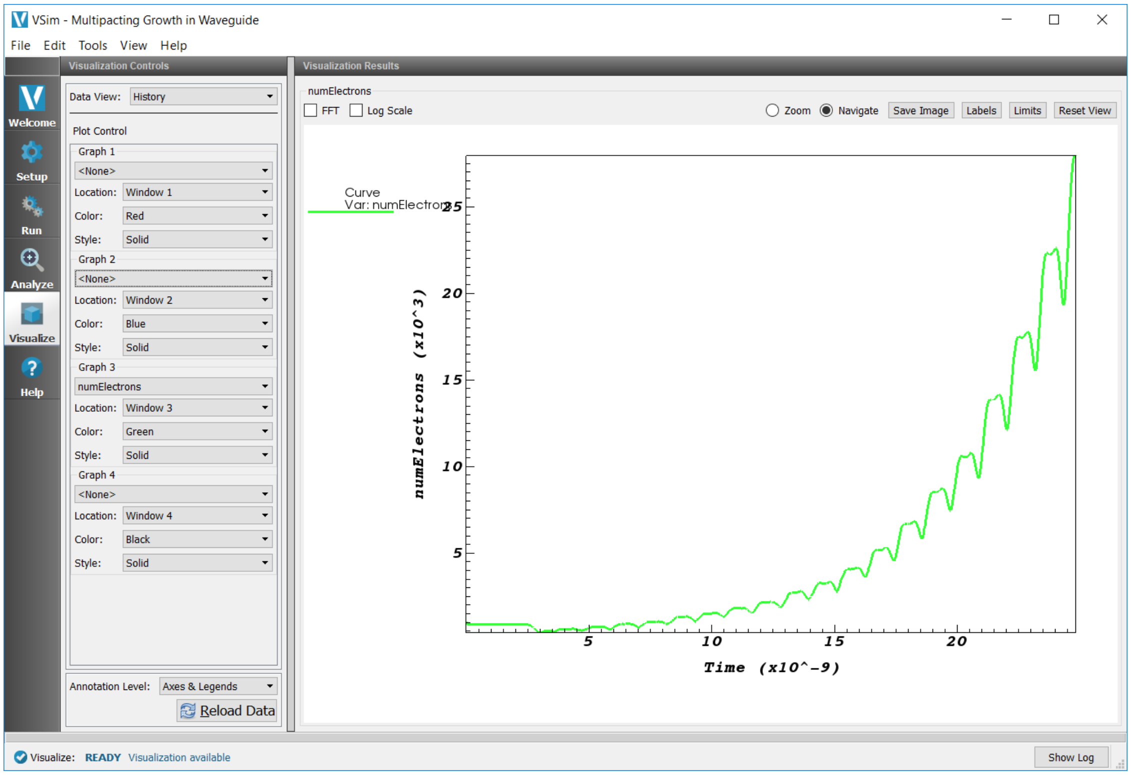
Additional Run Options
Restart at Dump Number
☑ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options
☑ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
Domain: Dumping histories at 2018-06-06-12:59:03.416.
Domain: Histories dumped at 2018-06-06-12:59:03.425.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-06-12:59:03.426.
Dumping E at 2018-06-06-12:59:03.426.
Dumped E at 2018-06-06-12:59:03.432.
Dumping B at 2018-06-06-12:59:03.432.
Dumped B at 2018-06-06-12:59:03.439.
Dumping J at 2018-06-06-12:59:03.439.
Dumped J at 2018-06-06-12:59:03.447.
Dumping nodalE at 2018-06-06-12:59:03.447.
Dumped nodalE at 2018-06-06-12:59:03.454.
Dumping nodalB at 2018-06-06-12:59:03.454.
Dumped nodalB at 2018-06-06-12:59:03.461.
All multiFields dumped at 2018-06-06-12:59:03.464.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-06-12:59:03.464.

Rank 0 processed a total of 3.25233e+006 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 1514 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 1 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-06-06-12:59:03.536.
NOTE: A VSimEM/VSimPD or VSimMD license was needed to run this simulation.
Lines from 'multipactingGrowth.pre' processed.
Finished with 'multipactingGrowth.pre'.
-------------------- END ENGINE OUTPUT ----------------

Warnings:

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 4.102: The Run Window at the end of execution.



Data View: History

Plot Control
Graph 1
<None>
Location: Window 1
Color: Red
Style: Solid
Graph 2
<None>
Location: Window 2
Color: Blue
Style: Solid
Graph 3
numElectrons
Location: Window 3
Color: Green
Style: Solid
Graph 4
<None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

□ FFT □ Log Scale

Visualize: **READY** Visualization available

Fig. 4.103: Visualization of the exponential growth of the electrons due to multipacting.

Further Experiments

Try changing the gap voltage or the frequency of the wave to see if one can take the simulation in and out of resonance.

4.5.2 Multipacting Resonances in Waveguide (multipactingResonances.sdf)

Keywords:

multipacting , multipactingResonances

Problem description

Multipacting, which is the resonant build up of secondary electrons, is often a concern in microwave devices. Anytime there is an oscillating electromagnetic field across a gap between two surfaces there exists the possibility that for the right voltage across the gap a resonance condition will exist allowing the exponential build up of secondary electrons. A coaxial waveguide is such a structure where these conditions can exist.

This simulation can be performed with the VSimMD license.

Opening the Simulation

The Multipacting Resonances example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Multipacting* option.
- Select “Multipacting Resonances in Waveguide” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is now shown with all the implemented physics and geometries, if applicable. See Fig. 4.104.

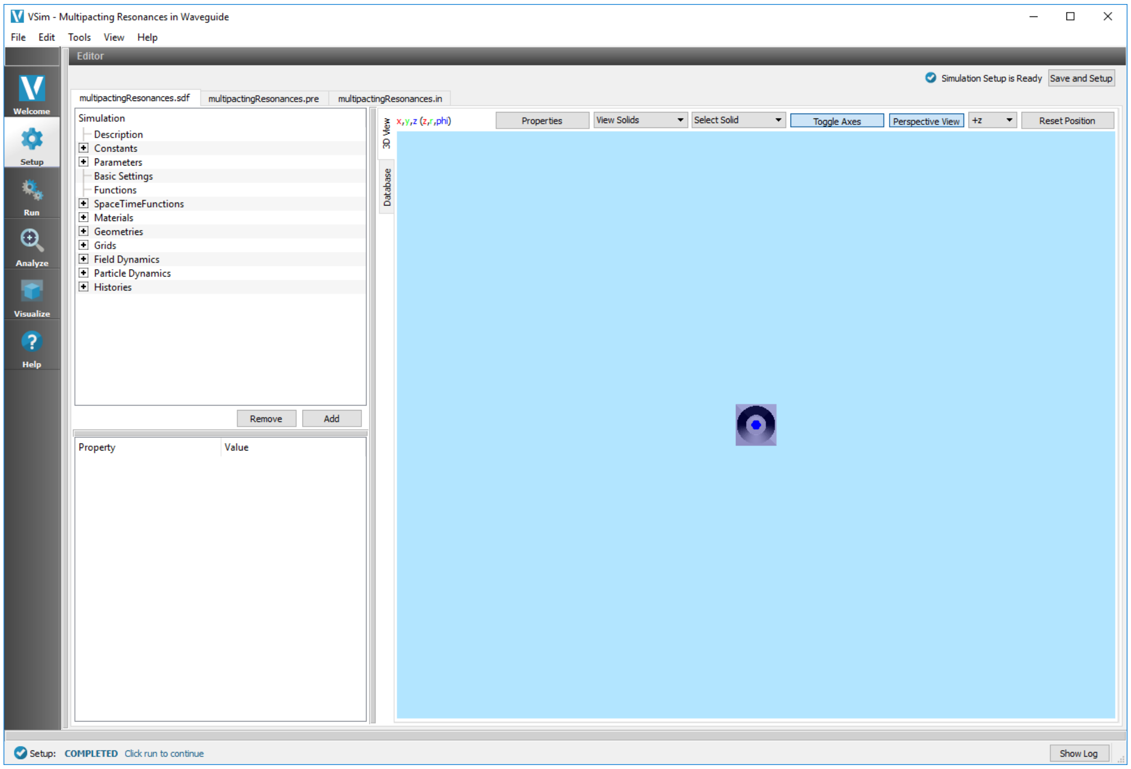
Simulation Properties

The input file sets the number of cells along the propagation (x) direction to resolve the wavelength. The electrons are seeded in the middle of the waveguide once the wave has passed. A special electron species is used that allows scans over power to be done in a single simulation. The time step is chosen to be at 90% of the CFL (stability) limit.

Running the simulation

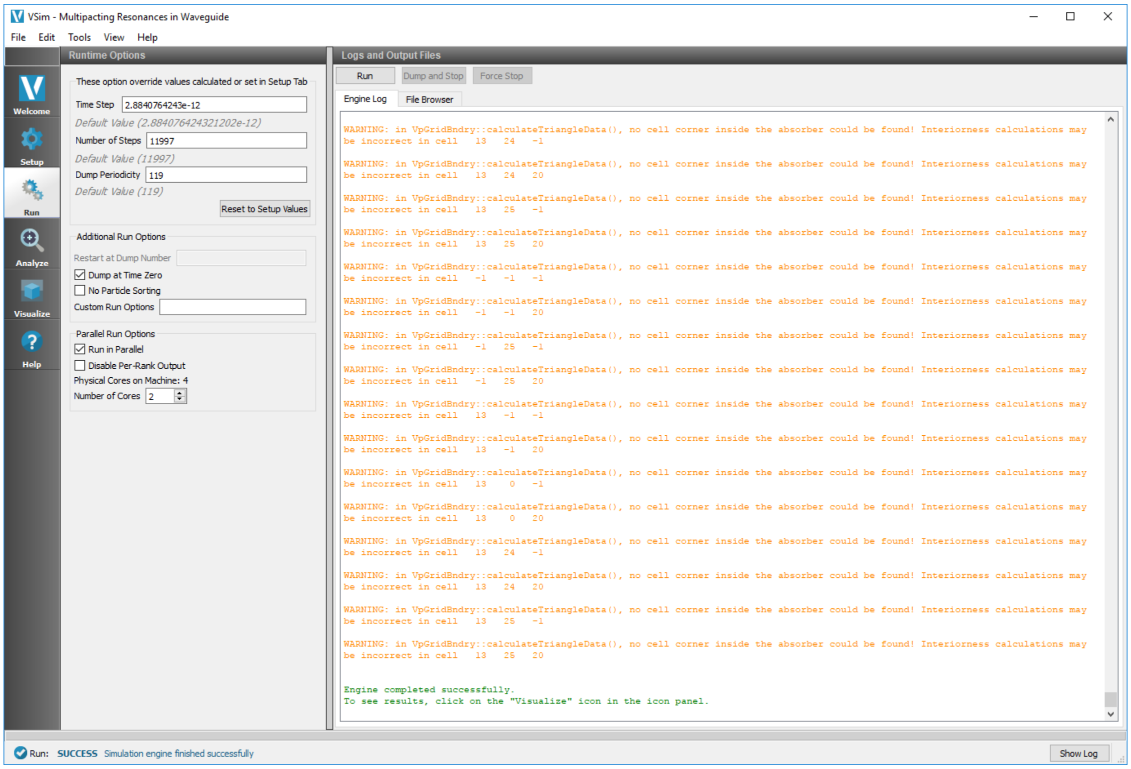
After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner. of the window. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.105.



Setup: COMPLETED Click run to continue

Fig. 4.104: Setup Window for the Multipacting Resonances example.



These option override values calculated or set in Setup Tab

Time Step 2.8840764243e-12

Default Value (2.884076424321202e-12)

Number of Steps 11997

Default Value (11997)

Dump Periodicity 119

Default Value (119)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 2

```
WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13   24   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13   24   20

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13   25   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13   25   20

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   -1   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   -1   -1   20

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   -1   25   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   -1   25   20

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13   -1   20

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13    0   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13    0   20

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13   24   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13   24   20

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13   25   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13   25   20


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.105: The Run Window at the end of execution.

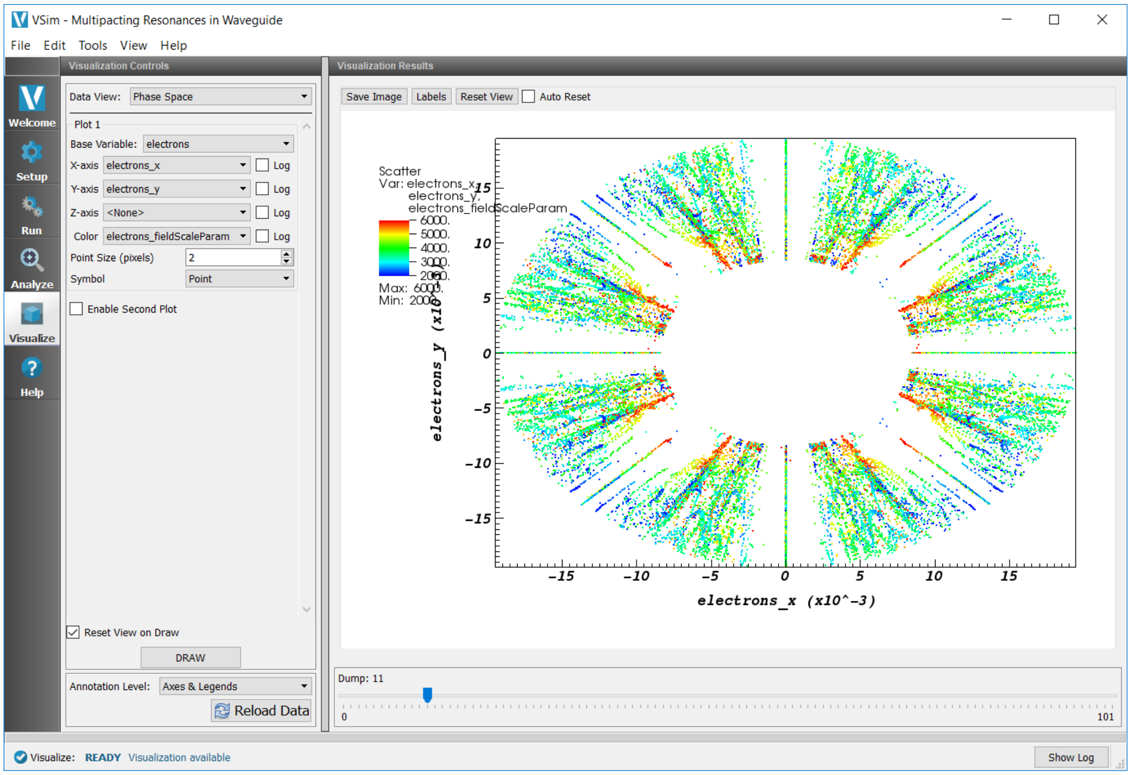
Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To track the electrons and their field scaled paramteres as shown in Fig. 4.106, proceed as follows:

- Select *Phase Space* from the *Data View* pull down menu (at the top of the *Controls* pane)
- Select *electrons_x* for the *X-axis*
- Select *electrons_y* for the *Y-axis*.
- Select *electrons_fieldScaleParam* for the *Color*.
- Move the *Dump* slider to *Dump: 11*
- Click on *Draw*.



Data View: Phase Space

Plot 1

Base Variable: electrons

X-axis electrons_x □ Log

Y-axis electrons_y □ Log

Z-axis <None> □ Log

Color electrons_fieldScaleParam □ Log

Point Size (pixels) 2

Symbol Point

□ Enable Second Plot

☑ Reset View on Draw

Annotation Level: Axes & Legends

□ Auto Reset

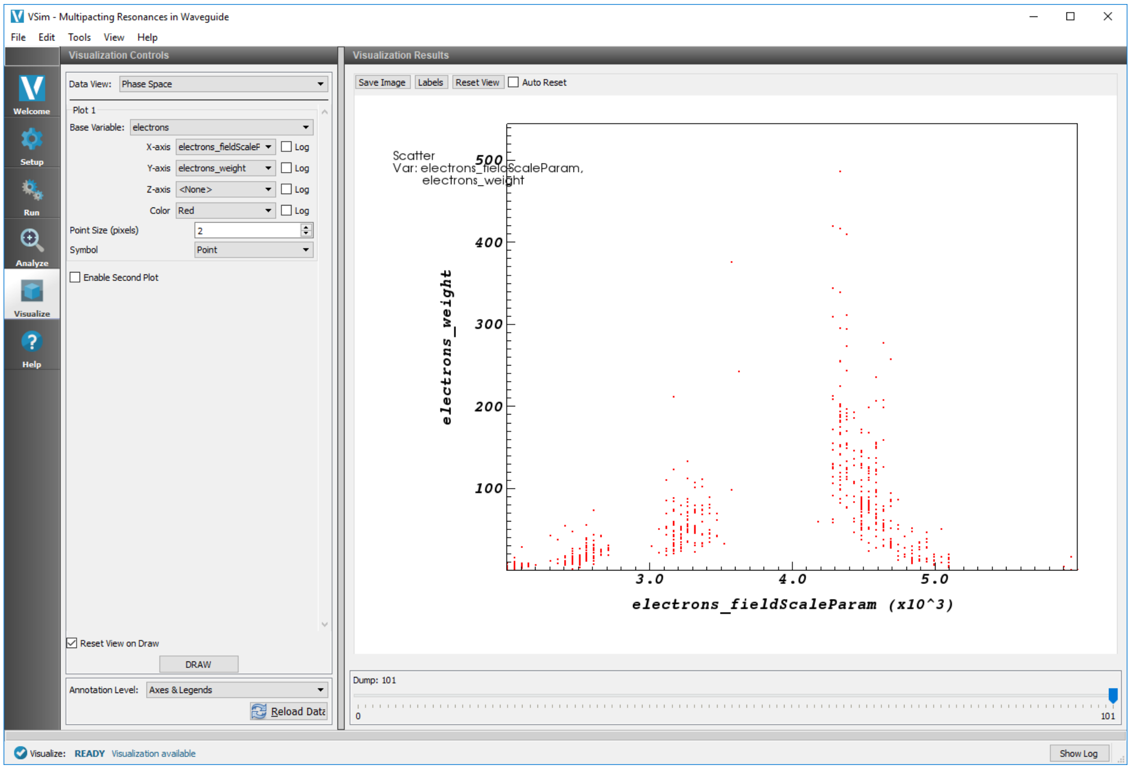
Dump: 11

Visualize: READY Visualization available

Fig. 4.106: Location of electrons and their field scaled parameters.

To view growth in the number of electrons, proceed as follows:

- Select *Phase Space* from the *Data View* pull down menu (at the top of the *Controls* pane)
- Select *electrons_fieldScaleParam* for the *X-axis*
- Select *electrons_weight* for the *Y-axis*.
- Move the *Dump* slider to the far right.
- Click on *Draw*.



Data View: Phase Space

Plot 1

Base Variable: electrons

X-axis electrons_fieldScaleF □ Log

Y-axis electrons_weight □ Log

Z-axis <None> □ Log

Color Red □ Log

Point Size (pixels): 2

Symbol: Point

□ Enable Second Plot

☑ Reset View on Draw

DRAW

Annotation Level: Axes & Legends

Reload Data

□ Auto Reset

Dump: 101

Visualize: READY Visualization available

Fig. 4.107: Visualization of the resonance bands.

Further Experiments

Try seeing how changing the gap voltage or the frequency of the wave changes the multipacting resonances.

4.5.3 Multipacting in Stripline (striplineMultipacting.sdf)

Keywords:

striplineMultipacting

Problem description

This VSimMD example launches a wave into a stripline multipacting geometry and demonstrates multipacting effects.

This example is run in a portion of a TEM transmission line CAD file, and demonstrates how multipacting occurs at specific drive voltages. This is done with the use of a Field Scaling Electron particle species.

This simulation can be performed with a VSimMD license.

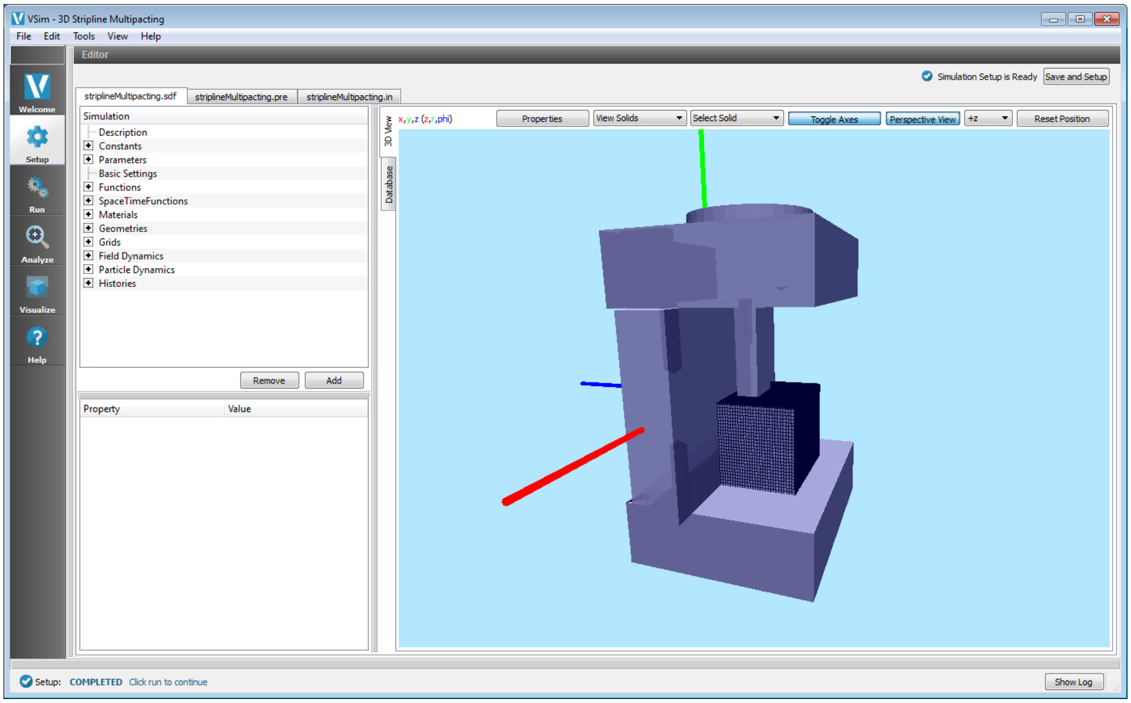
Opening the Simulation

The Stripline example is accessed from within VSimComposer by the following actions:

- Select the *New From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.

- Expand the *Multipacting* option.
- Select "3D Stripline Multipacting" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is now shown with all the implemented physics and geometries, if applicable. See Fig. 4.108. As shown in the image, *Field Scaling Electrons* are being used to be able to simulate multiple field powers in one simulation.



Simulation Setup is Ready

Setup: COMPLETED Click run to continue

Fig. 4.108: Setup Window for the Stripline example.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane on the right. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 4.109 below.

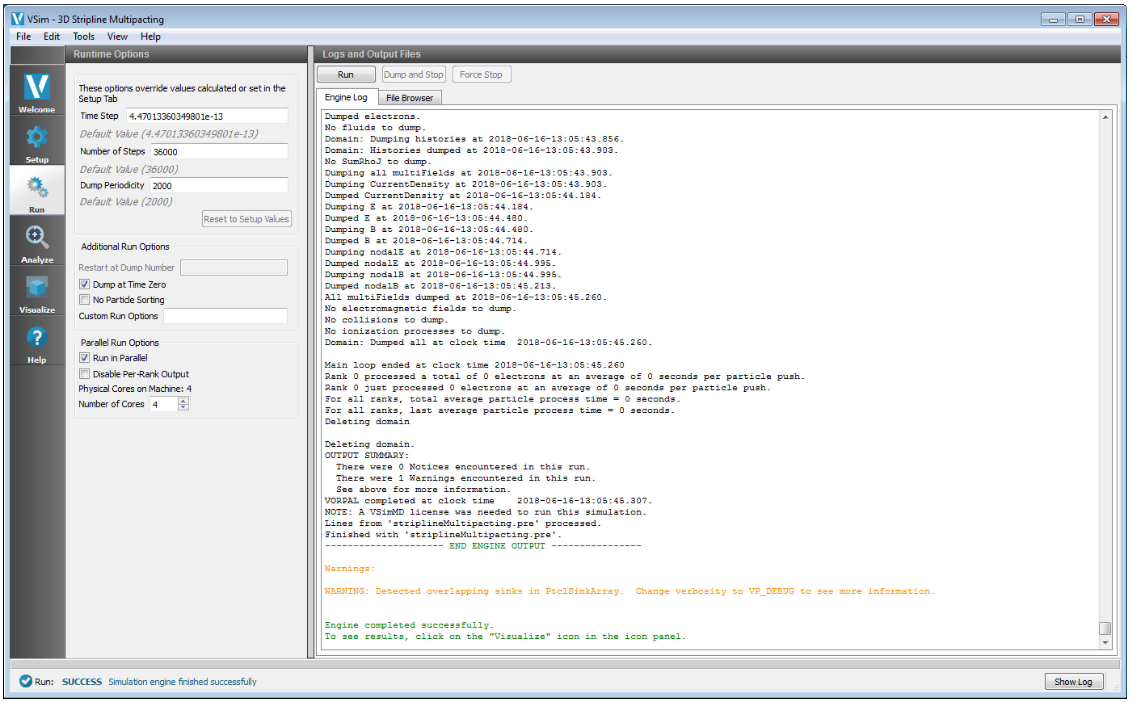
Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the exponential increase in electrons as in Fig. 4.110:

- Select *History* under the *Data View* drop down
- Set Graph 2 to *numRealElectrons*



These options override values calculated or set in the Setup Tab

Time Step 4.47013360349801e-13

Default Value (4.47013360349801e-13)

Number of Steps 36000

Default Value (36000)

Dump Periodicity 2000

Default Value (2000)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Dumped electrons.
No fluids to dump.
Domain: Dumping histories at 2018-06-16-13:05:43.856.
Domain: Histories dumped at 2018-06-16-13:05:43.903.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-16-13:05:43.903.
Dumping CurrentDensity at 2018-06-16-13:05:43.903.
Dumped CurrentDensity at 2018-06-16-13:05:44.184.
Dumping E at 2018-06-16-13:05:44.184.
Dumped E at 2018-06-16-13:05:44.480.
Dumping B at 2018-06-16-13:05:44.480.
Dumped B at 2018-06-16-13:05:44.714.
Dumping nodalE at 2018-06-16-13:05:44.714.
Dumped nodalE at 2018-06-16-13:05:44.995.
Dumping nodalB at 2018-06-16-13:05:44.995.
Dumped nodalB at 2018-06-16-13:05:45.213.
All multiFields dumped at 2018-06-16-13:05:45.260.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-16-13:05:45.260.

Main loop ended at clock time 2018-06-16-13:05:45.260
Rank 0 processed a total of 0 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 0 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 1 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-06-16-13:05:45.307.
NOTE: A VSimMD license was needed to run this simulation.
Lines from 'striplineMultipacting.pre' processed.
Finished with 'striplineMultipacting.pre'.
--------------------- END ENGINE OUTPUT ----------------

Warnings:

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.109: The Run Window at the end of execution.

- Set Graphs 3 & 4 to *None*

To view the voltage at which multipacting occurs in the stripline, as seen in Fig. 4.111:

- Select *Phase Space* under the *Data View* drop down
- Set the X-axis to *electrons_fieldScaleParameter*
- Set the Y-axis to *electrons_weight*
- Set the Color to *electrons_weight*
- Press *Draw*.
- Move the slide to the right to see how the multipacting growth is occurring.

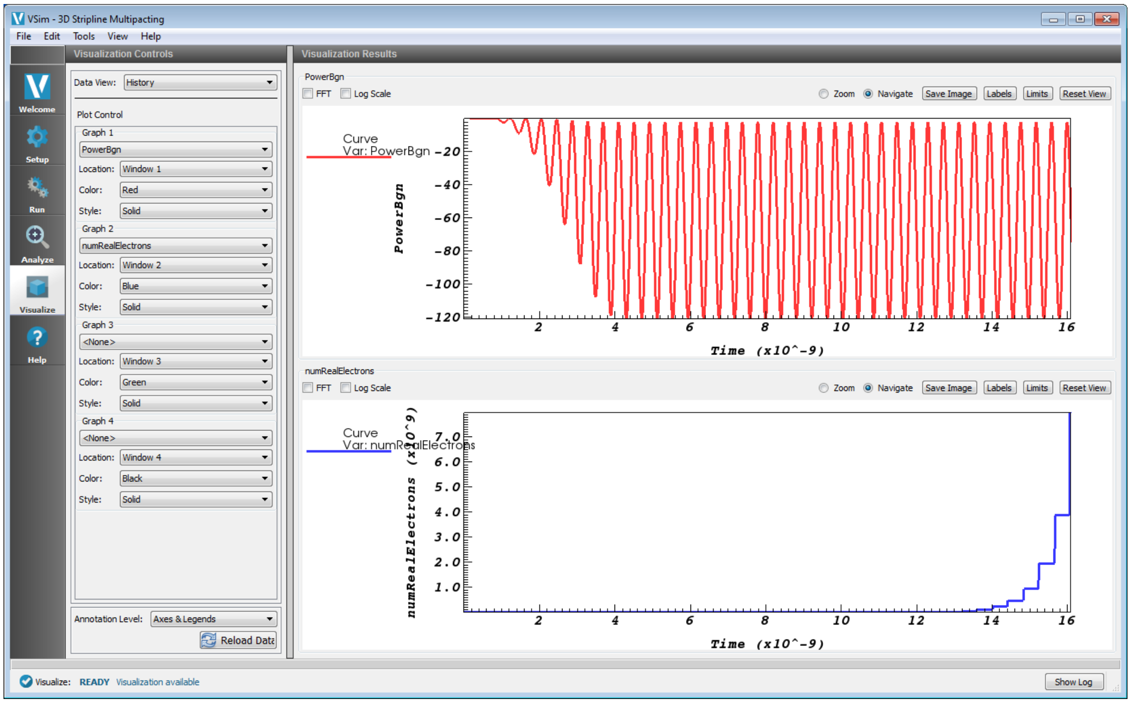
Further Experiments

Try varying the parameter VOLTAGE to observe when multipacting occurs. For example, multipacting does not occur for VOLTAGE = 45.0 Volts.

4.5.4 Multipacting Growth in Spherical PEC Cavity Using Prescribed Fields (multipactingGrowthPrescribedFields.sdf)

Keywords:

multipacting



Data View: History

Graph 1: PowerBgn
Location: Window 1
Color: Red
Style: Solid

Graph 2: numRealElectrons
Location: Window 2
Color: Blue
Style: Solid

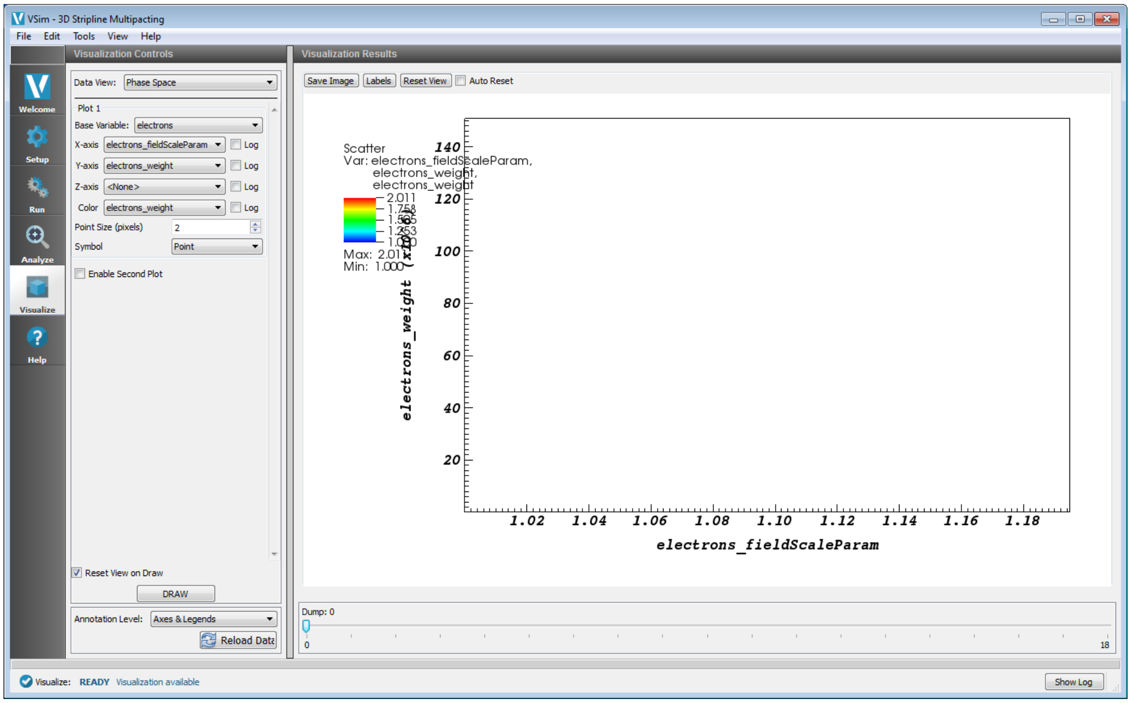
Graph 3: <None>
Location: Window 3
Color: Green
Style: Solid

Graph 4: <None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 4.110: The numRealElectrons history shows exponential growth in the number of electrons.



Data View: Phase Space

Plot 1
Base Variable: electrons
X-axis: electrons_fieldScaleParam □ Log
Y-axis: electrons_weight □ Log
Z-axis: <None> □ Log
Color: electrons_weight □ Log
Point Size (pixels): 2
Symbol: Point
□ Enable Second Plot

☑ Reset View on Draw

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 4.111: The multipacting occurs at a scaling value of around 2.5. You can use this to multiply the value of START_VOLTAGE to find out the voltage at which multipacting occurs. In this case that is around 60.0 Volts

Problem description

Multipacting, which is the resonant build up of secondary electrons, is often a concern in microwave devices. Anytime there is an oscillating electromagnetic field across a gap between two surfaces there exists the possibility that for the right voltage across the gap a resonance condition will exist allowing the exponential build up of secondary electrons. This example simulates multipacting growth in a spherical PEC cavity. The fundemental mode profile for the spherical PEC cavity is imported onto the VSim grid, then advanced in time by a single frequency time signal.

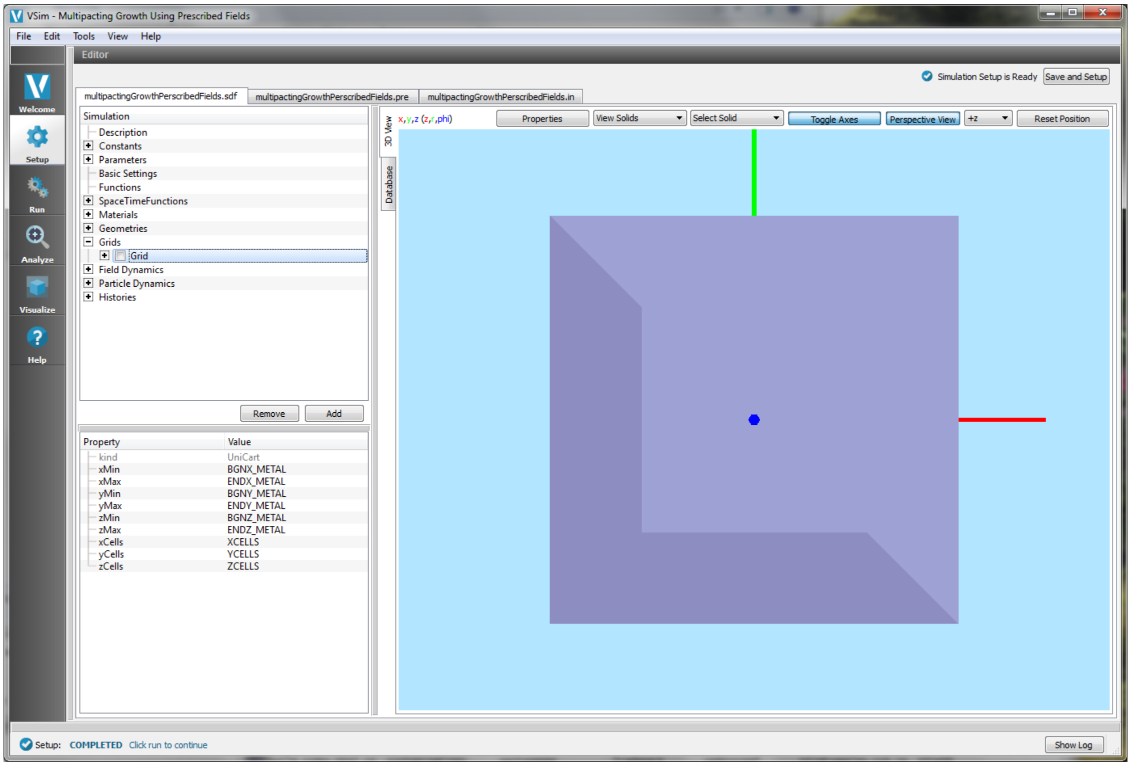
This simulation can be performed with the VSimMD or VSimPD license.

Opening the Simulation

The Multipacting Growth Prescribed Fields example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Multipacting* option.
- Select "Multipacting Growth Using Prescribed Fields" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 4.112. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



Property	Value
kind	UniCart
xMin	BGNX_METAL
xMax	ENDX_METAL
yMin	BGNY_METAL
yMax	ENDY_METAL
zMin	BGNZ_METAL
zMax	ENDZ_METAL
xCells	XCELLS
yCells	YCELLS
zCells	ZCELLS

Setup: COMPLETED Click run to continue

Fig. 4.112: Setup Window for the Multipacting Growth example.

Simulation Properties

This example contains a number of *Constants* to allow for easy manipulation of the device. Those include:

- **SPHERE_RADIUS**: radius of spherical cavity
- **LENGTH_METAL**: Length of metal box in each dimension (must be larger than 2*SPHERE_RADIUS)
- **RESOLUTION**: The number of cells per wavelength in each dimension

SpaceTimeFunctions are used to create expressions defining the drive frequency and amplitude of the applied field.

The amplitude and frequency of this driving function is defined in *Parameters*:

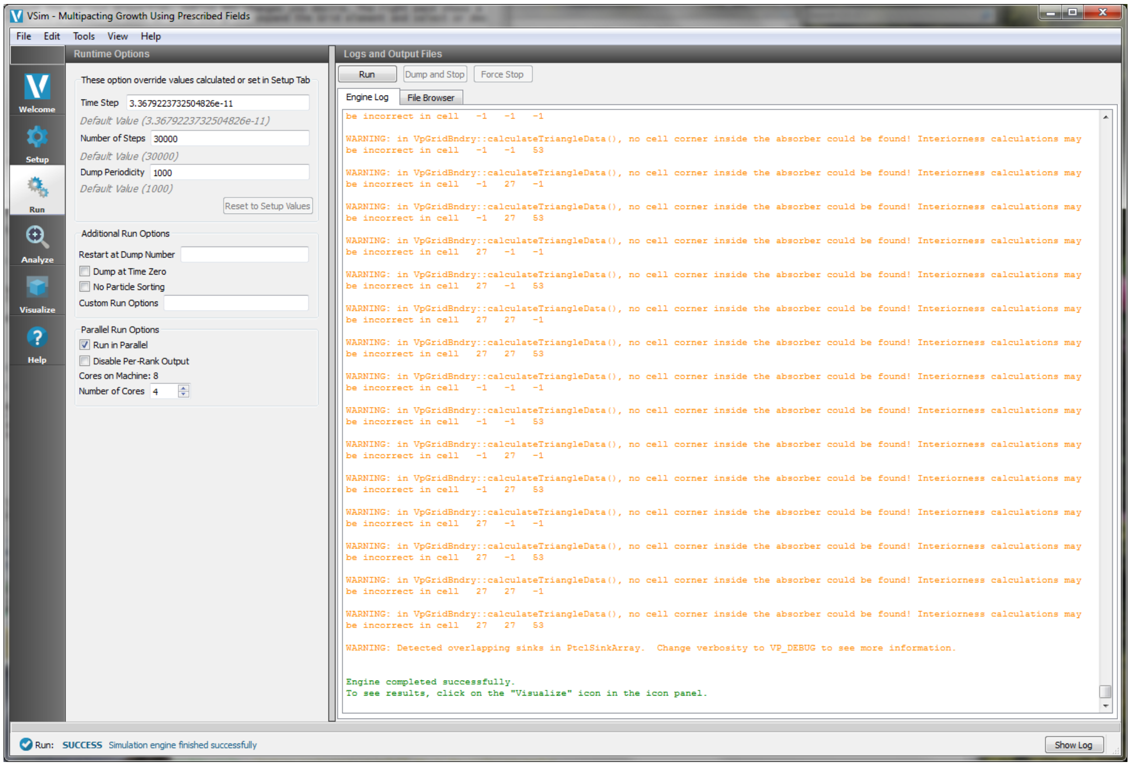
- **MODE_FREQ**: frequency at which the mode profile oscillates.
- **MODE_AMP**: amplitude that is applied to the mode profile each time step.

CSG is used to create the spherical PEC cavity by subtracting sphere from a cube.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner. of the window. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.113



These option override values calculated or set in Setup Tab

Time Step 3.3679223732504826e-11
Default Value (3.3679223732504826e-11)
Number of Steps 30000
Default Value (30000)
Dump Periodicity 1000
Default Value (1000)

Additional Run Options

Restart at Dump Number
☐ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
☐ Disable Per-Rank Output
Cores on Machine: 8
Number of Cores 4

```
be incorrect in cell   -1   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   -1   -1   53

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   -1   27   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   -1   27   53

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   27   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   27   -1   53

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   27   27   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   27   27   53

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   -1   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   -1   -1   53

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   -1   27   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   -1   27   53

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   27   -1   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   27   -1   53

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   27   27   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   27   27   53

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.113: The Run Window at the end of execution.

Visualizing the results

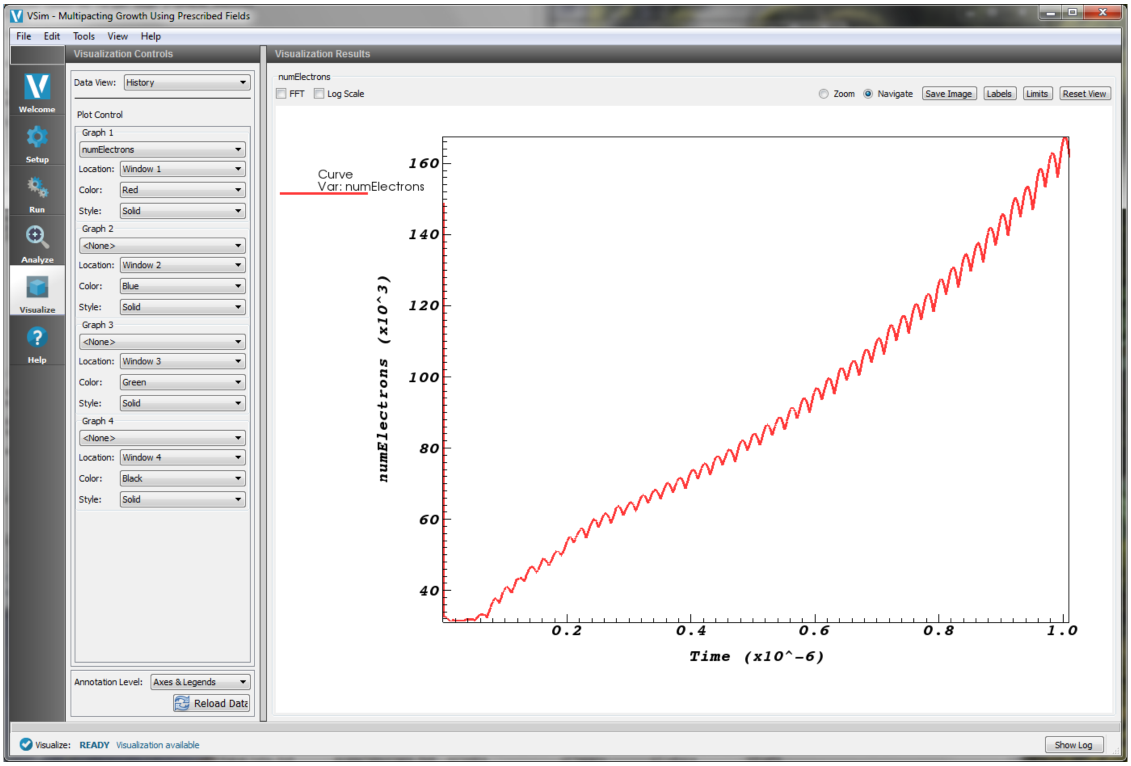
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view growth in the number of electrons, as shown in Fig. 4.114, do the following:

- Select *History* from the *Data View* pull down menu
- Set Graphs 2-4 to “None”
- Set Graph 1 to *numElectrons*

The overall trend in the number of electrons is an exponential growth with an ocsillatory signal that corresponds to the frequency of the time signal.



Data View: History

Plot Control

Graph 1
numElectrons
Location: Window 1
Color: Red
Style: Solid

Graph 2
<None>
Location: Window 2
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 3
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 4.114: Visualization of the exponential growth of the electrons due to multipacting.

Further Experiments

Try changing the parameters **MODE_AMP** and **MODE_FREQ** to see if one can take the simulation in and out of resonance.

4.6 Multipacting (text-based setup)

4.6.1 Multipacting Resonances in Waveguide (multipactingResonancesT.pre)

Keywords:

`multipacting , multipactingResonancesT`

Problem description

Multipacting, which is the resonant build up of secondary electrons, is often a concern in microwave devices. Anytime there is an oscillating electromagnetic field across a gap between two surfaces there exists the possibility that for the right voltage across the gap a resonance condition will exist allowing the exponential build up of secondary electrons. A coaxial waveguide is such a structure where these conditions can exist.

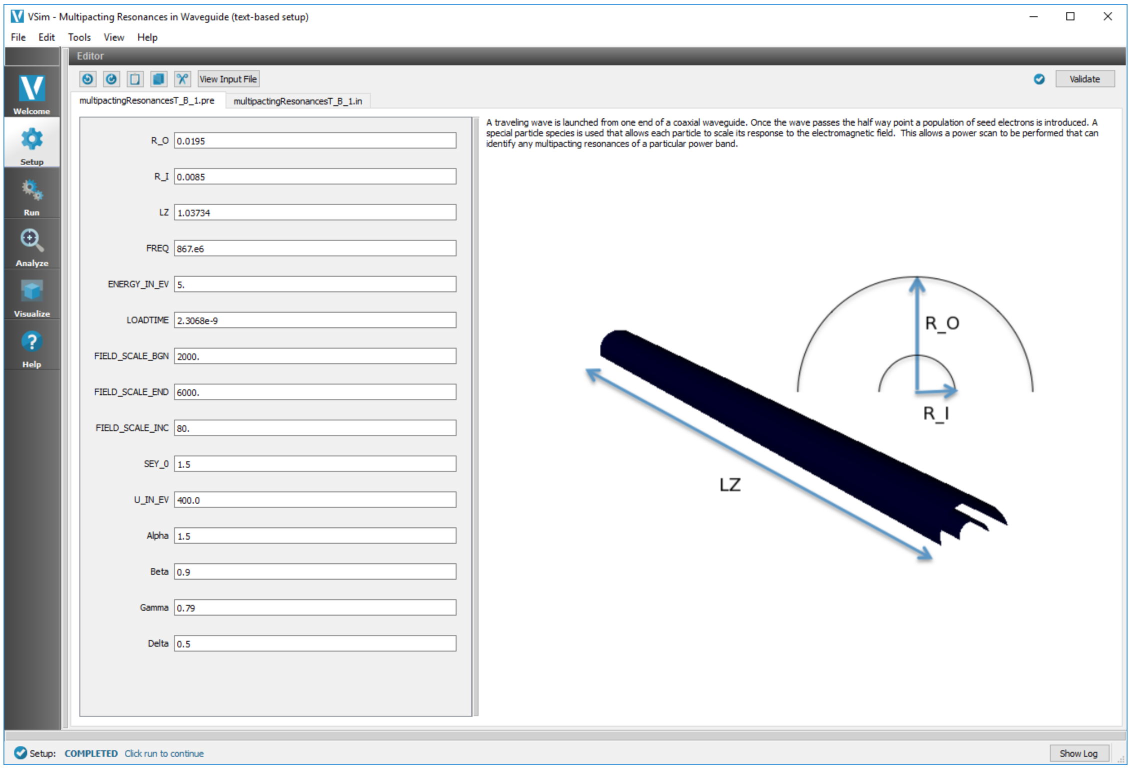
This simulation can be performed with the VSimMD license.

Opening the Simulation

The Multipacting Resonances example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Multipacting (text-based setup)* option.
- Select "Multipacting Resonances in Waveguide (text-based setup)" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 4.115.


VSim - Multipacting Resonances in Waveguide (text-based setup)
File
Edit
Tools
View
Help
Editor
Welcome
Setup
Run
Analyze
Visualize
Help
View Input File
Validate
multipactingResonancesT_B_1.pre
multipactingResonancesT_B_1.in
R_O
R_I
LZ
Show Log

R_O	0.0195
R_I	0.0085
LZ	1.03734
FREQ	867.e6
ENERGY_IN_EV	5.
LOADTIME	2.3068e-9
FIELD_SCALE_BGN	2000.
FIELD_SCALE_END	6000.
FIELD_SCALE_INC	80.
SEY_0	1.5
U_IN_EV	400.0
Alpha	1.5
Beta	0.9
Gamma	0.79
Delta	0.5

A traveling wave is launched from one end of a coaxial waveguide. Once the wave passes the half way point a population of seed electrons is introduced. A special particle species is used that allows each particle to scale its response to the electromagnetic field. This allows a power scan to be performed that can identify any multipacting resonances of a particular power band.

Setup: COMPLETED Click run to continue

Fig. 4.115: Setup Window for the Multipacting Resonances example.

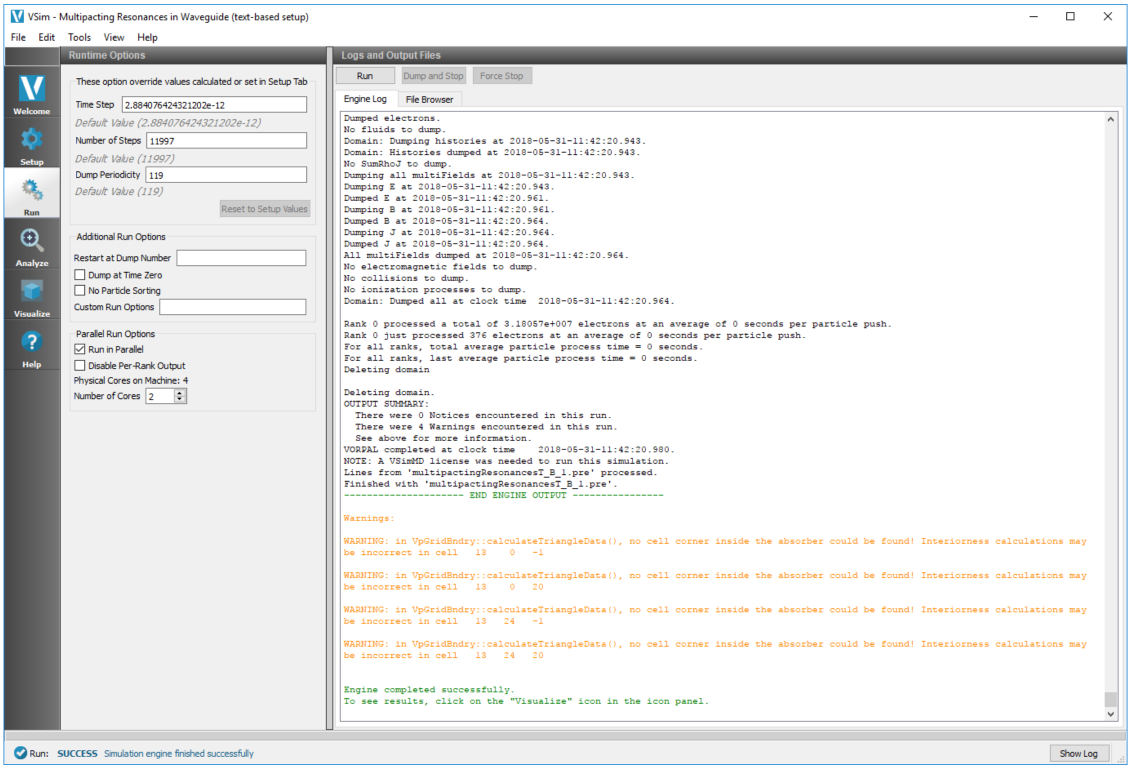
Input File Features

The input file sets the number of cells along the propagation (x) direction to resolve the wavelength. The electrons are seeded in the middle of the waveguide once the wave has passed. A special electron species is used that allows scans over power to be done in a single simulation. The time step is chosen to be at 90% of the CFL (stability) limit.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the **Run** button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner. of the window. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 4.116.



These option override values calculated or set in Setup Tab

Time Step 2.884076424321202e-12
Default Value (2.884076424321202e-12)
Number of Steps 11997
Default Value (11997)
Dump Periodicity 119
Default Value (119)

Additional Run Options

Restart at Dump Number
☐ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
☐ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 2

```
Dumped electrons.
No fluids to dump.
Domain: Dumping histories at 2018-05-31-11:42:20.943.
Domain: Histories dumped at 2018-05-31-11:42:20.943.
No SumRhoJ to dump.
Dumping all multiFields at 2018-05-31-11:42:20.943.
Dumping E at 2018-05-31-11:42:20.943.
Dumped E at 2018-05-31-11:42:20.961.
Dumping B at 2018-05-31-11:42:20.961.
Dumped B at 2018-05-31-11:42:20.964.
Dumping J at 2018-05-31-11:42:20.964.
Dumped J at 2018-05-31-11:42:20.964.
All multiFields dumped at 2018-05-31-11:42:20.964.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-05-31-11:42:20.964.

Rank 0 processed a total of 3.18057e+007 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 376 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 4 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-05-31-11:42:20.980.
NOTE: A VSimMD license was needed to run this simulation.
Lines from 'multipactingResonancesT_B_1.pre' processed.
Finished with 'multipactingResonancesT_B_1.pre'.
--------------------- END ENGINE OUTPUT ----------------

Warnings:

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13    0   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13    0   20

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13   24   -1

WARNING: in VpGridBndry::calculateTriangleData(), no cell corner inside the absorber could be found! Interiorness calculations may
be incorrect in cell   13   24   20


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.116: The **Run** window at the end of execution.

Visualizing the results

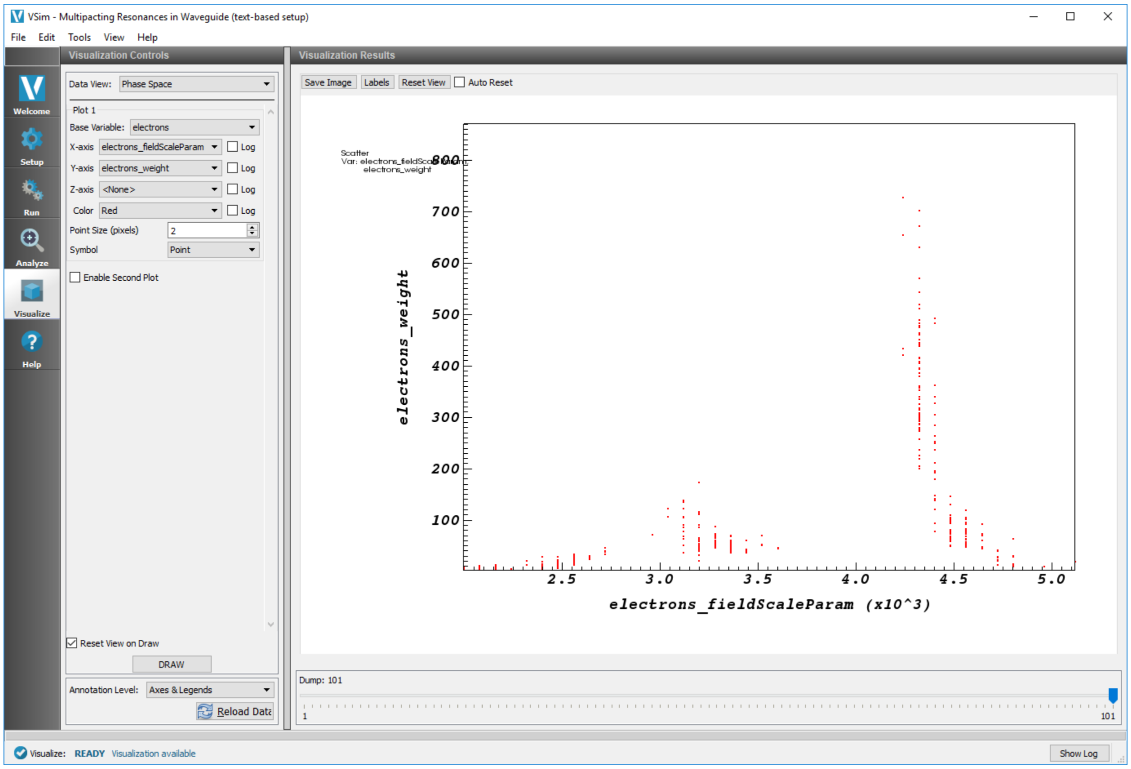
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view growth in the number of electrons:

- Select *Phase Space* from the *Data View* drop down
- Select *scaleFactor* for the *X-axis*
- Select *weight* forthe *Y-axis*

- Move the *Dump* slider to the far right then click on *Draw*.



Data View: Phase Space

Base Variable: electrons

X-axis: electrons_fieldScaleParam □ Log

Y-axis: electrons_weight □ Log

Z-axis: <None> □ Log

Color: Red □ Log

Point Size (pixels): 2

Symbol: Point

□ Enable Second Plot

☑ Reset View on Draw

Annotation Level: Axes & Legends

Dump: 101

Visualize: READY Visualization available

Fig. 4.117: Visualization of the resonance bands.

Further Experiments

Try seeing how changing the gap voltage or the frequency of the wave changes the multipacting resonances.

4.6.2 3D Stripline Multipacting (striplineMultipactingT.pre)

Keywords:

striplineMultipactingT

Problem description

This VSimMD example launches a wave into a stripline geometry and demonstrates multipacting effects. The use of "field scaled" particles allows the scanning of multiple power levels in one run. Each particle has a scaling parameter that multiplies the electromagnetic field, allowing multiple power or voltage levels to exist simultaneously. This feature allows for quick finding of the cut-off resonance in the stripline.

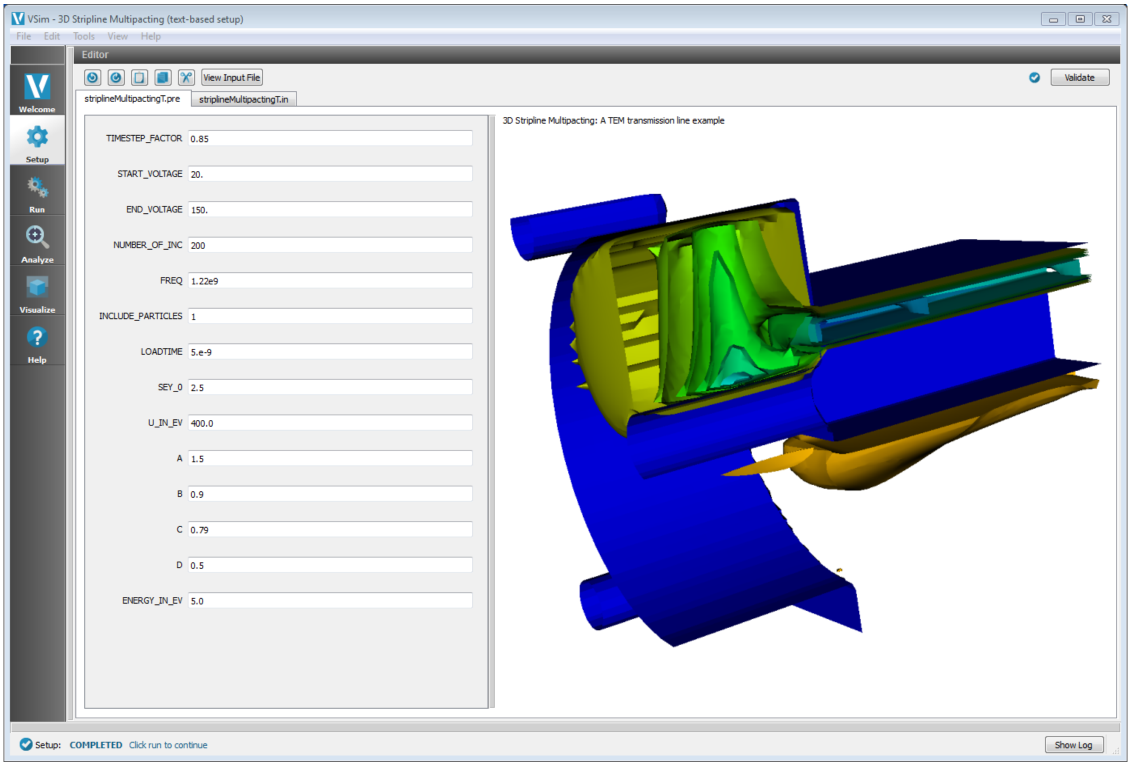
This simulation can be performed with a VSimMD license.

Opening the Simulation

The Stripline Multipacting example is accessed from within VSimComposer by the following actions:

- Select the *New From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Multipacting (text-based setup)* option.
- Select “3D Stripline Multipacting (text-based setup)” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 4.118.



VSim - 3D Stripline Multipacting (text-based setup)

TIMESTEP_FACTOR 0.85

START_VOLTAGE 20.

END_VOLTAGE 150.

NUMBER_OF_INC 200

FREQ 1.22e9

INCLUDE_PARTICLES 1

LOADTIME 5.e-9

SEY_0 2.5

U_IN_EV 400.0

A 1.5

B 0.9

C 0.79

D 0.5

ENERGY_IN_EV 5.0

3D Stripline Multipacting: A TEM transmission line example

Setup: COMPLETED Click run to continue

Fig. 4.118: Setup Window for the Stripline Multipacting example.

Input File Features

The Stripline Multipacting example has the following input parameters:

- TIMESTEP_FACTOR multiplies the numerical stability limit on the time step to compute the actual time step. TIMESTEP_FACTOR should be a real number greater than 0 and less than 1.
- START_VOLTAGE specifies the starting voltage for the field scale particles in Volts across the plates in the stripline. START_VOLTAGE should be a positive real number.
- END_VOLTAGE specifies the ending voltage for the field scale particles in Volts across the plates in the stripline. END_VOLTAGE should be a positive real number.
- NUMBER_OF_INC specifies the number of increments to use for the voltage between START_VOLTAGE and END_VOLTAGE.
- FREQ specifies the frequency in Hertz of the signal in the stripline. FREQ should be a positive real number.

- INCLUDE_PARTICLES = 1 loads electrons into the simulation and INCLUDE_PARTICLES = 0 does not load electrons into the simulation.
- LOADTIME specifies the time in seconds at which electrons are loaded into the simulation. LOADTIME should be a non-negative real number.

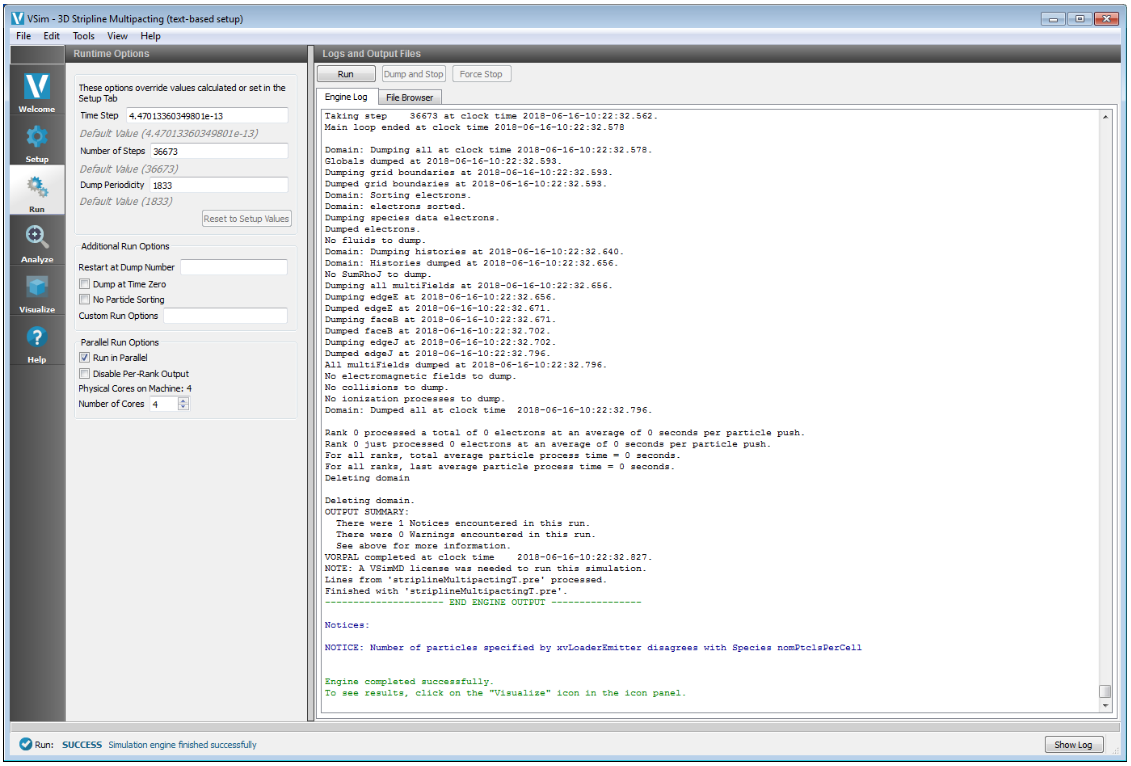
There are also several input parameters to specify the secondary electron yield based on the equation:

$SEY_0 * (1 - exp(-A * (x/U_0)^B))/C/((x/U_0)^D)$

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the window. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 4.119 below.



These options override values calculated or set in the Setup Tab

Time Step 4.47013360349801e-13

Default Value (4.47013360349801e-13)

Number of Steps 36673

Default Value (36673)

Dump Periodicity 1833

Default Value (1833)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Taking step    36673 at clock time 2018-06-16-10:22:32.562.
Main loop ended at clock time 2018-06-16-10:22:32.578

Domain: Dumping all at clock time 2018-06-16-10:22:32.578.
Globals dumped at 2018-06-16-10:22:32.593.
Dumping grid boundaries at 2018-06-16-10:22:32.593.
Dumped grid boundaries at 2018-06-16-10:22:32.593.
Domain: Sorting electrons.
Domain: electrons sorted.
Dumping species data electrons.
Dumped electrons.
No fluids to dump.
Domain: Dumping histories at 2018-06-16-10:22:32.640.
Domain: Histories dumped at 2018-06-16-10:22:32.656.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-16-10:22:32.656.
Dumping edgeE at 2018-06-16-10:22:32.656.
Dumped edgeE at 2018-06-16-10:22:32.671.
Dumping faceB at 2018-06-16-10:22:32.671.
Dumped faceB at 2018-06-16-10:22:32.702.
Dumping edgeJ at 2018-06-16-10:22:32.702.
Dumped edgeJ at 2018-06-16-10:22:32.796.
All multiFields dumped at 2018-06-16-10:22:32.796.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-16-10:22:32.796.

Rank 0 processed a total of 0 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 0 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 1 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-06-16-10:22:32.827.
NOTE: A VSimMD license was needed to run this simulation.
Lines from 'striplineMultipactingT.pre' processed.
Finished with 'striplineMultipactingT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: Number of particles specified by xvLoaderEmitter disagrees with Species nomPtclsPerCell


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.119: The Run Window at the end of execution.

Visualizing the results

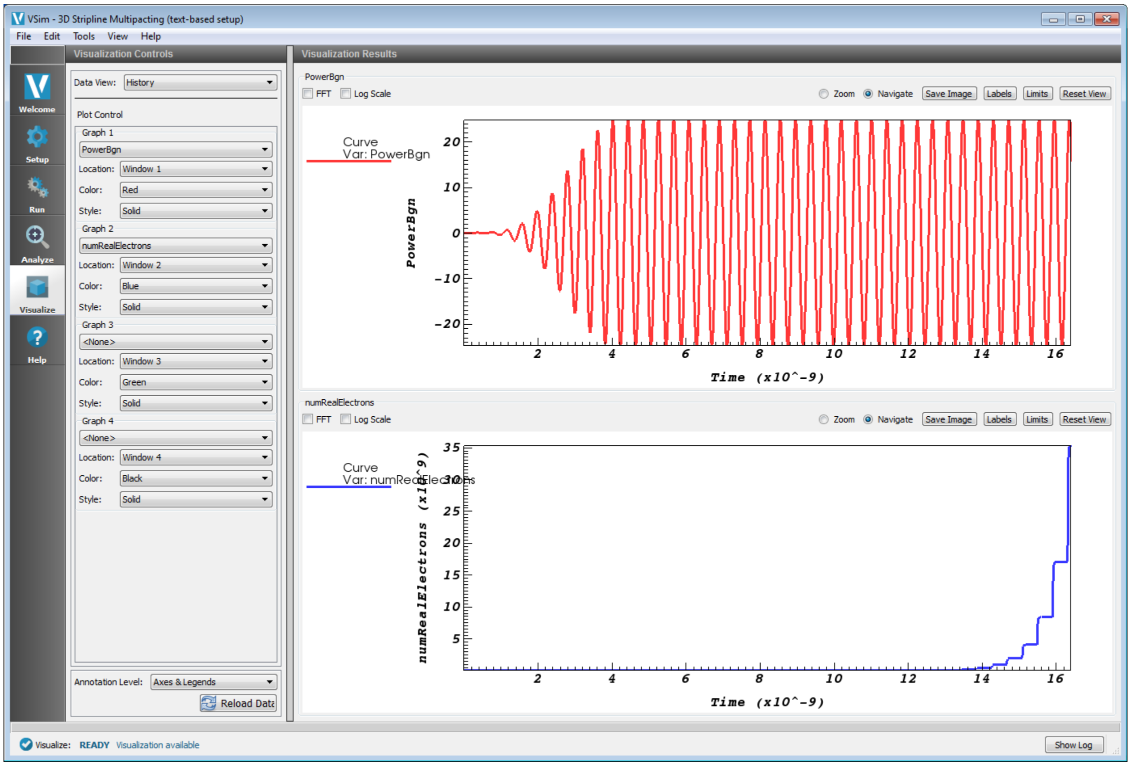
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the exponential increase in electrons as in Fig. 4.120:

- Select *History* under the *Data View* drop down

- Set Graph 2 to *numRealElectrons*
- Set Graphs 3 & 4 to *None*



Data View: History

Plot Control

Graph 1
PowerBgn
Location: Window 1
Color: Red
Style: Solid

Graph 2
numRealElectrons
Location: Window 2
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 3
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 4.120: The numRealElectrons history shows exponential growth in the number of electrons.

To view the voltage at which multipacting occurs in the stripline, as seen in Fig. 4.121:

- Select *Phase Space* under the *Data View* drop down
- Set the X-axis to *electrons_fieldScaleParameter*
- Set the Y-axis to *electrons_weight*
- Set the Color to *electrons_weight*
- Press *Draw*.
- Move the slide to the right to see how the multipacting growth is occurring.

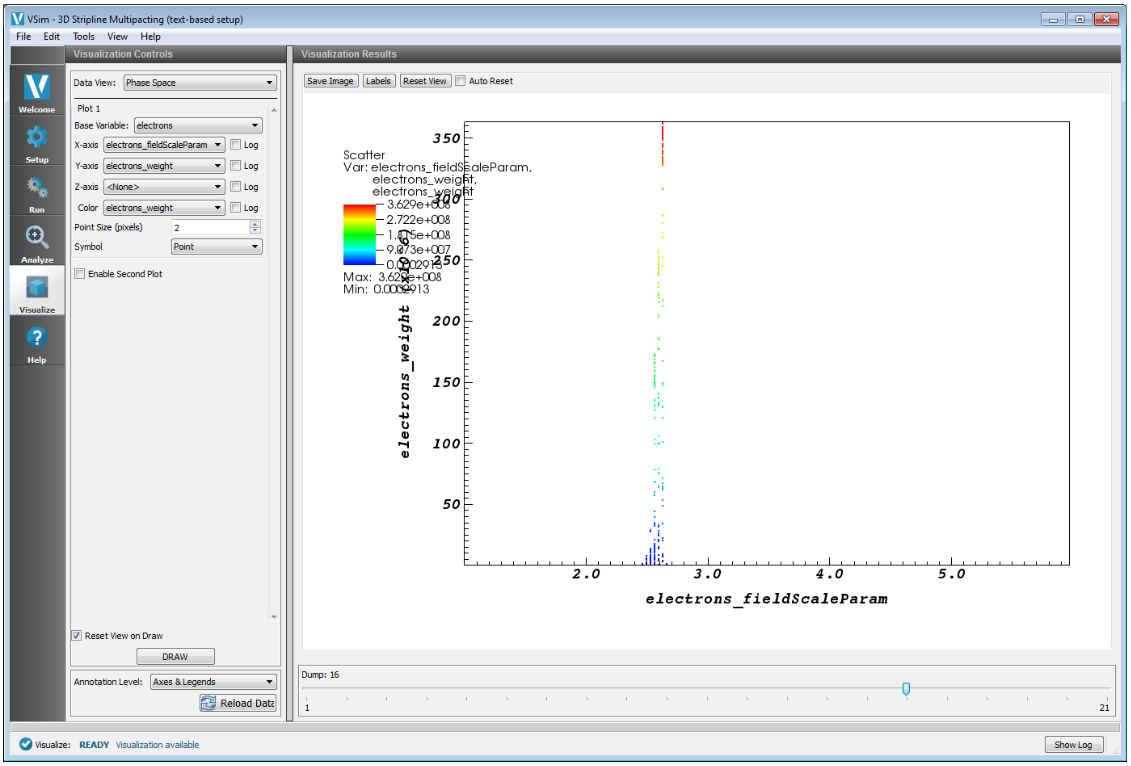
Further Experiments

Try varying the starting and ending voltages as well as the number of increments to see if multipacting occurs in different ranges.

4.7 Other

4.7.1 Electron Gun (electronGun.sdf)

Keywords:



VSim - 3D Stripline Multipacting (text-based setup)

Data View: Phase Space

Plot 1

Base Variable: electrons

X-axis electrons_fieldScaleParam □ Log

Y-axis electrons_weight □ Log

Z-axis <None> □ Log

Color electrons_weight □ Log

Point Size (pixels) 2

Symbol Point

□ Enable Second Plot

☑ Reset View on Draw

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 16

Visualize: READY Visualization available

Fig. 4.121: The multipacting occurs at a scaling value of around 2.5. You can use this to multiply the value of START_VOLTAGE to find out the voltage at which multipacting occurs. In this case that is around 60.0 Volts

electron, gun, beam, collimate

Problem description

Electron guns are devices that are often found in vacuum electronics as well as in more advanced technologies such as klystrons, electron microscopes, and particle accelerators. They produce narrow, collimated beams of electrons with precisely tuned kinetic energies. They were often found in cathode ray tubes at the heart of television sets prior to the digital revolution. Electron guns are composed of a cathode, an anode, and repulsive rings. A DC or RF signal is applied to the cathode to produce electrons via thermionic emission. The electrodes produce electric fields that focus the electron beam. Often an additional anode is placed between the cathode and the main anode to act as a repulsive ring that focuses the beam into a small hole in the main anode. The small hole in the main anode acts to collimate the beam.

This example is a specialized electron gun for klystrons and TWTs. It is characterized by high power, a consequence of which is that electrons not successfully collimated can damage the device. To minimize this effect, the gun includes a focusing anode cone, the angle of which is conducive to laminar flow of the electron beam.

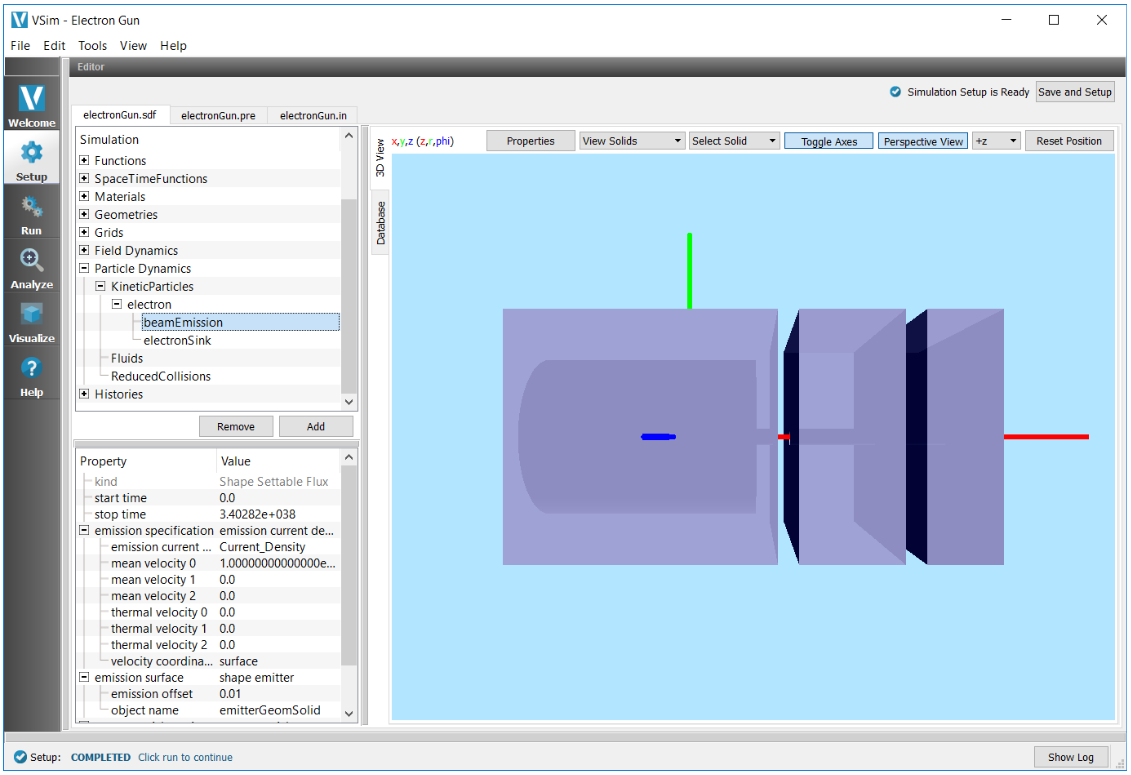
Opening the Simulation

The Electron Gun example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Other MD* option.

- Select “Electron Gun” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is now shown with all the implemented physics and geometries, if applicable. See Fig. 4.122.



Property	Value
kind	Shape Settable Flux
start time	0.0
stop time	3.40282e+038
emission specification	emission current de...
emission current ...	Current_Density
mean velocity 0	1.00000000000000e...
mean velocity 1	0.0
mean velocity 2	0.0
thermal velocity 0	0.0
thermal velocity 1	0.0
thermal velocity 2	0.0
velocity coordina...	surface
emission surface	shape emitter
emission offset	0.01
object name	emitterGeomSolid

Setup: COMPLETED Click run to continue

Fig. 4.122: Setup Window for the Electron Gun example.

Simulation Properties

The input parameters give you total flexibility in in defining the geometry of the example. Along with these one can define the nominal cell size, the driving voltages, the strength of the magnetic field, and the beam current.

Running the Simulation

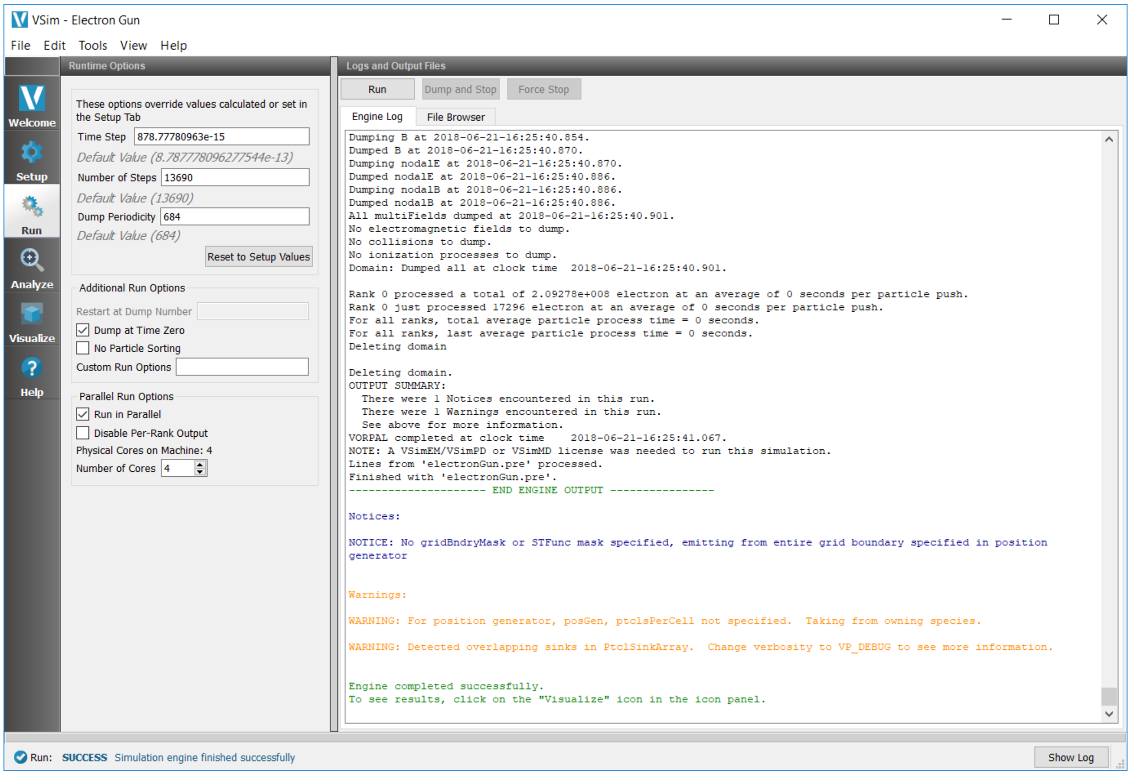
After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.123.

Visualizing the Results

To reproduce Fig. 4.124 proceed as follows:

- Expand *Particle Data*.



These options override values calculated or set in the Setup Tab

Time Step 878.77780963e-15

Default Value (8.787778096277544e-13)

Number of Steps 13690

Default Value (13690)

Dump Periodicity 684

Default Value (684)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Dumping B at 2018-06-21-16:25:40.854.
Dumped B at 2018-06-21-16:25:40.870.
Dumping nodalE at 2018-06-21-16:25:40.870.
Dumped nodalE at 2018-06-21-16:25:40.886.
Dumping nodalB at 2018-06-21-16:25:40.886.
Dumped nodalB at 2018-06-21-16:25:40.886.
All multiFields dumped at 2018-06-21-16:25:40.901.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-21-16:25:40.901.

Rank 0 processed a total of 2.09278e+008 electron at an average of 0 seconds per particle push.
Rank 0 just processed 17296 electron at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 1 Notices encountered in this run.
  There were 1 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-06-21-16:25:41.067.
NOTE: A VSimEM/VSimPD or VSimMD license was needed to run this simulation.
Lines from 'electronGun.pre' processed.
Finished with 'electronGun.pre'.
-------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: No gridBndryMask or STFunc mask specified, emitting from entire grid boundary specified in position
generator


Warnings:

WARNING: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 4.123: The Run Window at the end of execution.

- Expand *electron*.
- Select *electron* in red.
- Expand *Scalar Data*
- Expand *E*.
- Select “E_magnitude”.
- Expand *Geometries*.
- Select “poly (electronGunPecShapes)”.
- In the lower part of the left pane select “Display Contours”.
- Select “Clip All Plots”.
- In the lower part of the right pane, move the Dump slider to dump 14.

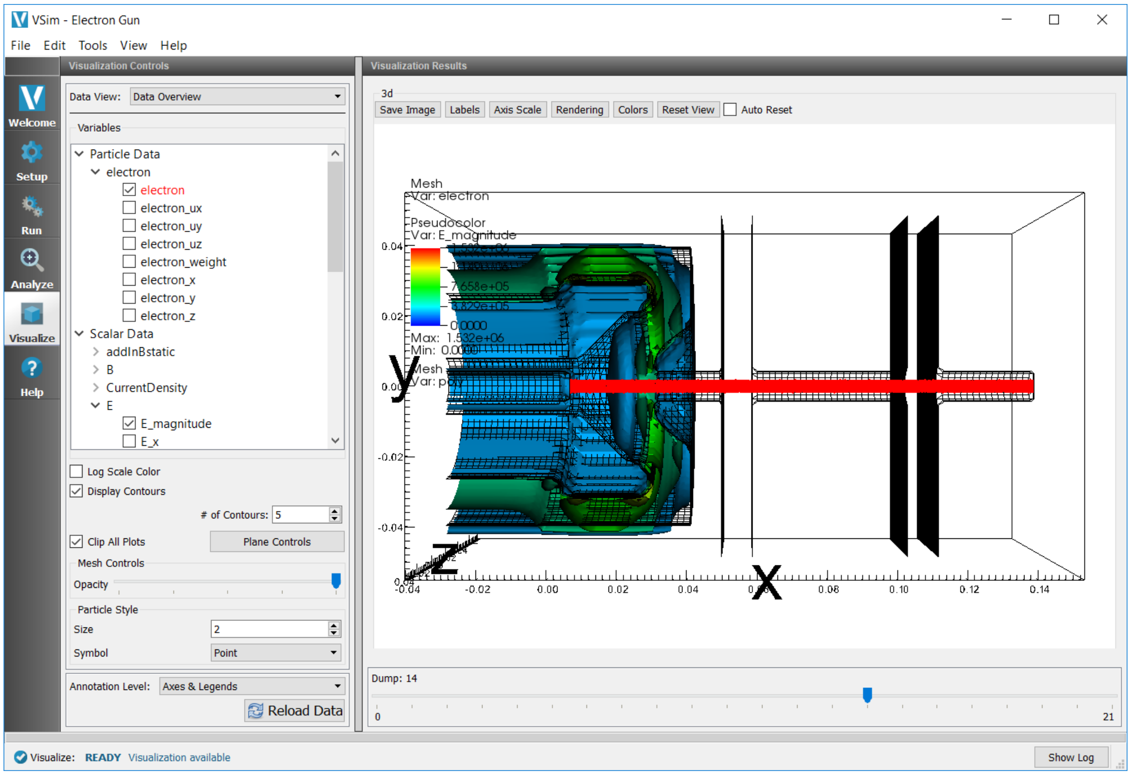
This will show the electron beam and the electric and magnetic fields.

The phase space diagram can also be viewed by choosing *Phase Space* in the *Data View* drop down menu.

The voltages and currents at key locations in the simulation are recorded in Histories and can be viewed by selecting the *History* data view.

Further Experiments

The geometry is extremely important for proper functionality in this example. For example, the angle that the focusing cone makes with the beam axis determines whether the beam will be laminar. If the beam intersects and diverges, the



Data View: Data Overview

□ Log Scale Color

☑ Display Contours

# of Contours: 5

☑ Clip All Plots

Opacity

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 14

Visualize: READY Visualization available

Fig. 4.124: The electron beam and electric field.

gun can be damaged by its own power. Try altering the dimensions of the the geometry and see the effect on the electron beam.

4.7.2 Multistage Collector (multistageCollector.sdf)

Keywords:

`electromagnetics, multistageCollector`

Problem description

Multistage Depressed Collectors (MDCs) are used to recover energy from a spent beam in linear type microwave tubes such as traveling wave tubes (TWTs) and klystrons. VSim provides the capability to simulate these collectors shaped with arbitrarily complex geometries and depressed with different time-dependent voltage profiles to optimize the recovery efficiency of a design. To demonstrate this capability, we show in this example a 4-stage depressed collector. One can adjust the depressed potentials at each electrode individually to see how the performance of the collector is affected.

This simulation can be performed with a VSimMD, VSimEM or VSimPD license.

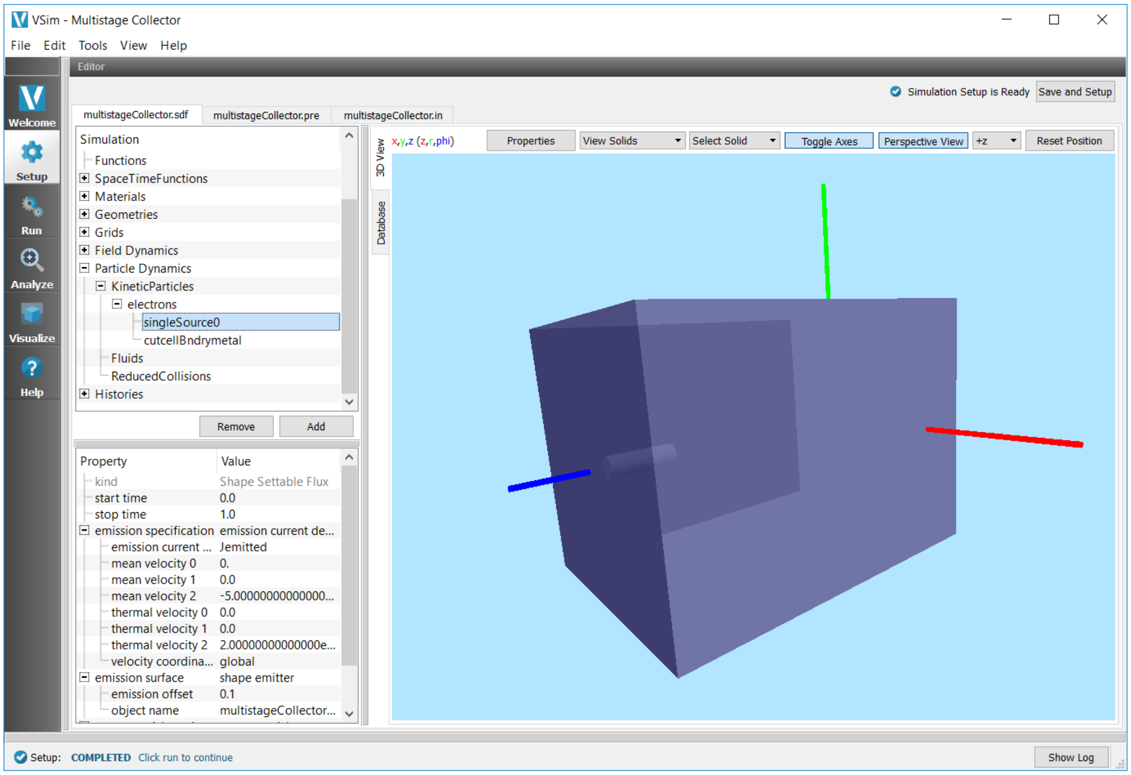
Opening the Simulation

The Multistage Collector example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example. . .* menu item in the *File* menu.

- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Other MD* option.
- Select "Multistage Collector" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is now shown with all the implemented physics and geometries, if applicable. See Fig. 4.7.2.



Property	Value
kind	Shape Settable Flux
start time	0.0
stop time	1.0
emission specification	emission current de...
emission current ...	Jemitted
mean velocity 0	0.
mean velocity 1	0.0
mean velocity 2	-5.00000000000000...
thermal velocity 0	0.0
thermal velocity 1	0.0
thermal velocity 2	2.00000000000000e...
velocity coordina...	global
emission surface	shape emitter
emission offset	0.1
object name	multistageCollector...

Setup: COMPLETED Click run to continue

Setup Window for the Multistage Collector example.

Simulation Properties

The simulation geometry consisting of an S-band 4-stage depressed collector is imported into the computational engine from CAD files in stl format. One can easily create new geometry using any CAD program and output or convert the CAD files into stl files for a new simulation design. The detailed import method is provided in the input file. The spent beam profile is taken from a TWT simulation provided by Prof. H. Song at University of Colorado at Colorado Springs.

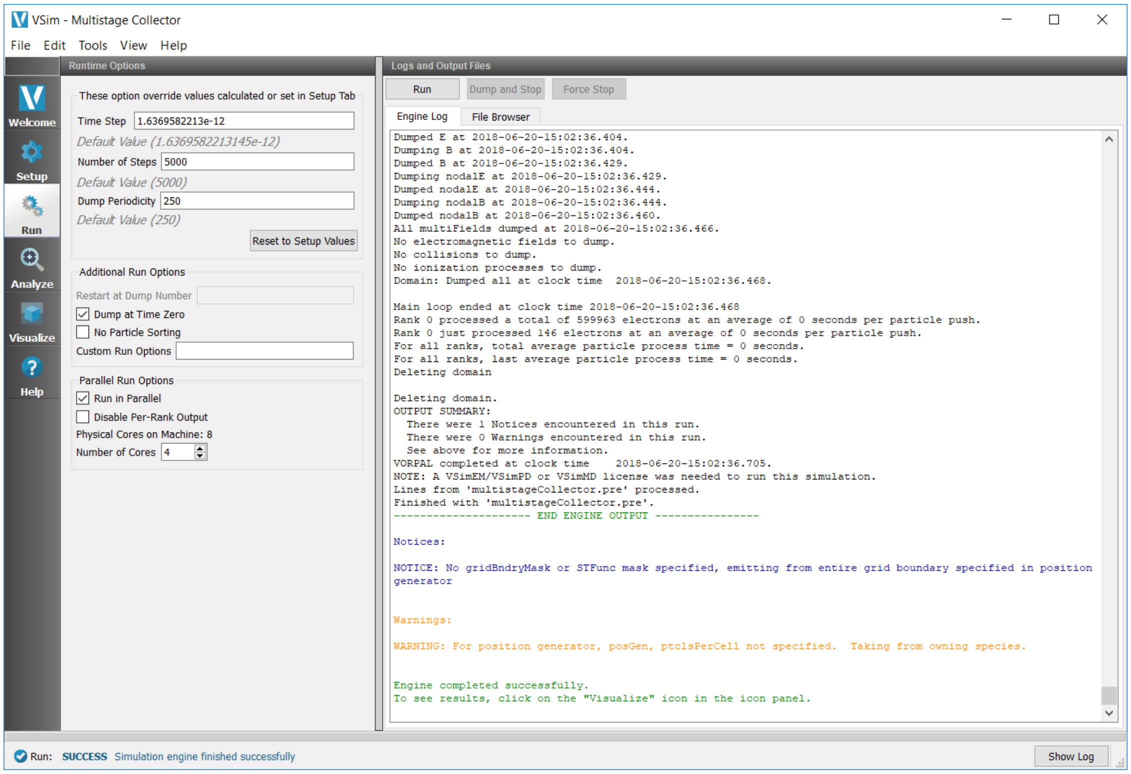
An optimized design for a MPM module can be found in reference [1]. Users can set preferred spent beam profiles by employing different emission methods or import data in dat format as in this example. A main feature of this input file is that the depressed voltage profiles are time-dependent and are stablized with a new external circuit model based on special feedback algorithms only available in VSim. Interested users may refer to the publication for more a detailed description and validation. In addition, the convergence of this example is carefully tested.

In this example, the Z coordinate is the direction aligned with the beam axis of the MDC, and the 4 different voltages can be easily assigned at the input panel. Since it is a time domain simulation, the Dey-Mittra algorithm is employed and the accuracy is second-order for the complex boundaries.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.7.2.



These option override values calculated or set in Setup Tab

Time Step 1.6369582213e-12
Default Value (1.6369582213145e-12)
Number of Steps 5000
Default Value (5000)
Dump Periodicity 250
Default Value (250)

Additional Run Options

Restart at Dump Number
☑ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
☐ Disable Per-Rank Output
Physical Cores on Machine: 8
Number of Cores 4

```
Dumped E at 2018-06-20-15:02:36.404.
Dumping B at 2018-06-20-15:02:36.404.
Dumped B at 2018-06-20-15:02:36.429.
Dumping nodalE at 2018-06-20-15:02:36.429.
Dumped nodalE at 2018-06-20-15:02:36.444.
Dumping nodalB at 2018-06-20-15:02:36.444.
Dumped nodalB at 2018-06-20-15:02:36.460.
All multiFields dumped at 2018-06-20-15:02:36.466.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-20-15:02:36.468.

Main loop ended at clock time 2018-06-20-15:02:36.468
Rank 0 processed a total of 599963 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 146 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 1 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-06-20-15:02:36.705.
NOTE: A VSimEM/VSimPD or VSimMD license was needed to run this simulation.
Lines from 'multistageCollector.pre' processed.
Finished with 'multistageCollector.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: No gridBndryMask or STFunc mask specified, emitting from entire grid boundary specified in position
generator


Warnings:

WARNING: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

The Run Window at the end of execution.

Visualizing the results

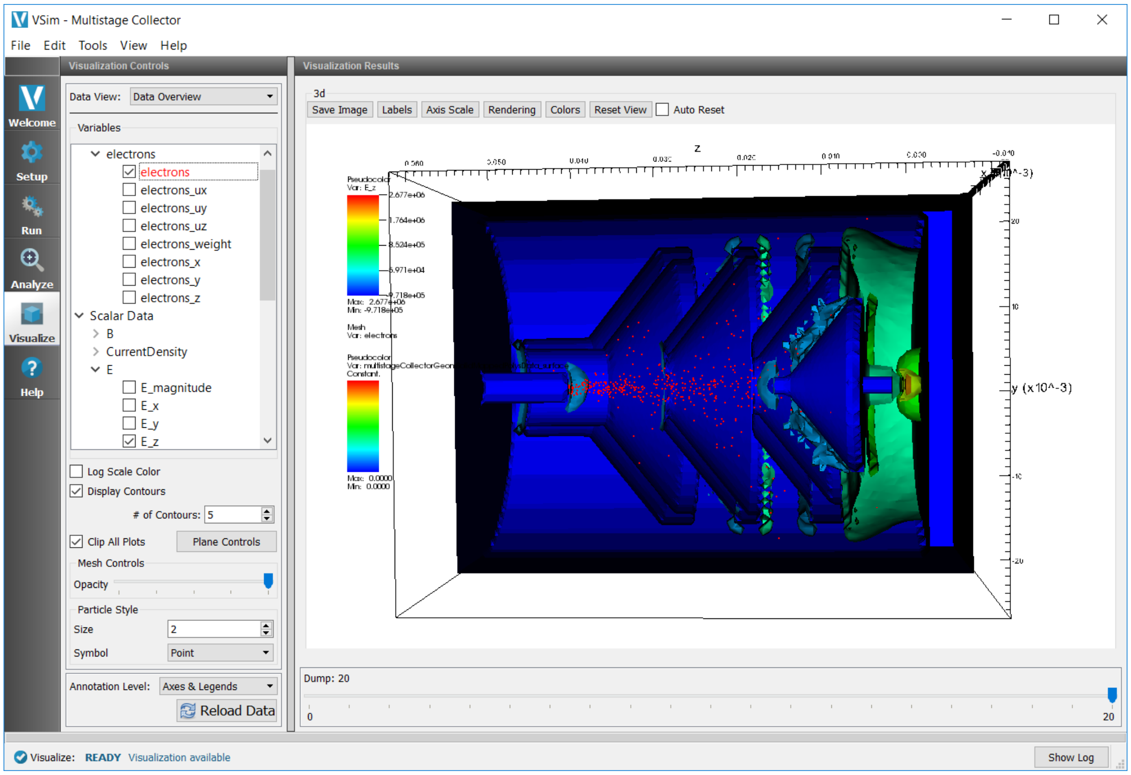
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The results are then read from the Data Overview in the Visualize Window:

- Expand *Particle Data*.
- Expand *electrons*.
- Select *electrons* in red.
- Expand *Scalar Data*.
- Expand *E*.
- Select *E_z*.
- Expand *Geometries*

- Select the second option from the top: *multistageCollectorGeomSolidMappedPolysData_surface*.
- Select *Display Cotours* in the bottom left corner.
- Select *Clip All Plots* in the bottom left corner.
- Click on *Plane Controls* and in the *Clip Plane Control* window, under *Clip Plane Normal* select *X (plane normal to x-axis)*, then click *Ok*.
- Move the Dump slider all the way to the end.
- Use the cursor to grab the image and rotate it from right to left to see the image in Fig. 4.7.2.



Data View: Data Overview

- electrons
  - ☑ electrons
  - □ electrons_ux
  - □ electrons_uy
  - □ electrons_uz
  - □ electrons_weight
  - □ electrons_x
  - □ electrons_y
  - □ electrons_z
- Scalar Data
  - B
  - CurrentDensity
  - E
    - □ E_magnitude
    - □ E_x
    - □ E_y
    - ☑ E_z

□ Log Scale Color

☑ Display Contours

# of Contours: 5

☑ Clip All Plots   Plane Controls

Opacity

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 20

Visualize: READY Visualization available

Visualization of the MDC model with a color contour plot of electric fields and electrons in red.

Further Experiments

The depressed voltages or beam current/radius can be varied in the input panel for testing runs. One can also change the grid cell numbers to see the convergence of the simulations.

References

[1] **M. C. Lin, P. H. Stoltz, D. N. Smithe, H. Song, H. J. Kim, J. J. Choi, S. J. Kim,** and S. H. Jang, “Design and Modeling of Multistage Depressed Collectors Using 3D Conformal Finite-Difference Time-Domain Particle-In-Cell Simulations ”, J. Korean Phys. Soc. 60, 731-738 (2012).

4.8 Other (text-based setup)

4.8.1 Electron Gun (electronGunT.pre)

Keywords:

`electron, gun, beam, collimate`

Problem description

Electron guns are devices that are often found in vacuum electronics as well as in more advanced technologies such as klystrons, electron microscopes, and particle accelerators. They produce narrow, collimated beams of electrons with precisely tuned kinetic energies. They were often found in cathode ray tubes at the heart of television sets prior to the digital revolution. Electron guns are composed of a cathode, an anode, and repulsive rings. A DC or RF signal is applied to the cathode to produce electrons via thermionic emission. The electrodes produce electric fields that focus the electron beam. Often an additional anode is placed between the cathode and the main anode to act as a repulsive ring that focuses the beam into a small hole in the main anode. The small hole in the main anode acts to collimate the beam.

This example is a specialized electron gun for klystrons and TWTs. It is characterized by high power, a consequence of which is that electrons not successfully collimated can damage the device. To minimize this effect, the gun includes a focusing anode cone, the angle of which is conducive to laminar flow of the electron beam.

Opening the Simulation

The Electron Gun example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Other MD (text-based setup)* option.
- Select "Electron Gun (text-based setup)" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

  The *Setup Window* is now shown with all the implemented physics and geometries, if applicable. See Fig. 4.125.

Input File Features

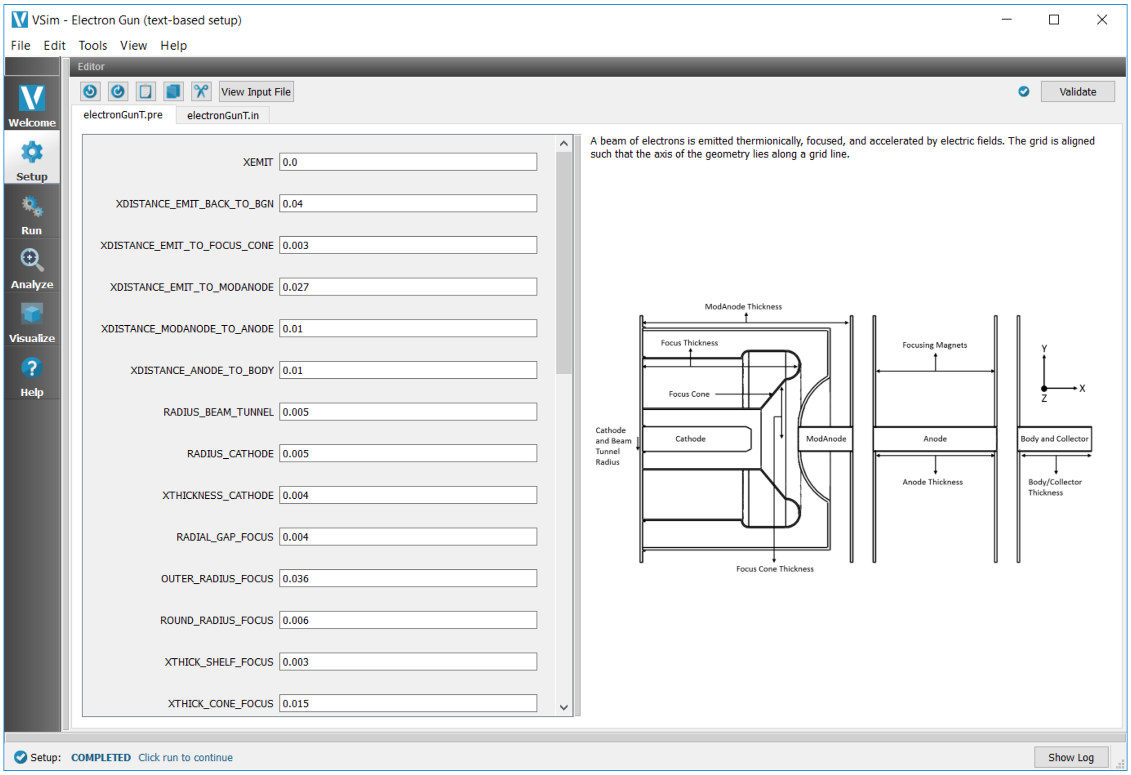
The input parameters give you total flexibility in in defining the geometry of the example. Along with these one can define the nominal cell size, the driving voltages, the strength of the magnetic field, and the beam current.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the window. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in the window below.

After performing the above actions, continue as follows:



XEMIT	0.0
XDISTANCE_EMIT_BACK_TO_BGN	0.04
XDISTANCE_EMIT_TO_FOCUS_CONE	0.003
XDISTANCE_EMIT_TO_MODANODE	0.027
XDISTANCE_MODANODE_TO_ANODE	0.01
XDISTANCE_ANODE_TO_BODY	0.01
RADIUS_BEAM_TUNNEL	0.005
RADIUS_CATHODE	0.005
XTHICKNESS_CATHODE	0.004
RADIAL_GAP_FOCUS	0.004
OUTER_RADIUS_FOCUS	0.036
ROUND_RADIUS_FOCUS	0.006
XTHICK_SHELF_FOCUS	0.003
XTHICK_CONE_FOCUS	0.015

A beam of electrons is emitted thermionically, focused, and accelerated by electric fields. The grid is aligned such that the axis of the geometry lies along a grid line.

Setup: COMPLETED Click run to continue

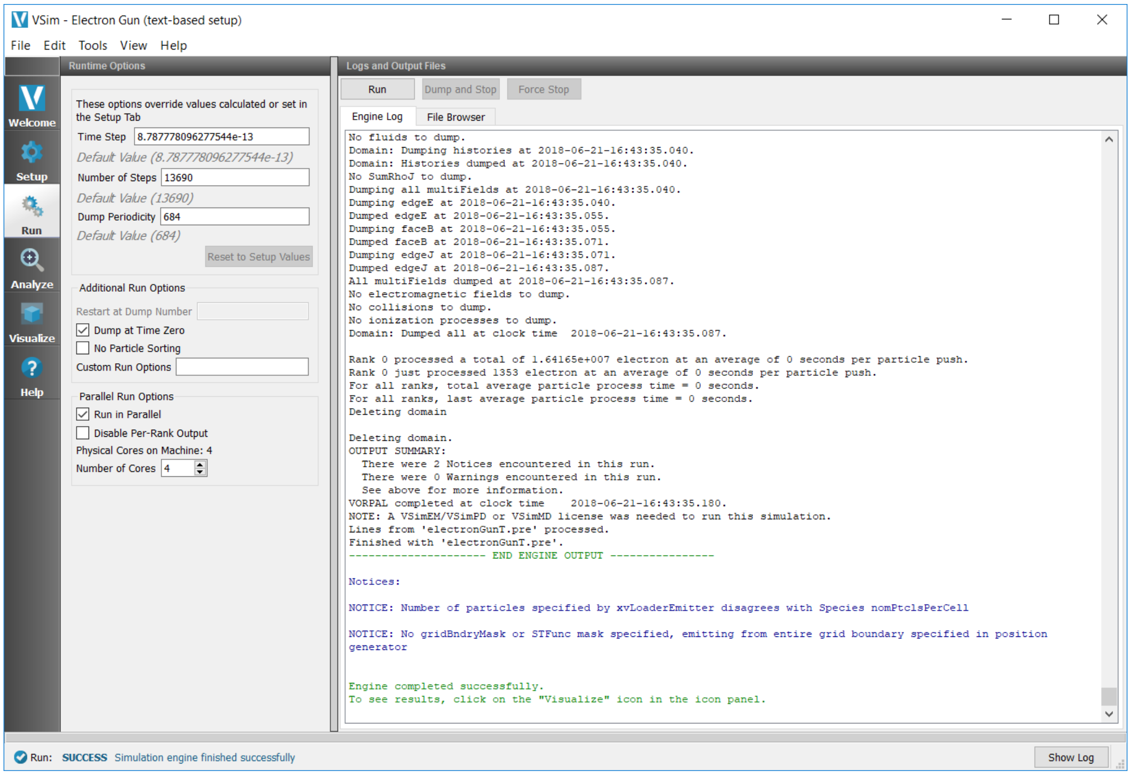
Fig. 4.125: Setup Window for the Electron Gun example.

- Proceed to the run window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.126.

Visualizing the Results

The stream of electrons can be visualized along with the electric field and the geometry by doing the following:

- Expand *Particle Data*.
- Expand *electron*.
- Select *electron*.
- Expand *Scalar Data*.
- Expand *edgeE*.
- Select *edgeE_magnitude*.
- Expand *Geometries*.
- Select *poly (electrodes)*
- Select *Display Contours*
- In the lower part of the left pane select “Display Contours”.
- Select “Clip All Plots”.



These options override values calculated or set in the Setup Tab

Time Step 8.787778096277544e-13
Default Value (8.787778096277544e-13)
Number of Steps 13690
Default Value (13690)
Dump Periodicity 684
Default Value (684)

Additional Run Options
Restart at Dump Number
☑ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options
☑ Run in Parallel
☐ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
No fluids to dump.
Domain: Dumping histories at 2018-06-21-16:43:35.040.
Domain: Histories dumped at 2018-06-21-16:43:35.040.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-21-16:43:35.040.
Dumping edgeE at 2018-06-21-16:43:35.040.
Dumped edgeE at 2018-06-21-16:43:35.055.
Dumping faceB at 2018-06-21-16:43:35.055.
Dumped faceB at 2018-06-21-16:43:35.071.
Dumping edgeJ at 2018-06-21-16:43:35.071.
Dumped edgeJ at 2018-06-21-16:43:35.087.
All multiFields dumped at 2018-06-21-16:43:35.087.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-21-16:43:35.087.

Rank 0 processed a total of 1.64165e+007 electron at an average of 0 seconds per particle push.
Rank 0 just processed 1353 electron at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 2 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-06-21-16:43:35.180.
NOTE: A VSimEM/VSimPD or VSimMD license was needed to run this simulation.
Lines from 'electronGunT.pre' processed.
Finished with 'electronGunT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: Number of particles specified by xvLoaderEmitter disagrees with Species nomPtclsPerCell

NOTICE: No gridBndryMask or STFunc mask specified, emitting from entire grid boundary specified in position
generator


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 4.126: The Run Window at the end of execution.

- In the lower part of the right pane, move the Dump slider to dump 12.

The phase space diagram can also be viewed by choosing *Phase Space* in the *Data View* drop down menu.

The voltages and currents at key locations in the simulation are recorded in Histories and can be viewed by selecting the *History* data view.

This will show the electron beam and the electric and magnetic fields.

Further Experiments

The geometry is extremely important for proper functionality in this example. For example, the angle that the focusing cone makes with the beam axis determines whether the beam will be laminar. If the beam intersects and diverges, the gun can be damaged by its own power. Try altering the dimensions of the the geometry and see the effect on the electron beam.

4.8.2 Multistage Collector (multistageCollectorT.pre)

Keywords:

electromagnetics, multistageCollectorT

Warning: Due to a known issue in the STL reader during parallel runs, this example may run into trouble if run on too many cores. We suggest limiting this run to 4 cores, though more may still work.

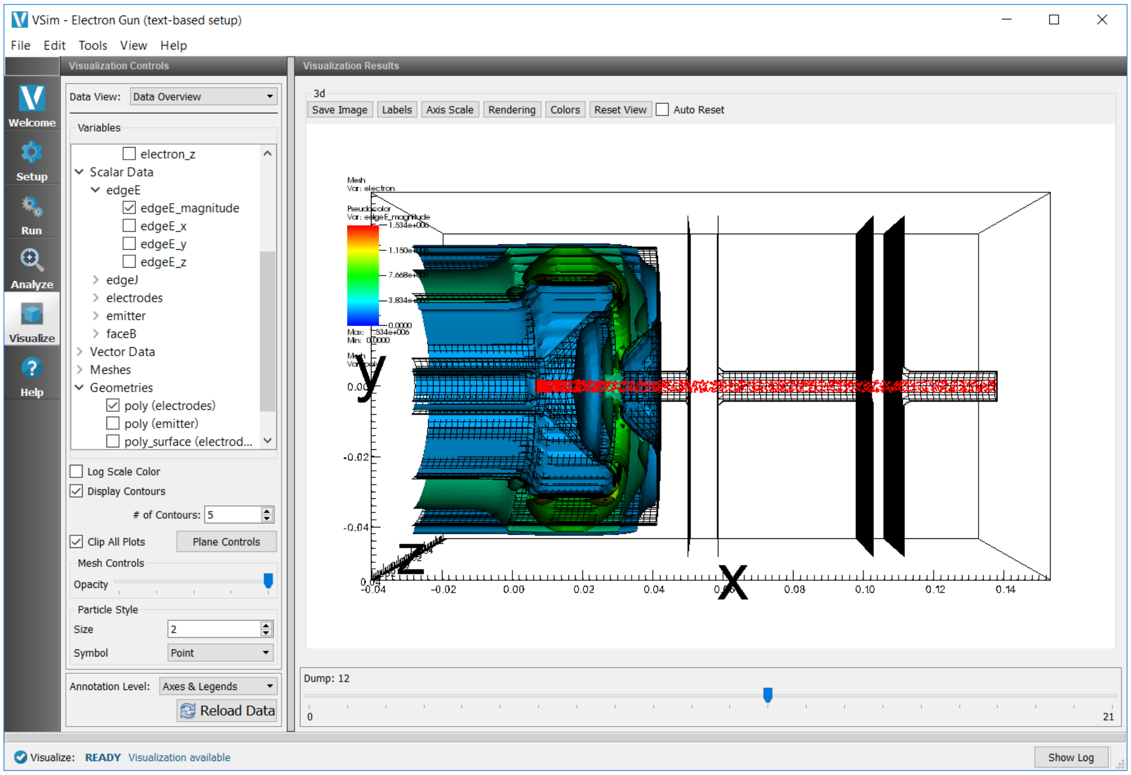


Fig. 4.127: The electron beam and electric field.

Problem description

Multistage Depressed Collectors (MDCs) are used to recover energy from a spent beam in linear type microwave tubes such as traveling wave tubes (TWTs) and klystrons. VSim provides the capability to simulate these collectors shaped with arbitrarily complex geometries and depressed with different time-dependent voltage profiles to optimize the recovery efficiency of a design. To demonstrate this capability, we show in this example a 4-stage depressed collector. One can adjust the depressed potentials at each electrode individually to see how the performance of the collector is affected.

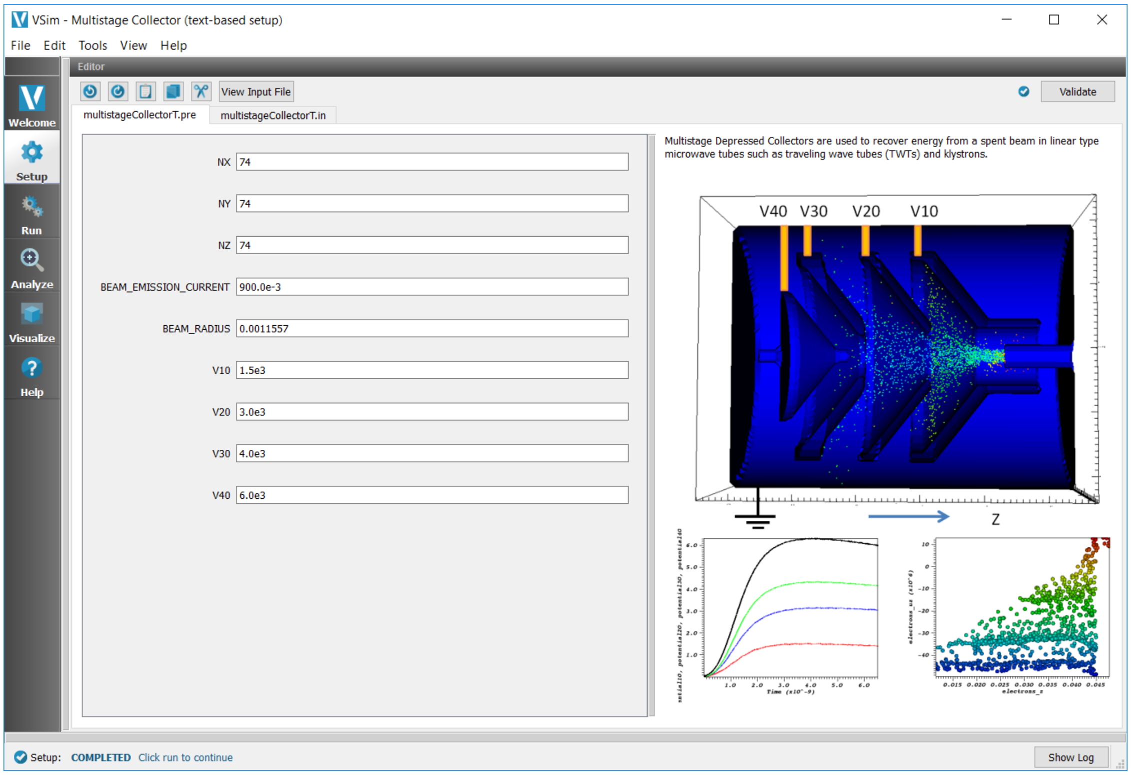
This simulation can be performed with a VSimMD or VSimPD license.

Opening the Simulation

The Multistage Collector example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Microwave Devices* option.
- Expand the *Other MD (text-based setup)* option.
- Select “Multistage Collector (text-based setup)” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is now shown with all the implemented physics and geometries, if applicable. See Fig. 4.128.



NX 74

NY 74

NZ 74

BEAM_EMISSION_CURRENT 900.0e-3

BEAM_RADIUS 0.0011557

V10 1.5e3

V20 3.0e3

V30 4.0e3

V40 6.0e3

Multistage Depressed Collectors are used to recover energy from a spent beam in linear type microwave tubes such as traveling wave tubes (TWTs) and klystrons.

Setup: COMPLETED Click run to continue

Fig. 4.128: Setup Window for the Multistage Collector example.

Input File Features

The simulation geometry consisting of an S-band 4-stage depressed collector is imported into the computational engine from CAD files in stl format. One can easily create new geometry using any CAD program and output or convert the CAD files into stl files for a new simulation design. The detailed import method is provided in the input file. The spent beam profile is taken from a TWT simulation provided by Prof. H. Song at University of Colorado at Colorado Springs.

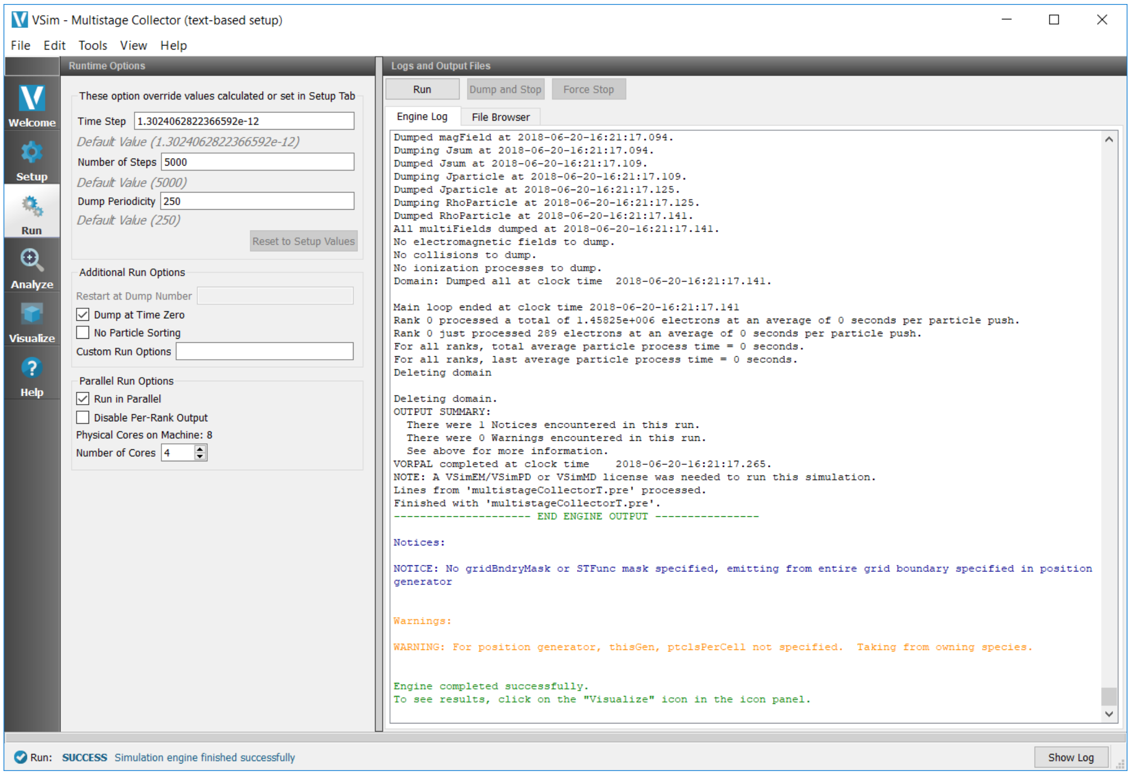
An optimized design for a MPM module can be found in reference [1]. Users can set preferred spent beam profiles by employing different emission methods or import data in dat format as in this example. A main feature of this input file is that the depressed voltage profiles are time-dependent and are stablized with a new external circuit model based on special feedback algorithms only available in VSim. Interested users may refer to the publication for more a detailed description and validation. In addition, the convergence of this example is carefully tested.

In this example, the Z coordinate is the direction aligned with the beam axis of the MDC, and the 4 different voltages can be easily assigned at the input panel. Since it is a time domain simulation, the Dey-Mittra algorithm is employed and the accuracy is second-order for the complex boundaries.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 4.129.



These option override values calculated or set in Setup Tab

Time Step 1.3024062822366592e-12
Default Value (1.3024062822366592e-12)
Number of Steps 5000
Default Value (5000)
Dump Periodicity 250
Default Value (250)

Additional Run Options

Restart at Dump Number
☑ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 8
Number of Cores 4

```
Dumped magField at 2018-06-20-16:21:17.094.
Dumping Jsum at 2018-06-20-16:21:17.094.
Dumped Jsum at 2018-06-20-16:21:17.109.
Dumping Jparticle at 2018-06-20-16:21:17.109.
Dumped Jparticle at 2018-06-20-16:21:17.125.
Dumping RhoParticle at 2018-06-20-16:21:17.125.
Dumped RhoParticle at 2018-06-20-16:21:17.141.
All multiFields dumped at 2018-06-20-16:21:17.141.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-20-16:21:17.141.

Main loop ended at clock time 2018-06-20-16:21:17.141
Rank 0 processed a total of 1.45825e+006 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 289 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 1 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-06-20-16:21:17.265.
NOTE: A VSimEM/VSimPD or VSimMD license was needed to run this simulation.
Lines from 'multistageCollectorT.pre' processed.
Finished with 'multistageCollectorT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: No gridBndryMask or STFunc mask specified, emitting from entire grid boundary specified in position
generator


Warnings:

WARNING: For position generator, thisGen, ptclsPerCell not specified.  Taking from owning species.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 4.129: The Run Window at the end of execution.

Visualizing the results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The results are then read from the Data Overview in the Visualize Window:

- Expand *Particle Data*.
- Expand *electrons*.
- Select *electrons* in red.
- Expand *Scalar Data*.
- Expand *elecField*.
- Select *elecField_z*.
- Expand *Geometries*.
- Select the second option from the top: *poly_surface*.
- Select *Display Cotours* in the bottom left corner.
- Select *Clip All Plots* in the bottom left corner.
- Click on *Plane Controls* and in the *Clip Plane Control* window, under *Clip Plane Normal* select *X (plane normal to x-axis)*, then click *Ok*.
- Use the cursor to grab the image and rotate it from right to left.

The time dependent evolution of the MDC can be seen by sliding the Dump number from left to right, as shown in Fig. 4.130.

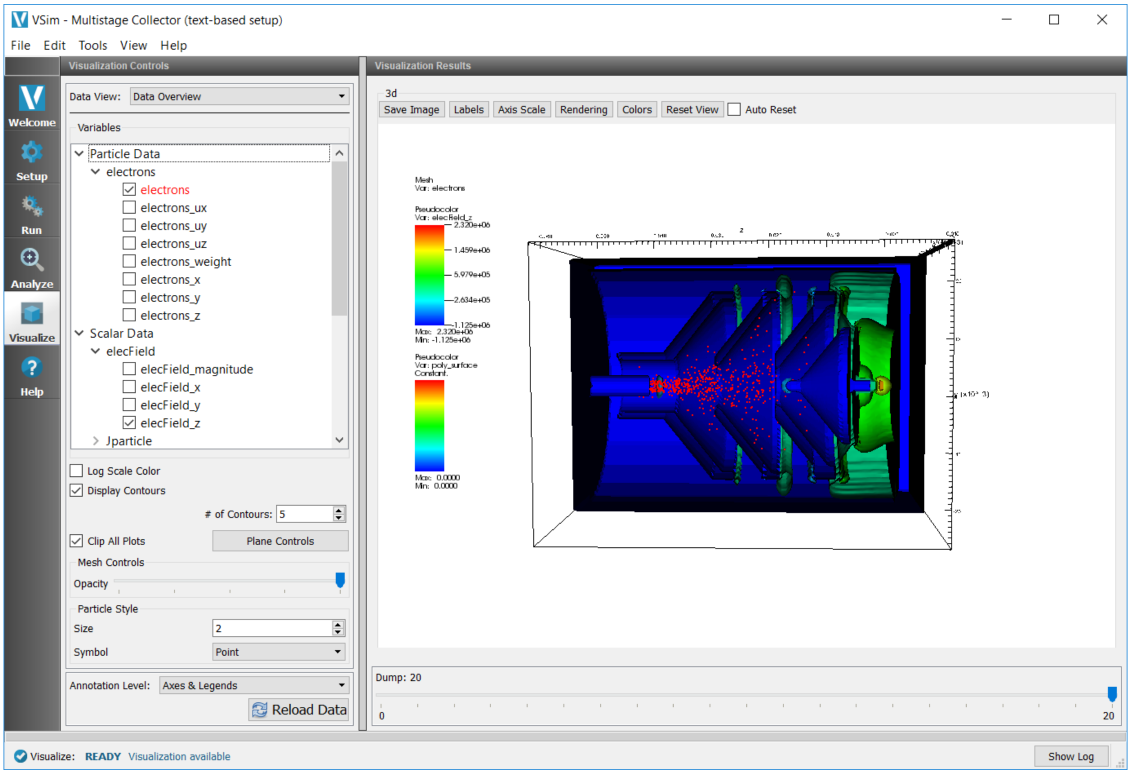


Fig. 4.130: Visualization of the MDC model with a color contour plot of electric fields and electrons in red.

The potential of each of collector sufraces is recorded using a history. To visualize these values as shown in Fig. 4.131, do the following:

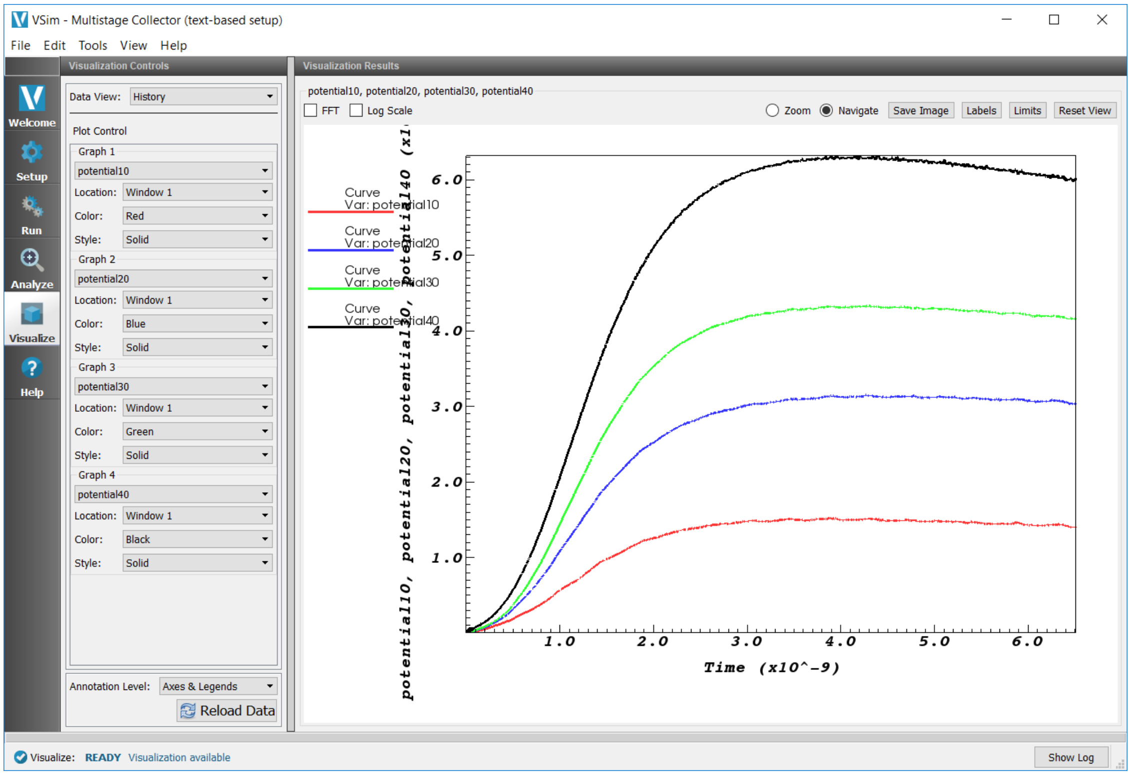
- Switch *Data View* to “History”.
- In the left pane, set Graphs 1-4 to each of the different potential histories: “potential10”, “potential20”, “potential30”, and “potential40”, respectively (see Fig. 4.131).
- Set the *Location* of each graph to the “Window 1”.

Further Experiments

The depressed voltages or beam current/radius can be varied in the input panel for testing runs. One can also change the grid cell numbers to see the convergence of the simulations.

References

[1] **M. C. Lin, P. H. Stoltz, D. N. Smithe, H. Song, H. J. Kim, J. J. Choi, S. J. Kim,** and S. H. Jang, “Design and Modeling of Multistage Depressed Collectors Using 3D Conformal Finite-Difference Time-Domain Particle-In-Cell Simulations ”, J. Korean Phys. Soc. 60, 731-738 (2012).



Data View: History

Plot Control

Graph 1

potential10

Location: Window 1

Color: Red

Style: Solid

Graph 2

potential20

Location: Window 1

Color: Blue

Style: Solid

Graph 3

potential30

Location: Window 1

Color: Green

Style: Solid

Graph 4

potential40

Location: Window 1

Color: Black

Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 4.131: The value of the potential on each of the collectors.

CHAPTER

FIVE

VSIM FOR PLASMA ACCELERATION EXAMPLES

These examples illustrate how to solve complex problems in plasma acceleration.

These examples can be run with a VSimPA license.

5.1 Beam Driven (text-based setup)

5.1.1 Electron Beam Driven Plasma Wakefield (electronBeamDrivenPlasmaT.pre)

Keywords:

`electron driven, plasma wakefield, CLARA, PARS, AWAKE`

Problem description

This example demonstrates a method to simulate an electron beam driven plasma wakefield accelerator. The electron beam initializes the field using a speed of light frame Poisson equation solve, then the fields and particles are evolved using FDTD EMPIC. We launch the electron beam from x=0 in the positive x direction using the Lorentz boosted Poisson fields to ensure that the simulation is self-consistent from start. The primary bunch generates a region of high field into which one might inject and accelerate a second bunch of charged particles. The example simulation uses parameters that are appropriate to the plasma acceleration research station (PARS) at the CLARA accelerator at Daresbury Laboratory in the UK.

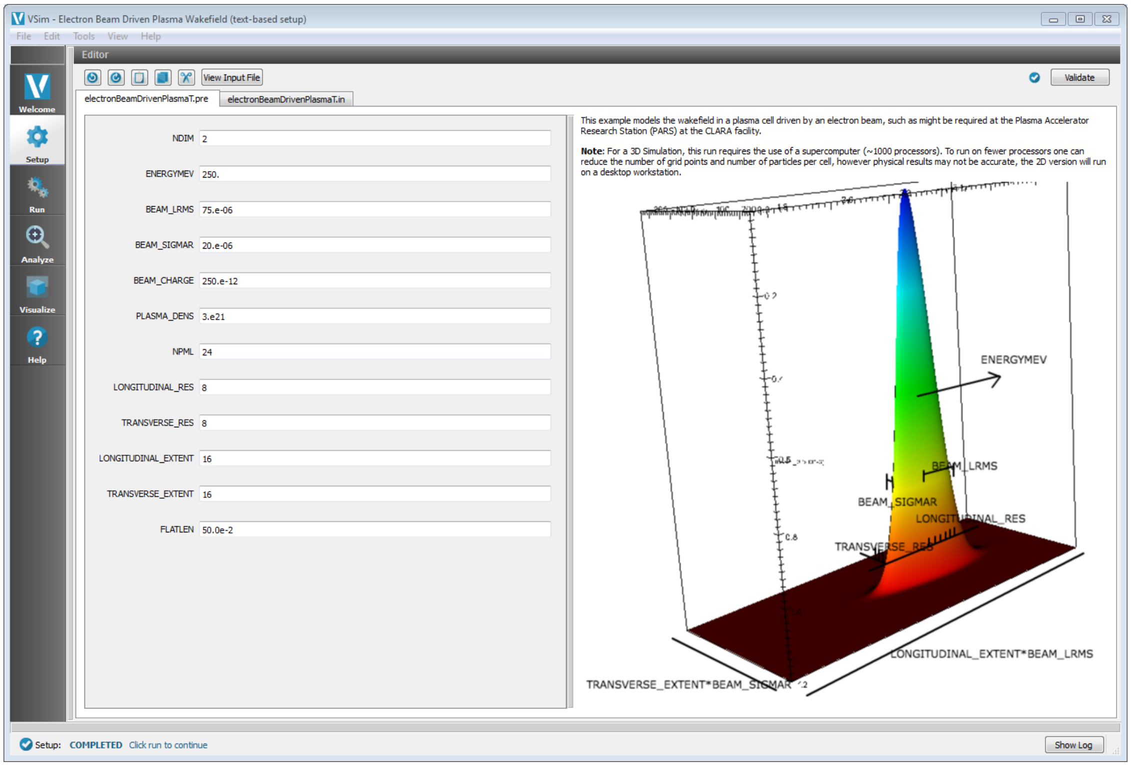
Opening the Simulation

The Electron Beam Driven Plasma Wakefield example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Acceleration* option.
- Expand the *Beam Drive Acceleration (text-based setup)* option.
- Select *Electron Beam Driven Plasma Wakefield (text-based setup)* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 5.1.

At this stage one can choose parameters such as `LONGITUDINAL_RES` and `TRANSVERSE_RES` which represent the longitudinal and transverse number of cells per RMS bunch size. The minimum of 6 or default of 8, generates a


VSim - Electron Beam Driven Plasma Wakefield (text-based setup)
File
Edit
Tools
View
Help
Editor
View Input File
Validate
Welcome
Setup
Run
Analyze
Visualize
Help
electronBeamDrivenPlasmaT.pre
electronBeamDrivenPlasmaT.in
ENERGYMEV
BEAM_LRMS
BEAM_SIGMAR
LONGITUDINAL_RES
TRANSVERSE_RES
LONGITUDINAL_EXTENT*BEAM_LRMS
TRANSVERSE_EXTENT*BEAM_SIGMAR
Show Log

NDIM	2
ENERGYMEV	250.
BEAM_LRMS	75.e-06
BEAM_SIGMAR	20.e-06
BEAM_CHARGE	250.e-12
PLASMA_DENS	3.e21
NPML	24
LONGITUDINAL_RES	8
TRANSVERSE_RES	8
LONGITUDINAL_EXTENT	16
TRANSVERSE_EXTENT	16
FLATLEN	50.0e-2

This example models the wakefield in a plasma cell driven by an electron beam, such as might be required at the Plasma Accelerator Research Station (PARS) at the CLARA facility.

Note: For a 3D Simulation, this run requires the use of a supercomputer (~1000 processors). To run on fewer processors one can reduce the number of grid points and number of particles per cell, however physical results may not be accurate, the 2D version will run on a desktop workstation.

Setup: COMPLETED Click run to continue

Fig. 5.1: Setup Window for the Electron Driven Plasma Wake example.

simulation that will complete reasonably quickly, but is not adequate to generate good results. Consider 12 or 16 cells in both dimensions to avoid a “checkerboard” pattern of numerical noise from developing.

Input File Features

The simulation setup consists of an electromagnetic solver using the Yee algorithm and uses the initBeam macro to set up the inital beam properties. This takes the beam of variable weight particles and calculates self-consistent fields with which to initialise the simulation. As this beam travels near the speed of light, a moving window that co-propogates with the beam is employed. MALs are used on the transverse sides of the window to absorb outgoing waves. The plasma is represented by macro-particles, and both beam and plasma are moved using the Boris push. The particles in the plasma are variably weighted to represent the density ramp. It is assumed the plasma consists of pre-ionized heavy ions, which do not move in the time frame of the simulations.

One can specify the size of the region to be simulated through `LONGITUDINAL_EXTENT` and `TRANSVERSE_EXTENT`, which are measured relative to the longitudinal RMS size `BEAM_LRMS` and transverse RMS size `BEAM_SIGMAR` of the beam. The number of cells is determined by the settings of `LONGITUDINAL_RES` and `TRANSVERSE_RES`, as shown in the figure.

The plasma density is ramped up using a flat top cosine function, by default, over a quarter of the longitudinal size of the simulation window. This can be modified by viewing the input file and editing the `STARTRAMP` and `RAMPLEN` variables.

Running the Simulations

After performing the above actions, continue as follows:

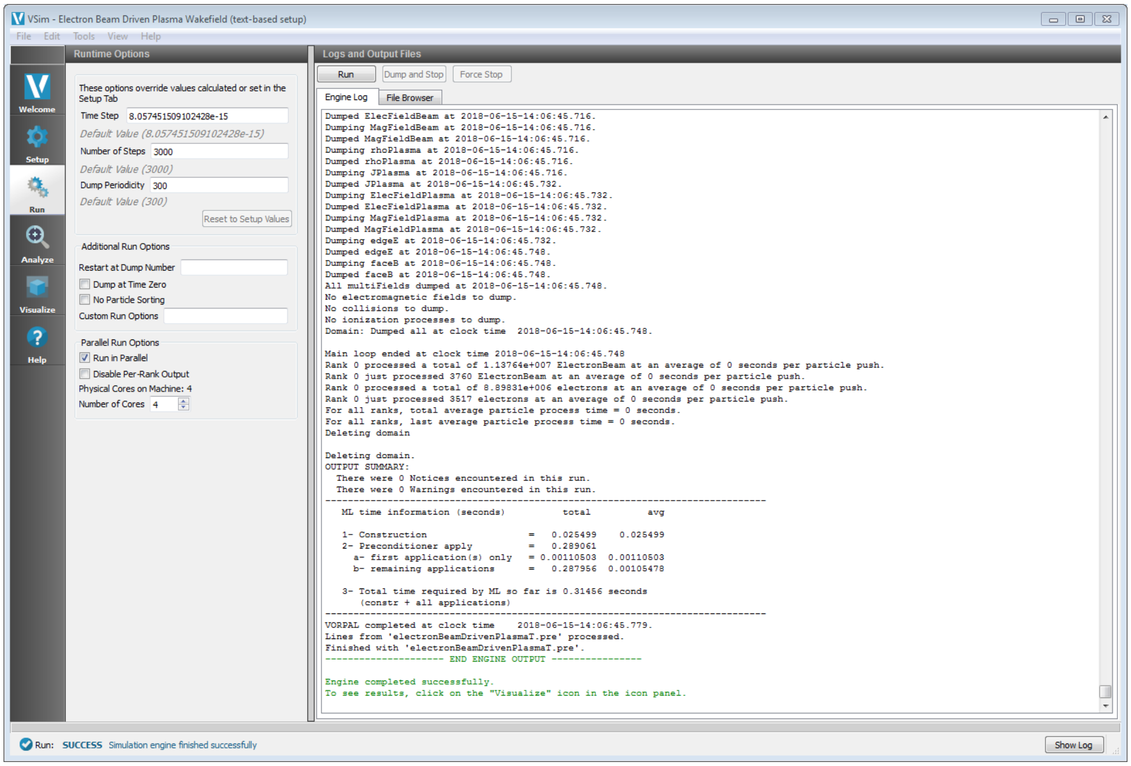
- Proceed to the Run Window by pressing the Run button in the left column of buttons.

- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane. You will see the output of the run in the right pane. This is shown in Fig. 5.2. The run has completed when you see the output, “Engine output has completed succesfully.”

Running in 2D, you can expect a 2 core laptop to take a few minutes at the default resolution and run time.

To produce real significant results, a higher resolution is required by changing LONGITUDINAL_RES and TRANSVERSE_RES. Doing so will greatly increase the amount of time to run this simulation. It should also be run for a longer time than the default 3000 steps.

The 3D simulation is about 100 times bigger, so 256 cores for a few hours is needed.



These options override values calculated or set in the Setup Tab

Time Step 8.057451509102428e-15

Default Value (8.057451509102428e-15)

Number of Steps 3000

Default Value (3000)

Dump Periodicity 300

Default Value (300)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Dumped ElecFieldBeam at 2018-06-15-14:06:45.716.
Dumping MagFieldBeam at 2018-06-15-14:06:45.716.
Dumped MagFieldBeam at 2018-06-15-14:06:45.716.
Dumping rhoPlasma at 2018-06-15-14:06:45.716.
Dumped rhoPlasma at 2018-06-15-14:06:45.716.
Dumping JPlasma at 2018-06-15-14:06:45.716.
Dumped JPlasma at 2018-06-15-14:06:45.732.
Dumping ElecFieldPlasma at 2018-06-15-14:06:45.732.
Dumped ElecFieldPlasma at 2018-06-15-14:06:45.732.
Dumping MagFieldPlasma at 2018-06-15-14:06:45.732.
Dumped MagFieldPlasma at 2018-06-15-14:06:45.732.
Dumping edgeE at 2018-06-15-14:06:45.732.
Dumped edgeE at 2018-06-15-14:06:45.748.
Dumping faceB at 2018-06-15-14:06:45.748.
Dumped faceB at 2018-06-15-14:06:45.748.
All multiFields dumped at 2018-06-15-14:06:45.748.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-15-14:06:45.748.

Main loop ended at clock time 2018-06-15-14:06:45.748
Rank 0 processed a total of 1.13764e+007 ElectronBeam at an average of 0 seconds per particle push.
Rank 0 just processed 3760 ElectronBeam at an average of 0 seconds per particle push.
Rank 0 processed a total of 8.89831e+006 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 3517 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
------------------------------------------------------------------------------
   ML time information (seconds)          total          avg

   1- Construction                  =   0.025499    0.025499
   2- Preconditioner apply          =   0.289061
     a- first application(s) only   = 0.00110503  0.00110503
     b- remaining applications      =   0.287956  0.00105478

   3- Total time required by ML so far is 0.31456 seconds
      (constr + all applications)
------------------------------------------------------------------------------
VORPAL completed at clock time    2018-06-15-14:06:45.779.
Lines from 'electronBeamDrivenPlasmaT.pre' processed.
Finished with 'electronBeamDrivenPlasmaT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 5.2: The Run Window.

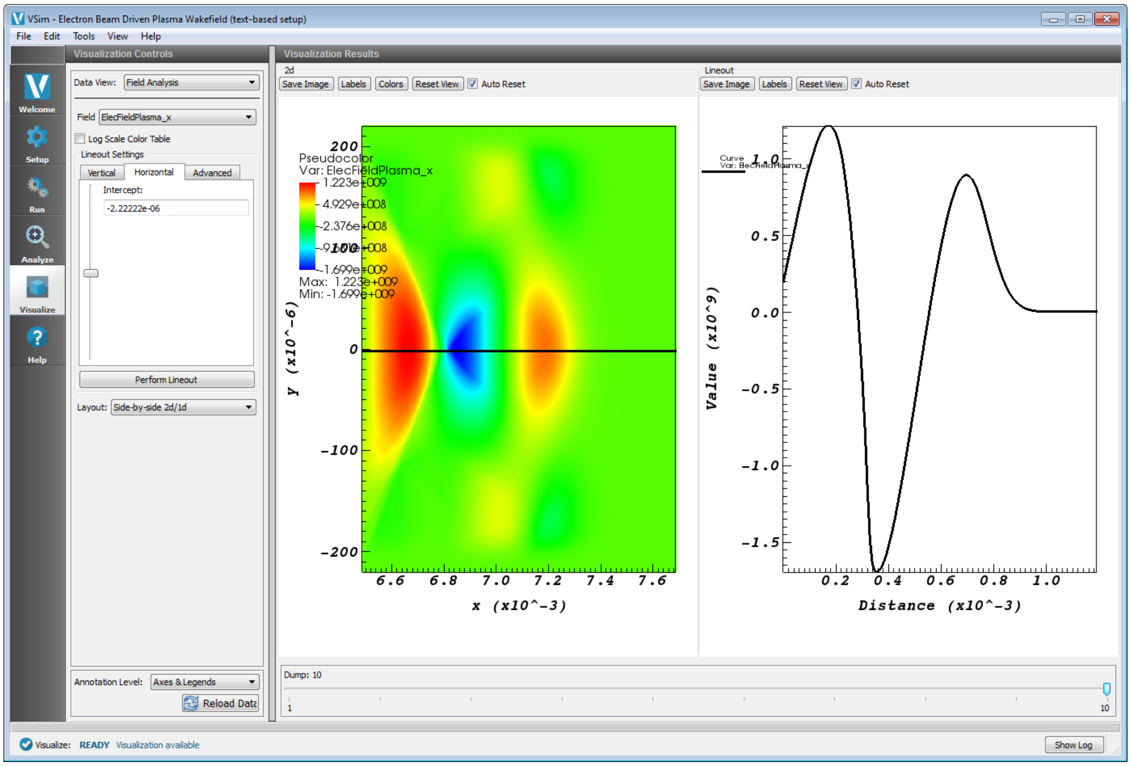
Visualizing the Output

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

View the electric field generated by the plasma as shown in Fig. 5.3 by doing the following:

- Switch to *Field Analysis* in the *Data View* Controls pane
- Set the *Field* to *ElecFieldPlasma_x*
- Choose the *Horizontal* tab in *Lineout Settings* set the intercept to zero, and click *Perform Lineout*
- Check the *Auto Reset* buttons on both the 2d and the Lineout plots. Sometimes it is necessary to expand the plot size in order for the box to appear. You can do this by pulling the divider between “Visualization Controls” and “Visualization Results” to the left and hiding it. Both the 2d and the Lineout plots should be larger now.
- Move the dump slider forward in time.



Data View: Field Analysis

Field: ElecFieldPlasma_x

☐ Log Scale Color Table

Lineout Settings: Vertical | Horizontal | Advanced

Intercept: -2.22222e-06

Layout: Side-by-side 2d/1d

Annotation Level: Axes & Legends

☑ Auto Reset

Dump: 10

Visualize: READY Visualization available

Fig. 5.3: Visualization of the longitudinal electric field as a color contour plot and longitudinal lineout.

The plasma density can be seen as shown in Fig. 5.4 by doing the following:

- Switch to *Data Overview* in the *Data View* drop down
- Expand *Scalar Data*
- Select *rhoPlasma*
- Click *Auto Reset*
- Move slider all the way to the right.

5.2 Laser Driven

5.2.1 Laser Plasma Accelerator (laserPlasmaAccel.sdf)

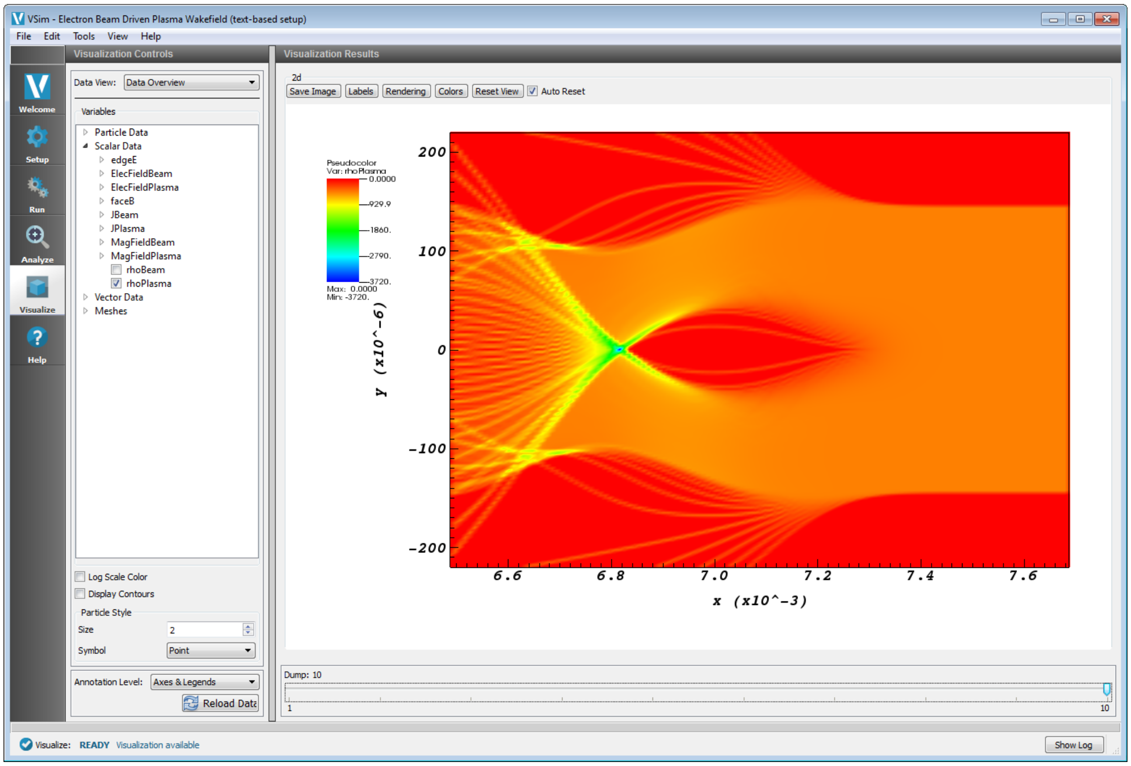
Keywords:

`Laser Plasma Accelerator`

Problem description

This example demonstrates the use of VSim to simulate a simple laser-plasma accelerator problem using the full PIC algorithm.

An intense, short laser pulse propagating through a plasma can lead to the separation of electrons and ions capable of producing accelerating electric fields of hundreds of GV/m *[GTVT+04]*. VSim is capable of simulating laser plasma



Data View: Data Overview

☐ Log Scale Color
☐ Display Contours

Size: 2
Symbol: Point

Annotation Level: Axes & Legends

☑ Auto Reset

Dump: 10

Visualize: READY Visualization available

Fig. 5.4: Visualization of the longitudinal plasma density field as a color contour plot.

accelerators (laserPlasmaAccel) using several different models: envelope, fluid and full particle-in-cell (PIC).

Here we look at the full PIC model with a 1-mm long plasma with uniform density of $1.e25$ m^{-3}. A gaussian laser pulse, defined by the transverse electric field

$$E_y = E_0(0.5 - 0.5\cos(\pi t/T))H(T-t)\exp(-(y^2+z^2)/(w_0^2 D_y D_z))\cos(\omega t + \phi_y + \phi_z)$$

where w_0 is the radius at which the wave amplitude drops to $1/e$, T is the temporal duration of the pulse,

$$D_y = 1 + (F/Z)^2$$
$$D_z = 1 + (F/Z)^2$$

are the squares of amplitude reductions from being the launching at a distance, F, from the focus, Z is the Rayleigh length, and

$$\phi_y = -(F/Z)(y/w_0)^2/D_y$$
$$\phi_z = -(F/Z)(z/w_0)^2/D_z$$

are the Gouy phases.

The laser is launched from the left side of the box. The laser amplitude is determined through the normalized vector potential $A_0 = eE_0/\omega m_e c$, where ω is the laser angular frequency.

The simulation setup consists of an electromagnetic solver using the Yee algorithm. The laser pulse is launched from the left side of the window using the pre-defined gaussian pulse launcher at the left boundary. Simple conducting boundary conditions are used at the top and sides. As such, one must ensure that waves reflected off the top or bottom do not get into the simulation, and that no waves hit the right boundary to be reflected back into the simulation.

The plasma is represented by macro-particles which are moved using the Boris push. The particles are variably weighted to represent the density ramp.

The input file allows one to set up plasma and laser parameters. The simulation box size is determined as a function of the laser length and spot size. The resolution was set to have about 24 cells per wavelength longitudinally and 3 transversely. The time step is chosen to be very close to the courant condition limit in order to have good dispersion.

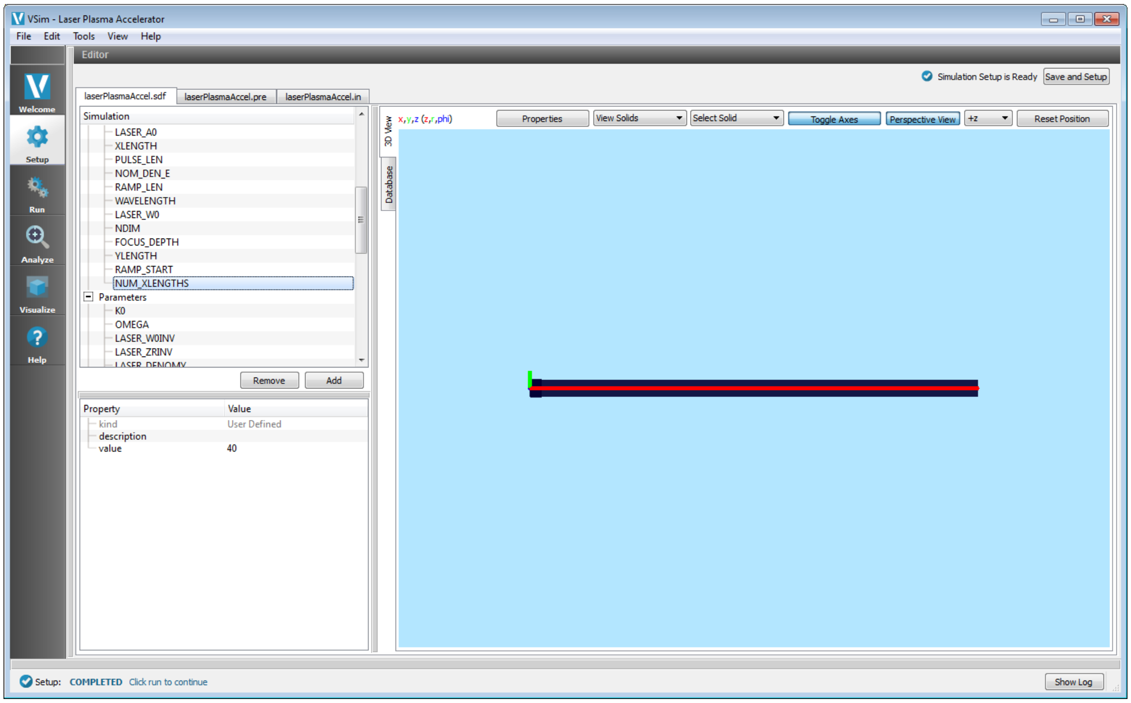
This simulation can be performed with a VSimPA license.

Opening the Simulation

The Laser Plasma Accelerator example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Acceleration* option.
- Expand the *Introductory Examples* option.
- Select "Laser Plasma Acceleration" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 5.5. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.



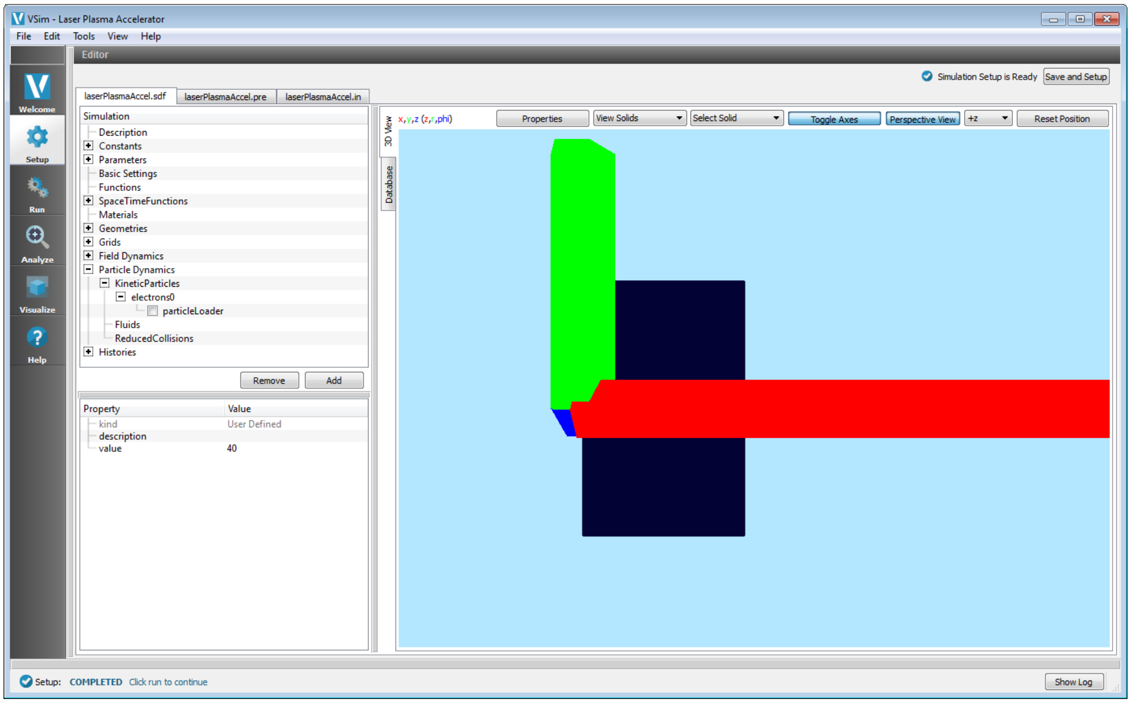
Property	Value
kind	User Defined
description	
value	40

Setup: COMPLETED Click run to continue

Fig. 5.5: Setup Window for the Laser Plasma Acceleration example.

The Setup Window shows a very long simulation. However, the full length is not simulated at any one time. Instead a moving window is used to simulate only the region where dynamics is occurring. The length seen in Fig. 5.6 is that of the electron loader.

To see the grid size, one can expand *Particle Dynamics* → *KineticParticles* → *electrons0* and unclick *particleLoader*. If one wishes to simulate this for longer distances, one can set the Parameter, `NUM_XLENGTHSS`, to a larger number; 400 is more than enough.



Property	Value
kind	User Defined
description	
value	40

Fig. 5.6: Zoom of Setup Window for the Laser Plasma Acceleration example showing size of grid.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper right corner of the window. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” See Fig. 5.7.

Running in 2D, this simulation uses around 225,000 cells and nearly 200,000 particles for 20,000 time steps. The run takes about an hour on a 4-core 2.5 GHz I7.

Visualizing the results

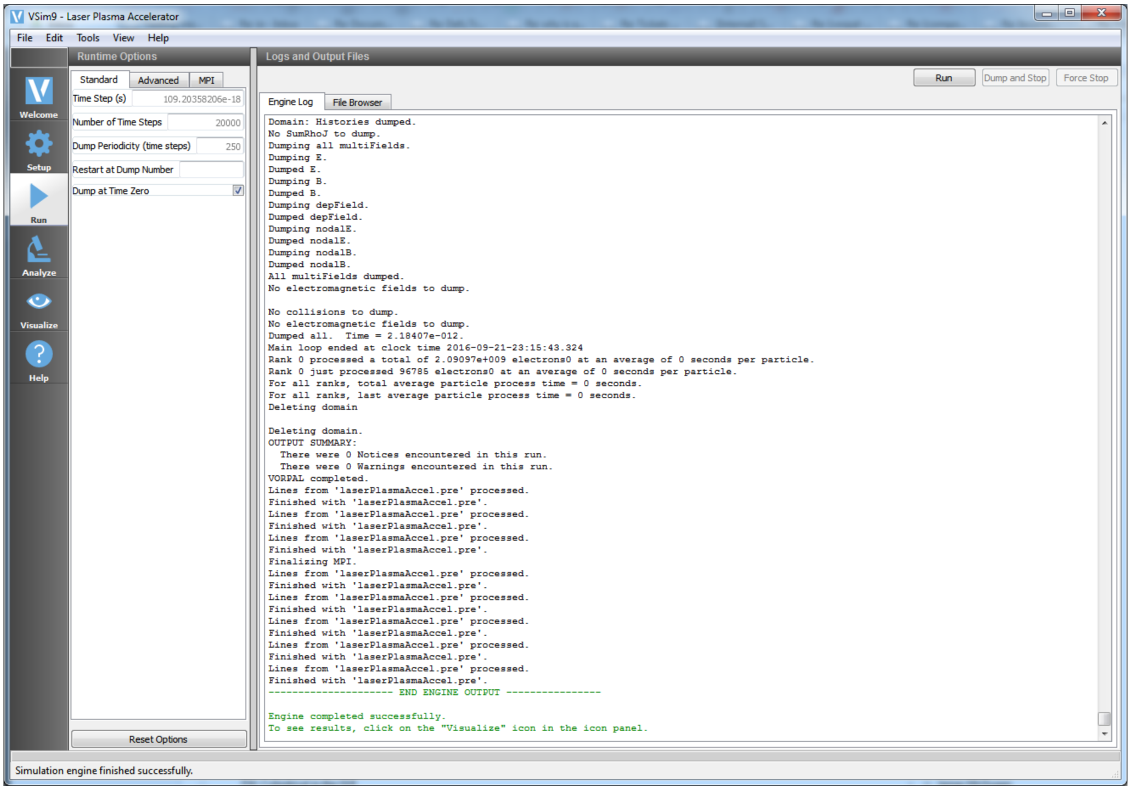
After performing the above actions, click on *Visualize* in the column of buttons at the left. For all plots, it is useful to keep Auto Reset on so that the window moves with the data.

To view the electric field, switch to Field Analysis in the Data View drop-down menu. From the Field drop-down menu, choose the desired component of the electric field, `E`. The `depField` field is the current density. See Fig. 5.8.

The acceleration of the particles can be seen by viewing the (x) component of the velocity. To do this, switch to the Phase Space Data View in VSimComposer and set the X-axis variable to electrons0_x, and the Y-axis variable to electrons0_ux. Then click Draw. You will see a color-coded picture of particle velocity like in Fig. 5.9.

Further Experiments

Try increasing or decreasing the intensity of the laser pulse through the parameter `A_0` and see the effect on the shape of the plasma wakefield.

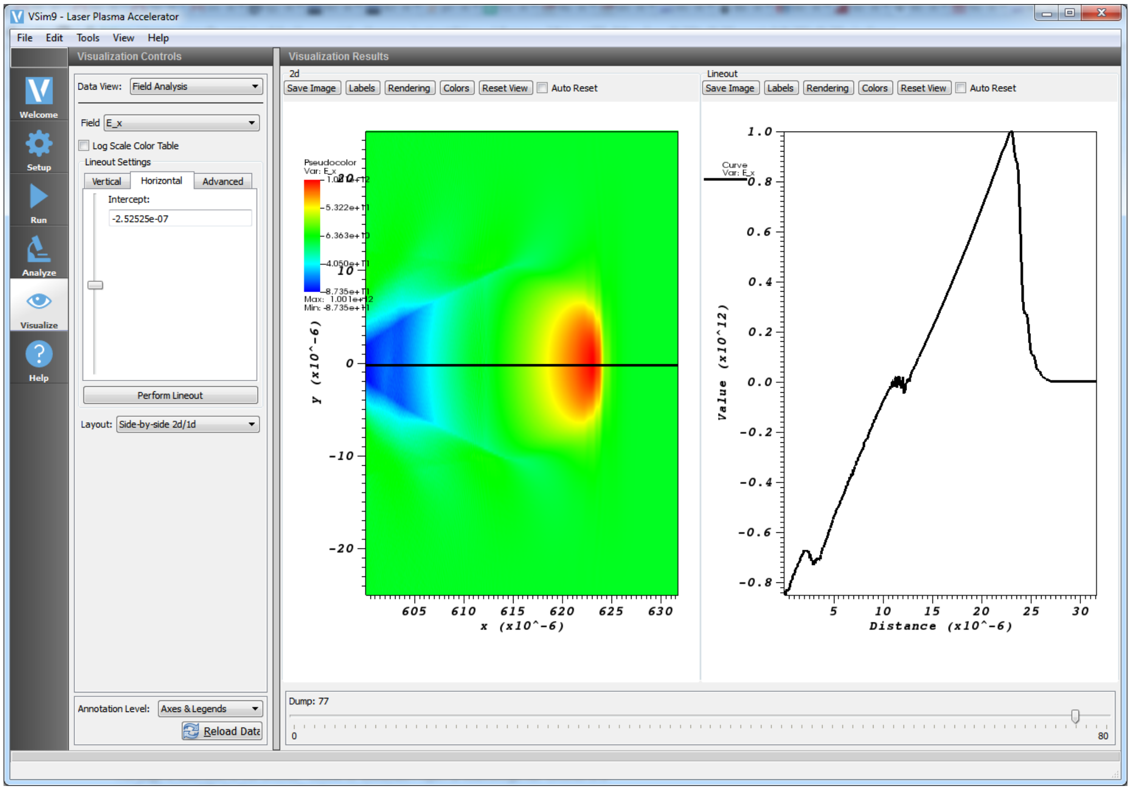


Time Step (s) 109.20358206e-18
Number of Time Steps 20000
Dump Periodicity (time steps) 250
Restart at Dump Number
Dump at Time Zero ☑

```
Domain: Histories dumped.
No SumRhoJ to dump.
Dumping all multiFields.
Dumping E.
Dumped E.
Dumping B.
Dumped B.
Dumping depField.
Dumped depField.
Dumping nodalE.
Dumped nodalE.
Dumping nodalB.
Dumped nodalB.
All multiFields dumped.
No electromagnetic fields to dump.

No collisions to dump.
No electromagnetic fields to dump.
Dumped all.  Time = 2.18407e-012.
Main loop ended at clock time 2016-09-21-23:15:43.324
Rank 0 processed a total of 2.09097e+009 electrons0 at an average of 0 seconds per particle.
Rank 0 just processed 96785 electrons0 at an average of 0 seconds per particle.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed.
Lines from 'laserPlasmaAccel.pre' processed.
Finished with 'laserPlasmaAccel.pre'.
Lines from 'laserPlasmaAccel.pre' processed.
Finished with 'laserPlasmaAccel.pre'.
Lines from 'laserPlasmaAccel.pre' processed.
Finished with 'laserPlasmaAccel.pre'.
Finalizing MPI.
Lines from 'laserPlasmaAccel.pre' processed.
Finished with 'laserPlasmaAccel.pre'.
Lines from 'laserPlasmaAccel.pre' processed.
Finished with 'laserPlasmaAccel.pre'.
Lines from 'laserPlasmaAccel.pre' processed.
Finished with 'laserPlasmaAccel.pre'.
Lines from 'laserPlasmaAccel.pre' processed.
Finished with 'laserPlasmaAccel.pre'.
Lines from 'laserPlasmaAccel.pre' processed.
Finished with 'laserPlasmaAccel.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

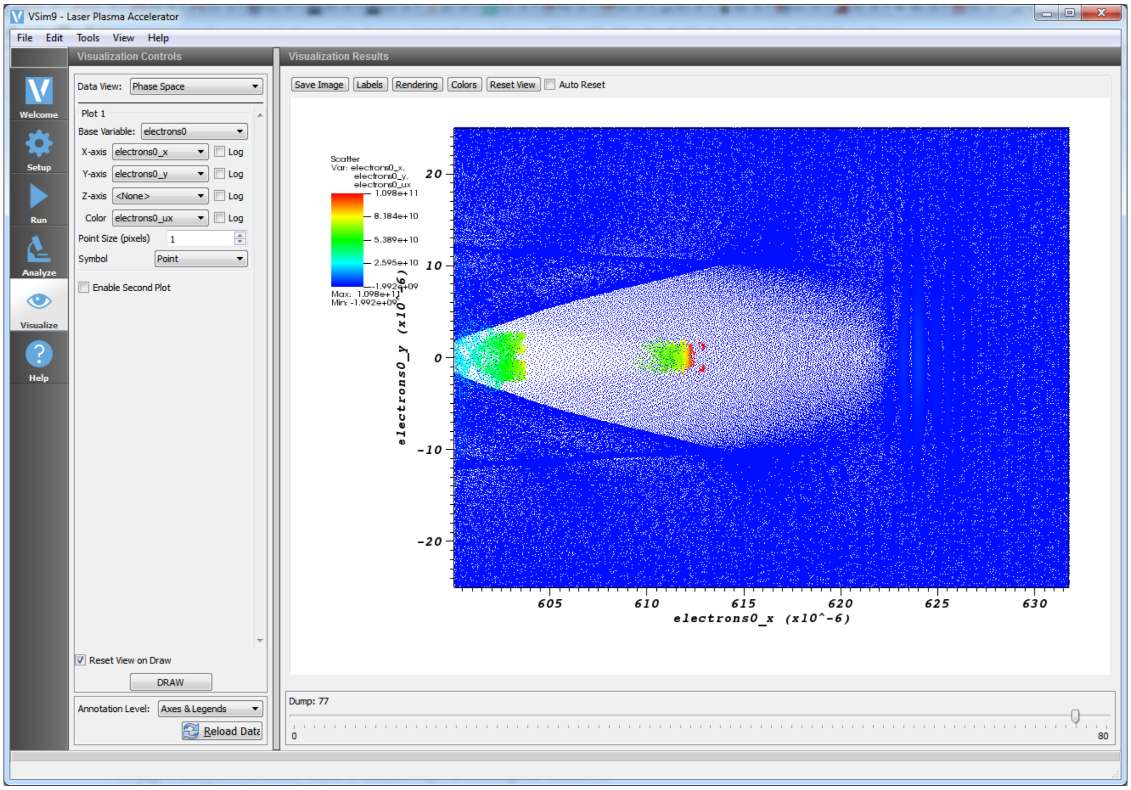
Simulation engine finished successfully.

Fig. 5.7: The Run Window at the end of execution.



Data View: Field Analysis
Field E_x
□ Log Scale Color Table
Lineout Settings
Intercept: -2.52525e-07
Layout: Side-by-side 2d/1d
Annotation Level: Axes & Legends
Dump: 77

Fig. 5.8: The output of the run shows the accelerating field E_x after about 2 picoseconds.



Data View: Phase Space

Plot 1

Base Variable: electrons0

X-axis electrons0_x □ Log

Y-axis electrons0_y □ Log

Z-axis <None> □ Log

Color electrons0_ux □ Log

Point Size (pixels) 1

Symbol Point

□ Enable Second Plot

☑ Reset View on Draw

DRAW

Annotation Level: Axes & Legends

Reload Data

Dump: 77

Fig. 5.9: Phase-space plot of plasma electrons at t = 2.1 picoseconds.

5.3 Laser Driven (text-based setup)

5.3.1 Colliding Pulse Injection (collidingPulseInjT.pre)

Keywords:

`laser plasma accelerator, controlled injection, colliding laser pulses`

Problem description

This example demonstrates the use of VSim to simulate controlled injection in a laser-plasma accelerator using colliding laser pulses *[CMRB+10]*. Two laser pulses are launched from opposite sides (one from the left side and the other one from the right side of the box) and propagate in opposite directions. The laser pulse coming from the left side is the main pulse that drives the plasma wake. The laser pulse coming from the right is the collider pulse, with much lower intensity than the main pulse. It can also propagate with a small angle with respect to the main pulse propagation axis. When the two lasers collide they create a slow beat wave, which allows electrons of the background plasma to be trapped and be accelerated by the wakefield driven by the main pulse.

In this example, the laser pulses are polarized in the y direction and both have a Gaussian profile defined by

$$E_y = E_{\mathrm{pump\_(L,R)}} \exp(-x^2/\mathrm{LPUMP\_(L,R)}^2) \exp(-(y^2+z^2)/\mathrm{W0\_(L,R)}^2)$$

where L and R refer to the left and right pulse respectively. The laser intensity is defined through the normalized vector potential

$$\mathrm{APUMP\_(L,R)} = eE_{\mathrm{pump\_(L,R)}}/\omega_{\mathrm{(L,R)}}m_e c$$

where $\omega = 2\pi c/\mathrm{WAVELENGTH\_(L,R)}$ is the laser frequency.

The pulses enter a plasma channel with density on axis `DENSITY0` through a density ramp of length 20 μ m.

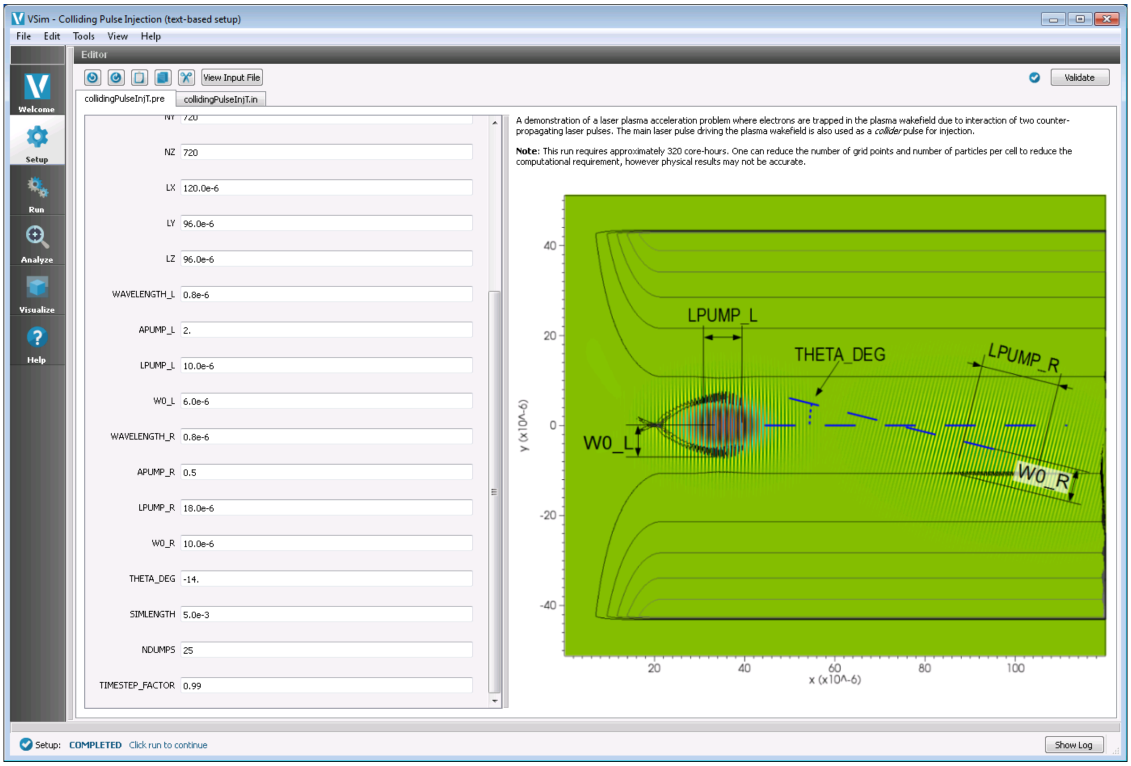
This simulation can be performed with the VSimPA or VSimPD license.

Opening the Simulation

The Colliding Pulse Injection example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Acceleration* option.
- Expand the *Laser Driven Acceleration (text-based setup)* option.
- Select *Colliding Pulse Injection (text-based setup)* and press the *Choose* utton.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem can now be changed via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 5.10.



NY 720

NZ 720

LX 120.0e-6

LY 96.0e-6

LZ 96.0e-6

WAVELENGTH_L 0.8e-6

APUMP_L 2.

LPUMP_L 10.0e-6

W0_L 6.0e-6

WAVELENGTH_R 0.8e-6

APUMP_R 0.5

LPUMP_R 18.0e-6

W0_R 10.0e-6

THETA_DEG -14.

SIMLENGTH 5.0e-3

NDUMPS 25

TIMESTEP_FACTOR 0.99

A demonstration of a laser plasma acceleration problem where electrons are trapped in the plasma wakefield due to interaction of two counter-propagating laser pulses. The main laser pulse driving the plasma wakefield is also used as a *collider* pulse for injection.

Note: This run requires approximately 320 core-hours. One can reduce the number of grid points and number of particles per cell to reduce the computational requirement, however physical results may not be accurate.

Setup: COMPLETED Click run to continue

Fig. 5.10: Setup Window for the Colliding Pulse Injection example.

Input File Features

The simulation setup consists of an electromagnetic solver using the Yee algorithm. Two laser launchers are used, one from the left edge and the other from the right edge of the window. PMLs are used on the transverse sides of the window to absorb outgoing waves. The plasma is represented by macroparticles which are moved using the Boris push. The particles are variably weighted to represent the density ramp, and they have a unique tag. The current deposited by the particles is smoothed using four passes of the 1-2-1 filter and subsequently applying a compensator.

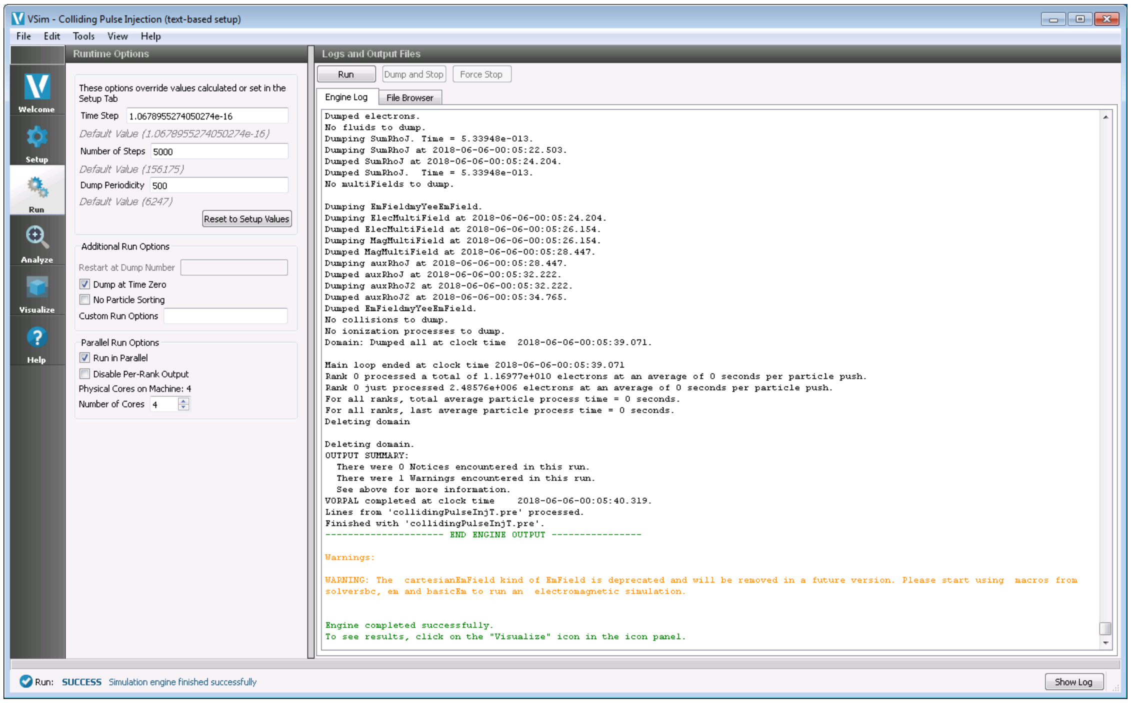
The input file allows one to set up both lasers, plasma and grid parameters.

Running the Simulations

Running in 2D, this simulation uses around 2,600,000 cells and nearly 10^7 particles. This run requires bout 320 core-hores for the full 156,000 steps on a 2.5 GHz I7. On less powerful hardware, one can reduce the number of steps to 15000 see just the collision or one can reduce the number of grid points and number of particles per cell to see more of the evolution, but physical results may not be accurate.

To run on local hardware do

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- Set the number of steps to 5000 and the dump periodicity to 500 in order to see the initial evolution. The collision occurs at step 4500 (dump 9).
- Run in parallel with as many physical cores as are on your machine, because this is a computationally intense problem. Even with the reduced number of steps, this run can take up to 7 hours on four cores for 5000 steps, depending on the processor.
- To run the file, click on the *Run* button in the upper left This is shown in Fig. 5.11. The run has completed when you see the output, “Engine completed successfully.” in this same pane.



These options override values calculated or set in the Setup Tab

Time Step 1.0678955274050274e-16

Default Value (1.0678955274050274e-16)

Number of Steps 5000

Default Value (156175)

Dump Periodicity 500

Default Value (6247)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Dumped electrons.
No fluids to dump.
Dumping SumRhoJ. Time = 5.33948e-013.
Dumping SumRhoJ at 2018-06-06-00:05:22.503.
Dumped SumRhoJ at 2018-06-06-00:05:24.204.
Dumped SumRhoJ.  Time = 5.33948e-013.
No multiFields to dump.

Dumping EmFieldmyYeeEmField.
Dumping ElecMultiField at 2018-06-06-00:05:24.204.
Dumped ElecMultiField at 2018-06-06-00:05:26.154.
Dumping MagMultiField at 2018-06-06-00:05:26.154.
Dumped MagMultiField at 2018-06-06-00:05:28.447.
Dumping auxRhoJ at 2018-06-06-00:05:28.447.
Dumped auxRhoJ at 2018-06-06-00:05:32.222.
Dumping auxRhoJ2 at 2018-06-06-00:05:32.222.
Dumped auxRhoJ2 at 2018-06-06-00:05:34.765.
Dumped EmFieldmyYeeEmField.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-06-00:05:39.071.

Main loop ended at clock time 2018-06-06-00:05:39.071
Rank 0 processed a total of 1.16977e+010 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 2.48576e+006 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 1 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-06-06-00:05:40.319.
Lines from 'collidingPulseInjT.pre' processed.
Finished with 'collidingPulseInjT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Warnings:

WARNING: The  cartesianEmField kind of EmField is deprecated and will be removed in a future version. Please start using  macros from
solversbc, em and basicEm to run an  electromagnetic simulation.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 5.11: The Run Window.

Alternatively, copy collidingPulseInjT.pre to your more powerful hardware and run it through the command line or submit it to your job queue.

Visualizing the Output

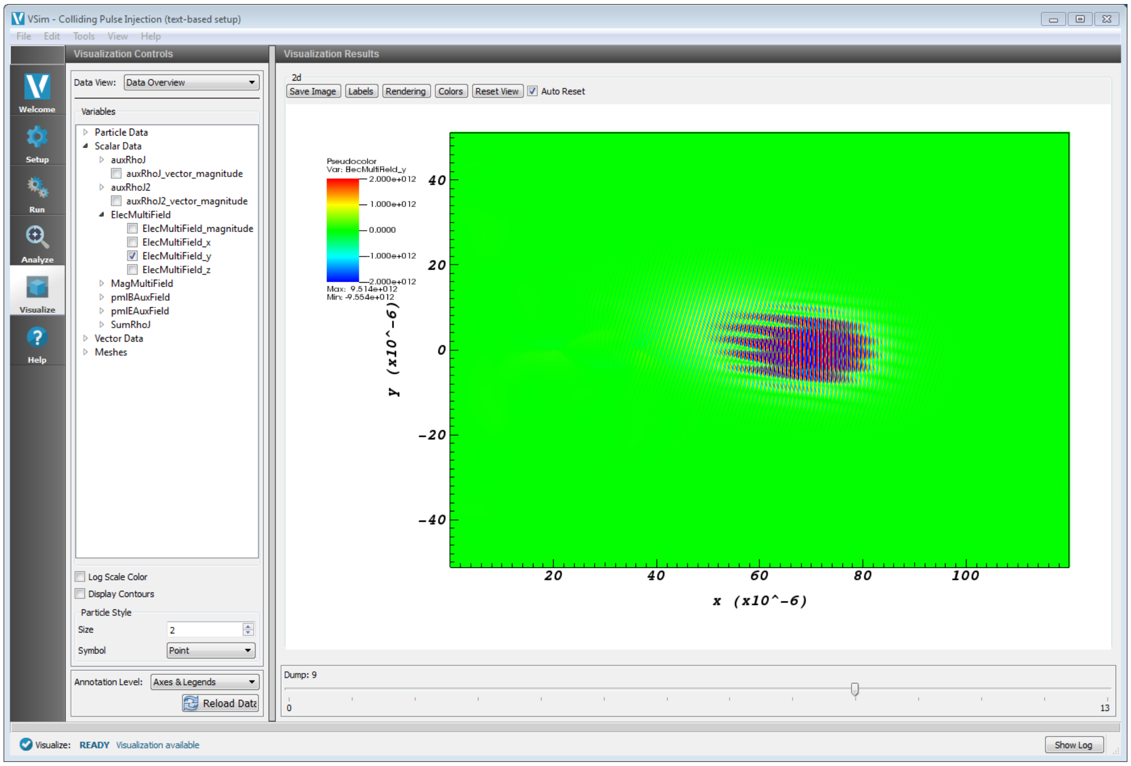
If you have run the job on a remote computer, you would now need to copy back the files that you want to visualize locally into the local directory in which one has the input file open. E.g.,

```
for dmpnum in 0 8 9 10; do
  scp mybigcomputer.mydomain:myspace/collidingPulseInjT_*_${dmpnum}.h5 .
done
```

Then

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the transverse electric field, switch to the *Data Overview* in the Visualization Controls pane. From the Field drop down menu, choose the y component of the `ElecMultiField`. Set the scale by clicking on the *Colors* button, then setting the minimum to -2e12 and the maximum to 2e12. Click the Auto Reset check box. Move the dump slider to position 8, then 9, then 10 to see the pulses collide. The collision of the pulses is then seen as shown in Fig. 5.12 below.



Data View: Data Overview

☐ Log Scale Color
☐ Display Contours

Particle Style
Size 2
Symbol Point

Annotation Level: Axes & Legends

☑ Auto Reset

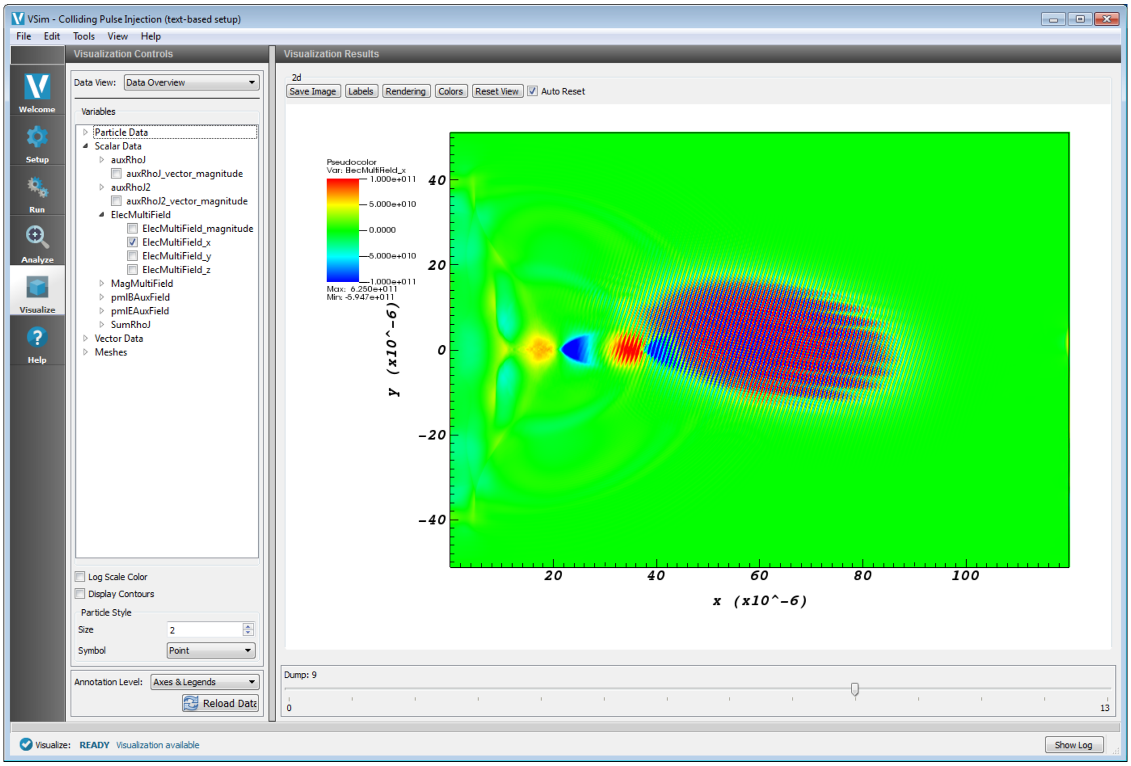
Dump: 9

Visualize: READY Visualization available

Fig. 5.12: Visualization of the transverse electric field as a color contour plot and longitudinal lineout.

The x-component shows the wake field of the left incoming pulse and some of the electromagnetic field of the incoming collider pulse. The wake field can be better seen by clicking on the *Colors* button and setting the min and max to be -+ 1.e11. The plasma density can be seen in the zeroth component of the SumRhoJ field.

Particle phase-space can be seen by switching to the *Phase Space Data View* in the Controls pane. Fig. 5.13 shows the particle longitudinal momentum as a function of the longitudinal coordinate just after the collision.



Data View: Data Overview

☐ auxRhoJ_vector_magnitude
☐ auxRhoJ2_vector_magnitude
☐ ElecMultiField_magnitude
☑ ElecMultiField_x
☐ ElecMultiField_y
☐ ElecMultiField_z

☐ Log Scale Color
☐ Display Contours

Size: 2
Symbol: Point

Annotation Level: Axes & Legends

☑ Auto Reset

Dump: 9

Visualize: READY Visualization available

Fig. 5.13: Visualization of the particle longitudinal phase-space during collision. Particles kicked up into the trapped region by the colliding pulses. One can see the acceleration of particle to high energy in later dumps.

Continuing the simulation

The simulation to this point has allowed one to study the initial injection of particles up to high energy, so that they can be trapped by the wake. One can now continue this simulation to study the acceleration in the wake. Since this simulation stopped at dump 10, one can now set the Addition Run Options to unclick *Dump at Time Zero* and then set *Restart at Dump Number* to 10. Since at this point, the evolution changes more slowly, one can set the *Number of Steps* to 10000 and the *Dump Periodicity* to 5000. Again hit *Run*.

At any time one sees that another data dump has occurred, one can switch over to the Visualize pane and hit *Reload Data* to view the new available data, any of the fields or particles as before.

5.3.2 Ionization Injection (fieldIonizeT.pre)

Keywords:

`laser plasma accelerator, controlled injection, ionization of high-Z gas`

Problem description

This example demonstrates the use of VSim to simulate ionization-induced injection in a laser plasma accelerator *[CES+12]*. An intense laser pulse propagates up a plasma density ramp into a uniform plasma, which creates a wakefield. Neutral nitrogen atoms are added to the pre-ionized gas at the beginning of the plasma, where the laser pulse field ionizes them. If the electrons released from the nitrogen ionization are at the correct position relative to the wakefield phase, they can be trapped and accelerated to high energy *[CCMG+13]*.

The laser envelope has a Gaussian profile defined at the waist position by (X_0_LASER):

$$E_z = E_0 \exp\left(-x^2/LPUMP^2\right) \exp\left(-(y^2+z^2)/W\_0^2\right) \sin\left(\omega_0 t\right)$$

where $\omega_0 = 2\pi c$ / WAVELENGTH is the laser frequency. The laser amplitude is defined through the normalized vector potential A_0 = $eE_0/\omega_0 m_e c$.

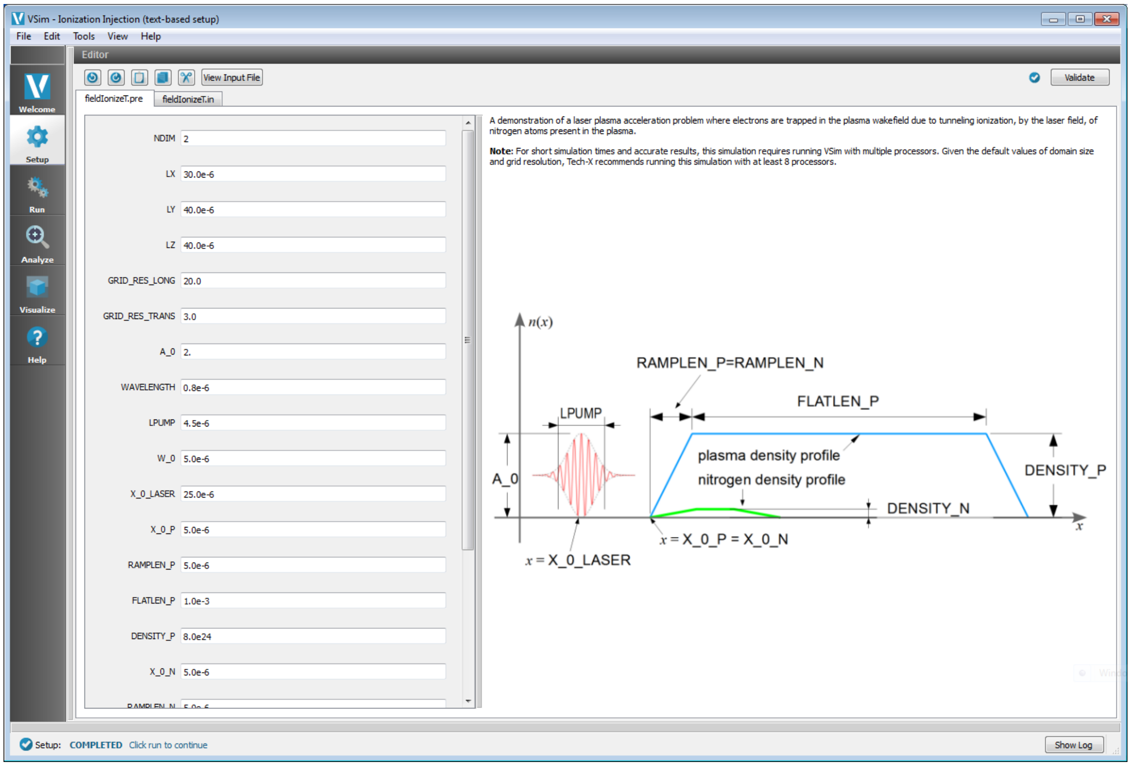
This simulation can be performed with a VSimPA license.

Opening the Simulation

The Ionization Injection example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Acceleration* option.
- Expand the *Laser Driven Acceleration (text-based setup)* option.
- Select "Ionization injection (text-based setup)" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 5.14.



NDIM 2

LX 30.0e-6

LY 40.0e-6

LZ 40.0e-6

GRID_RES_LONG 20.0

GRID_RES_TRANS 3.0

A_0 2.

WAVELENGTH 0.8e-6

LPUMP 4.5e-6

W_0 5.0e-6

X_0_LASER 25.0e-6

X_0_P 5.0e-6

RAMPLEN_P 5.0e-6

FLATLEN_P 1.0e-3

DENSITY_P 8.0e24

X_0_N 5.0e-6

RAMPLEN_N 5.0e-6

A demonstration of a laser plasma acceleration problem where electrons are trapped in the plasma wakefield due to tunneling ionization, by the laser field, of nitrogen atoms present in the plasma.

Note: For short simulation times and accurate results, this simulation requires running VSim with multiple processors. Given the default values of domain size and grid resolution, Tech-X recommends running this simulation with at least 8 processors.

Setup: COMPLETED Click run to continue

Fig. 5.14: Setup Window for the Ionization injection in Laser Plasma Accelerator example.

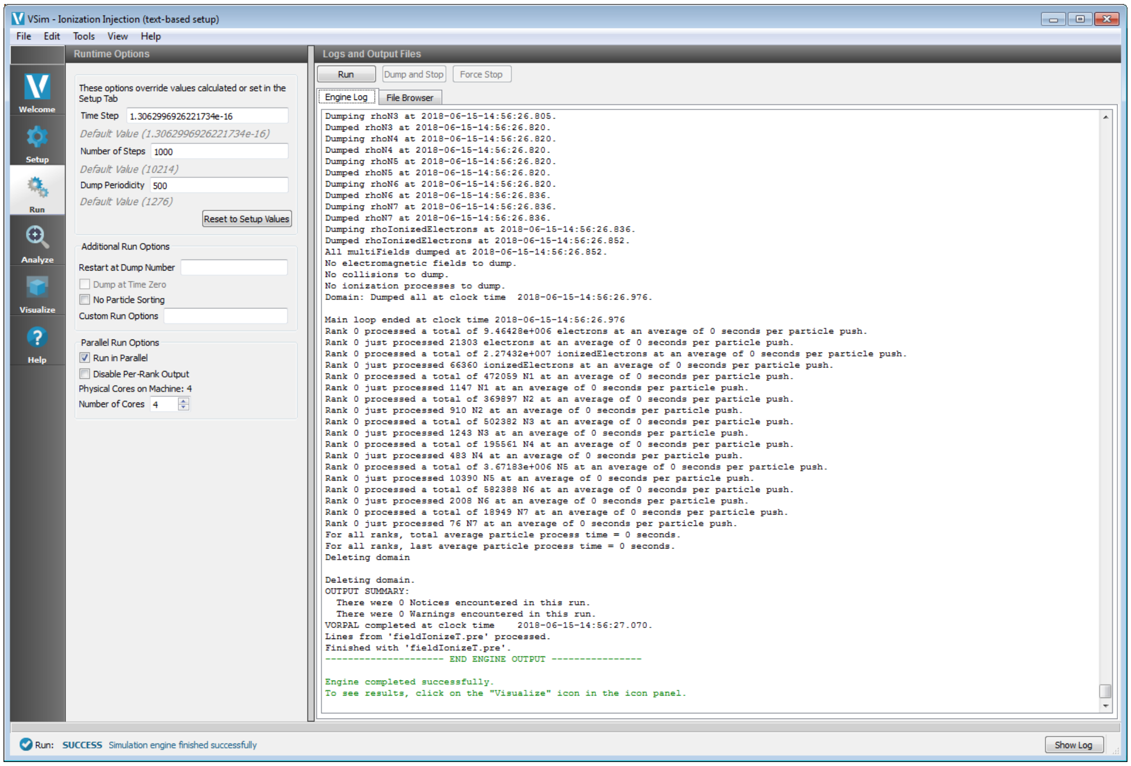
Input File Features

The simulation setup consists of an electromagnetic solver using the Yee algorithm. The laser pulse is launched from the left side of the window using an expression launcher at the boundary. MALs are used on the transverse sides of the window to absorb outgoing waves. The plasma is represented by macro-particles which are moved using the Boris push. The particles are variably weighted to represent the density ramp. The nitrogen atoms are represented using a fluid neutral gas. The different excited levels of the nitrogen and electrons product of the ionization are represented through variably weighted macro-particles. The ionization process takes place in MonteCarlo interactions, using the modified time-resolved ADK formula *[CES+12]*.

Running the Simulations

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- This run is computationally intensive, so you click *Run in Parallel* and select a number of cores equal to the number of physical cores on your machine.
- To see the initial evolution, set the *Number of Steps* to 1000 and the *Dump Periodicity* to 500.
- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane on the right. of the window. You will see the output of the run in that same pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 5.15.



These options override values calculated or set in the Setup Tab

Time Step 1.3062996926221734e-16

Default Value (1.3062996926221734e-16)

Number of Steps 1000

Default Value (10214)

Dump Periodicity 500

Default Value (1276)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Dumping rhoN3 at 2018-06-15-14:56:26.805.
Dumped rhoN3 at 2018-06-15-14:56:26.820.
Dumping rhoN4 at 2018-06-15-14:56:26.820.
Dumped rhoN4 at 2018-06-15-14:56:26.820.
Dumping rhoN5 at 2018-06-15-14:56:26.820.
Dumped rhoN5 at 2018-06-15-14:56:26.820.
Dumping rhoN6 at 2018-06-15-14:56:26.820.
Dumped rhoN6 at 2018-06-15-14:56:26.836.
Dumping rhoN7 at 2018-06-15-14:56:26.836.
Dumped rhoN7 at 2018-06-15-14:56:26.836.
Dumping rhoIonizedElectrons at 2018-06-15-14:56:26.836.
Dumped rhoIonizedElectrons at 2018-06-15-14:56:26.852.
All multiFields dumped at 2018-06-15-14:56:26.852.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-15-14:56:26.976.

Main loop ended at clock time 2018-06-15-14:56:26.976
Rank 0 processed a total of 9.46428e+006 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 21303 electrons at an average of 0 seconds per particle push.
Rank 0 processed a total of 2.27432e+007 ionizedElectrons at an average of 0 seconds per particle push.
Rank 0 just processed 66360 ionizedElectrons at an average of 0 seconds per particle push.
Rank 0 processed a total of 472059 N1 at an average of 0 seconds per particle push.
Rank 0 just processed 1147 N1 at an average of 0 seconds per particle push.
Rank 0 processed a total of 369897 N2 at an average of 0 seconds per particle push.
Rank 0 just processed 910 N2 at an average of 0 seconds per particle push.
Rank 0 processed a total of 502382 N3 at an average of 0 seconds per particle push.
Rank 0 just processed 1243 N3 at an average of 0 seconds per particle push.
Rank 0 processed a total of 195561 N4 at an average of 0 seconds per particle push.
Rank 0 just processed 483 N4 at an average of 0 seconds per particle push.
Rank 0 processed a total of 3.67183e+006 N5 at an average of 0 seconds per particle push.
Rank 0 just processed 10390 N5 at an average of 0 seconds per particle push.
Rank 0 processed a total of 582388 N6 at an average of 0 seconds per particle push.
Rank 0 just processed 2008 N6 at an average of 0 seconds per particle push.
Rank 0 processed a total of 18949 N7 at an average of 0 seconds per particle push.
Rank 0 just processed 76 N7 at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-15-14:56:27.070.
Lines from 'fieldIonizeT.pre' processed.
Finished with 'fieldIonizeT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 5.15: The Run Window at the end of the first execution.

At this point, one can skip ahead to the visualization section to see whether the fields look reasonable. If they do, you can restart:

- Set the *Number of Steps* to 9000 and *Restart at Dump Number* to 2.

- Click on the *Run* button. The run has completed when you see the output, “Engine completed successfully.”

This run takes about 70 minutes on a 4 core, 2.5 GHz Intel I7. To run on less powerful hardware one can reduce the number of grid points and number of particles per cell, however physical results may not be as accurate.

Visualizing the Results

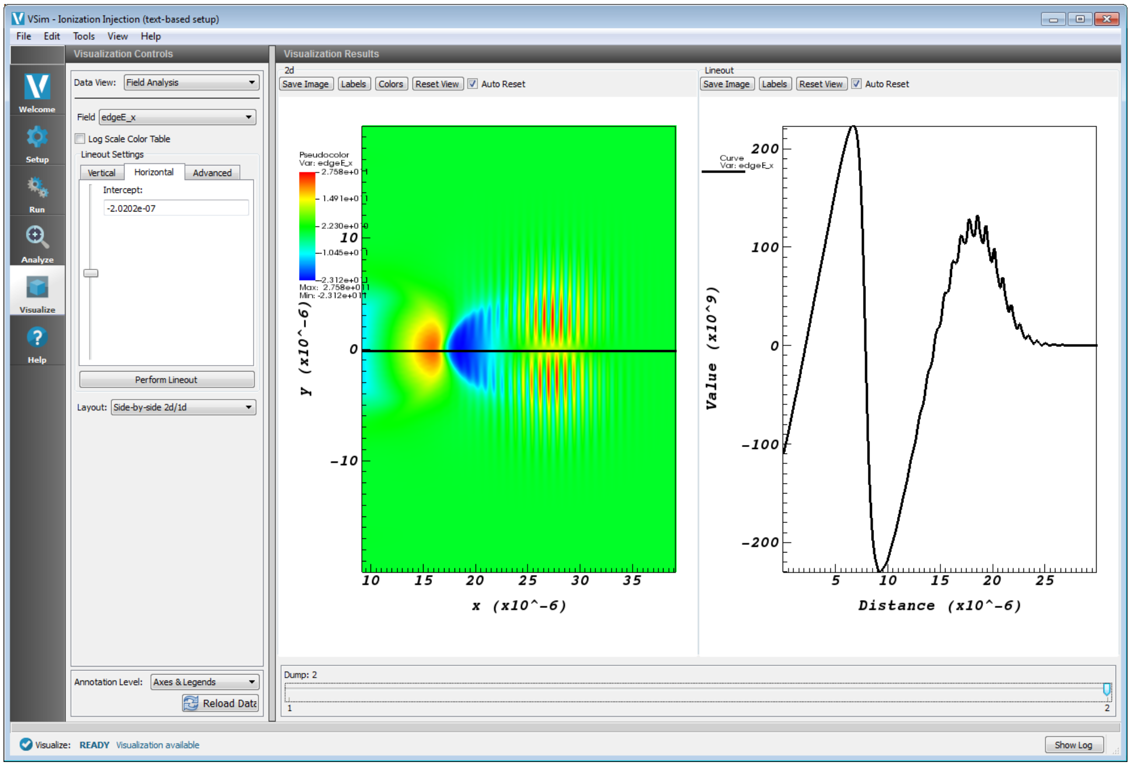
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The laser pulse is the z component of the field, while the accelerating field is the x component. The plasma density can be seen in `rho`.

Fig. 5.16 shows the longitudinal laser field along the beam axis. To reproduce:

- Set *Data View* to *Field Analysis*
- Click *Auto Reset* on both the pseudocolor and lineout plots so that the window updates the plot region as one moves the slider. You may need to expand your visualization window for the *Auto Reset* checkbox to appear.
- Select *edgeE_x* from the Field drop down menu
- Select the *Horizontal* tab in the lineout settings
- Set the intercept to `0`
- Click “Perform Lineout
- Move the dump slider forward in time



VSim - Ionization Injection (text-based setup)

Data View: Field Analysis

Field edgeE_x

☐ Log Scale Color Table

Intercept: -2.0202e-07

Layout: Side-by-side 2d/1d

Annotation Level: Axes & Legends

☑ Auto Reset

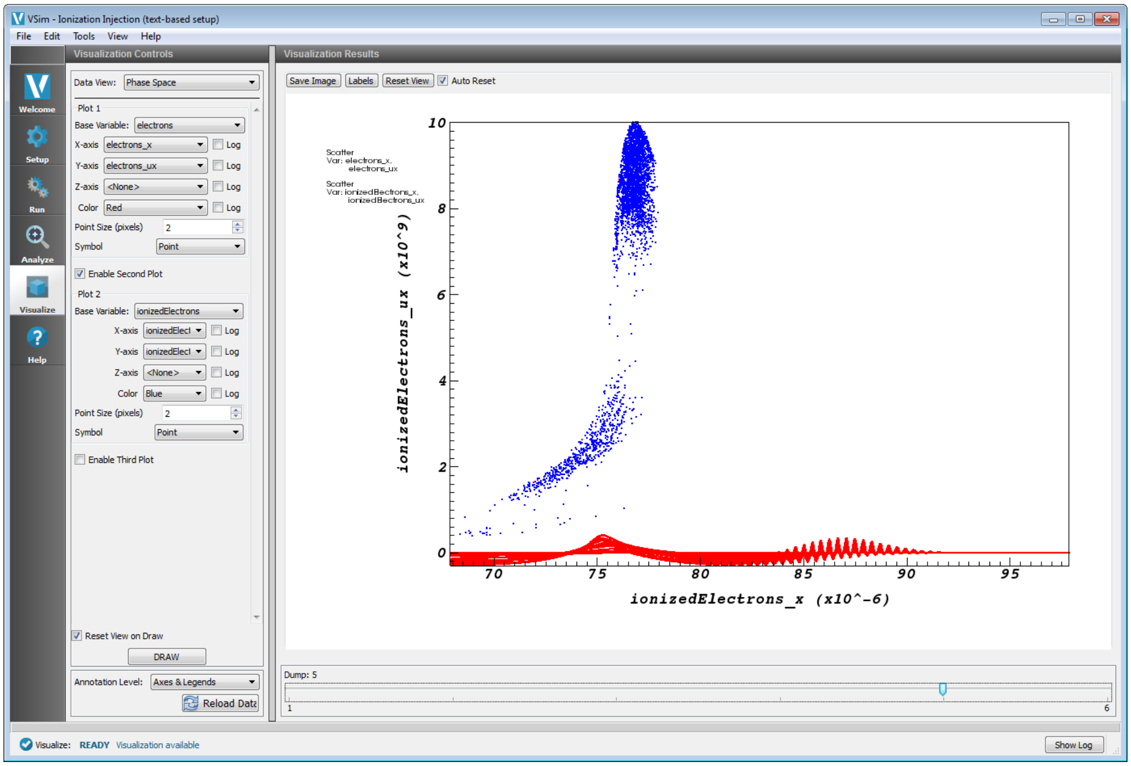
Dump: 2

Visualize: READY Visualization available

Fig. 5.16: Left: Longitudinal electric field $E_x(x, y)$ at t=1.3 picoseconds. Right: Line-out of field plot at $y = 0$.

The acceleration of the particles can be seen by viewing the (x) component of the velocity as shown in Fig. 5.17

- Set *Data View* to *Phase Space*
- Set *Base Variable* to *electrons*
- Set the X-axis variable to *electrons_x*, the Y-axis variable to *electrons_ux*
- Check *Enable Second Plot*
- Set *Base Varible* to *ionzedElectrons*
- Set the X-axis variable to *ionizedElectrons_x*, the Y-axis variable to *ionizedElectrons_ux*
- Click *Draw*
- Click *Reset View*



VSim - Ionization Injection (text-based setup)

Data View: Phase Space

Plot 1
Base Variable: electrons
X-axis electrons_x □ Log
Y-axis electrons_ux □ Log
Z-axis <None> □ Log
Color Red □ Log
Point Size (pixels) 2
Symbol Point

☑ Enable Second Plot

Plot 2
Base Variable: ionizedElectrons
X-axis ionizedElect □ Log
Y-axis ionizedElect □ Log
Z-axis <None> □ Log
Color Blue □ Log
Point Size (pixels) 2
Symbol Point

□ Enable Third Plot

☑ Reset View on Draw

Annotation Level: Axes & Legends

☑ Auto Reset

Dump: 5

Visualize: READY Visualization available

Fig. 5.17: Phase-space plot (x, $\gamma\ v_{\mathrm{x}}$) of plasma electrons at t=0.33 picoseconds.

CHAPTER

SIX

VSIM FOR PLASMA DISCHARGES EXAMPLES

These examples illustrate how to solve complex problems in plasma discharge modelling.

These examples can be run with a VSimPD license.

6.1 Capacitively Coupled

6.1.1 1D Capacitive Plasma Chamber (capacitivelyCoupledPlasma1D.sdf)

Keywords:

`CCP discharge, secondary emission, elastic collision, excitation, ionization.`

Problem description

The capacitively coupled plasma (CCP) is one of the most common types of industrial plasma sources. The discharges usually take place between metal electrodes in a reaction chamber and are driven by a radio-frequency (RF) or DC power supply. The plasma is sustained by ohmic heating in the main body and stochastic heating through a capacitive sheath.

This example demonstrates the generation of a capacitively coupled plasma inside two parallel conducting plates separated by 0.05 m. A background Ar neutral gas at approximately 6 mTorr a number density of approximately $2.0 \times 10^{20}\ m^{-3}$) is filled between the electrodes. The right electrode is grounded, while the left one is connected to a voltage source of 200 V at 60 MHz.

This simulation can be performed with a VSimPD license.

Opening the Simulation

The 1D Capacitively Coupled Plasma Discharge example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Capacitively Coupled Plasmas* option.
- Select “1D Capacitive Plasma Chamber” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

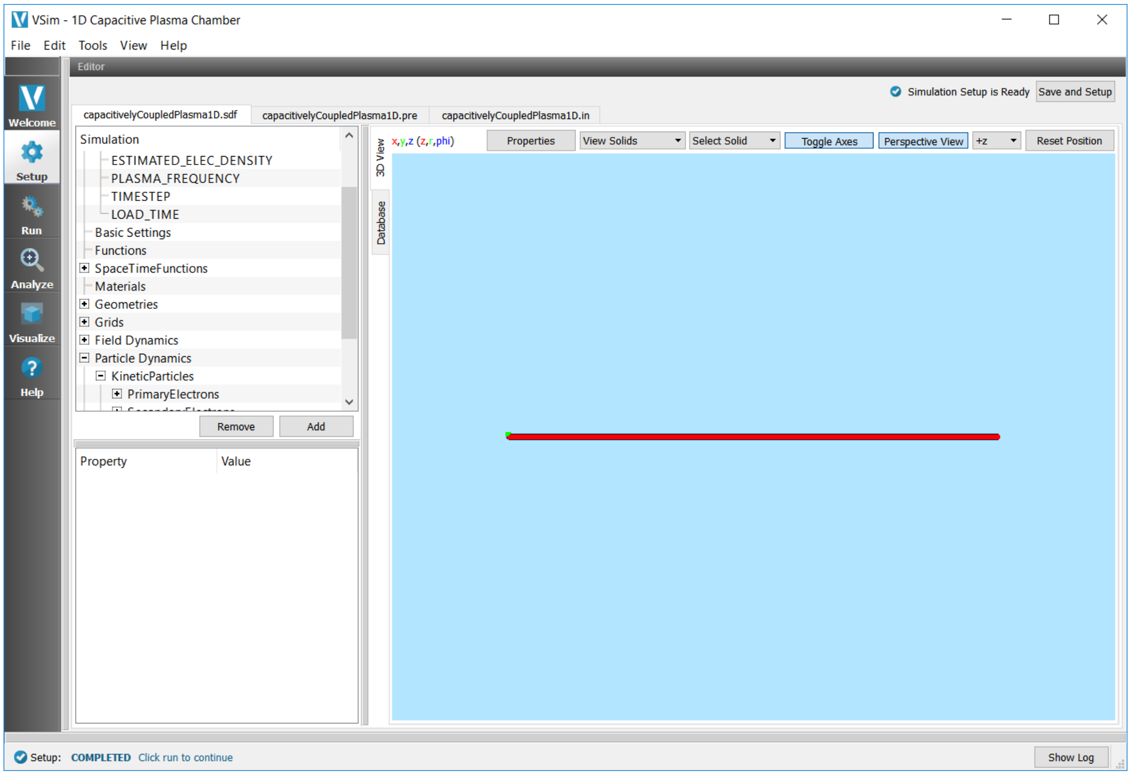


Fig. 6.1: Setup Window for the 1D Capacitively Coupled Plasma Discharge example.

The *Setup Window* and elements tree with all the implemented physics and geometries, is shown below in Fig. 6.1.

The time step DT should sufficiently resolve the plasma frequency and collision frequency. The default time step used in this example is TIMESTEP_FACTOR * (0.1 / Plasma frequency) to ensure stability. The initial primary electrons are gradually loaded into the simulation domain over a period of LOADSTEPS timesteps, which has a default value of 5000.

Simulation Properties

This simulation includes some constants and parameters for easy adjustment of the simulation properties. These include:

Constants

- NEUTRAL_ARGON_DENSITY: number density of the background neutral argon gas (number/m^3).
- FREQUENCY: sets the frequency of the driving voltage set on the lower X boundary.
- VOLTAGE: sets the amplitude of the driving voltage set on the lower X boundary.
- NOMINAL_DENSITY: this adjusts the number of physical particles loaded into the simulation.
- LOADSTEPS: Timestep when particle loading will end.
- NSTEPS: How many timesteps to simulate.
- STEPS_PER_DUMP: number of steps to take between data dumps.
- BMAG: sets the strength of the magnetic field (default = 0T).

Time-dependent Dirichlet boundary conditions are used to set up the boundaries of electric fields around the reaction chamber walls, and are set in *Field Dynamics -> FieldBoundaryConditions*. The self-consistent electric field is solved from Poisson's equation by the Generalized Minimum Residual (gmres) electrostatic solver in Cartesian coordinates. This solver is chosen under *Field Dynamics -> PoissonSolver.*

The plasma is represented by macroparticles which are moved using the Boris pusher in Cartesian coordinates and interact with the background neutral argon gas through collisions set up with the Reactions framework. The particles, background gas, and collisions are set up in the *Particle Dynamics* Element.

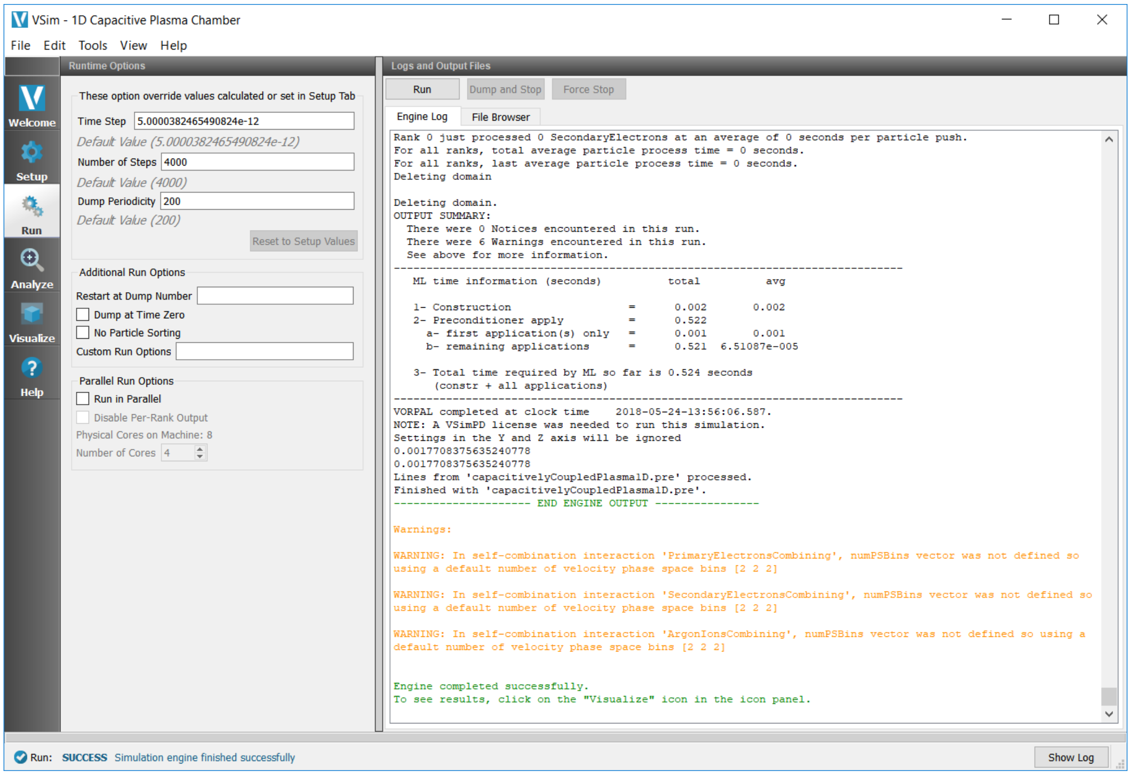
The simulation includes two electron species: Primary electrons which are electrons loaded into the simulation, and Secondary electrons which are created through physical processes. Both species are managed weight particle species, which will combine or split macro particles based on user choices.

Elastic collisions between electrons and the background gas, excitation collisions in which an electron will lose energy to the background gas, and ionization collision in which electrons create argon ions from the background gas are all included. The cross-sections for this collisions are imported from 2-column data files.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in this pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 6.1.



These option override values calculated or set in Setup Tab

Time Step 5.0000382465490824e-12

Default Value (5.0000382465490824e-12)

Number of Steps 4000

Default Value (4000)

Dump Periodicity 200

Default Value (200)

Additional Run Options

Restart at Dump Number

☐ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☐ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 8

Number of Cores 4

```
Rank 0 just processed 0 SecondaryElectrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 6 Warnings encountered in this run.
  See above for more information.
-------------------------------------------------------------------------------
   ML time information (seconds)          total         avg

   1- Construction                  =      0.002       0.002
   2- Preconditioner apply          =      0.522
     a- first application(s) only   =      0.001       0.001
     b- remaining applications      =      0.521   6.51087e-005

   3- Total time required by ML so far is 0.524 seconds
      (constr + all applications)
-------------------------------------------------------------------------------
VORPAL completed at clock time    2018-05-24-13:56:06.587.
NOTE: A VSimPD license was needed to run this simulation.
Settings in the Y and Z axis will be ignored
0.0017708375635240778
0.0017708375635240778
Lines from 'capacitivelyCoupledPlasma1D.pre' processed.
Finished with 'capacitivelyCoupledPlasma1D.pre'.
--------------------- END ENGINE OUTPUT ----------------

Warnings:

WARNING: In self-combination interaction 'PrimaryElectronsCombining', numPSBins vector was not defined so
using a default number of velocity phase space bins [2 2 2]

WARNING: In self-combination interaction 'SecondaryElectronsCombining', numPSBins vector was not defined so
using a default number of velocity phase space bins [2 2 2]

WARNING: In self-combination interaction 'ArgonIonsCombining', numPSBins vector was not defined so using a
default number of velocity phase space bins [2 2 2]


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 6.2: The Run Window at the end of execution.

Visualizing the results

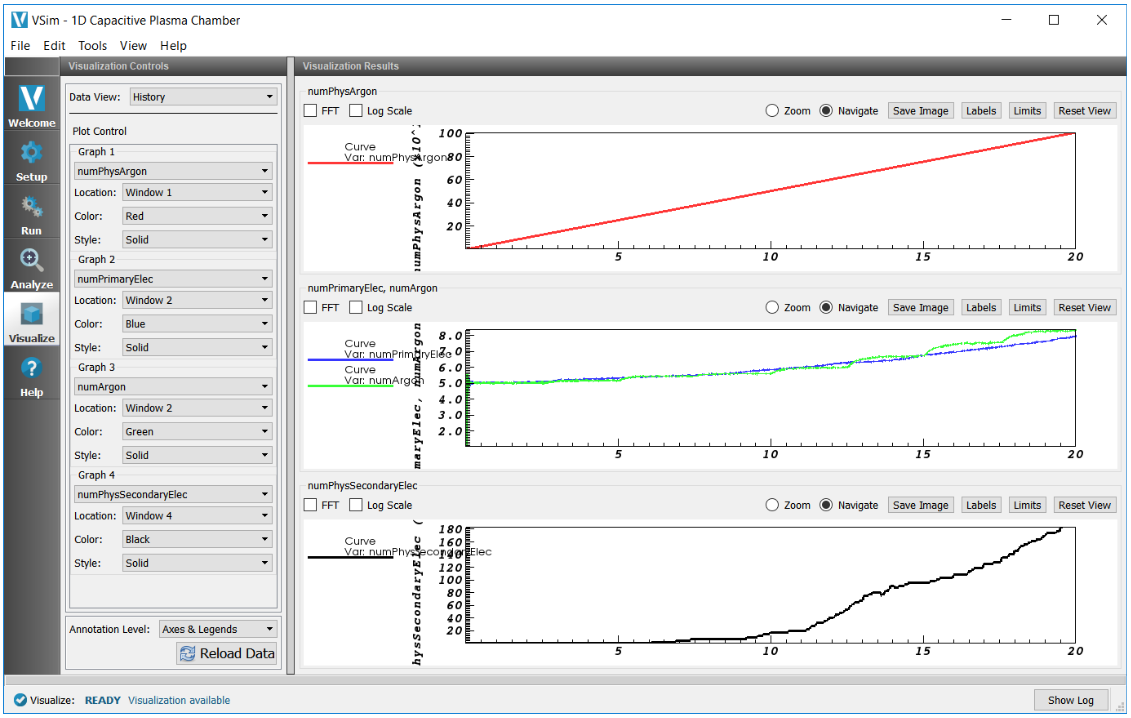
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

From the "Data View" option, select "History". There are 6 histories that can be plotted in this window: the number of physical particles and the number of macro particles for each of the three particle species (argon ions, primary electrons, and secondary electrons). To produce the plot in Fig. 6.14

- Plot the 'numArgon' history in Graph 1.
- Plot the 'numPrimaryElec' history in Graph 2.
- Plot the 'numArgon' history in Graph 3 and change the *Location* to Window 2
- Plot the 'numPhysSecondaryElec" history in Graph 4.

The simulation converges as the number of secondary electrons approaches a constant, indicating a steady state plasma. With the default number of time steps (4000, or 2 nanoseconds), the simulation does not reach steady state (see the black, numPhysSecondaryElec history curve). To reach steady state, the simulation must run for approximately 100 microseconds.



VSim - 1D Capacitive Plasma Chamber

Data View: History

Plot Control

Graph 1
numPhysArgon
Location: Window 1
Color: Red
Style: Solid

Graph 2
numPrimaryElec
Location: Window 2
Color: Blue
Style: Solid

Graph 3
numArgon
Location: Window 2
Color: Green
Style: Solid

Graph 4
numPhysSecondaryElec
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 6.3: Visualization of number histories of ion, primary electron, and secondary electrons for at 4000 steps or 20 nanoseconds.

Further Experiments

Set up a History that records the electron current flowing into the left and right sides of the simulation. Right click on the "Histories" element and under "Add ParticleHistory" select "Absorbed Particle Current." You can change the name of the history by double clicking on the new "absorbedPtclCurrent0" element in the tree. Then be sure to pick the particle absorber from which you would like to collect data.

The Reactions framework allows one to set up collision interactions flexibly. The collisions involved in this example are electron-neutral collisions that lead to ionization and ohmic heating. As a further experiment, ion-neutral collisions, such as elastic scattering and charge exchange, can also be added to the simulation.

The VSim interface can import any cross sections that are in a 2-column format. There should be NO headings in the data file. The LXcat scattering database (https://fr.lxcat.net/data/set_type.php) and EEDL cross section database contain cross section data for around one hundred different materials. As another experiment, change the cross-section used in the simulation or change the species of the background gas and import new cross-sections.

6.1.2 Turner Case 2 (Turner.sdf)

Keywords:

capacitively coupled plasma, CCP, discharge, steady state, Turner

Problem Description

In this example we demonstrate VSim's ability to simulate capacitively coupled plasmas, using the benchmark cases of Turner et al. *[TDD+13]*. Turner's work documents the successful benchmarking of five independently developed particle-in-cell codes (not including VSim) for four different capacitive coupling scenarios at various background pressures.

Here, we consider the second of the Turner scenarios, though the input file can be readily modified to simulate the others. In addition to being able to accurately reproduce the Turner results, VSim can also employ physics-based initialization methods to enable more rapid convergence of the simulations to their steady-state. The use of such methods will also be explained below.

This simulation can be performed with a VSimPD license.

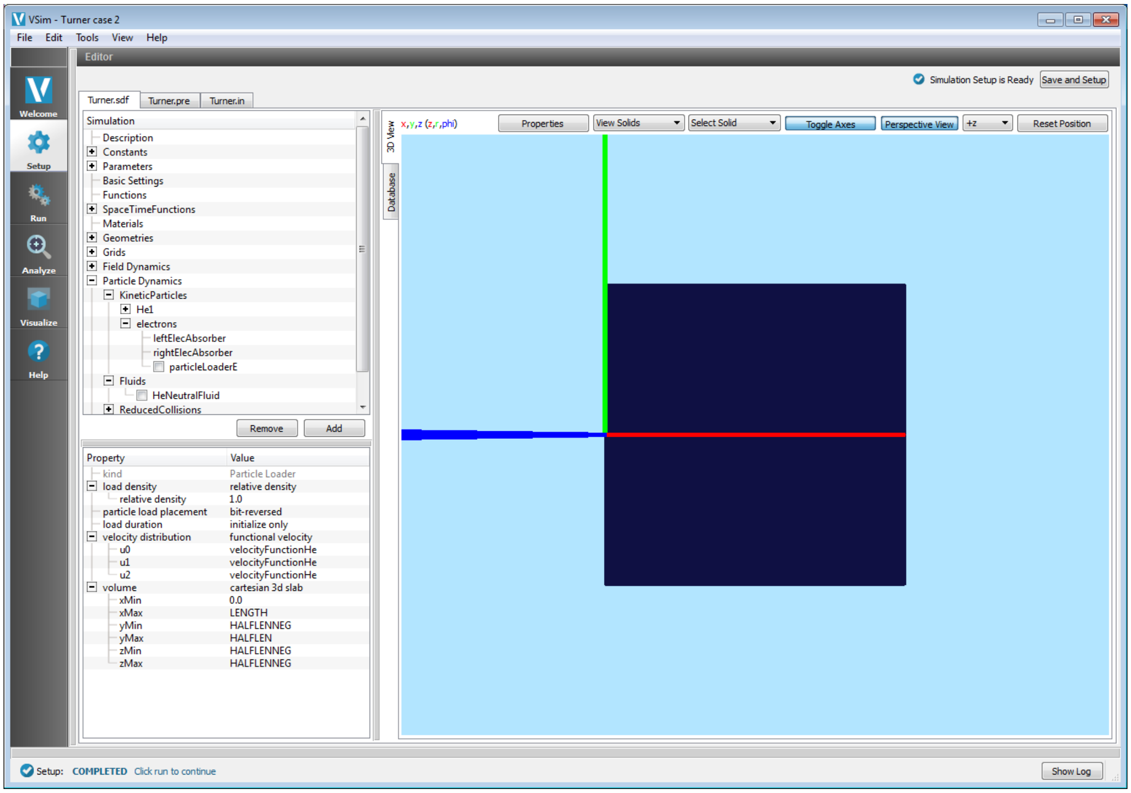
Opening the Simulation

The Turner example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item from the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Introductory Examples* option.
- Select *Turner* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 6.4. In this image, we have unclicked the electrons' particleLoaderE and the He1NeutralFluid so that they will not hide the basic grid You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`. This is a one-dimensional problem, which is shown by having the grid have only a single cell above and below the x-axis.

Clicking the electrons' particleLoaderE shows that the electron loader is defined to exist over a cartesian 3d slab, even though this is a one-dimensional simulation. The dimensions that do not apply are ignored, with the coordinate set to zero, but this allows easy conversion from a 1D simulation to a 2D simulation.



Property	Value
kind	Particle Loader
load density	relative density
relative density	1.0
particle load placement	bit-reversed
load duration	initialize only
velocity distribution	functional velocity
u0	velocityFunctionHe
u1	velocityFunctionHe
u2	velocityFunctionHe
volume	cartesian 3d slab
xMin	0.0
xMax	LENGTH
yMin	HALFLENNEG
yMax	HALFLEN
zMin	HALFLENNEG
zMax	HALFLENNEG

Setup: COMPLETED Click run to continue

Fig. 6.4: Setup Window for the Turner example.

Simulation Properties

The basic physics of this simulation is a balance between collisional processes and wall losses; a one-dimensional box of length 6.7 cm contains neutral helium gas at room temperature (300 K) and density $3.21e211/m^3$ (1 Torr of pressure at that temperature). The gas is weakly ionized, resulting in a population of free electrons and singly ionized helium atoms at density $5.12e141/m^3$. The helium ions are also at room temperature, while the electrons are considerably hotter (30,000 K). The left wall of the box is grounded, while the right wall oscillates with a bias voltage of 200 V at frequency 13.56 MHz.

Charged particles are lost upon collision with the wall and are replenished by ionization of the background neutral gas by the hot electrons; the latter process repopulates both the electrons and helium ions in the plasma (the background neutral gas is treated as an infinite source). Plasma sheaths form near the walls, containing electric fields which are strong relative to those elsewhere in the plasma; the particle density profiles adjust in response to the fields in the sheath. The sheath transit time, for ions, is much longer than the period of the oscillating potential; thus, multiple RF cycles occur while an ion crosses the sheath. A steady state is attained when the loss rate of particles to the wall comes into balance with the ionization rate for a particular profile shape.

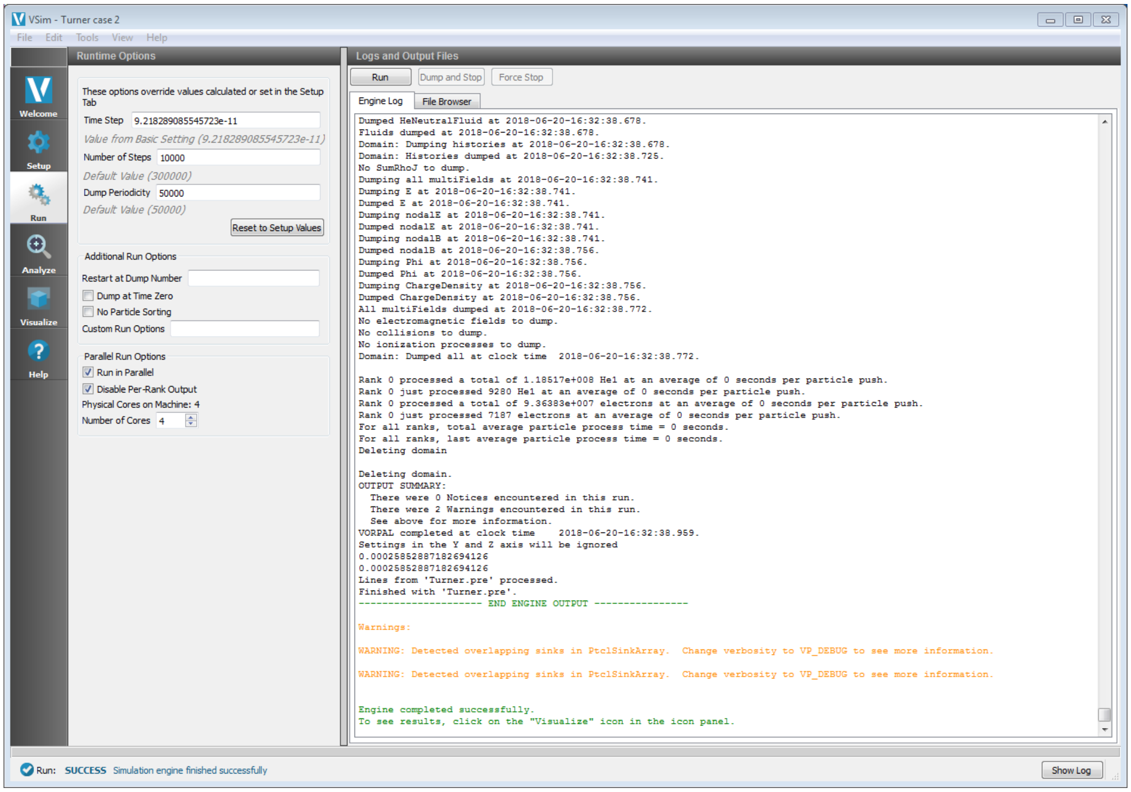
In our initial run we are not going to model the full evolution of the discharge to its steady-state parameters; rather, we will explore the basic physics of the discharge and modify the simulation accordingly (with the aim of ultimately hastening convergence to this steady state, while exploring VSim capabilities).

Running the Simulation

The original runs by Turner were for about 4,000,000 steps. However, the asymptotic state is reached after about 300,000 steps. To illustrate how to run this problem, we will run for only 50,000 steps, which takes about 5 minutes on a 4-core i7 Windows workstation.

To run the simulation, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- Change Number of Time Steps to 50000 (approximately 1/80 of the length of Turner's run)
- Consider clicking 'Disable Communication Text Files' (yes, if desired - reduces clutter) in and 'Run with MPI' (yes, if desired - set 'Number of Cores' corresponding to your VSim license) in the *MPI* tab at the left side of the run panel.
- To run the file, click on the *Run* button in the upper left corner of the window. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." A snapshot of the simulation run during execution is shown in Fig. 6.5.



These options override values calculated or set in the Setup Tab

Time Step: 9.218289085545723e-11

Value from Basic Setting (9.218289085545723e-11)

Number of Steps: 10000

Default Value (300000)

Dump Periodicity: 50000

Default Value (50000)

Additional Run Options

Restart at Dump Number:

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options:

Parallel Run Options

☑ Run in Parallel

☑ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores: 4

```
Dumped HeNeutralFluid at 2018-06-20-16:32:38.678.
Fluids dumped at 2018-06-20-16:32:38.678.
Domain: Dumping histories at 2018-06-20-16:32:38.678.
Domain: Histories dumped at 2018-06-20-16:32:38.725.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-20-16:32:38.741.
Dumping E at 2018-06-20-16:32:38.741.
Dumped E at 2018-06-20-16:32:38.741.
Dumping nodalE at 2018-06-20-16:32:38.741.
Dumped nodalE at 2018-06-20-16:32:38.741.
Dumping nodalB at 2018-06-20-16:32:38.741.
Dumped nodalB at 2018-06-20-16:32:38.756.
Dumping Phi at 2018-06-20-16:32:38.756.
Dumped Phi at 2018-06-20-16:32:38.756.
Dumping ChargeDensity at 2018-06-20-16:32:38.756.
Dumped ChargeDensity at 2018-06-20-16:32:38.756.
All multiFields dumped at 2018-06-20-16:32:38.772.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-20-16:32:38.772.

Rank 0 processed a total of 1.18517e+008 He1 at an average of 0 seconds per particle push.
Rank 0 just processed 9280 He1 at an average of 0 seconds per particle push.
Rank 0 processed a total of 9.36383e+007 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 7187 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 2 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-06-20-16:32:38.959.
Settings in the Y and Z axis will be ignored
0.00025852887182694126
0.00025852887182694126
Lines from 'Turner.pre' processed.
Finished with 'Turner.pre'.
--------------------- END ENGINE OUTPUT ----------------

Warnings:

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 6.5: The Run Window during execution.

Analyzing the Results

We are going to run a postprocessing script, computePtclNumDensity.py, which builds density profiles from the particle data generated by VSim, so that we can look at these profiles and their evolution. To do so, we do the following:

- Click the *Analyze Window*
- Click *Show All Analyzers* and choose *computePtclNumDensity.py*. Then click *Open*.
- Fill in the text boxes
  - The simulationName should be already filled in, but if it is not, type in the name of the .sdf file without the .sdf extension.
  - For the speciesName, type in 'electrons' without the quotes.

- Click *Analyze* (in the Analyze Window); this will generate the electron density profiles.
- Now replace 'electrons' in the speciesName box with 'He1', for the helium ions.
- Click *Analyze* (again in the Analyze Window) to generate the ion density profiles.

The resulting data will be visualizable as electronsDensity and He1Density under the 1-D Fields *Data View* in the *Visualize* Tab.

Visualizing the Results

Now that we've got all of our data, let's look at it.

- Click the *Visualize Window*

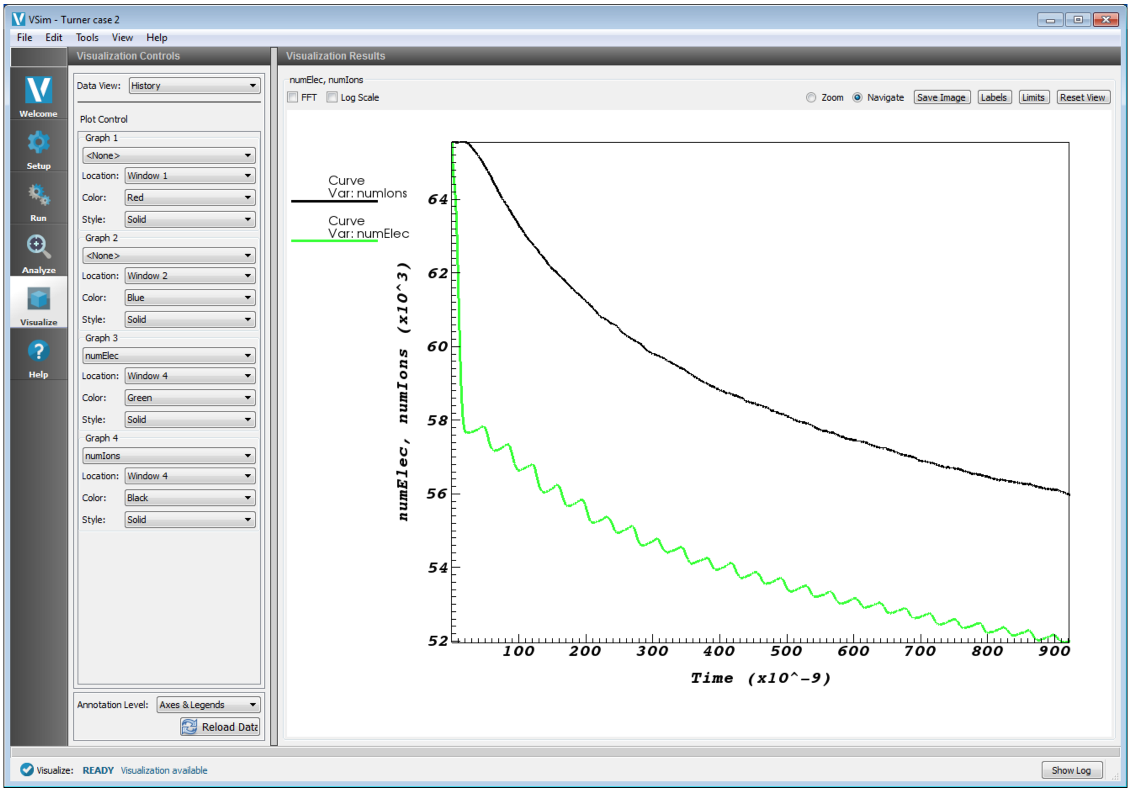
After a brief moment the visualization options for this data should appear.

We'll first look at the time evolution of some fundamental one-dimensional quantities. From the *Data View* pulldown menu on the top left, select *History*. The default view here should contain four plots, namely, the electron and ion currents to the left wall and the number of electron and ion macroparticles in the simulation. A number of notable physics effects can be seen here:

- **After a sharp initial decrease in the electron population, both ion and electron populations decline at approximately the same rate.** This is not as apparent from the separate numElec and numIons plots, but clicking on the "Location" drop-down window in Graph 3 and selecting "Window 4" as the new rendering destination, places both ion and electron populations in the same plot. (Select "<None>" in the plot variable (the topmost menu) for both Graph 1 and Graph 2 to resize the electron/ion plot.) The initial decrease in electron population arises when rapid electron wall losses create a charge imbalance in the plasma and establish plasma sheaths near the walls. Thereafter, this charge imbalance is preserved and the transport of both electrons and ions to the wall becomes ambipolar. A history of the particle popuations can be seen in Fig. 6.6
- **The electron wall currents are quasi-periodic.** The oscillating potential drives the highly mobile electrons alternately into the left and right walls. In the plot variable menu, change "numElec" to "leftElecCurr" in Graph 3 and "numIons" to "rightElecCurr" in Graph 4. The impacts of the electron cloud on the left and right walls, and their phasing in time, can be seen in response to the potential oscillations. A history of the electron currents can be seen in Fig. 6.7
- **The ion currents are non-periodic.** Ions, being much heavier than the electrons, exhibit relatively little response to the oscillating potentials. In the plot variable menu, change the Graph 1 quantity "None" to "leftIonCurrent" and the location to "Window 4", then change the Graph 2 quantity "None" to "rightIonCurrent" and the location again to "Window 4". The ion currents do not have the quasi-periodic structure of the electron currents; rather, ions diffuse outward to the walls in response to the DC sheath potentials, which are established by the initial departure of electrons and may also be rectified by the RF. A history of all the particle currents can be seen in Fig. 6.8
- **Ion losses are negligible before the initial establishment of the sheath.** Change the plot quantity in Graph 3 from "leftElecCurr" to "None". Change the plot quantity in Graph 4 from "rightElecCurr" to "numElec" and the "Location" to "Window 3". It is clear that the dominant loss of ions to the wall only begins after the initial decrease in electron population (which corresponds to the establishment of the sheath). A history of the electron popluation against the ion currents can be seen in Fig. 6.9

We can also look at the plasma sheath and the ensuing changes in density profiles directly. In the "Data View" menu at the top left of the Composer window, select "1-D fields". The plot controls here are similar to those of the history window. Select "E_x" for the plot variable in Graph 1. Select "Phi" for the plot variable in Graph 2. Select "electronsDensity" for the plot variable in Graph 3. Select "He1Density" for the plot variable in Graph 4, and select "Window 3" for the location of this plot. The evolution of the discharge in time can be viewed by moving the time slider below the plots.

A number of additional physics features can be seen:



Data View: History

Plot Control

Graph 1
<None>
Location: Window 1
Color: Red
Style: Solid

Graph 2
<None>
Location: Window 2
Color: Blue
Style: Solid

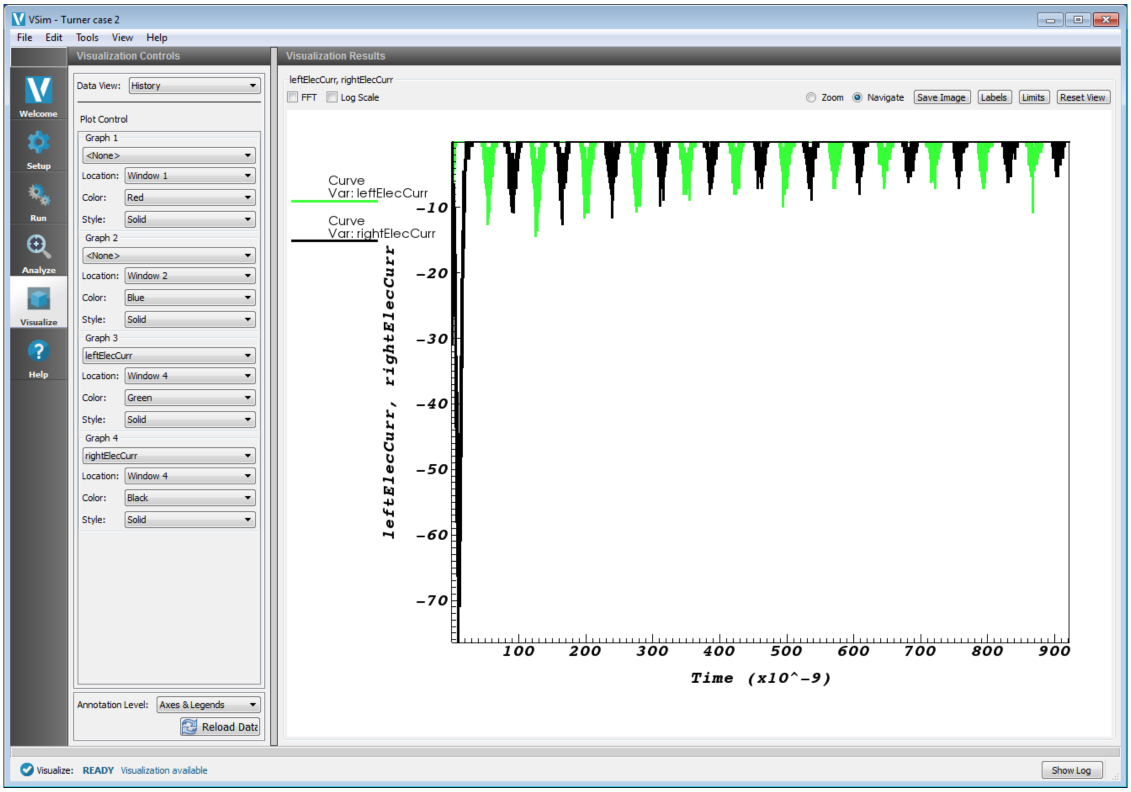
Graph 3
numElec
Location: Window 4
Color: Green
Style: Solid

Graph 4
numIons
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 6.6: The electron and ion populations versus time.



Data View: History

Plot Control

Graph 1
<None>
Location: Window 1
Color: Red
Style: Solid

Graph 2
<None>
Location: Window 2
Color: Blue
Style: Solid

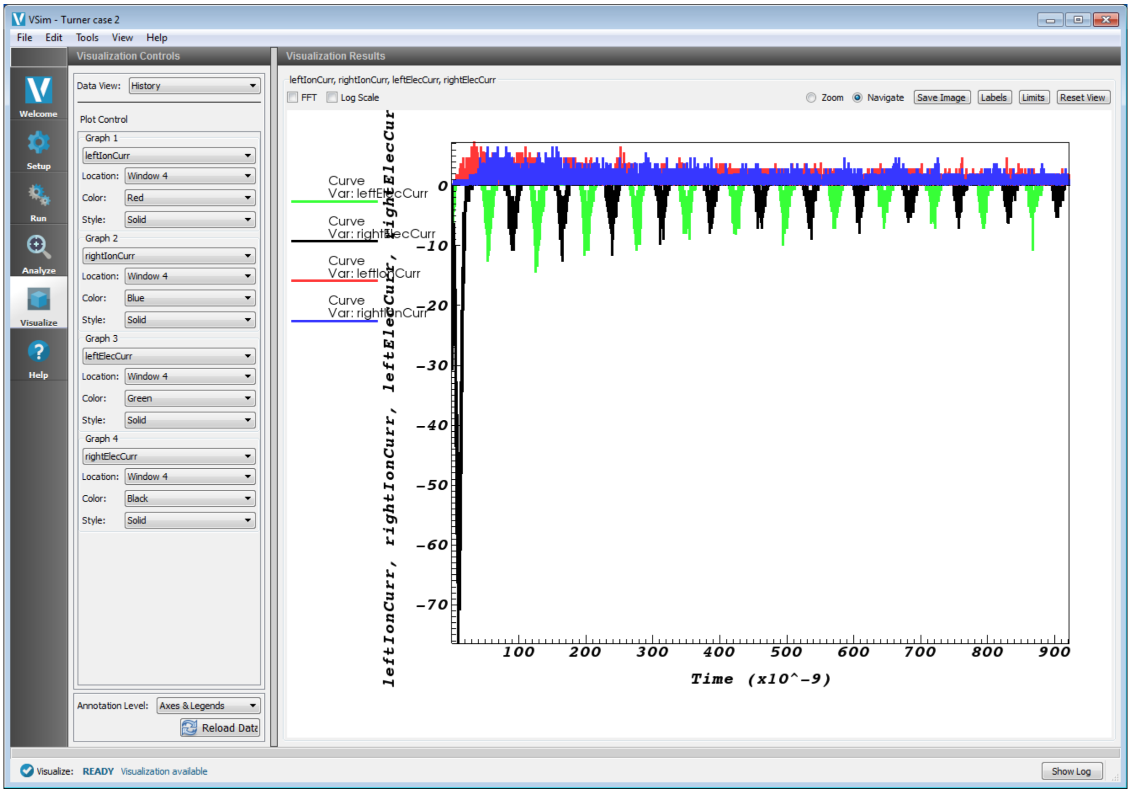
Graph 3
leftElecCurr
Location: Window 4
Color: Green
Style: Solid

Graph 4
rightElecCurr
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 6.7: Electron currents on the left (green) and right (black) walls versus time



Data View: History

Plot Control

Graph 1
leftIonCurr
Location: Window 4
Color: Red
Style: Solid

Graph 2
rightIonCurr
Location: Window 4
Color: Blue
Style: Solid

Graph 3
leftElecCurr
Location: Window 4
Color: Green
Style: Solid

Graph 4
rightElecCurr
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

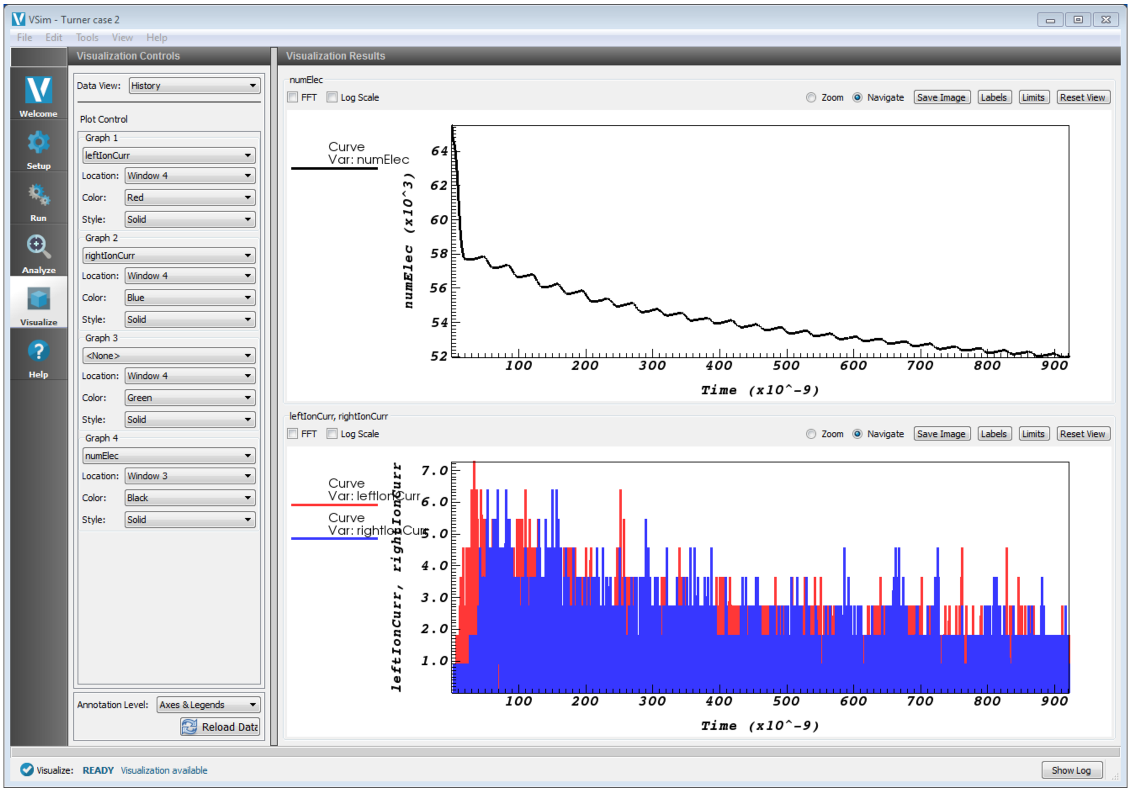
Fig. 6.8: Electron and ion currents on the left and right walls versus time

- **Sheath effects are present.** Regions of sharp potential variation, corresponding to strong electric fields, arise near the walls, but such fields are screened out in the bulk plasma. Moving the time slider, it is clear that this sheath behavior persists regardless of the phase of the oscillating wall potential.
- **Electron profiles are altered much faster than ion profiles.** Both ions and electron profiles are initially constant (5.12e14 1/m^3), but by the time the first nontrivial dump file is produced (at time dumpPeriodicity * dt, approximately 1/3 of the way through the period of the first wall oscillation), electron-poor regions corresponding to the sheaths have already been established in the electron profile, while the ions have barely begun to respond to the presence of the sheath. Moving the slider forward in time, one observes that the electron profile predominantly oscillates in response to the wall potential, while the ion profile evolves considerably more slowly, particularly outside the sheath regions. The 1-D fields can be seen in Fig. 6.10

Further Experiments

Now that we understand some of the basic physics of the discharge, we are in position to apply physics-based particle loading methods to hasten its eventual convergence to steady-state. The underlying principle here is to identify the 'slow' processes involved in the evolution of the discharge toward steady state, and then alter the loading to more closely mimic the state to which the plasma is being driven. While we cannot entirely predict the parameters of the steady-state, it is not difficult to at least get some idea of how the simulation is evolving and adjust the particle loads accordingly. We have already observed a number of physical processes of possible relevance:

- initial electron loss and the establishment of ambipolarity
- the slow decay of the total ion and electron population following the initial electron loss
- the rapid response of electrons to applied electric fields, particularly in the sheath region



Data View: History

Plot Control

Graph 1
leftIonCurr
Location: Window 4
Color: Red
Style: Solid

Graph 2
rightIonCurr
Location: Window 4
Color: Blue
Style: Solid

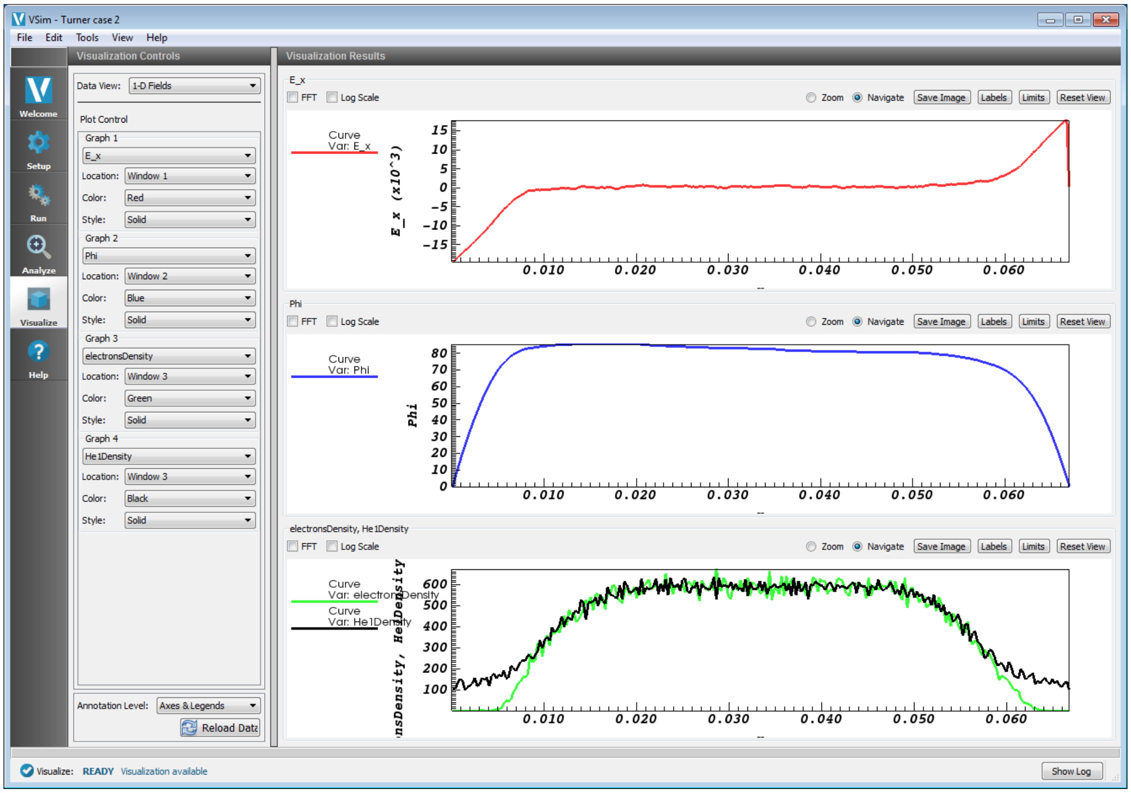
Graph 3
<None>
Location: Window 4
Color: Green
Style: Solid

Graph 4
numElec
Location: Window 3
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 6.9: History plots showing the majority of the ion current to the walls only begins after initial decrease in electron population


VSim - Turner case 2

Data View: 1-D Fields

Plot Control

Graph 1
E_x
Location: Window 1
Color: Red
Style: Solid

Graph 2
Phi
Location: Window 2
Color: Blue
Style: Solid

Graph 3
electronsDensity
Location: Window 3
Color: Green
Style: Solid

Graph 4
He1Density
Location: Window 3
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 6.10: Plots of various 1-D field quantities showing the final state of the run.

- the slow evolution of ion density profiles.

Of these, we will primarily consider the ion profiles; the high mobility of the electrons suggests that electron profiles will adjust correspondingly on much shorter timescales. Additionally, since the strong electric fields in the plasma sheath region are screened out via Debye shielding as we move away from the walls, it seems clear that profile adjustments in the bulk plasma (where the driving electric fields are weakest) will ensue more slowly than in the plasma edge. We therefore concentrate our attention first on obtaining an approximately correct value for the ion density at the center of the domain.

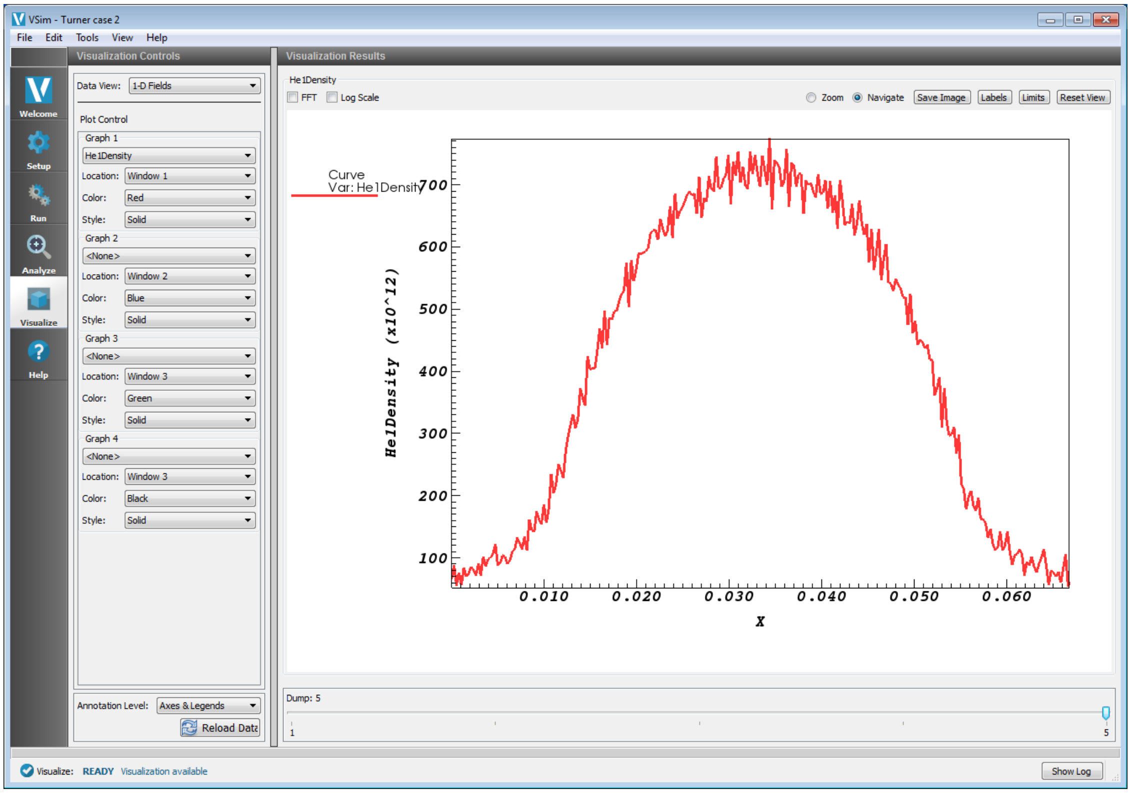
From the 'Data View' menu at the top, select '1-D Fields' again, and set the plot variable to '<None>' in plots 2, 3, and 4. In Graph 1, set the plot variable to 'He1Density' and again move the timeslider on the bottom right of the window. The central ion density steadily rises; from its initial value of 5.12e14 1/m^3, it rises to 7.5e14 1/m^3 by the end of our comparatively short run. In addition, a rapid decrease in density near the walls (associated with the plasma sheath) has lowered the edge densities to less than 1e14 1/m^3. The ion density can be seen in Fig. 6.11

Additional Studies

It is possible to try other techniques to converge to steady state faster. These techniques are outlined in detail in the text-based version of this example. In summary:

Another thing you can try are by loading the particle with a non-uniform profile that better resembles the outcome.

Yet another is to leave a gap near the walls when loading electrons and ions. What happens in the discharge? The electrons, being highly mobile, rush to fill the gap, but rather than immediately being lost to the wall, they instead produce strong electric fields at the plasma edge which begin to modify the ion profile and bring about ambipolarity. If the gap is sufficiently large, the collisional production of ions and electrons will begin before appreciable wall losses ensue, and we can thus assess the relative rates of production and loss fairly early in the simulation. Since the electrons



Data View: 1-D Fields

Plot Control

Graph 1
He1Density
Location: Window 1
Color: Red
Style: Solid

Graph 2
<None>
Location: Window 2
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 3
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 3
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Dump: 5

Visualize: READY Visualization available

Fig. 6.11: The He ion density at the end of the run

are highly mobile, let's treat the average electron population as a measure of how well we've achieved this balance; net electron production as we move into the ambipolar phase means that our gap is too low (we have removed too much density), while net losses mean that our gap is insufficiently wide. As the profile shapes near the walls tend to adjust themselves fairly quickly (due to the larger electric fields in this region), we can in this manner obtain approximately correct values for the total ion and electron populations at the simulation outset.

6.2 Capacitively Coupled (text-based setup)

6.2.1 1D Capacitive Plasma Chamber (capacitivelyCoupledPlasma1DT.pre)

Keywords:

`CCP discharge, secondary emission, elastic collision, excitation, ionization.`

Problem description

The capacitively coupled plasma (CCP) is one of the most common types of industrial plasma sources. The discharges usually take place between metal electrodes in a reaction chamber and are driven by a radio-frequency (RF) or DC power supply. The plasma is sustained by ohmic heating in the main body and stochastic heating through a capacitive sheath.

This example demonstrates the generation of a capacitively coupled plasma inside two parallel conducting plates separated by 0.05 m. A background Ar neutral gas at approximately 6 mTorr $\left(2.0 \times 10^{20}\ m^{-3}\right)$ is filled between the electrodes. The right electrode is grounded, while the left one is connected to a voltage source of 200 V at 60 MHz.

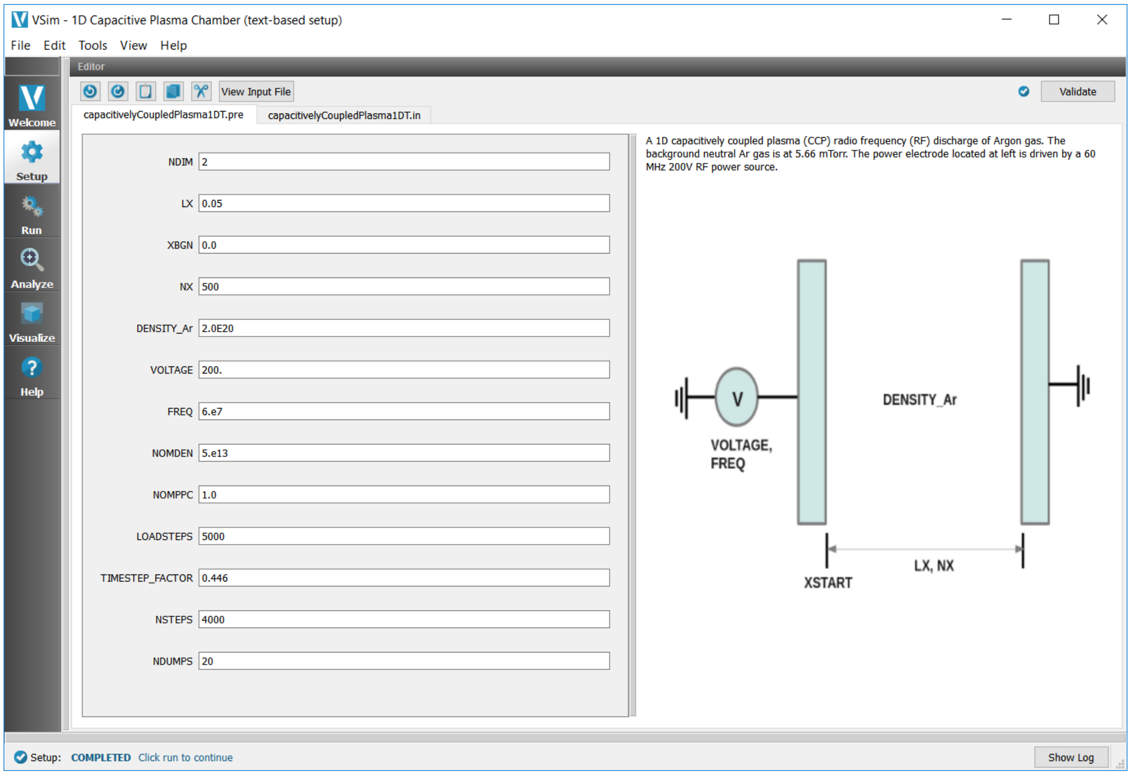
This simulation can be performed with a VSimPD license.

Opening the Simulation

The 1D Capacitively Coupled Plasma Discharge example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window, expand the *VSim for Plasma Discharges* option.
- Expand the *Capacitively Coupled Plasmas (text-based setup)* option.
- Select “1D Capacitive Plasma Chamber (text-based setup)” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 6.12.



VSim - 1D Capacitive Plasma Chamber (text-based setup)

capacitivelyCoupledPlasma1DT.pre

NDIM	2
LX	0.05
XBGN	0.0
NX	500
DENSITY_Ar	2.0E20
VOLTAGE	200.
FREQ	6.e7
NOMDEN	5.e13
NOMPPC	1.0
LOADSTEPS	5000
TIMESTEP_FACTOR	0.446
NSTEPS	4000
NDUMPS	20

A 1D capacitively coupled plasma (CCP) radio frequency (RF) discharge of Argon gas. The background neutral Ar gas is at 5.66 mTorr. The power electrode located at left is driven by a 60 MHz 200V RF power source.

Setup: COMPLETED Click run to continue

Fig. 6.12: Setup Window for the 1D Capacitively Coupled Plasma Discharge example.

The time step DT should sufficiently resolve the plasma frequency and collision frequency. The default time step used in this example is TIMESTEP_FACTOR * (0.1 / Plasma frequency) to ensure stability. The initial primary electrons are gradually loaded into the simulation domain over a period of LOADSTEPS timesteps, which has a default value of 5000.

Input File Features

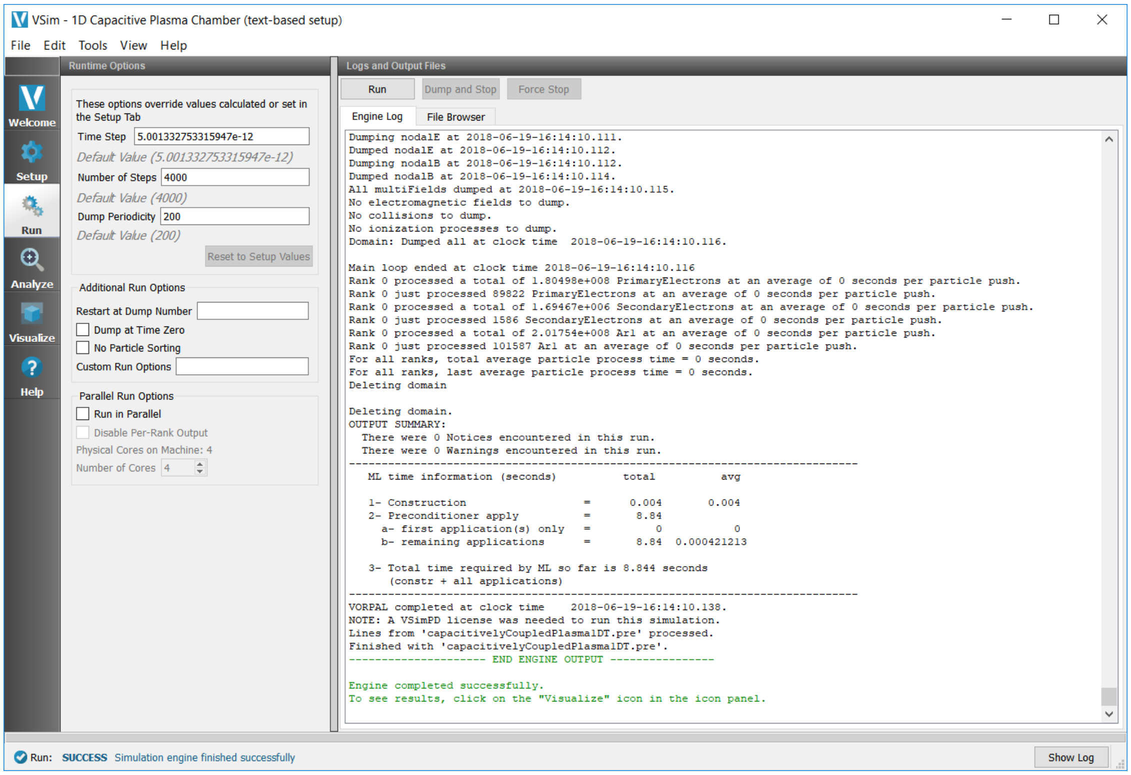
The self-consistent electric field is solved from Poisson’s equation by the electrostatic solver in cylindrical coordinates. Time-dependent Dirichlet boundary conditions are used to set up the boundaries of electric fields around the reaction chamber walls.

The plasma is represented by macroparticles which are moved using the Boris pusher in cylindrical coordinates. Various types of elastic and inelastic collisions of the particles are calculated.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in this pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in the window Fig. 6.13 below.



These options override values calculated or set in the Setup Tab

Time Step 5.001332753315947e-12
Default Value (5.001332753315947e-12)
Number of Steps 4000
Default Value (4000)
Dump Periodicity 200
Default Value (200)

Additional Run Options
Restart at Dump Number
□ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options
□ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
Dumping nodalE at 2018-06-19-16:14:10.111.
Dumped nodalE at 2018-06-19-16:14:10.112.
Dumping nodalB at 2018-06-19-16:14:10.112.
Dumped nodalB at 2018-06-19-16:14:10.114.
All multiFields dumped at 2018-06-19-16:14:10.115.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-19-16:14:10.116.

Main loop ended at clock time 2018-06-19-16:14:10.116
Rank 0 processed a total of 1.80498e+008 PrimaryElectrons at an average of 0 seconds per particle push.
Rank 0 just processed 89822 PrimaryElectrons at an average of 0 seconds per particle push.
Rank 0 processed a total of 1.69467e+006 SecondaryElectrons at an average of 0 seconds per particle push.
Rank 0 just processed 1586 SecondaryElectrons at an average of 0 seconds per particle push.
Rank 0 processed a total of 2.01754e+008 Ar1 at an average of 0 seconds per particle push.
Rank 0 just processed 101587 Ar1 at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
------------------------------------------------------------------------------
   ML time information (seconds)          total          avg

   1- Construction                  =      0.004       0.004
   2- Preconditioner apply          =       8.84
     a- first application(s) only   =          0           0
     b- remaining applications      =       8.84  0.000421213

   3- Total time required by ML so far is 8.844 seconds
      (constr + all applications)
------------------------------------------------------------------------------
VORPAL completed at clock time    2018-06-19-16:14:10.138.
NOTE: A VSimPD license was needed to run this simulation.
Lines from 'capacitivelyCoupledPlasma1DT.pre' processed.
Finished with 'capacitivelyCoupledPlasma1DT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 6.13: The Run Window at the end of execution.

Visualizing the results

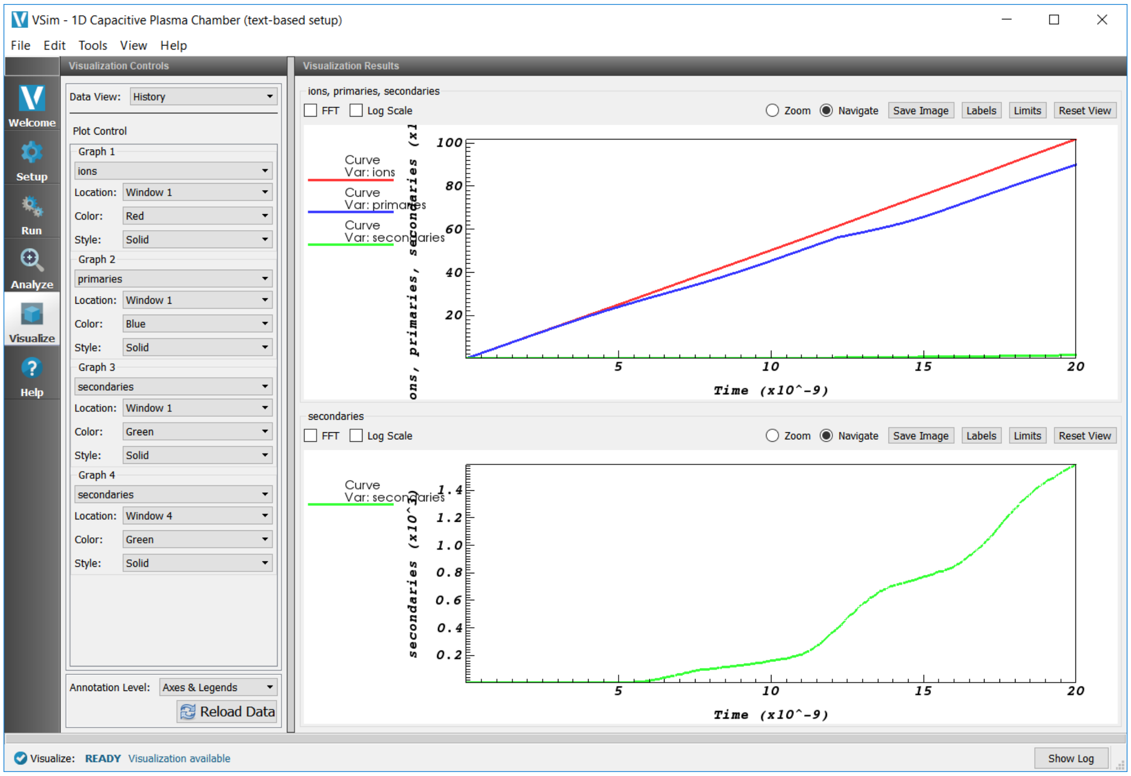
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

The ion, primary electron, and secondary electron particle number histories can be shown as in Fig. 6.14 as follows:

- From the “Data View” option, select “History”
- Set the Location of Graphs 2 and 3 to *Window 1*
- Set Graph 4 to *secondaries* with color *Green*

After around 10ns (2000 time steps), the numbers of ions, primaries, and secondary electrons are each increasing. The simulation converges as the number of secondary electrons approaches a constant, indicating a steady state plasma. At 20ns, when this example ends, the simulation has not yet reached steady state.



Data View: History

Graph 1: ions; Location: Window 1; Color: Red; Style: Solid

Graph 2: primaries; Location: Window 1; Color: Blue; Style: Solid

Graph 3: secondaries; Location: Window 1; Color: Green; Style: Solid

Graph 4: secondaries; Location: Window 4; Color: Green; Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 6.14: Visualization of number histories of ion, primary electron and secondary electron macroparticles for around 4000 steps or 20 nanoseconds.

Further Experiments

Vorpal allows one to set up collision interactions with considerable flexibility. The collisions involved in this example are electron-neutral collisions that produce ionization and ohmic heating. As a further experiment, ion-neutral collisions, such as elastic scattering and charge exchange, can also be added to the simulation.

This example uses sample cross-section files (sampleElasticCrossSection.dat, sampleExcitationCrossSection.dat, and sampleIonizationCrossSection.dat) for the neutral argon gas. While the data in these cross-section files is not necessarily physically correct, these files can easily be modified by users to make use of more accurate scattering datasets (EEDL, LXcat, etc.) VSim can directly import scattering data from the LXcat database, which contains cross section data for around one hundred different materials. As another further experiment, the background argon gas and its cross section data can be replaced with other materials in the LXcat database. This enables one to easily switch the background gas in a CCP simulation.

The format of a user-defined cross-sectional data file (e.g. the sample cross-section files used in this example), is formatted as follows:

- Line 1: Process specifier as a capitalized text string, either `ELASTIC`, `EXCITATION` or `IONIZATION`.
- Line 2: Threshold, the minimum energy in eV needed for this process to occur, as a float
- Line 3: Number of table entries, as an integer

- Lines 4 - end: A two-column table in which the first column is the collision energy in eV, and the second column is the cross-section in square meters.

As an example, the cross-sectional data file sampleElasticCrossSection.dat is shown below:

```
ELASTIC
0.000
7
0.000 0.00e-20
10.50 16.7e-20
25.00 7.75e-20
40.00 4.45e-20
70.00 2.25e-20
100.0 1.50e-20
150.0 1.00e-20
```

Users may import data from other sources as text files with this format, enabling precision control over the collision processes used in their model.

6.2.2 2D Capacitive Plasma Chamber (capacitivelyCoupledPlasma2DT.pre)

Keywords:

`capacitively coupled plasma discharge under RF and DC voltage in 2D cylindrical system.`

Problem description

The capacitively coupled plasma (CCP) is one of the most common types of industrial plasma sources. These plasma discharges typically take place between metal electrodes in a reaction chamber and are driven by a radio frequency (RF) or direct current (DC) power supply. The plasma is sustained by ohmic heating in the main body and stochastic heating through the capacitive sheath.

This example demonstrates the generation of a capacitively coupled plasma inside an axially symmetric reaction chamber with a 50 mm radius and 50 mm length. The top and side walls are grounded at zero potential. A target located at the bottom of the chamber is connected to a 60 MHz AC voltage source at 200 V. There is a small gap of 5 mm between the target and the grounded wall. The chamber is filled with a background gas of Argon at about 0.005 Torr $\left(2.0 \times 10^{20}\ \mathrm{m}^{-3}\right)$. An initial electron density at $10^{12}\ \mathrm{m}^{-3}$ is seeded to start the discharge.

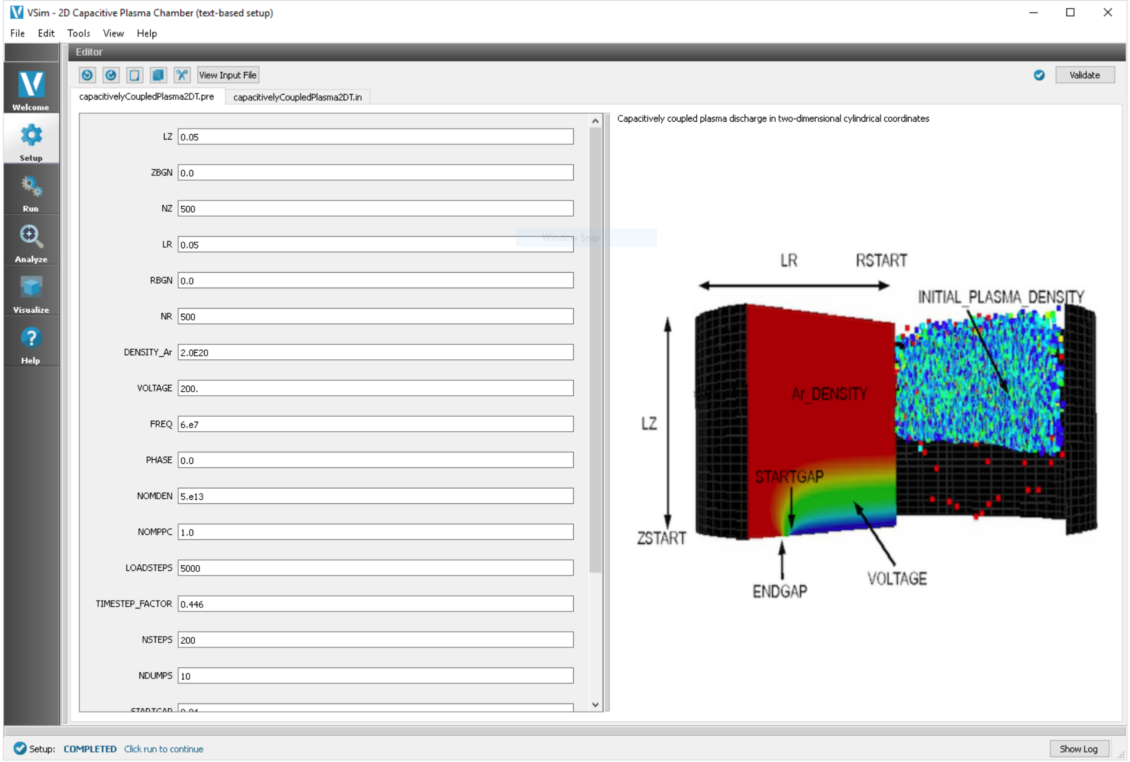
This simulation can be performed with a VSimPD license.

Opening the Simulation

The Capacitively Coupled Plasma 2D example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Capacitively Coupled Plasmas (text-based setup)* option.
- Select "2D Capacitive Plasma Chamber (text-based setup)" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in figure Fig. 6.15.


VSim - 2D Capacitive Plasma Chamber (text-based setup)
File Edit Tools View Help
Editor
View Input File
Validate
capacitivelyCoupledPlasma2DT.pre
capacitivelyCoupledPlasma2DT.in
Welcome
Setup
Run
Analyze
Visualize
Help
LR
RSTART
INITIAL_PLASMA_DENSITY
LZ
Ar_DENSITY
STARTGAP
ZSTART
ENDGAP
VOLTAGE
Show Log

Capacitively coupled plasma discharge in two-dimensional cylindrical coordinates

LZ 0.05
ZBGN 0.0
NZ 500
LR 0.05
RBGN 0.0
NR 500
DENSITY_Ar 2.0E20
VOLTAGE 200.
FREQ 6.e7
PHASE 0.0
NOMDEN 5.e13
NOMPPC 1.0
LOADSTEPS 5000
TIMESTEP_FACTOR 0.446
NSTEPS 200
NDUMPS 10

Setup: COMPLETED Click run to continue

Fig. 6.15: Setup Window for the Capacitively Coupled Plasma 2D example.

Input File Features

The self-consistent electric field is solved from Poisson's equation by an electrostatic solver in cylindrical coordinates. Time-dependent Dirichlet boundary conditions are used to set up the boundaries of electric fields around the reaction chamber walls.

The plasma is simulated with macroparticles which are moved using the Boris pusher in cylindrical coordinates. Various types of elastic and inelastic collisions of the particles are calculated.

The Setup Window has various parameters available for easy manipulation including the density of the argon background gas (DENSITY_Ar), the voltage (VOLTAGE), and the frequency (FREQ).

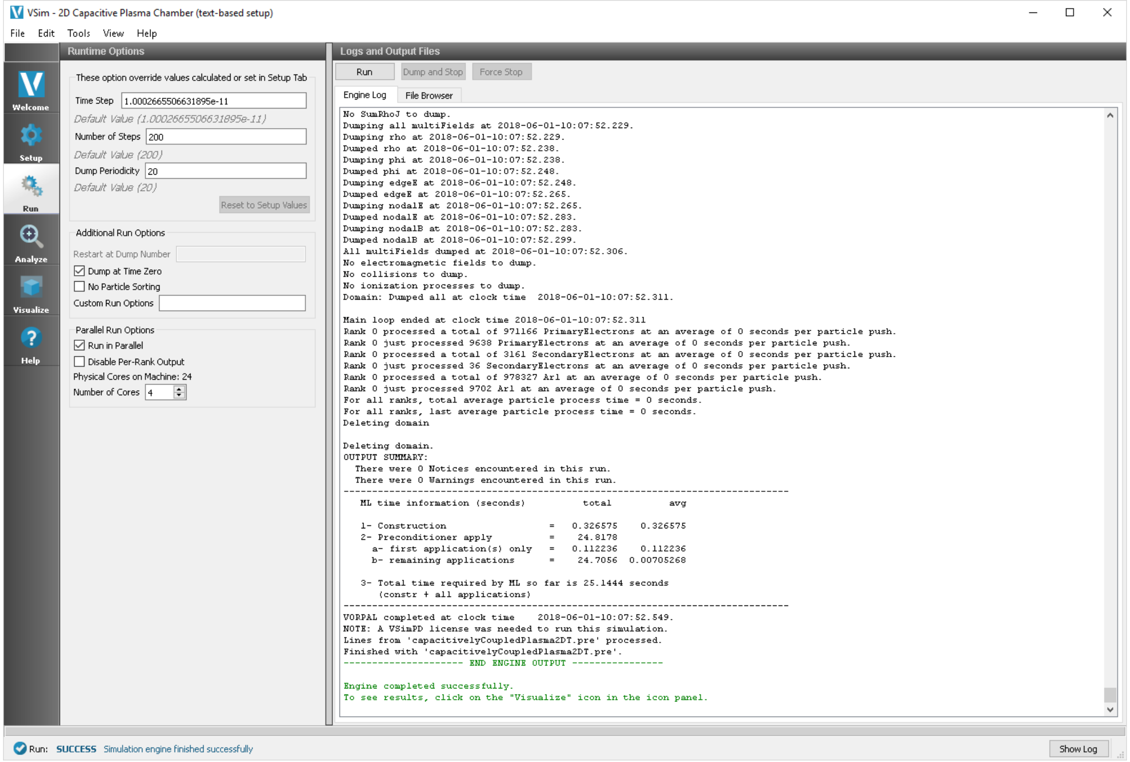
Running the simulation

Once finished with the problem setup, continue as follows:

- Proceed to the Run Window by clicking the **Run** button in the left column of buttons.
- Choose your desired parallel computing options under *Parallel Run Options*.
- To run the file, click on the *Run* button in the upper left corner of the Logs and Output Files pane. You will see the output of the run in that pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 6.16.

Visualizing the results

After performing the above actions, continue as follows:



These option override values calculated or set in Setup Tab

Time Step 1.0002665506631895e-11
Default Value (1.0002665506631895e-11)
Number of Steps 200
Default Value (200)
Dump Periodicity 20
Default Value (20)

Additional Run Options

Restart at Dump Number
☑ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 24
Number of Cores 4

```
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-01-10:07:52.229.
Dumping rho at 2018-06-01-10:07:52.229.
Dumped rho at 2018-06-01-10:07:52.238.
Dumping phi at 2018-06-01-10:07:52.238.
Dumped phi at 2018-06-01-10:07:52.248.
Dumping edgeE at 2018-06-01-10:07:52.248.
Dumped edgeE at 2018-06-01-10:07:52.265.
Dumping nodalE at 2018-06-01-10:07:52.265.
Dumped nodalE at 2018-06-01-10:07:52.283.
Dumping nodalB at 2018-06-01-10:07:52.283.
Dumped nodalB at 2018-06-01-10:07:52.299.
All multiFields dumped at 2018-06-01-10:07:52.306.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-01-10:07:52.311.

Main loop ended at clock time 2018-06-01-10:07:52.311
Rank 0 processed a total of 971166 PrimaryElectrons at an average of 0 seconds per particle push.
Rank 0 just processed 9638 PrimaryElectrons at an average of 0 seconds per particle push.
Rank 0 processed a total of 3161 SecondaryElectrons at an average of 0 seconds per particle push.
Rank 0 just processed 36 SecondaryElectrons at an average of 0 seconds per particle push.
Rank 0 processed a total of 978327 Arl at an average of 0 seconds per particle push.
Rank 0 just processed 9702 Arl at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
------------------------------------------------------------------------------
   ML time information (seconds)          total          avg

   1- Construction                  =   0.326575    0.326575
   2- Preconditioner apply          =    24.8178
     a- first application(s) only   =   0.112236    0.112236
     b- remaining applications      =    24.7056  0.00705268

   3- Total time required by ML so far is 25.1444 seconds
      (constr + all applications)
------------------------------------------------------------------------------
VORPAL completed at clock time    2018-06-01-10:07:52.549.
NOTE: A VSimPD license was needed to run this simulation.
Lines from 'capacitivelyCoupledPlasma2DT.pre' processed.
Finished with 'capacitivelyCoupledPlasma2DT.pre'.
-------------------- END ENGINE OUTPUT ---------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 6.16: The Run Window at the end of execution.

- Proceed to the Visualize Window by clicking the **Visualize** button in the left column of buttons.

To plot the potential:

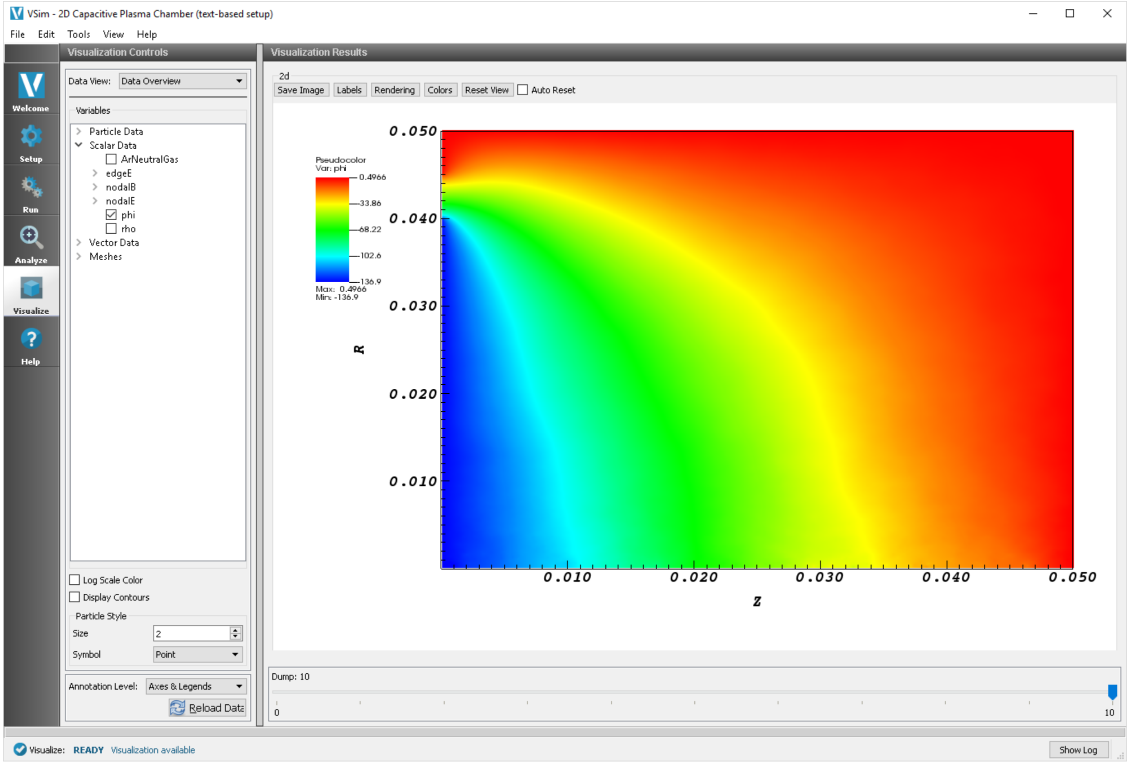
- In the *Variables* section of the *Visualization Controls* pane, expand *Scalar Data*
- Select *phi*
- Move the dump slider at the bottom of the *Visualization Results* pane to the right to move forward in time. Once the slider is selected, the left and right arrow keys will control the slider position as well.

Further Experiments

With a time-step of 10^{-11} seconds, running this simulation for the default 200 time-steps will only capture part of the first oscillation. With the frequency set to 6×10^7 Hz, the oscillation period is 1.667×10^{-8} seconds, which corresponds to 1,667 time-steps. To see the approximate steady-state behavior of this example, set the number of time-steps to 10,000 or more and restart or re-run the simulation. When running in parallel on 4 processors, this should take approximately two hours to complete.

After 5×10^{-8} seconds, or 500 dumps, the plasma sheath starts to exhibit oscillating steady-state behavior. To view the behavior of the oscillating plasma sheath, take the following steps:

- In the *Variables* section of the *Visualization Controls* pane, expand *Scalar Data*
- Select *rho*
- At the top of the *Visualization Results* pane, click *Colors* to open the *Color Options* window
- Select "Fix Minimum" and set the minimum to `-0.0005`, then select "Fix Maximum" and set the Maximum to `0.0005` (or experiment with minimum and maximum values for best results) and click *OK*



Data View: Data Overview

☑ phi

□ Log Scale Color

□ Display Contours

Particle Style

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

Dump: 10

Visualize: READY Visualization available

Fig. 6.17: Visualization of the electric potential in r-z coordinates.

- Move the dump slider at the bottom of the *Visualization Results* pane to dump 250. Once the slider is selected, the left and right arrow keys will control the slider position.

After 250 time-steps, the plasma density should appear as shown in Fig. 6.18. The green areas at approximately zero charge density denote the quasi-neutral plasma bulk, while the red areas (positive charge density) denote the non-neutral the plasma sheath.

To view the plasma sheath potential profile, take the following steps:

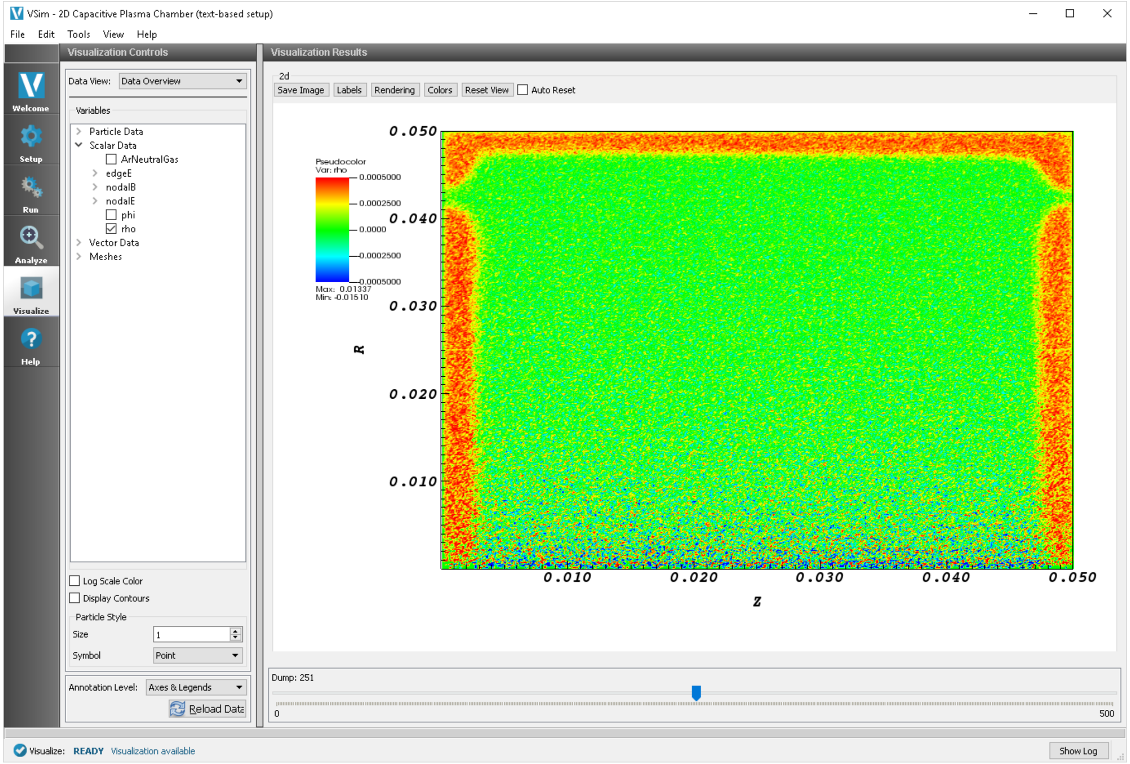
- In the *Variables* section of the *Visualization Controls* pane, select *Field Analysis* from the *Data View* drop-down menu
- In the *Field* drop-down menu, select *phi*
- Under *Lineout Settings* click on the *Horizontal* tab, and change the intercept value if desired
- Click *Perform Lineout*

The electric potential as well as the axial potential profile should now be visible as shown in Fig. 6.19. Move the slider to the right to see how the plasma sheath potential oscillates in time.

6.2.3 TurnerT Case 2 (TurnerT.pre)

Keywords:

`capacitively coupled plasma, CCP, discharge, steady state, TurnerT`



VSim - 2D Capacitive Plasma Chamber (text-based setup)

Data View: Data Overview

Variables
- Particle Data
- Scalar Data
  - □ ArNeutralGas
  - edgeE
  - nodalB
  - nodalE
  - □ phi
  - ☑ rho
- Vector Data
- Meshes

□ Log Scale Color
□ Display Contours

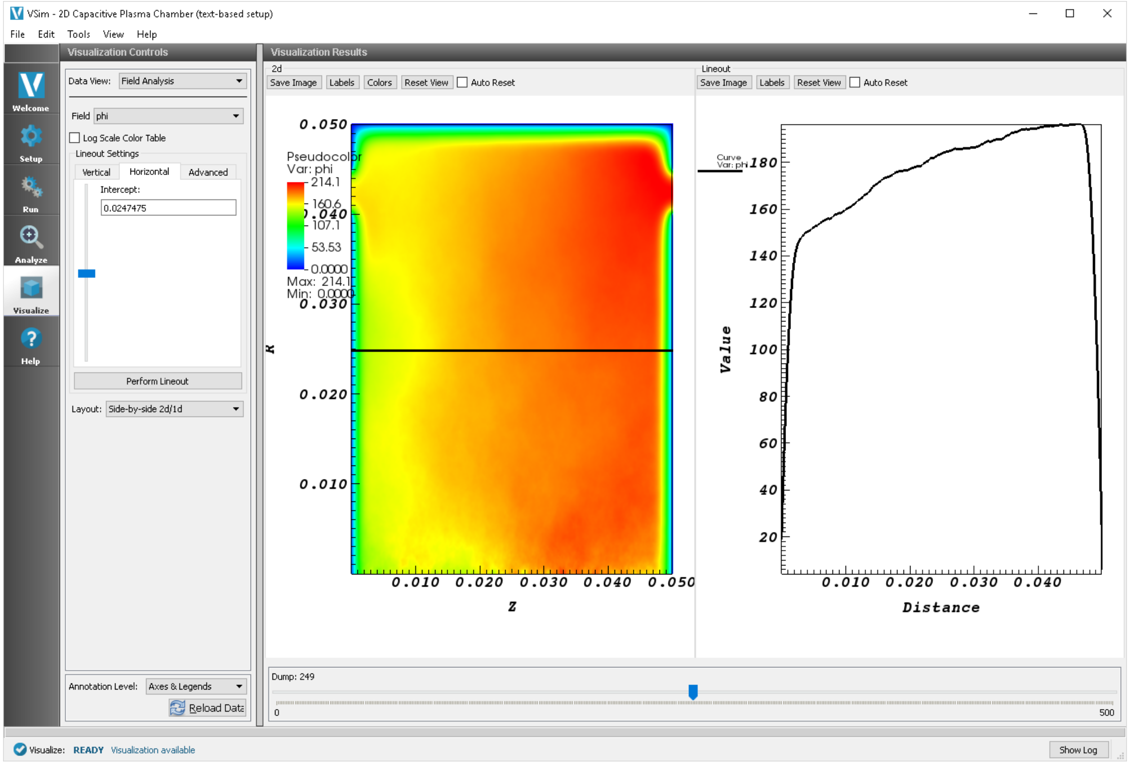
Particle Style
Size: 1
Symbol: Point

Annotation Level: Axes & Legends

Dump: 251

Visualize: READY Visualization available

Fig. 6.18: Visualization of the plasma sheath via the charge density.



VSim - 2D Capacitive Plasma Chamber (text-based setup)

Data View: Field Analysis

Field: phi

□ Log Scale Color Table

Lineout Settings: Vertical | Horizontal | Advanced

Intercept: 0.0247475

Perform Lineout

Layout: Side-by-side 2d/1d

Annotation Level: Axes & Legends

Dump: 249

Visualize: READY Visualization available

Fig. 6.19: Visualization of the plasma sheath via the charge density.

Problem Description

In this example we demonstrate VSim's ability to simulate capacitively coupled plasmas, using the benchmark cases of Turner et al. *[TDD+13]*. Turner's work documents the successful benchmarking of five independently developed particle-in-cell codes (not including VSim) for four different capacitively coupled discharges at various background pressures.

Here, we consider the second of the Turner scenarios, though the input file can be readily modified to simulate the others. In addition to being able to accurately reproduce the Turner results, VSim can also employ physics-based initialization methods to enable more rapid convergence of the simulations to their steady-state. The use of such methods will also be explained below.

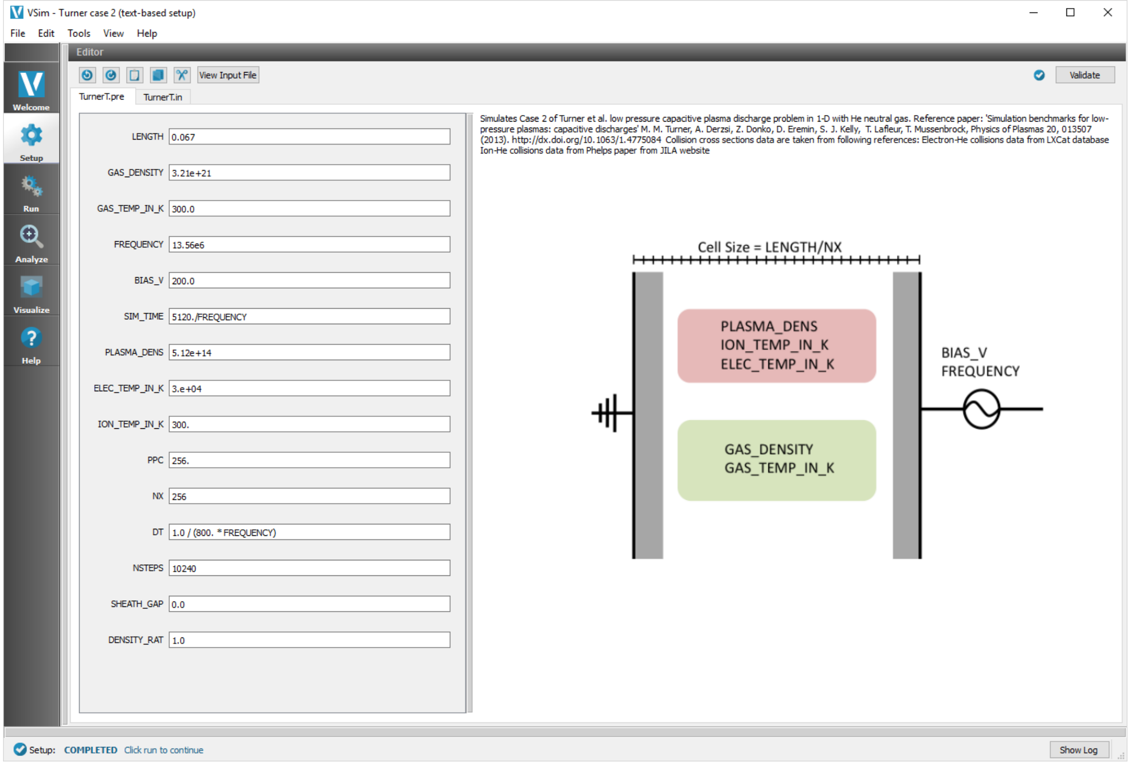
This simulation can be performed with a VSimPD license.

Opening the Simulation

The TurnerT example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item from the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Capacitively Coupled Plasmas (text-based setup)* option.
- Select *Turner case 2 (text-based setup)* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 6.20.



LENGTH 0.067

GAS_DENSITY 3.21e+21

GAS_TEMP_IN_K 300.0

FREQUENCY 13.56e6

BIAS_V 200.0

SIM_TIME 5120./FREQUENCY

PLASMA_DENS 5.12e+14

ELEC_TEMP_IN_K 3.e+04

ION_TEMP_IN_K 300.

PPC 256.

NX 256

DT 1.0 / (800. * FREQUENCY)

NSTEPS 10240

SHEATH_GAP 0.0

DENSITY_RAT 1.0

Simulates Case 2 of Turner et al. low pressure capacitive plasma discharge problem in 1-D with He neutral gas. Reference paper: 'Simulation benchmarks for low-pressure plasmas: capacitive discharges' M. M. Turner, A. Derzsi, Z. Donko, D. Eremin, S. J. Kelly, T. Lafleur, T. Mussenbrock, Physics of Plasmas 20, 013507 (2013). http://dx.doi.org/10.1063/1.4775084 Collision cross sections data are taken from following references: Electron-He collisions data from LXCat database Ion-He collisions data from Phelps paper from JILA website

Setup: COMPLETED Click run to continue

Fig. 6.20: Setup Window for the TurnerT example.

Input File Features

The basic physics of this simulation is a balance between collisional processes and wall losses; a one-dimensional box of length 6.7 cm contains neutral helium gas at room temperature (300 K) and density 3.21e21 1/m^3. The gas is weakly ionized, resulting in a population of free electrons and singly ionized helium atoms at density 5.12e14 1/m^3. The helium ions are also at room temperature, while the electrons are considerably hotter (30,000 K). The left wall of the box is grounded, while the right wall oscillates with a bias voltage of 200 V at frequency 13.56 MHz.

Charged particles are lost upon collision with the wall, and are replenished by ionization of the background neutral gas by the hot electrons; the latter process repopulates both the electrons and helium ions in the plasma (the background neutral gas is treated as an infinite source). Plasma sheaths form near the walls, containing electric fields which are strong relative to those elsewhere in the plasma; the particle density profiles adjust in response to the fields in the sheath. The sheath transit time, for ions, is much longer than the period of the oscillating potential; thus, multiple RF cycles occur while an ion crosses the sheath. A steady state is attained when the loss rate of particles to the wall comes into balance with the ionization rate for a particular profile shape.

In our initial run we are not going to model the full evolution of the discharge to its steady-state parameters; rather, we will explore the basic physics of the discharge and modify the simulation accordingly (with the aim of ultimately hastening convergence to this steady state, while exploring VSim capabilities).

Running the Simulation

To run the simulation, continue as follows:

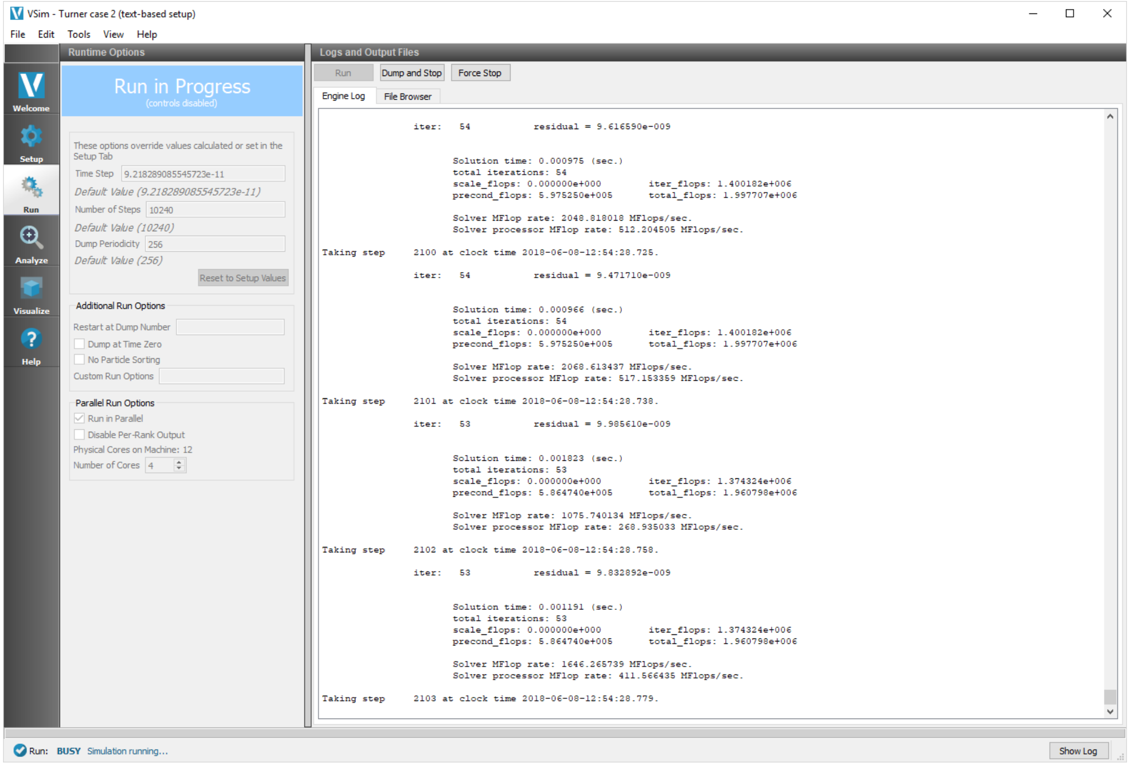
- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- The Number of Time Steps is 10240 (1/400 of the length of Turner's run) for this example, and can be modified if desired.
- Consider checking the 'Run In Parallel' box if desired (set a 'Number of Cores' appropriate for your VSim license).
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." A snapshot of the simulation run during execution is shown in Fig. 6.21.

Note: 10240 steps takes roughly ten minutes on a MacBook Pro in serial, and less than four minutes on a four-core Windows machine.

Analyzing the Results

We are going to run a postprocessing script, computePtclNumDensity.py, which builds density profiles from the particle data generated by VSim, so that we can look at these profiles and their evolution. To do so, we do the following:

- Click the *Analyze Window*
- Check the *Show All Analyzers* box
- Select *computePtclNumDensity.py* from the list of analyzers and click the *Open* button at the bottom of the left panel
- Fill in the text boxes
  - The simulationName should already be filled in, but if it is not, type in the name of the .pre file without the .pre extension, which is TurnerT.
  - For the speciesName, type in 'electrons' without the quotes.



These options override values calculated or set in the Setup Tab

Time Step 9.218289085545723e-11

Default Value (9.218289085545723e-11)

Number of Steps 10240

Default Value (10240)

Dump Periodicity 256

Default Value (256)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

Physical Cores on Machine: 12

Number of Cores 4

```
                iter:   54          residual = 9.616590e-009


                       Solution time: 0.000975 (sec.)
                       total iterations: 54
                       scale_flops: 0.000000e+000        iter_flops: 1.400182e+006
                       precond_flops: 5.975250e+005      total_flops: 1.997707e+006

                       Solver MFlop rate: 2048.818018 MFlops/sec.
                       Solver processor MFlop rate: 512.204505 MFlops/sec.

Taking step     2100 at clock time 2018-06-08-12:54:28.725.

                iter:   54          residual = 9.471710e-009


                       Solution time: 0.000966 (sec.)
                       total iterations: 54
                       scale_flops: 0.000000e+000        iter_flops: 1.400182e+006
                       precond_flops: 5.975250e+005      total_flops: 1.997707e+006

                       Solver MFlop rate: 2068.613437 MFlops/sec.
                       Solver processor MFlop rate: 517.153359 MFlops/sec.

Taking step     2101 at clock time 2018-06-08-12:54:28.738.

                iter:   53          residual = 9.985610e-009


                       Solution time: 0.001823 (sec.)
                       total iterations: 53
                       scale_flops: 0.000000e+000        iter_flops: 1.374324e+006
                       precond_flops: 5.864740e+005      total_flops: 1.960798e+006

                       Solver MFlop rate: 1075.740134 MFlops/sec.
                       Solver processor MFlop rate: 268.935033 MFlops/sec.

Taking step     2102 at clock time 2018-06-08-12:54:28.758.

                iter:   53          residual = 9.832892e-009


                       Solution time: 0.001191 (sec.)
                       total iterations: 53
                       scale_flops: 0.000000e+000        iter_flops: 1.374324e+006
                       precond_flops: 5.864740e+005      total_flops: 1.960798e+006

                       Solver MFlop rate: 1646.265739 MFlops/sec.
                       Solver processor MFlop rate: 411.566435 MFlops/sec.

Taking step     2103 at clock time 2018-06-08-12:54:28.779.
```

Run: BUSY Simulation running...

Fig. 6.21: The Run Window during execution.

- Click the *Analyze* button in the top right of the right panel; this will generate the electron density profiles.
- Now replace 'electrons' in the speciesName box with 'He1', for the helium ions.
- Click the *Analyze* button again to generate the ion density profiles.

The resulting data will be visualizable as electronsDensity and He1Density under the 1-D Fields *Data View* in the *Visualize* Tab.

Visualizing the Results

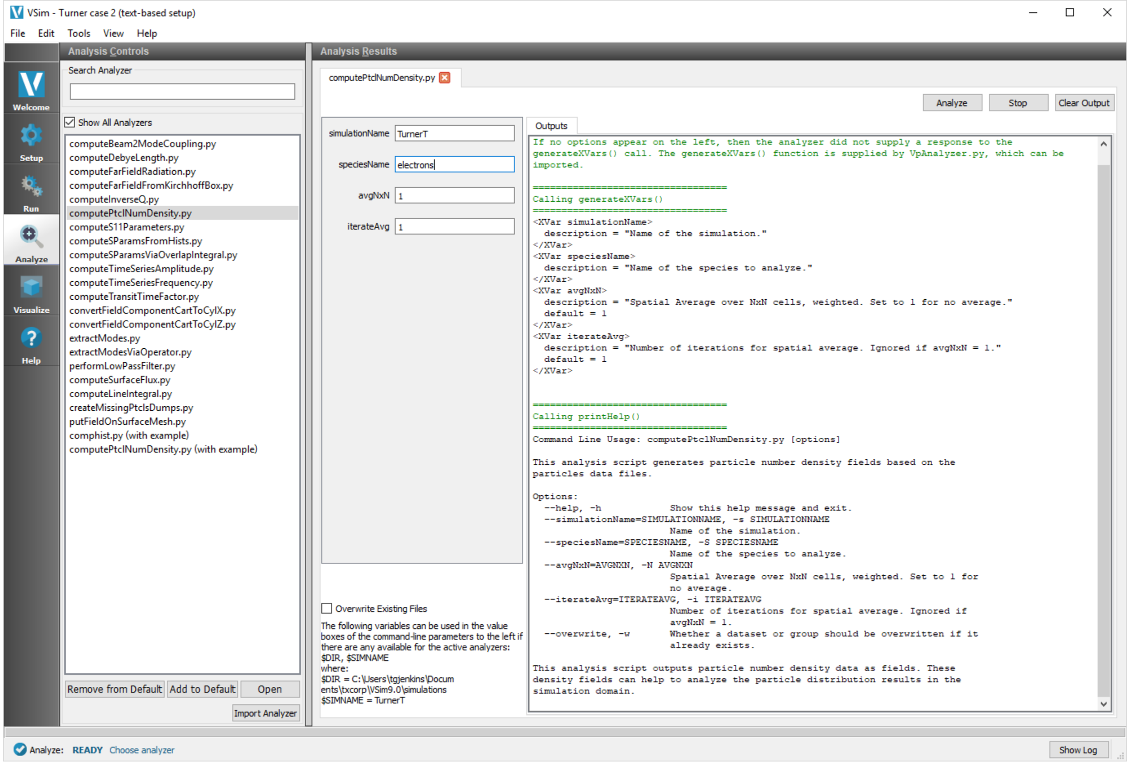
Now that we've got all of our data, let's look at it.

- Click the *Visualize Window*

After a brief moment the visualization options for this data should appear.

We'll first look at the time evolution of some fundamental one-dimensional quantities. From the *Data View* pulldown menu on the top left, select *History*. The default view here should contain four plots, namely, the electron and ion currents to the left wall and the number of electrons and ions in the simulation. A number of notable physics effects can be seen here:

- **After a sharp initial decrease in the electron population, both ion and electron populations decline at approximately the same rate.** This is not as apparent from the separate numElec and numIons plots, but clicking on the "Location" dropdown window in Graph 3 and selecting "Window 4" as the new rendering destination, places both ion and electron populations in the same plot. (Select "<None>" in the plot variable (the topmost menu) for both Graph 1 and Graph 2 to resize the electron/ion plot.) The initial decrease in electron population arises when rapid electron wall losses create a charge imbalance in the plasma and establish plasma


VSim - Turner case 2 (text-based setup)
File Edit Tools View Help
Analysis Controls
Search Analyzer
☑ Show All Analyzers
computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
computeSurfaceFlux.py
computeLineIntegral.py
createMissingPtclsDumps.py
putFieldOnSurfaceMesh.py
comphist.py (with example)
computePtclNumDensity.py (with example)
Remove from Default
Add to Default
Open
Import Analyzer
Welcome
Setup
Run
Analyze
Visualize
Help
Analysis Results
computePtclNumDensity.py
Analyze
Stop
Clear Output
Outputs
Analyze: READY Choose analyzer
Show Log

simulationName: TurnerT

speciesName: electrons

avgNxN: 1

iterateAvg: 1

□ Overwrite Existing Files

The following variables can be used in the value boxes of the command-line parameters to the left if there are any available for the active analyzers:
$DIR, $SIMNAME
where:
$DIR = C:\Users\tgjenkins\Documents\txcorp\VSim9.0\simulations
$SIMNAME = TurnerT

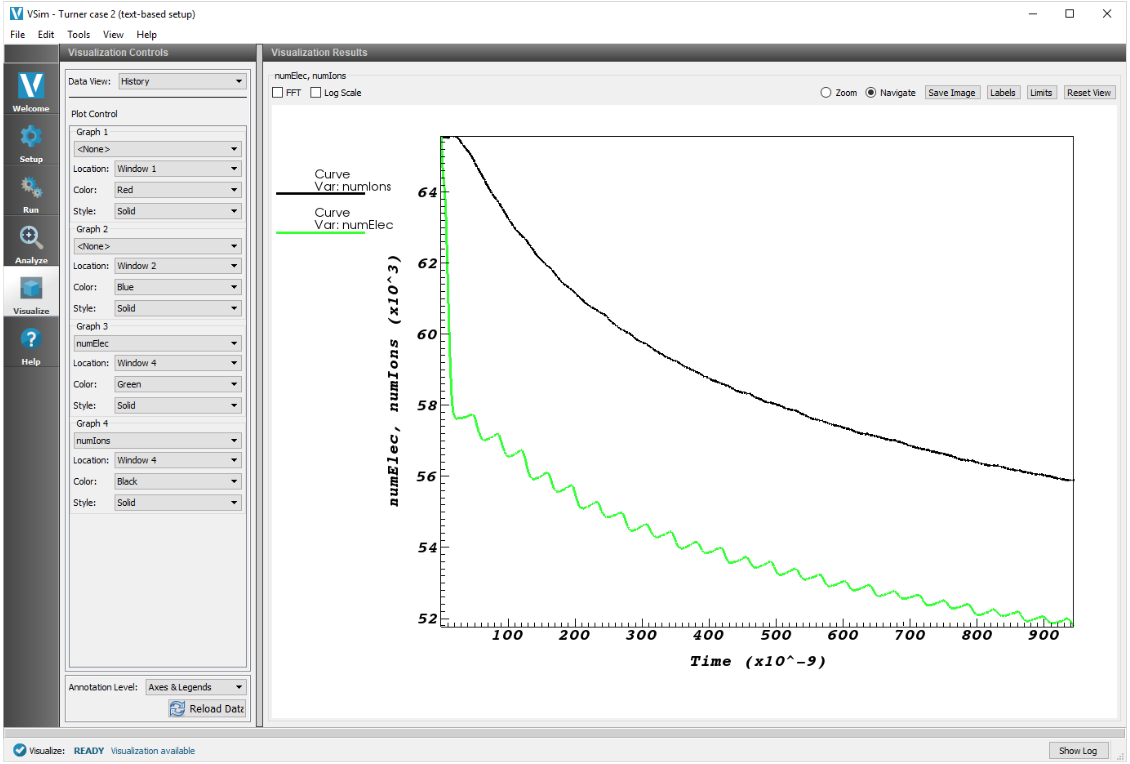
```
If no options appear on the left, then the analyzer did not supply a response to the
generateXVars() call. The generateXVars() function is supplied by VpAnalyzer.py, which can be
imported.

==================================
Calling generateXVars()
==================================
<XVar simulationName>
  description = "Name of the simulation."
</XVar>
<XVar speciesName>
  description = "Name of the species to analyze."
</XVar>
<XVar avgNxN>
  description = "Spatial Average over NxN cells, weighted. Set to 1 for no average."
  default = 1
</XVar>
<XVar iterateAvg>
  description = "Number of iterations for spatial average. Ignored if avgNxN = 1."
  default = 1
</XVar>


==================================
Calling printHelp()
==================================
Command Line Usage: computePtclNumDensity.py [options]

This analysis script generates particle number density fields based on the
particles data files.

Options:
  --help, -h            Show this help message and exit.
  --simulationName=SIMULATIONNAME, -s SIMULATIONNAME
                        Name of the simulation.
  --speciesName=SPECIESNAME, -S SPECIESNAME
                        Name of the species to analyze.
  --avgNxN=AVGNXN, -N AVGNXN
                        Spatial Average over NxN cells, weighted. Set to 1 for
                        no average.
  --iterateAvg=ITERATEAVG, -i ITERATEAVG
                        Number of iterations for spatial average. Ignored if
                        avgNxN = 1.
  --overwrite, -w       Whether a dataset or group should be overwritten if it
                        already exists.

This analysis script outputs particle number density data as fields. These
density fields can help to analyze the particle distribution results in the
simulation domain.
```

Fig. 6.22: Analysis window for the TurnerT example.

sheaths near the walls. Thereafter, this charge imbalance is preserved and the transport of both electrons and ions to the wall becomes ambipolar. A history of the particle popluations can be seen in Fig. 6.23.

- **The electron wall currents are quasi-periodic.** The oscillating potential drives the highly mobile electrons alternately into the left and right walls. In the plot variable menu, change "numElec" to "leftElecCurr" in Graph 3 and "numIons" to "rightElecCurr" in Graph 4. The impacts of the electron cloud on the left and right walls, and their phasing in time, can be seen in response to the potential oscillations. A history of the electron currents can be seen in Fig. 6.24.
- **The ion currents are non-periodic.** Ions, being much heavier than the electrons, exhibit relatively little response to the oscillating potentials. In the plot variable menu, change the Graph 1 quantity "None" to "leftIonCurrent" and the location to "Window 4", then change the Graph 2 quantity "None" to "rightIonCurrent" and the location again to "Window 4". The ion currents do not have the quasi-periodic structure of the electron currents; rather, ions diffuse outward to the walls in response to the DC sheath potentials, which are established by the initial departure of electrons and may also be rectified by the RF. A history of all the particle currents can be seen in Fig. 6.25.
- **Ion losses are negligible before the initial establishment of the sheath.** Change the plot quantity in Graph 3 from "leftElecCurr" to "None". Change the plot quantity in Graph 4 from "rightElecCurr" to "numElec" and the "Location" to "Window 3". It is clear that the dominant loss of ions to the wall only begins after the initial decrease in electron population (which corresponds to the establishment of the sheath). A history of the electron population against the ion currents can be seen in Fig. 6.26.

We can also look at the plasma sheath and the ensuing changes in density profiles directly. In the "Data View" menu at the top left of the Composer window, select "1-D fields". The plot controls here are similar to those of the history window. Select "YeeStaticElecFldTrilinos_x" for the plot variable in Graph 1. Select "YeeStaticElecFldTrilinosPotential" for the plot variable in Graph 2. Select "electronsDensity" for the plot variable in Graph 3. Select "He1Density" for the plot variable in Graph 4, and select "Window 3" for the location of this plot. The evolution of the discharge in time



Data View: History

Plot Control

Graph 1
<None>
Location: Window 1
Color: Red
Style: Solid

Graph 2
<None>
Location: Window 2
Color: Blue
Style: Solid

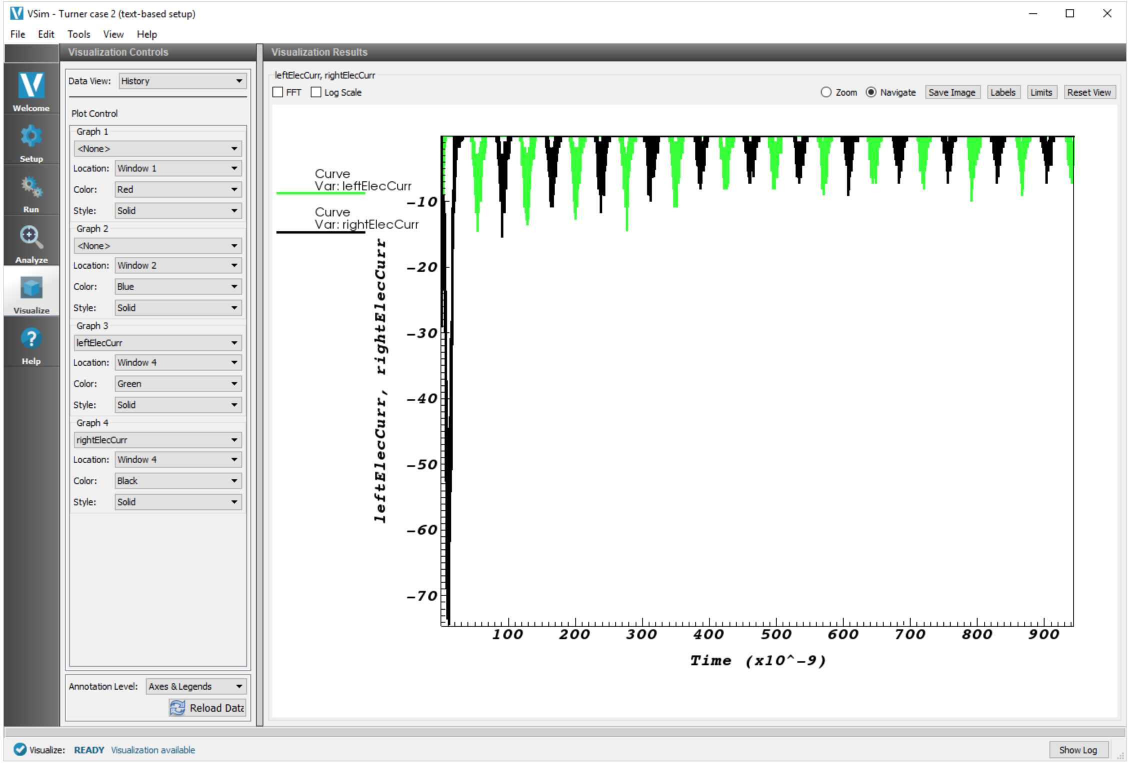
Graph 3
numElec
Location: Window 4
Color: Green
Style: Solid

Graph 4
numIons
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 6.23: The electron and ion populations versus time.



Data View: History

Plot Control

Graph 1
<None>
Location: Window 1
Color: Red
Style: Solid

Graph 2
<None>
Location: Window 2
Color: Blue
Style: Solid

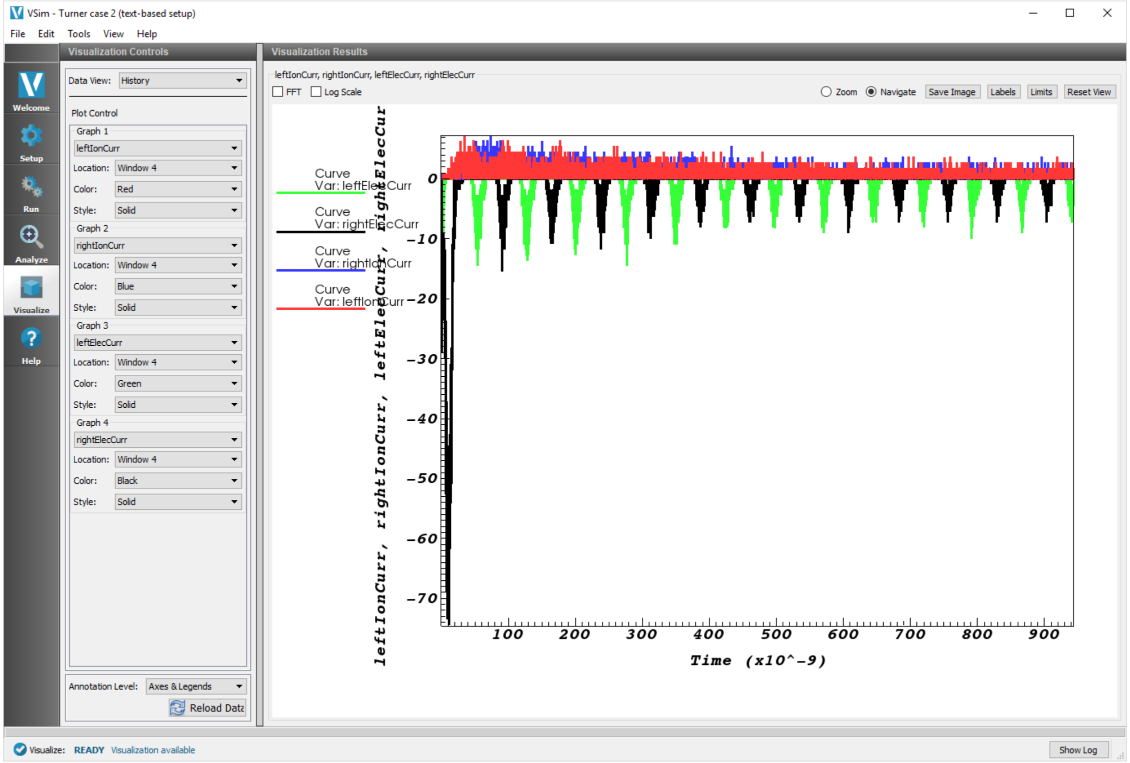
Graph 3
leftElecCurr
Location: Window 4
Color: Green
Style: Solid

Graph 4
rightElecCurr
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 6.24: Electron currents on the left (green) and right (black) walls versus time.



Data View: History

Plot Control

Graph 1
leftIonCurr
Location: Window 4
Color: Red
Style: Solid

Graph 2
rightIonCurr
Location: Window 4
Color: Blue
Style: Solid

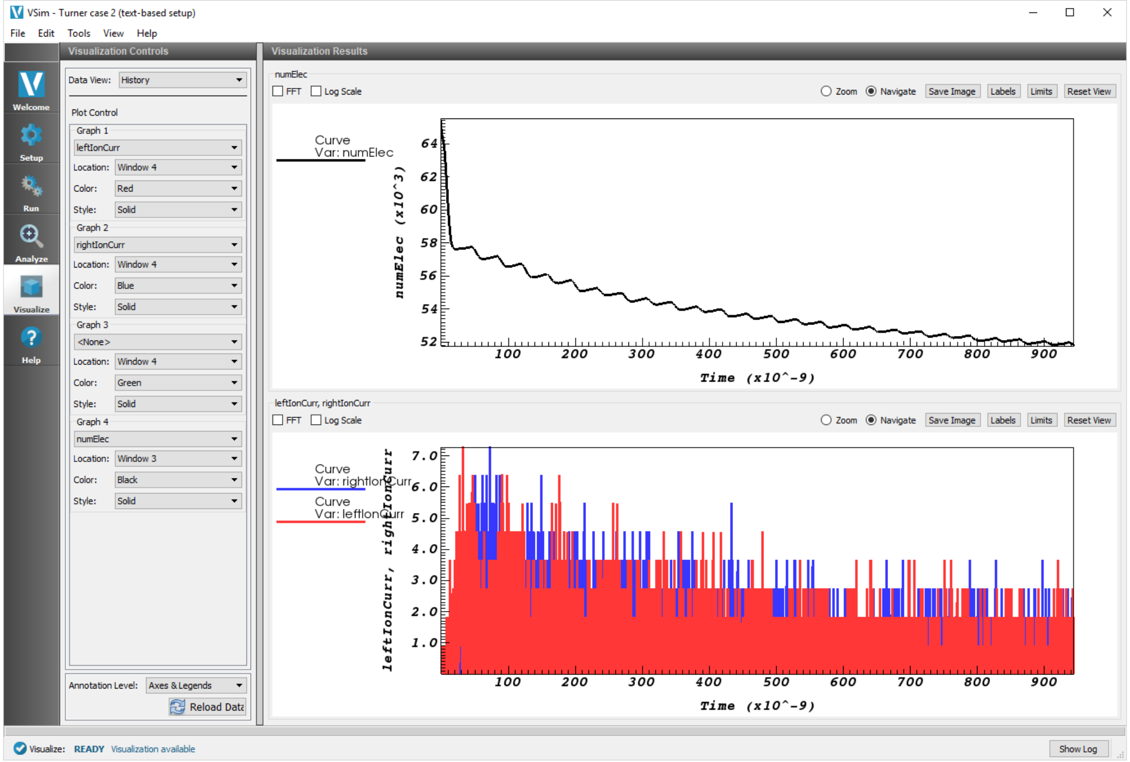
Graph 3
leftElecCurr
Location: Window 4
Color: Green
Style: Solid

Graph 4
rightElecCurr
Location: Window 4
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 6.25: Electron and ion currents on the left and right walls versus time.



Data View: History

Plot Control

Graph 1
leftIonCurr
Location: Window 4
Color: Red
Style: Solid

Graph 2
rightIonCurr
Location: Window 4
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 4
Color: Green
Style: Solid

Graph 4
numElec
Location: Window 3
Color: Black
Style: Solid

Annotation Level: Axes & Legends

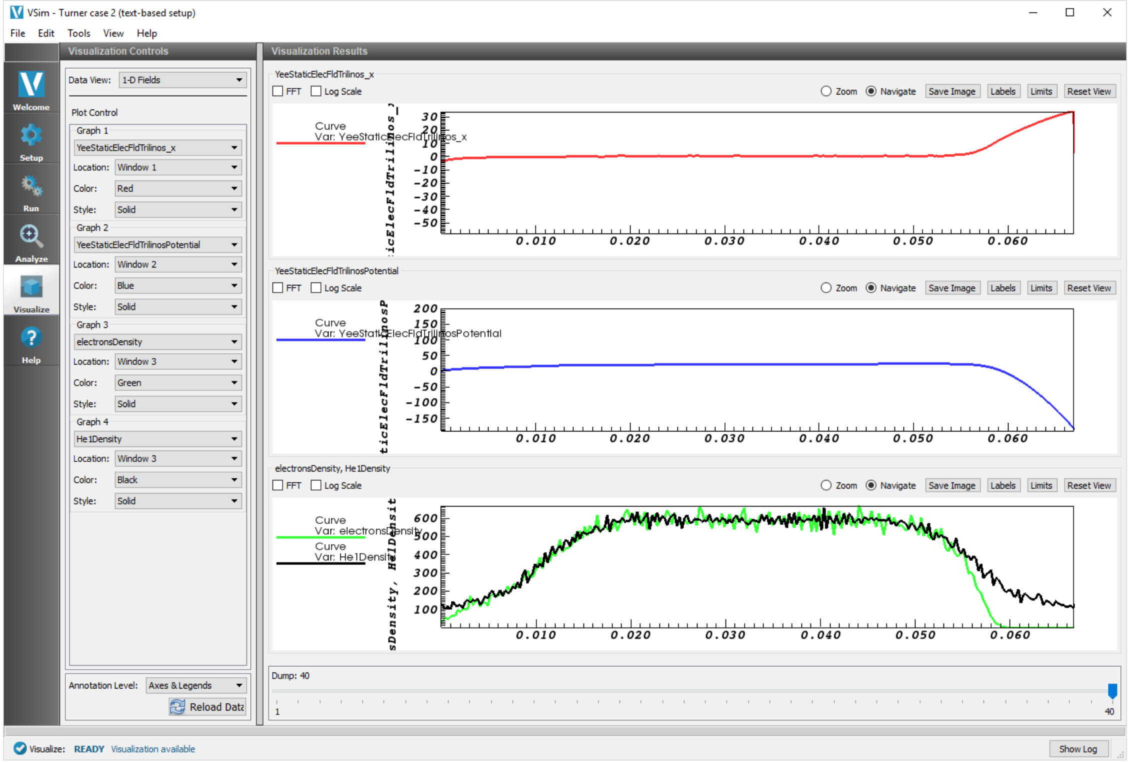
Visualize: READY Visualization available

Fig. 6.26: History plots showing the majority of the ion current to the walls only begins after initial decrease in electron population.

can be viewed by moving the time slider below the plots.

A number of additional physics features can be seen:

- **Sheath effects are present.** Regions of sharp potential variation, corresponding to strong electric fields, arise near the walls, but such fields are screened out in the bulk plasma. Moving the time slider, it is clear that this sheath behavior persists regardless of the phase of the oscillating wall potential.
- **Electron profiles are altered much faster than ion profiles.** Both ions and electron profiles are initially constant (5.12e14 1/m^3), but by the time the first nontrivial dump file is produced (at time dumpPeriodicity * dt, approximately 1/3 of the way through the period of the first wall oscillation), electron-poor regions corresponding to the sheaths have already been established in the electron profile, while the ions have barely begun to respond to the presence of the sheath. Moving the slider forward in time, one observes that the electron profile predominantly oscillates in response to the wall potential, while the ion profile evolves considerably more slowly, particularly outside the sheath regions. The 1-D fields can be seen in Fig. 6.27.



Data View: 1-D Fields

Graph 1
YeeStaticElecFldTrilinos_x
Location: Window 1
Color: Red
Style: Solid

Graph 2
YeeStaticElecFldTrilinosPotential
Location: Window 2
Color: Blue
Style: Solid

Graph 3
electronsDensity
Location: Window 3
Color: Green
Style: Solid

Graph 4
He1Density
Location: Window 3
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Dump: 40

Visualize: READY Visualization available

Fig. 6.27: Plots of various 1-D field quantities showing the final state of the run.

Further Experiments

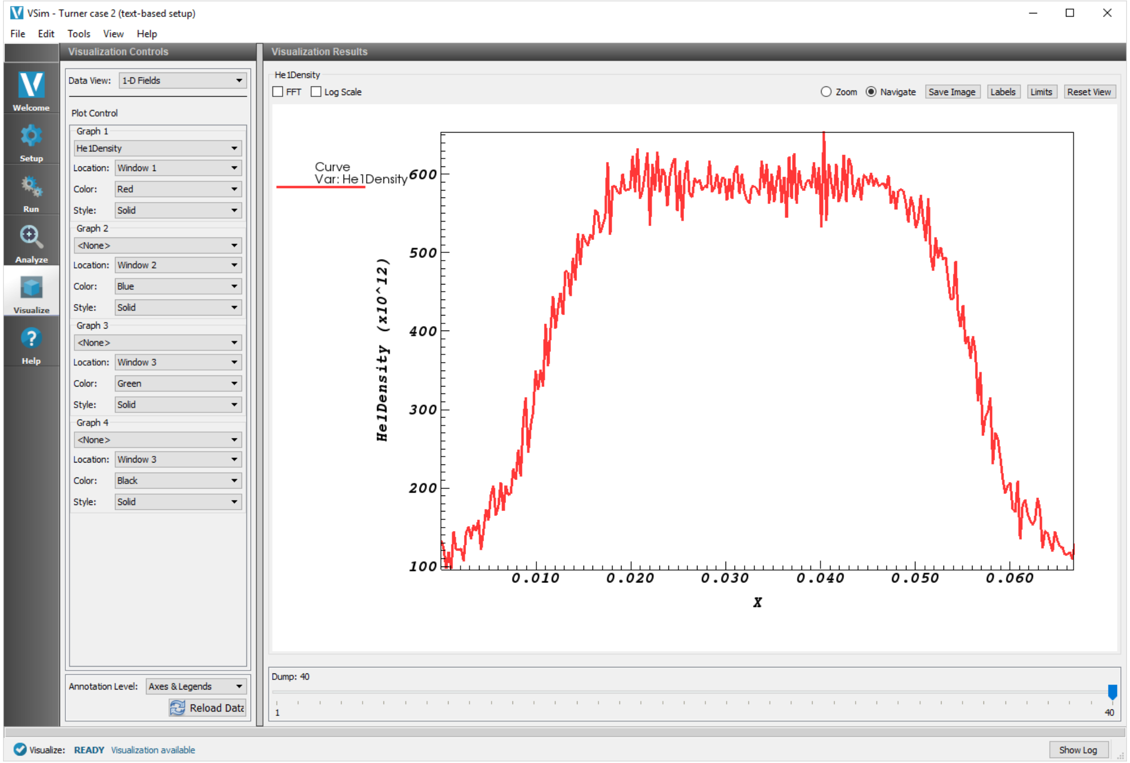
Now that we understand some of the basic physics of the discharge, we are in position to apply physics-based particle loading methods to hasten its eventual convergence to steady-state. The underlying principle here is to identify the ‘slow’ processes involved in the evolution of the discharge toward steady state, and then alter the loading to more closely mimic the state to which the plasma is being driven. While we cannot entirely predict the parameters of the steady-state, it is not difficult to at least get some idea of how the simulation is evolving and adjust the particle loads accordingly. We have already observed a number of physical processes of possible relevance:

- initial electron loss and the establishment of ambipolarity
- the slow decay of the total ion and electron population following the initial electron loss

- the rapid response of electrons to applied electric fields, particularly in the sheath region
- the slow evolution of ion density profiles.

Of these, we will primarily consider the ion profiles; the high mobility of the electrons suggests that electron profiles will adjust correspondingly on much shorter timescales. Additionally, since the strong electric fields in the plasma sheath region are screened out via Debye shielding as we move away from the walls, it seems clear that profile adjustments in the bulk plasma (where the driving electric fields are weakest) will ensue more slowly than in the plasma edge. We therefore concentrate our attention first on obtaining an approximately correct value for the ion density at the center of the domain.

From the 'Data View' menu at the top, select '1-D Fields' again, and set the plot variable to '<None>' in plots 2, 3, and 4. In Graph 1, set the plot variable to 'He1Density' and again move the timeslider on the bottom right of the window. Observe that the central ion density appears to be steadily but slowly rising; from its initial value of 5.12e14 1/m^3, it rises to 6.0e14 1/m^3 by the end of our comparatively short run. In addition, a rapid decrease in density near the walls (associated with the plasma sheath) has lowered the edge densities to around 1.2e14 1/m^3. The ion density can be seen in Fig. 6.28.



VSim - Turner case 2 (text-based setup)

Data View: 1-D Fields

Plot Control

Graph 1: He1Density; Location: Window 1; Color: Red; Style: Solid

Graph 2: <None>; Location: Window 2; Color: Blue; Style: Solid

Graph 3: <None>; Location: Window 3; Color: Green; Style: Solid

Graph 4: <None>; Location: Window 3; Color: Black; Style: Solid

Annotation Level: Axes & Legends

He1Density

□ FFT □ Log Scale ○ Zoom ● Navigate

Dump: 40

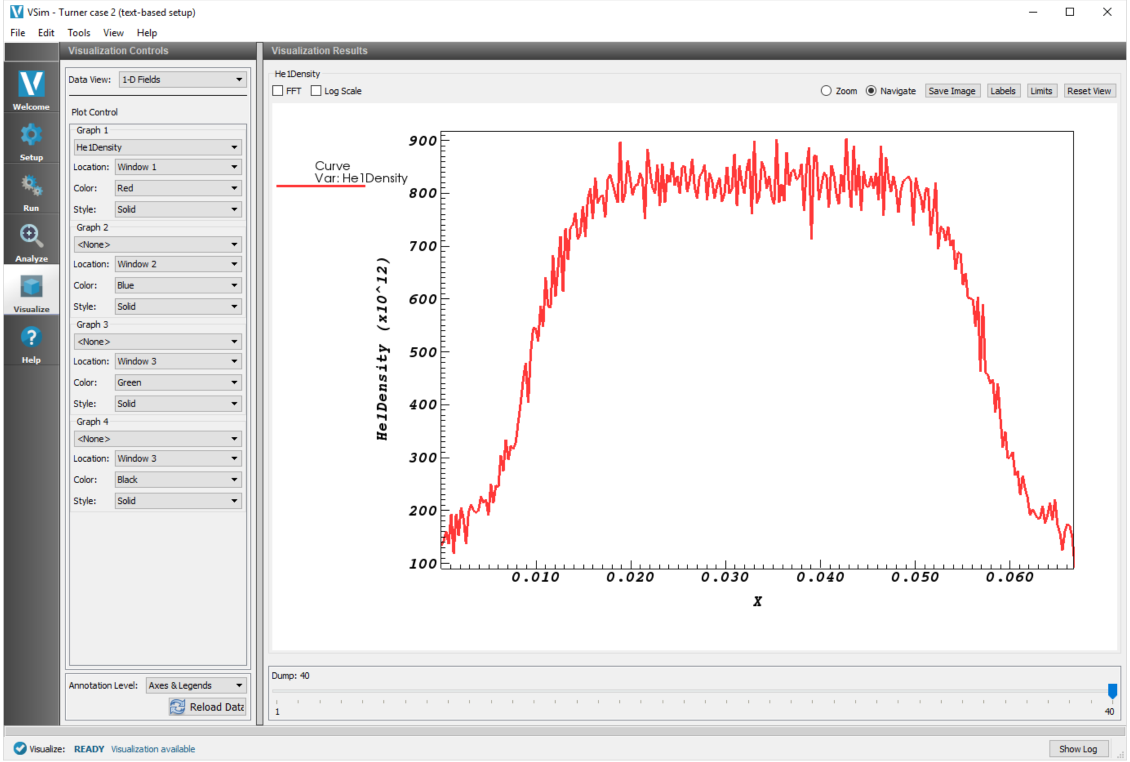
Visualize: READY Visualization available

Fig. 6.28: The He ion density at the end of the run.

Iteration 2

Now we'll create a new run which accounts for the central density buildup at the outset, by returning to the *Welcome* tab. The run we just did should appear in the list of *Recent Simulations*; we'll want to click on that entry in the list and then hit the *Copy Recent* button on the bottom right. In the *Select Files to be Copied* window, the default settings are fine; just hit *Select*, choose a name and location for your new run, and save the files. We are automatically taken to the *Setup Window*, where we can adjust the density. Let's try a value half again as big as the previous value; set PLASMA_DENS to 1.5 * 5.12e14 1/m^3 = 7.68e14 1/m^3 and Validate using the button at the upper right. Move to the *Run Window* and again adjust computation parameters (number of cores, etc.) as before before clicking *Run*.

When the simulation completes, return to the *Analyze Window* and rerun the 'computePtclNumDensity.py' script as before, inputting the new simulationName appropriate for the most recent run and again running the script for both the 'electrons' and 'He1' species. Then return to the *Visualize Window*. In the *Data View* menu, again choose *1-D Fields*, set the plot variable to '<None>' in plots 2, 3, and 4, and again select 'He1Density' in Graph 1. Moving the timeslider this time reveals that the central density continues to rise somewhat, but not as much as in the previous case. We could continue to raise PLASMA_DENS until this behavior ceases, but will forego that for the moment. We also observe that the edge density plummets sharply, ultimately nearing the 1.2e14 1/m^3 measurement we observed in the previous run. These observations suggest that a spatially dependent profile peaked at the center may be more appropriate. The ion density can be seen in Fig. 6.29.



Data View: 1-D Fields

Plot Control

Graph 1: He1Density; Location: Window 1; Color: Red; Style: Solid

Graph 2: <None>; Location: Window 2; Color: Blue; Style: Solid

Graph 3: <None>; Location: Window 3; Color: Green; Style: Solid

Graph 4: <None>; Location: Window 3; Color: Black; Style: Solid

Annotation Level: Axes & Legends

Dump: 40

Visualize: READY Visualization available

Fig. 6.29: The He ion density at the end of the run.

Iteration 3

Let's create yet another new run by returning to the *Welcome* tab and repeating the *Copy Recent* procedure we did a moment ago. When we are taken to the Setup Window, we now want to focus on the DENSITY_RAT parameter. Rather than loading the particles uniformly, we will now impose a symmetric, parabolic function (with value 1 at the domain center, and falling off to value DENSITY_RAT at the walls) as a probabilistic restriction of the particle load. Explicitly, this function has the form

$$f(x) = \frac{4x}{L}(1 - \frac{x}{L}) + D(1 - \frac{2x}{L})^2$$

where L = LENGTH (the extent of the spatial domain, as given in the input file) and D = DENSITY_RAT. The probability of generating a particle at point x is f(x); we thereby generate fewer particles near the walls. We'll choose DENSITY_RAT as 0.1667 (i.e. 1/6) to begin with and see how things evolve. Make this change and save in the *Setup Window*. In the runtime options in the *Run*, again adjust the computational parameters (number of cores, etc.) as before, and run the new simulation.

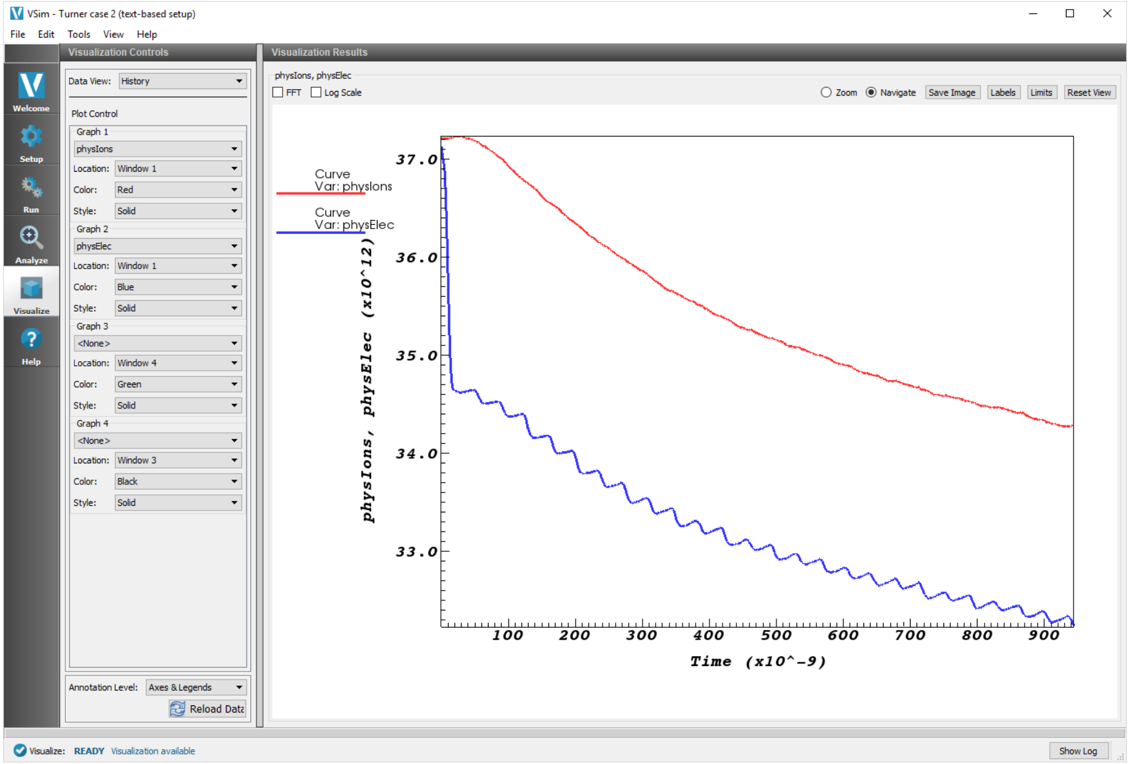
When the simulation completes, we will need to repeat a number of steps we have done before:

- Return to the *Analyze Window* and select the 'computePtclNumDensity.py' script, inputting the new simulationName appropriate for the most recent run and again running the script for both the 'electrons' and 'He1' species
- Return to the *Visualize Window*, again choosing *1-D Fields* in the *Data View* menu, setting the plot variable to '<None>' in plots 2, 3, and 4, and selecting 'He1Density' in Graph 1.

As we move the timeslider, the evolution of the ion density profile is now much less pronounced, suggesting that our initial particle loading more closely resembles the steady-state profile. Yet if we switch the *Data View* menu at the top left to look at *History*, we can see that the particle densities have not yet come to steady-state. This is done by again repeating steps we have performed previously:

- Set the plot variable to '<None>' in plots 3 and 4
- Set the plot variable to 'physIons' and 'physElec' in plots 1 and 2 respectively
- Set the *Location* variable in plots 1 and 2 to the same value, e.g. 'Window 1'.

Now we can see that both ion and electron densities decrease slightly as the simulation proceeds; a slight excess of plasma density is slowly being transported out of the system. This excess will persist until a balance is struck between the production rate and the rate of wall losses associated with a given profile; we are slightly over-predicting the density production and the plasma thus adjusts itself in such a way that more particles are expelled. The electron and ion populations can be seen in Fig. 6.30.



Data View: History

Plot Control

Graph 1
physIons
Location: Window 1
Color: Red
Style: Solid

Graph 2
physElec
Location: Window 1
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 4
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 3
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 6.30: The electron and ion populations versus time.

Lowering the PLASMA_DENS parameter to reduce the initial particle count is one way of dealing with this problem, but a more elegant solution which takes advantage of the more rapid physics processes occurring near the walls can also improve our initial particle loading. Recall that the initial behavior of the plasma, even before the ambipolar phase, centers on the establishment of the plasma sheath and the strong electric fields associated with the sheath. This behavior happens on a timescale much faster than any process in the ambipolar phase, and is characterized by high electron losses to the wall.

Iteration 4

Suppose that rather than loading particles all the way to the wall, we reduce the overall plasma density by leaving a gap near the wall where neither electrons nor ions are present. What happens in the discharge? The electrons, being highly mobile, rush to fill the gap, but rather than immediately being lost to the wall, they instead produce strong electric fields at the plasma edge which begin to modify the ion profile and bring about ambipolarity. If the gap is sufficiently large, the collisional production of ions and electrons will begin before appreciable wall losses ensue, and we can thus assess the relative rates of production and loss fairly early in the simulation. Since the electrons are highly mobile, let's treat the average electron population as a measure of how well we've achieved this balance; net electron production as we move into the ambipolar phase means that our gap is too low (we have removed too much density), while net losses mean that our gap is insufficiently wide. As the profile shapes near the walls tend to adjust themselves fairly quickly (due to the larger electric fields in this region), we can in this manner obtain approximately correct values for the total ion and electron populations at the simulation outset.

Return to the *Welcome* tab and create another new run, using the run we just carried out as a basis. In the *Setup Window*, the parameter 'SHEATH_GAP' describes the physical width of the particle-deficient region we'll introduce near the walls. We'll need to do a bit of work since our profile is now parabolic. Recall that the probability function we're using for the parabolic load has the form

$$f(x) = \frac{4x}{L}(1 - \frac{x}{L}) + D(1 - \frac{2x}{L})^2$$

where L=LENGTH and D=DENSITY_RAT. If we average f(x) across the simulation domain, we obtain

$$< f(x) >= \int_0^L \frac{f(x)}{L} dx = 1 - \frac{(1 - D)}{3}$$

The function <f(x)> is proportional to the average species population; introducing a gap of width S = SHEATH_GAP to reduce this average population modifies this calculation such that

$$< f(x) >= \int_S^{L-S} \frac{f(x)}{L} dx = (1 - \frac{2S}{L}) - (1 - \frac{2S}{L})^3 \frac{(1 - D)}{3}$$

Assuming we want the new <f(x)> to equal some fraction B of the original population, we can find the appropriate sheath gap by numerically solving the cubic equation

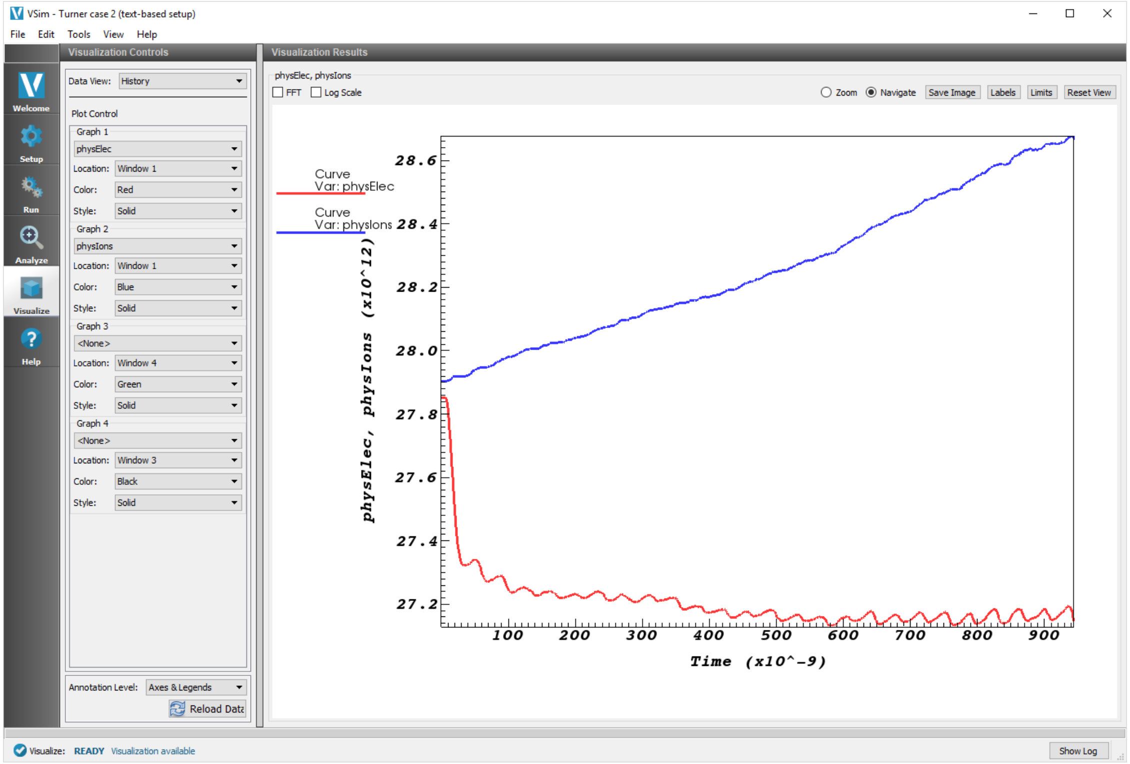
$$B(1 - \frac{(1 - D)}{3}) = (1 - \frac{2S}{L}) - (1 - \frac{2S}{L})^3 \frac{(1 - D)}{3}$$

for the SHEATH_GAP parameter S. (Alternatively, we could just make a guess for a sensible value and adjust accordingly.) For our case, with D = 1/6; letting B = 3/4 yields three real solutions to the cubic, but one is negative (and thus unphysical) while another is greater than L (and thus also unphysical). The remaining solution yields a value 0.0133 for SHEATH_GAP, so let's try that as a starting value.

- Run the simulation with the modified SHEATH_GAP parameter, with other parameters the same as were used previously
- Return to the *Analyze Window* and select the 'computePtclNumDensity.py' script, inputting the new simulationName appropriate for the most recent run and again running the script for both the 'electrons' and 'He1' species
- Return to the *Visualize Window*, again choosing *History* in the *Data View* menu, setting the plot variable to '<None>' in plots 3, and 4, and selecting 'physIons' and 'physElec' in plots 1 and 2 while giving the *Location* variable the same value.

The electron and ion populations can be seen in Fig. 6.31.

The average electron population, after the initial wall losses, remains relatively constant, so we have struck a good balance between ionization (an electron source) and wall loss (an electron sink). Modified loading techniques have enabled us to observe and balance the rapid physics processes which ensue as electrons rush to fill our imposed sheath gap S. Such balancing hastens the ultimate convergence of the simulation to steady-state.



Data View: History

Plot Control

Graph 1
physElec
Location: Window 1
Color: Red
Style: Solid

Graph 2
physIons
Location: Window 1
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 4
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 3
Color: Black
Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 6.31: The electron and ion populations versus time.

6.3 DC Plasmas

6.3.1 Drifting Electrons (driftingElectrons.sdf)

Keywords:

electron transport, electron mobility, monte carlo, electrostatic

Problem description

VSim may be used to model charged particles drifting in a background neutral gas. When charged particles, such as electrons, are injected into a background neutral gas, collisions between gas atoms and electrons eventually lead to thermal equilibrium, and electrons will reach the same temperature as the background gas. However, when an external electric field is applied across the neutral gas, the electron collisions and distribution will change due to this applied field. Electrons will gain energy from the applied electric field. The energy loss due to electron-atom collision is small, and most of the energy ends up heating the electrons. Assuming only elastic collisions take place between electrons and atoms, the electron mobility is defined as

$$\mu_e = (\frac{\pi\lambda}{2mE})^{\frac{1}{2}}$$

which describes the relation between electron drifting velocity and applied electric field.

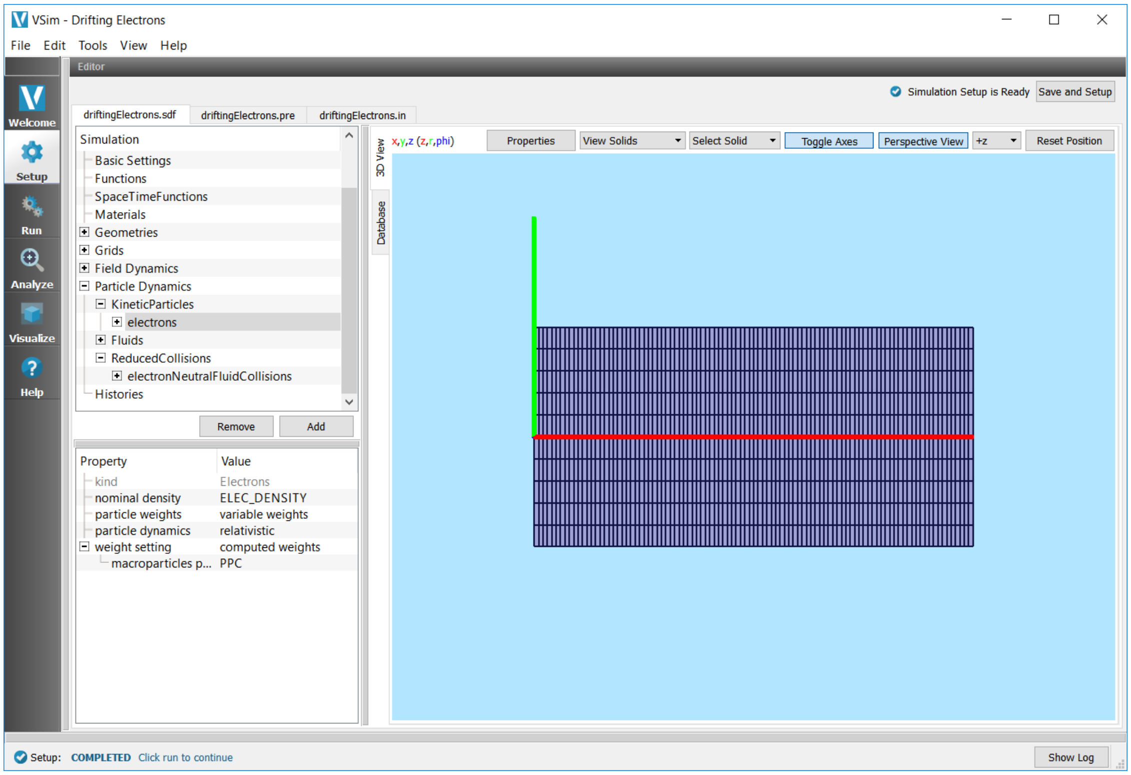
This simulation can be performed with a VSimPD license.

Opening the Simulation

The Electron Drifting example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Other PD* option.
- Select “Electron Drifting” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is now shown with all the implemented physics and geometries, if applicable. See Fig. 6.32.



Property	Value
kind	Electrons
nominal density	ELEC_DENSITY
particle weights	variable weights
particle dynamics	relativistic
weight setting	computed weights
macroparticles p...	PPC

Setup: COMPLETED Click run to continue

Fig. 6.32: Setup Window for the Electron Drifting example.

Simulation Properties

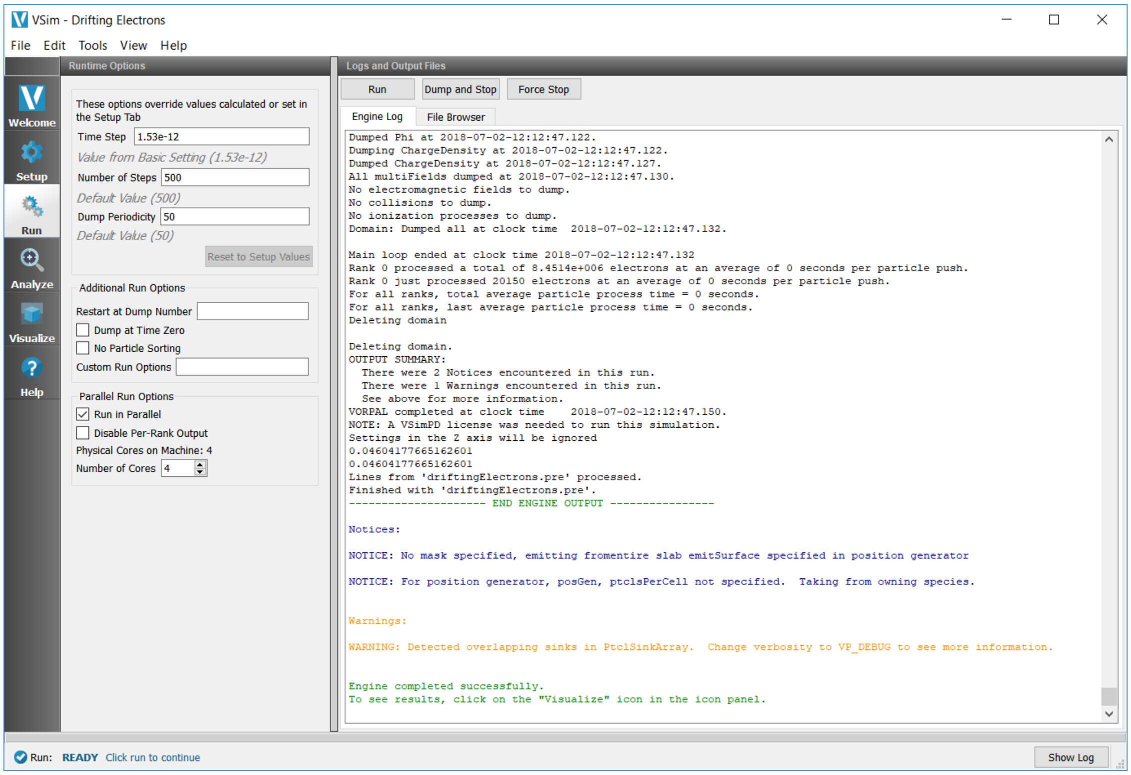
This input file contains electron as kinetic species as well as a background fluid description of a gas. Elastic collisions between kinetic particles and the background gas are described by Monte Carlo interaction blocks of kind impactIonization.

The fields are solved for electrostatically at each time step, including the fields due to all charged particles, subject to the boundary conditions specified in the input file. There are a number of histories that record the number of particles for electrons.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in this pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in the window below.



These options override values calculated or set in the Setup Tab

Time Step 1.53e-12

Value from Basic Setting (1.53e-12)

Number of Steps 500

Default Value (500)

Dump Periodicity 50

Default Value (50)

Additional Run Options

Restart at Dump Number

☐ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
Dumped Phi at 2018-07-02-12:12:47.122.
Dumping ChargeDensity at 2018-07-02-12:12:47.122.
Dumped ChargeDensity at 2018-07-02-12:12:47.127.
All multiFields dumped at 2018-07-02-12:12:47.130.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-02-12:12:47.132.

Main loop ended at clock time 2018-07-02-12:12:47.132
Rank 0 processed a total of 8.4514e+006 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 20150 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 2 Notices encountered in this run.
  There were 1 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-07-02-12:12:47.150.
NOTE: A VSimPD license was needed to run this simulation.
Settings in the Z axis will be ignored
0.04604177665162601
0.04604177665162601
Lines from 'driftingElectrons.pre' processed.
Finished with 'driftingElectrons.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: No mask specified, emitting fromentire slab emitSurface specified in position generator

NOTICE: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.


Warnings:

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: READY Click run to continue

Fig. 6.33: The Run Window at the end of execution.

Visualizing the Results

After run completion, continue as follows:

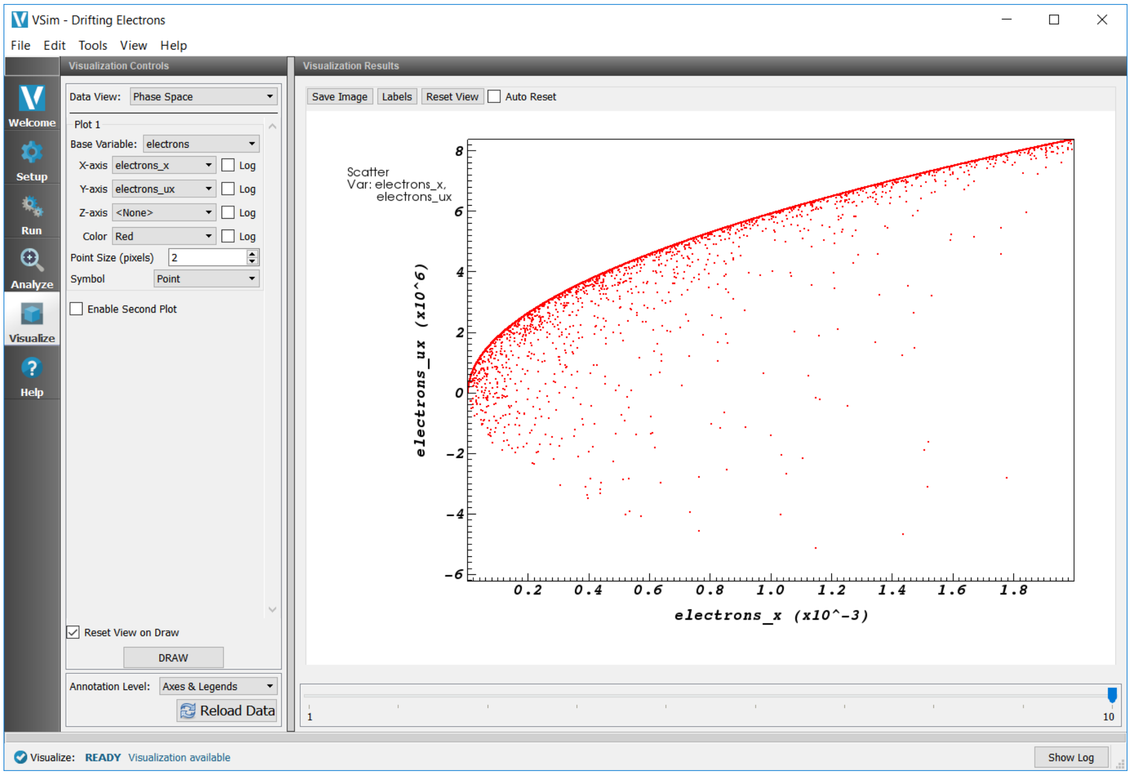
- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the phase space distribution for the drifting electrons, select *Phase Space* from the drop down *Data View* menu. In *Base Variable*, select *electrons*. Select *electrons_x* for *X-axis* and *electrons_ux* for *Y-aixs*. Click *DRAW* and move the *Dump* slider to view electron accelerating and scattering when they drift over the space. The electron phase space at dump number 10 is shown in Fig. 6.3.1.

Further Experiments

At lower applied electric fields, electrons are more collisional due to increased cross section. Try reducing the CATHODE_POTENTIAL, and observe more scattered electron distribution when drift over space.

At higher applied electric fields, not only elastic collisions, but also inelastic collisions will take place between electrons and atoms, which further reduce electron drifting velocity and mobility. For further experiments, try adding other collision types, such as excitation and ionization, and observe the effects to electron drifting velocity.



Data View: Phase Space

Plot 1

Base Variable: electrons

X-axis electrons_x □ Log

Y-axis electrons_ux □ Log

Z-axis <None> □ Log

Color Red □ Log

Point Size (pixels) 2

Symbol Point

□ Enable Second Plot

☑ Reset View on Draw

DRAW

Annotation Level: Axes & Legends

Reload Data

Visualize: READY Visualization available

6.3.2 Langmuir Probe (langmuirProbe.sdf)

Keywords:

electrostatics, particle in cell, sheath, box bounding, internal boundary

Problem description

This example computes the fields and particles in a box, with an interior probe, modeled as a particle absorber and a constant-voltage (Dirichlet) boundary condition. There is an immobile, background neutralizing charge density. The electrons move to the walls and the probe, creating sheaths at all interfaces.

This simulation can be performed with a VSimBase license.

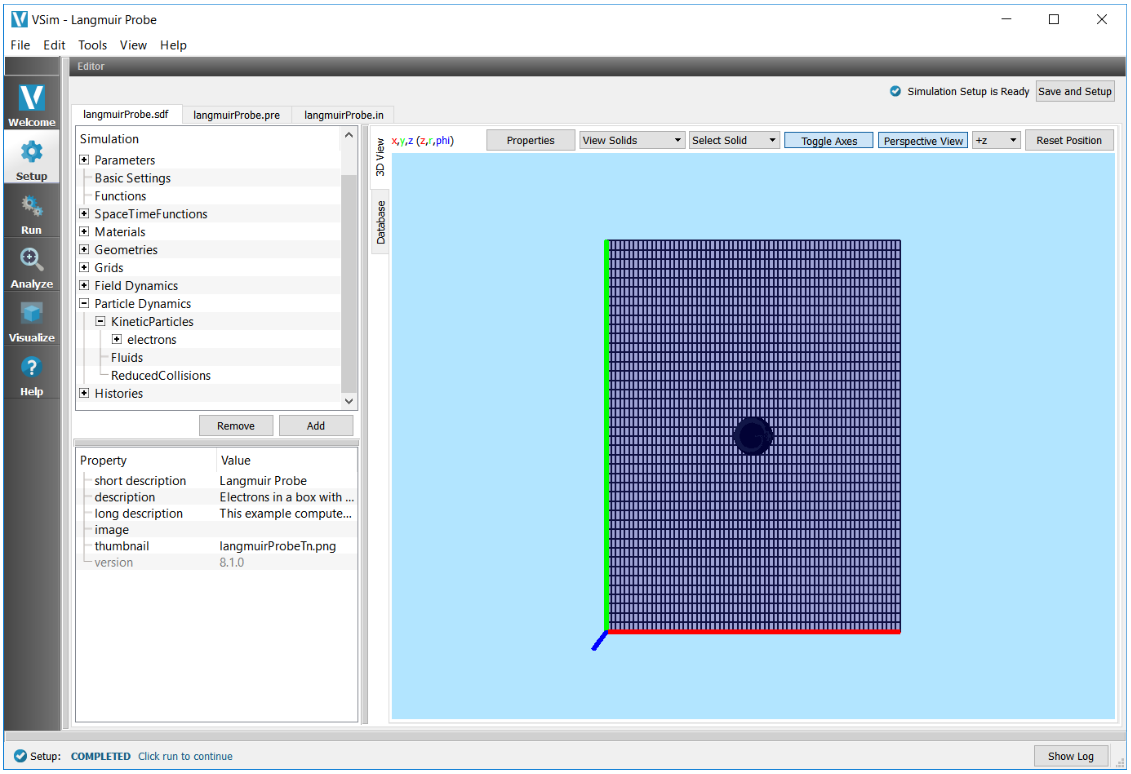
Opening the Simulation

The Langmuir Probe example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Other PD* option.
- Select “Langmuir Probe” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 6.34. You can expand the tree elements and navigate through the various properties, making any changes you desire.

The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to Grid.



Simulation Setup is Ready

- Simulation
  - Parameters
  - Basic Settings
  - Functions
  - SpaceTimeFunctions
  - Materials
  - Geometries
  - Grids
  - Field Dynamics
  - Particle Dynamics
    - KineticParticles
      - electrons
    - Fluids
    - ReducedCollisions
  - Histories

Property	Value
short description	Langmuir Probe
description	Electrons in a box with ...
long description	This example compute...
image	
thumbnail	langmuirProbeTn.png
version	8.1.0

Setup: COMPLETED Click run to continue

Fig. 6.34: Setup Window for the Langmuir Probe example.

Simulation Properties

Constants are set up to allow setting the electron temperature in eV (ELEC_TEMP_EV), the electron density (NOM_DENS_E), the number of cells (NCELLS_X, NCELLS_Y) in the x and y directions, the number of particles per cell (PPC), and the size of the simulation (LEN_X, LEN_Y) in the x and y directions.

Running the Simulation

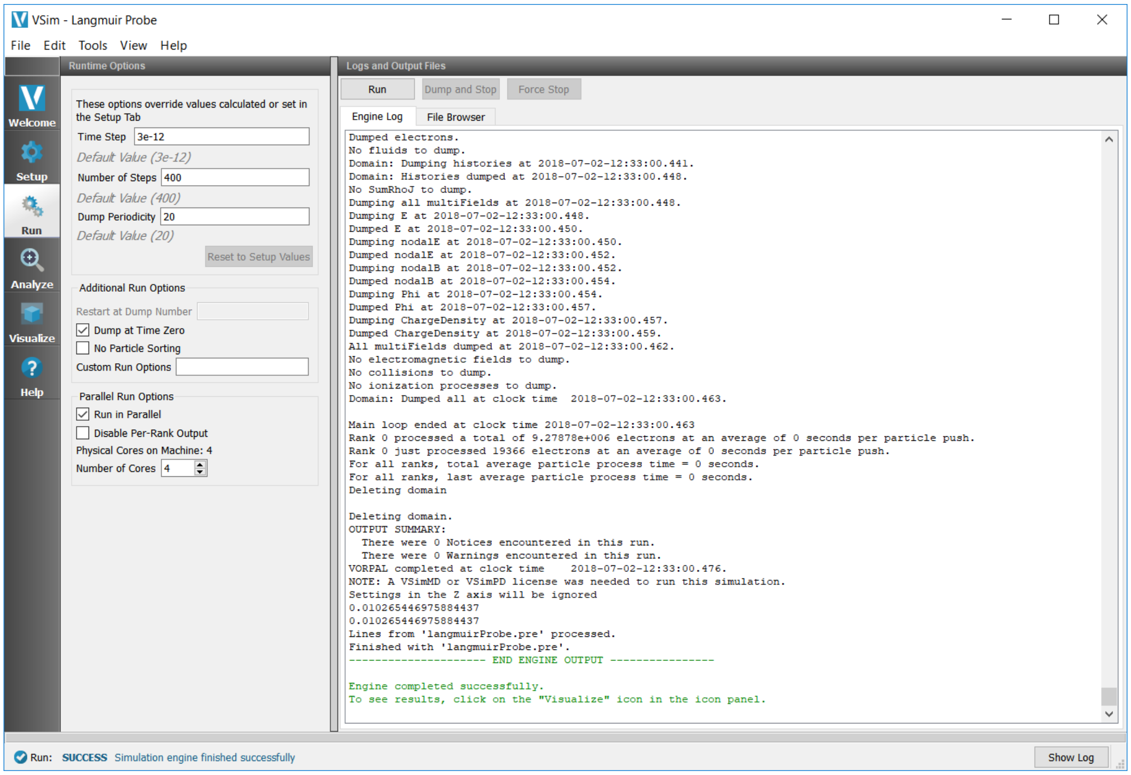
After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in this pane. The run has completed when you see the output, “Engine completed successfully.” A snapshot of the simulation run completion is shown in Fig. 6.35.

Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.



These options override values calculated or set in the Setup Tab

Time Step: 3e-12
Default Value (3e-12)
Number of Steps: 400
Default Value (400)
Dump Periodicity: 20
Default Value (20)

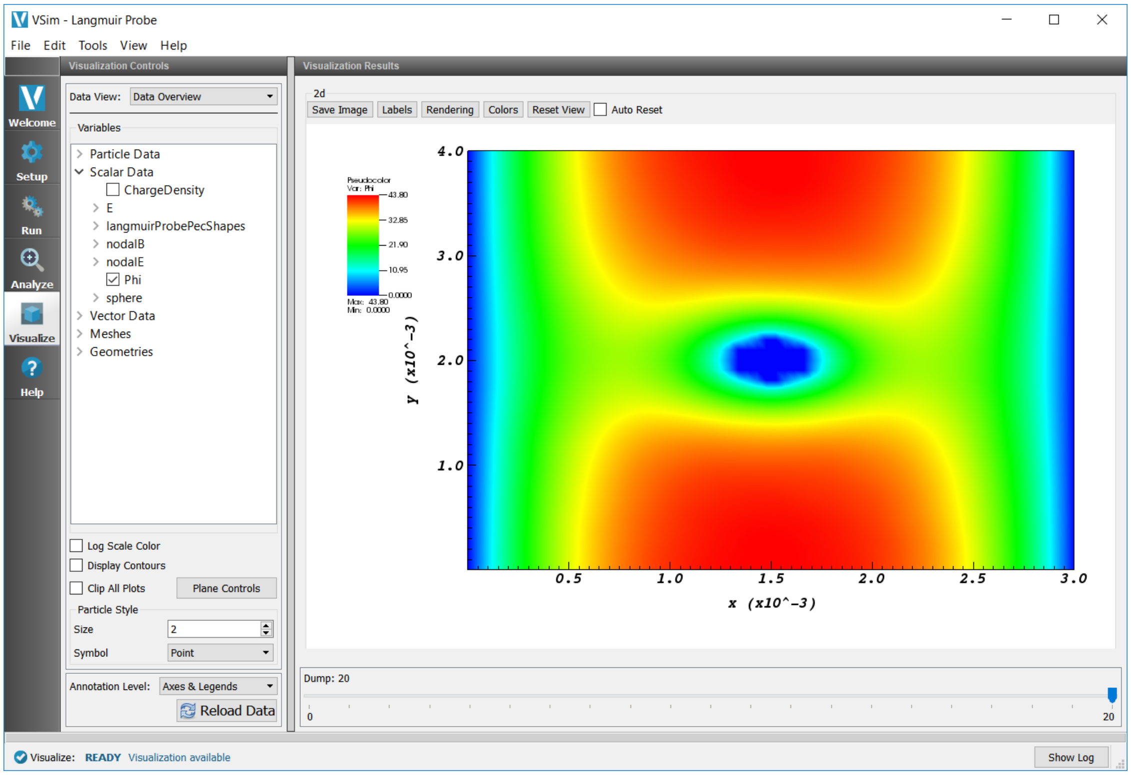
Additional Run Options
Restart at Dump Number:
☑ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options:

Parallel Run Options
☑ Run in Parallel
☐ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores: 4

```
Dumped electrons.
No fluids to dump.
Domain: Dumping histories at 2018-07-02-12:33:00.441.
Domain: Histories dumped at 2018-07-02-12:33:00.448.
No SumRhoJ to dump.
Dumping all multiFields at 2018-07-02-12:33:00.448.
Dumping E at 2018-07-02-12:33:00.448.
Dumped E at 2018-07-02-12:33:00.450.
Dumping nodalE at 2018-07-02-12:33:00.450.
Dumped nodalE at 2018-07-02-12:33:00.452.
Dumping nodalB at 2018-07-02-12:33:00.452.
Dumped nodalB at 2018-07-02-12:33:00.454.
Dumping Phi at 2018-07-02-12:33:00.454.
Dumped Phi at 2018-07-02-12:33:00.457.
Dumping ChargeDensity at 2018-07-02-12:33:00.457.
Dumped ChargeDensity at 2018-07-02-12:33:00.459.
All multiFields dumped at 2018-07-02-12:33:00.462.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-02-12:33:00.463.

Main loop ended at clock time 2018-07-02-12:33:00.463
Rank 0 processed a total of 9.27878e+006 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 19366 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-07-02-12:33:00.476.
NOTE: A VSimMD or VSimPD license was needed to run this simulation.
Settings in the Z axis will be ignored
0.0102654469758844437
0.0102654469758844437
Lines from 'langmuirProbe.pre' processed.
Finished with 'langmuirProbe.pre'.
-------------------- END ENGINE OUTPUT ---------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 6.35: The Run Window at the end of execution.



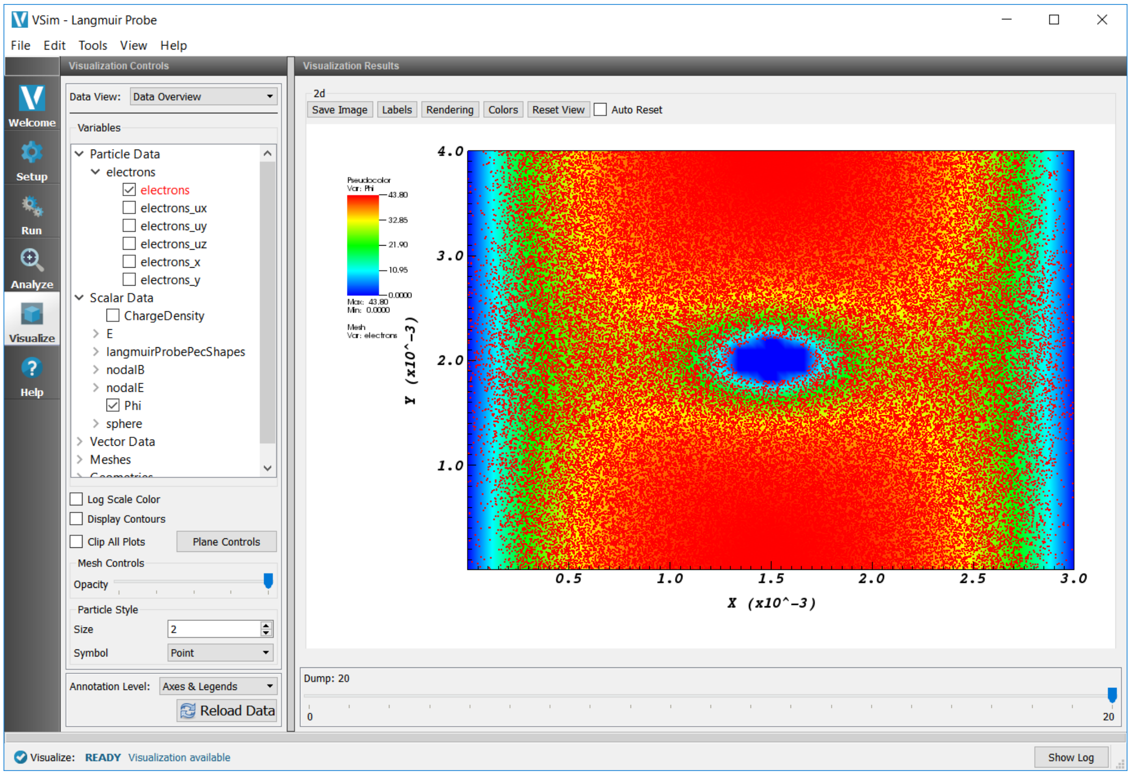
Data View: Data Overview
☐ ChargeDensity
☑ Phi
☐ Log Scale Color
☐ Display Contours
☐ Clip All Plots
Size: 2
Symbol: Point
Annotation Level: Axes & Legends
☐ Auto Reset
Dump: 20

Visualize: READY Visualization available

Fig. 6.36: The electrostatic potential

To view the electric potential, expand *Scalar Data* and select *Phi*. The potential in the visualization window resembles that shown in Fig. 6.36.

To view the electrons and sheaths, expand the *Particle Data*, expand *electrons* and select *electrons*. Move the dump slider forward in time to see the formation of the sheaths as seen in Fig. 6.37.



Data View: Data Overview

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

Dump: 20

Visualize: READY Visualization available

Fig. 6.37: The sheath formation

Further Experiments

Try adding in another geometry for inclusion of the support rod or try changing the geometry to represent a different probe.

6.4 Ion Sources

6.5 Processes

6.5.1 Laser Ionization (laserIonization.sdf)

Keywords:

`electromagnetic, particle in cell, field ionization, moving window`

Problem description

This example launches an electromagnetic laser pulse into a homogeneous volume of neutral argon gas. The field strength is significant enough to ionize the argon to multiple ionization states, which are included in the simulation. The neutral gas density is depleted as the ionization occurs, with layers of argon atoms at increasing ionization levels towards the center of the Gaussian beam.

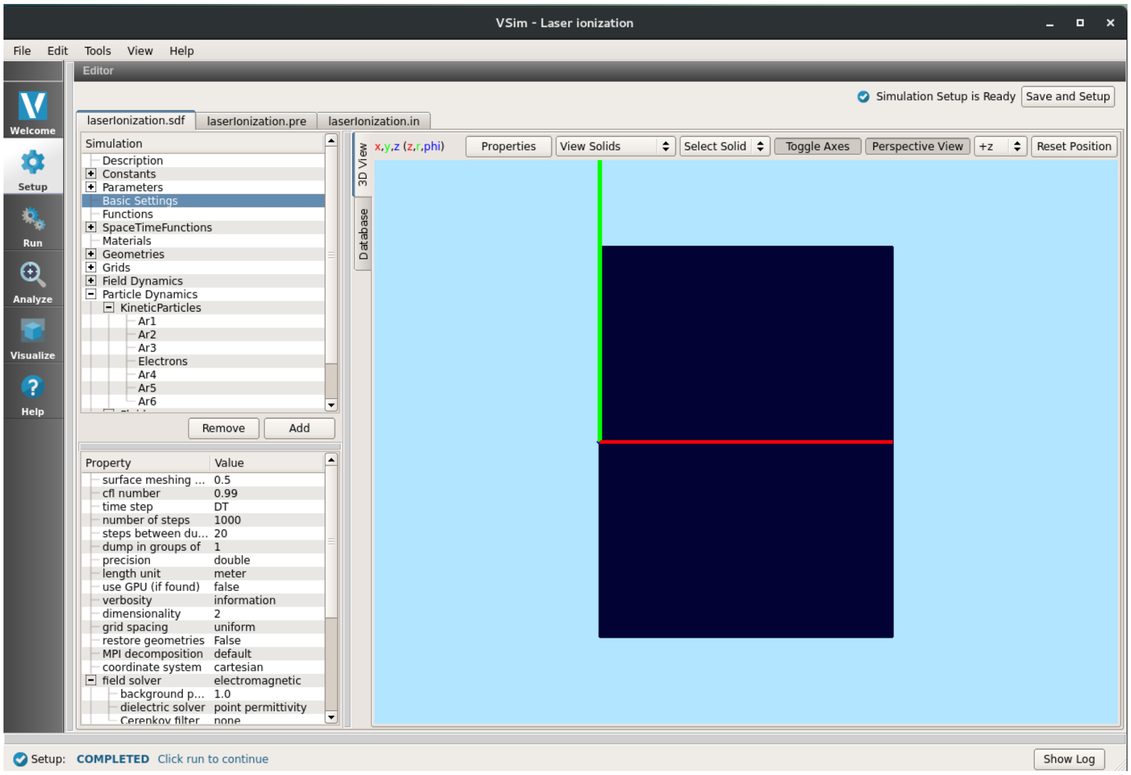
This simulation can be performed with a VSimPD license.

Opening the Simulation

The Laser Ionization example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Processes* option.
- Select "Laser Ionization" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 6.38. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to Grid.



Property	Value
surface meshing …	0.5
cfl number	0.99
time step	DT
number of steps	1000
steps between du…	20
dump in groups of	1
precision	double
length unit	meter
use GPU (if found)	false
verbosity	information
dimensionality	2
grid spacing	uniform
restore geometries	False
MPI decomposition	default
coordinate system	cartesian
field solver	electromagnetic
background p…	1.0
dielectric solver	point permittivity
Cerenkov filter	none

Setup: COMPLETED Click run to continue

Fig. 6.38: Setup Window for the Laser Ionization example.

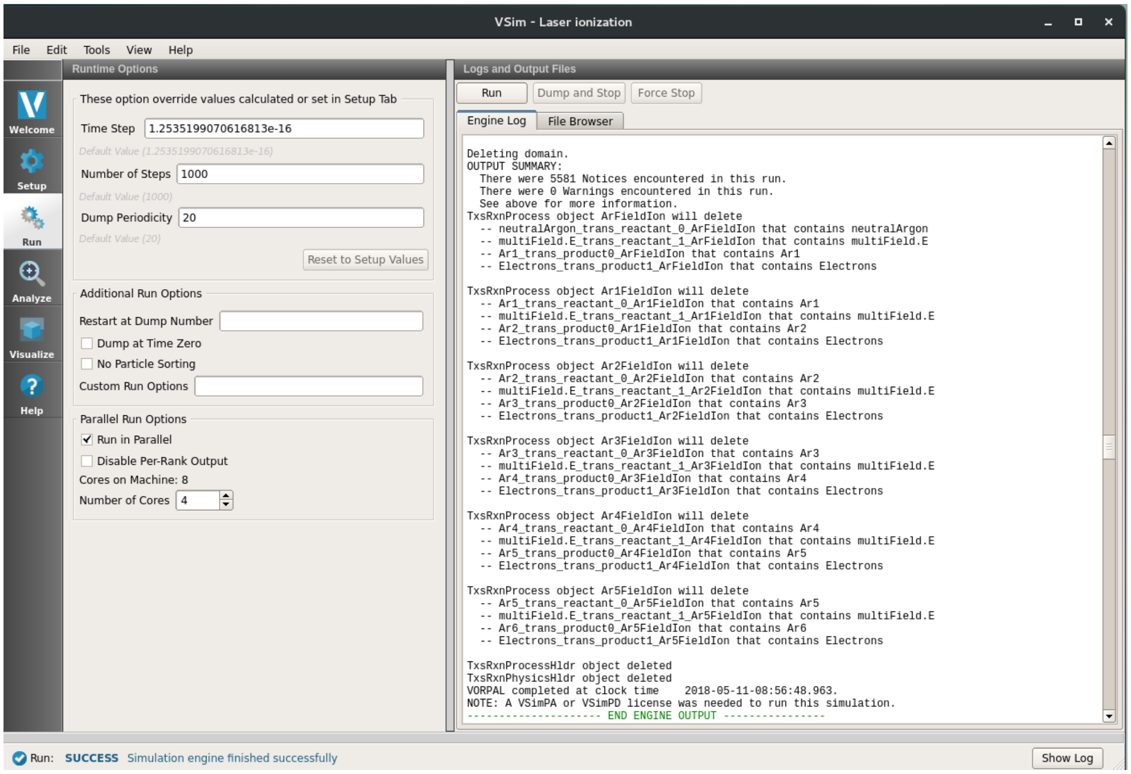
Simulation Properties

Constants are set up to allow setting the laser amplitude and the neutral argon density (1/m^3).

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." A snapshot of the simulation run completion is shown in Fig. 6.39.



These option override values calculated or set in Setup Tab

Time Step 1.2535199070616813e-16
Default Value (1.2535199070616813e-16)
Number of Steps 1000
Default Value (1000)
Dump Periodicity 20
Default Value (20)

Additional Run Options

Restart at Dump Number
□ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
□ Disable Per-Rank Output
Cores on Machine: 8
Number of Cores 4

```
Deleting domain.
OUTPUT SUMMARY:
  There were 5581 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
  See above for more information.
TxsRxnProcess object ArFieldIon will delete
  -- neutralArgon_trans_reactant_0_ArFieldIon that contains neutralArgon
  -- multiField.E_trans_reactant_1_ArFieldIon that contains multiField.E
  -- Ar1_trans_product0_ArFieldIon that contains Ar1
  -- Electrons_trans_product1_ArFieldIon that contains Electrons

TxsRxnProcess object Ar1FieldIon will delete
  -- Ar1_trans_reactant_0_Ar1FieldIon that contains Ar1
  -- multiField.E_trans_reactant_1_Ar1FieldIon that contains multiField.E
  -- Ar2_trans_product0_Ar1FieldIon that contains Ar2
  -- Electrons_trans_product1_Ar1FieldIon that contains Electrons

TxsRxnProcess object Ar2FieldIon will delete
  -- Ar2_trans_reactant_0_Ar2FieldIon that contains Ar2
  -- multiField.E_trans_reactant_1_Ar2FieldIon that contains multiField.E
  -- Ar3_trans_product0_Ar2FieldIon that contains Ar3
  -- Electrons_trans_product1_Ar2FieldIon that contains Electrons

TxsRxnProcess object Ar3FieldIon will delete
  -- Ar3_trans_reactant_0_Ar3FieldIon that contains Ar3
  -- multiField.E_trans_reactant_1_Ar3FieldIon that contains multiField.E
  -- Ar4_trans_product0_Ar3FieldIon that contains Ar4
  -- Electrons_trans_product1_Ar3FieldIon that contains Electrons

TxsRxnProcess object Ar4FieldIon will delete
  -- Ar4_trans_reactant_0_Ar4FieldIon that contains Ar4
  -- multiField.E_trans_reactant_1_Ar4FieldIon that contains multiField.E
  -- Ar5_trans_product0_Ar4FieldIon that contains Ar5
  -- Electrons_trans_product1_Ar4FieldIon that contains Electrons

TxsRxnProcess object Ar5FieldIon will delete
  -- Ar5_trans_reactant_0_Ar5FieldIon that contains Ar5
  -- multiField.E_trans_reactant_1_Ar5FieldIon that contains multiField.E
  -- Ar6_trans_product0_Ar5FieldIon that contains Ar6
  -- Electrons_trans_product1_Ar5FieldIon that contains Electrons

TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-05-11-08:56:48.963.
NOTE: A VSimPA or VSimPD license was needed to run this simulation.
-------------------- END ENGINE OUTPUT ----------------
```

Run: SUCCESS Simulation engine finished successfully

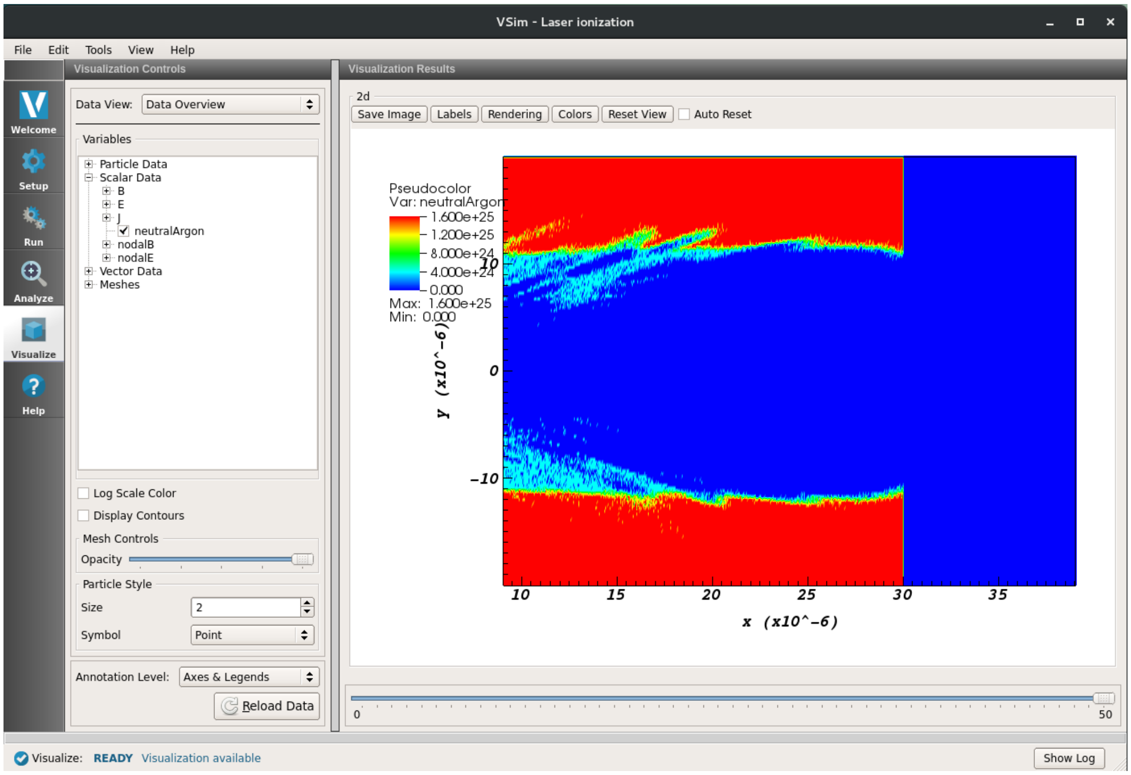
Fig. 6.39: The Run Window at the end of execution.

Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To view the electric field magnitude, expand *Scalar Data*, expand *E* and select *E_magnitude*. Scrolling through time (by moving the slider at the bottom of the window) will show the laser pulse propataing across the simluation domain. Next, untick the *E_magnitude* and instead tick *neutralArgon*. You can now see the depletion of the neutral background gas as the laser passes through. This will appear the same as in Fig. 6.40.



Data View: Data Overview

☑ neutralArgon

□ Log Scale Color

□ Display Contours

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

Var: neutralArgon

Max: 1.600e+25
Min: 0.000

Visualize: READY Visualization available

Fig. 6.40: The ionized charge states of argon during laser pulse propagation

Further Experiments

Try adding more charge states of Argon (past 6+) and find the limit of ionization that is achievable with this laser pulse.

6.5.2 Negative Ion Beam (negativeIonBeam.sdf)

Keywords:

`ion beam, beam transport, reactions, electrostatic`

Problem description

VSim may be used to model ion beam transport and particle dynamics where the beam is represented by kinetic simulation particles. Low density background gasses can cause instabilities in the beams due to collisions between the beam particles and the background gas.

In this simulation, a beam of H- ions propagates through a background H2 gas. Collisions between the beam ions and the background gas produce electrons, H2+, and neutral H through the following reactions:

$$H^- + H_2 \to H^- + H_2^+ + e^- \text{ (ion impact ionization)}$$

$$e^- + H_2 \to H_2^+ + 2e^- \text{ (electron impact ionization)}$$

$$H^- + H_2 \to H + H_2 + e^- \text{ (detachment)}$$

$$H^- + e^- \to H + 2e^- \text{ (stripping)}$$

There are other reactions that are not included in this tutorial simulation. Typically these reactions have low cross sections. Fig. 6.41 shows the cross sections for the above reactions as a function of incident energy.

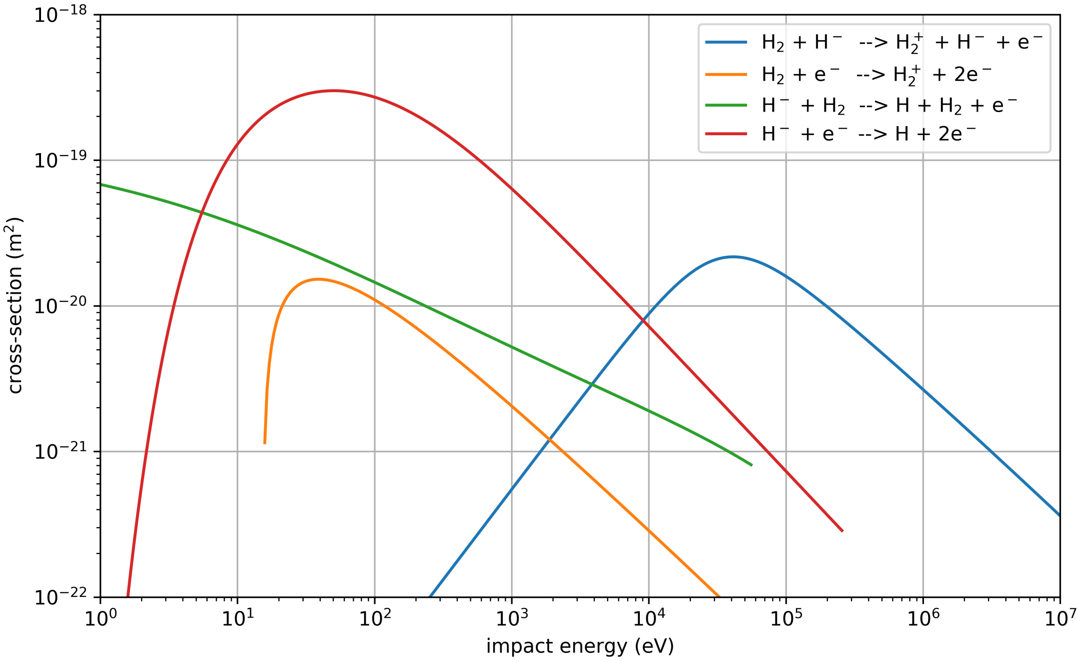


Fig. 6.41: Cross sections for the four collision reactions included in this example.

This simulation can be performed with a VSimPD license.

Opening the Simulation

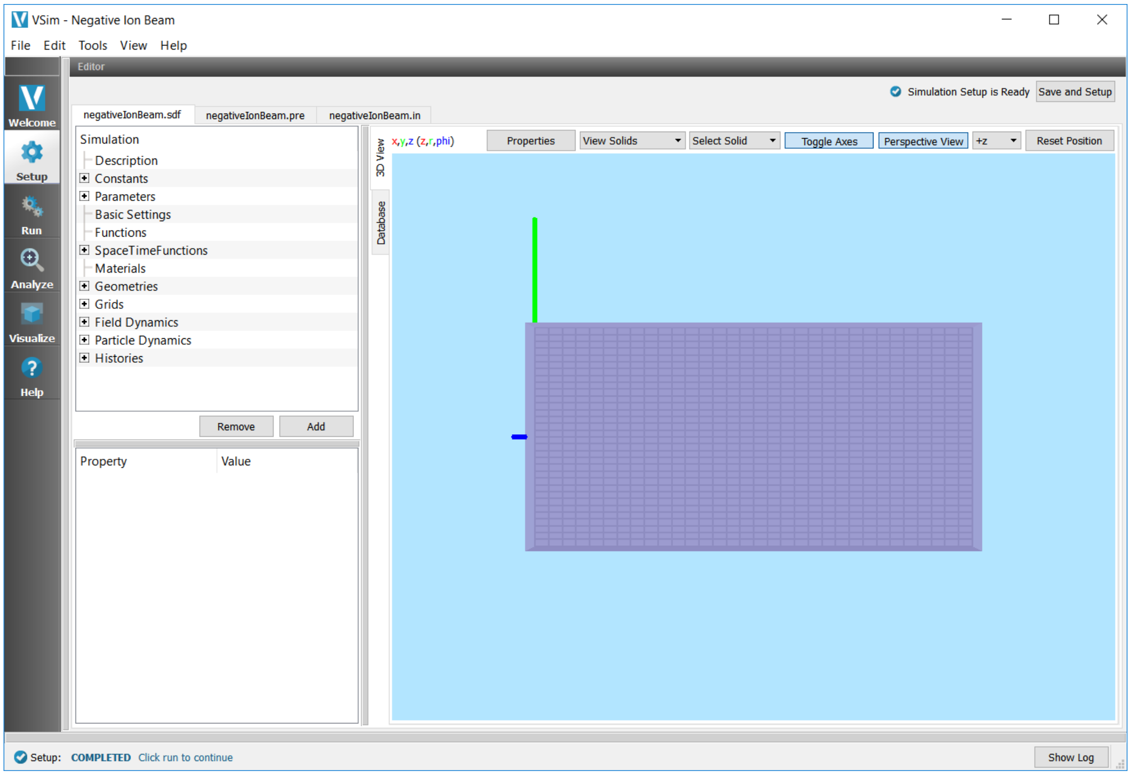
The Kinetic Collisions example is accessed from within VSimComposer by the following actions:

- Select the *New → From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Processes* option.
- Select “Negative Ion Beam” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is now shown with all the implemented physics and geometries, if applicable. See Fig. 6.42.

Simulation Properties

This input file contains a number of different kinetic species as well as a background fluid description of a gas. Ionization collisions between kinetic particles and the background gas are described by Monte Carlo interaction blocks of kind impactIonization, and detachment of electrons due to a collision with the background gas are of kind negativeIonDetachment. Collisions between kinetic particles and other kinetic particles are described in the input file by an interaction of kind binaryIonization.



Simulation Setup is Ready

Setup: COMPLETED Click run to continue

Fig. 6.42: Setup Window for the Negative Ion Beam example.

The fields are electrostatically solved for at each time step, including the fields due to all charged particles, subject to the boundary conditions specified in the input file. There are a number of histories that record the number of particles for different species, their energies, as well as currents absorbed at the boundaries.

Running the Simulation

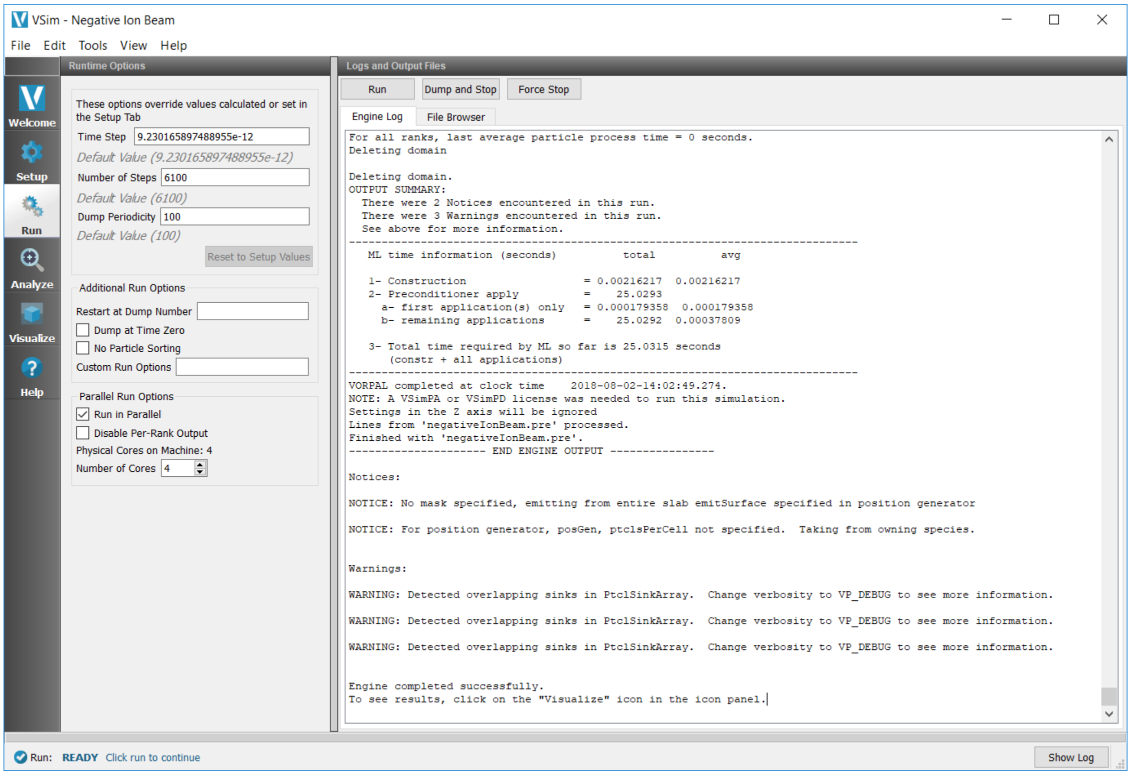
After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in this pane. The run has completed when you see the output, "Engine completed successfully." This is shown in the window below.

Analyzing the Results

If it is desired to calculate the density of the electrons the analysis script *computePtclNumDensity.py* must be used.

- First click on the *Analyze* Tab.
- Click *Show All Analyzers* and choose *computePtclNumDensity.py*. Then click *Open* at the bottom of the *Analysis Controls* pane.
- Ensure that the "simulationName" field is "negativeIonBeam" and enter "Electrons" in the "speciesName" field. Leave the "aveNxN" and "iterateAve" with their default values.
- Press the *Analyze* button on in the upper right corner of the window to run the analysis. Below, the Analyze Tab is shown at the end of a successful run.



These options override values calculated or set in the Setup Tab

Time Step 9.230165897488955e-12

Default Value (9.230165897488955e-12)

Number of Steps 6100

Default Value (6100)

Dump Periodicity 100

Default Value (100)

Additional Run Options

Restart at Dump Number

□ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

□ Disable Per-Rank Output

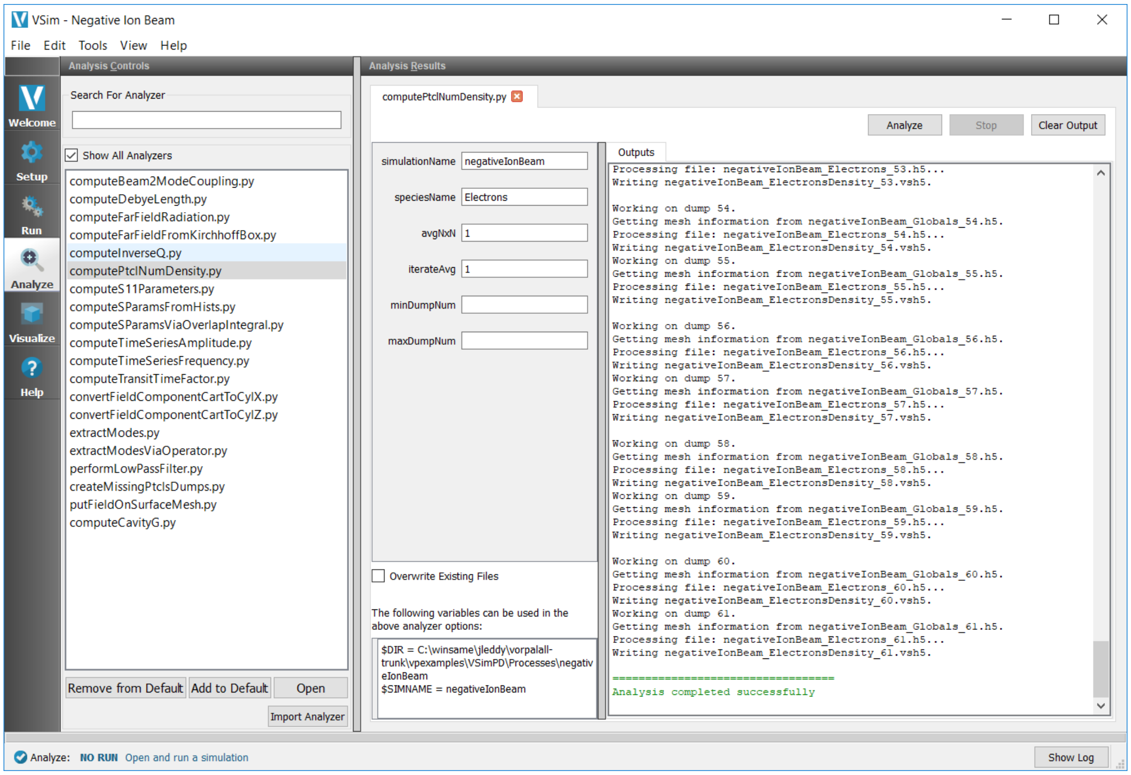
Physical Cores on Machine: 4

Number of Cores 4

```
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 2 Notices encountered in this run.
  There were 3 Warnings encountered in this run.
  See above for more information.
------------------------------------------------------------------------------
   ML time information (seconds)          total          avg

   1- Construction                  = 0.00216217  0.00216217
   2- Preconditioner apply          =    25.0293
     a- first application(s) only   = 0.000179358  0.000179358
     b- remaining applications      =    25.0292  0.00037809

   3- Total time required by ML so far is 25.0315 seconds
      (constr + all applications)
------------------------------------------------------------------------------
VORPAL completed at clock time    2018-08-02-14:02:49.274.
NOTE: A VSimPA or VSimPD license was needed to run this simulation.
Settings in the Z axis will be ignored
Lines from 'negativeIonBeam.pre' processed.
Finished with 'negativeIonBeam.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: No mask specified, emitting from entire slab emitSurface specified in position generator

NOTICE: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.


Warnings:

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **READY** Click run to continue

Fig. 6.43: The Run Window at the end of execution.



Search For Analyzer

☑ Show All Analyzers

computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
createMissingPtclsDumps.py
putFieldOnSurfaceMesh.py
computeCavityG.py

simulationName negativeIonBeam

speciesName Electrons

avgNxN 1

iterateAvg 1

minDumpNum

maxDumpNum

□ Overwrite Existing Files

The following variables can be used in the above analyzer options:

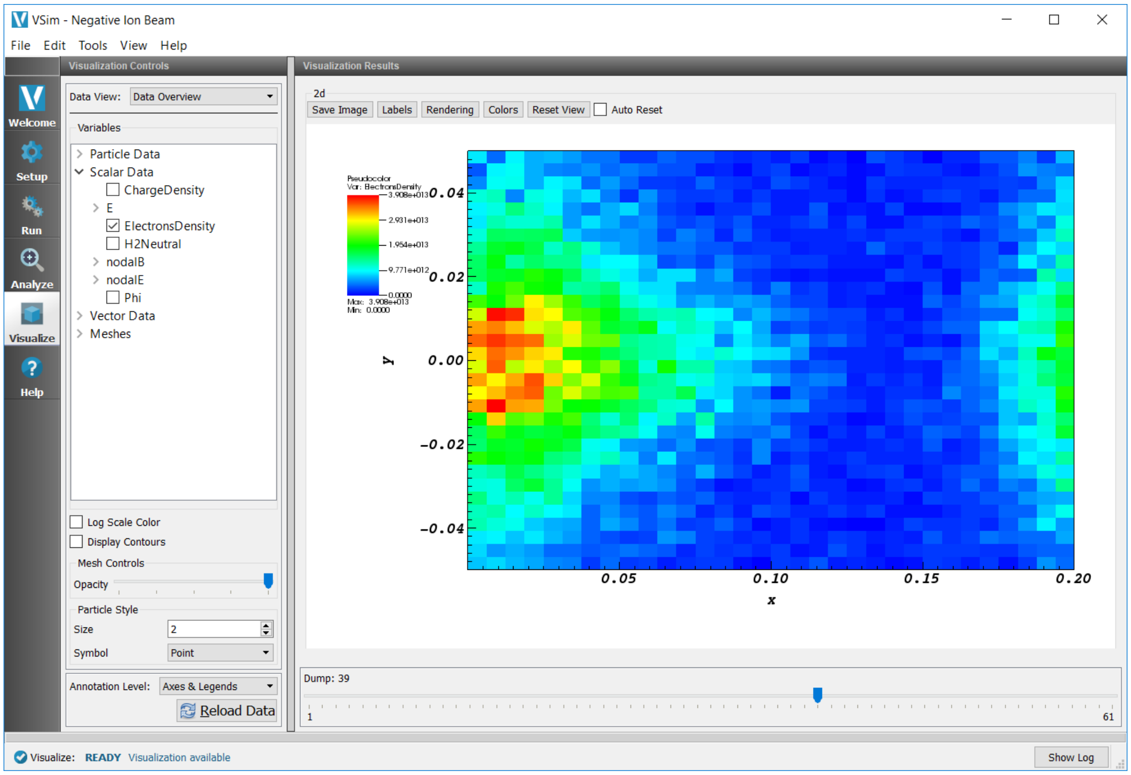
$DIR = C:\winsame\jleddy\vorpalall-trunk\vpexamples\VSimPD\Processes\negativeIonBeam
$SIMNAME = negativeIonBeam

```
Processing file: negativeIonBeam_Electrons_53.h5...
Writing negativeIonBeam_ElectronsDensity_53.vsh5.

Working on dump 54.
Getting mesh information from negativeIonBeam_Globals_54.h5.
Processing file: negativeIonBeam_Electrons_54.h5...
Writing negativeIonBeam_ElectronsDensity_54.vsh5.
Working on dump 55.
Getting mesh information from negativeIonBeam_Globals_55.h5.
Processing file: negativeIonBeam_Electrons_55.h5...
Writing negativeIonBeam_ElectronsDensity_55.vsh5.

Working on dump 56.
Getting mesh information from negativeIonBeam_Globals_56.h5.
Processing file: negativeIonBeam_Electrons_56.h5...
Writing negativeIonBeam_ElectronsDensity_56.vsh5.
Working on dump 57.
Getting mesh information from negativeIonBeam_Globals_57.h5.
Processing file: negativeIonBeam_Electrons_57.h5...
Writing negativeIonBeam_ElectronsDensity_57.vsh5.

Working on dump 58.
Getting mesh information from negativeIonBeam_Globals_58.h5.
Processing file: negativeIonBeam_Electrons_58.h5...
Writing negativeIonBeam_ElectronsDensity_58.vsh5.
Working on dump 59.
Getting mesh information from negativeIonBeam_Globals_59.h5.
Processing file: negativeIonBeam_Electrons_59.h5...
Writing negativeIonBeam_ElectronsDensity_59.vsh5.

Working on dump 60.
Getting mesh information from negativeIonBeam_Globals_60.h5.
Processing file: negativeIonBeam_Electrons_60.h5...
Writing negativeIonBeam_ElectronsDensity_60.vsh5.
Working on dump 61.
Getting mesh information from negativeIonBeam_Globals_61.h5.
Processing file: negativeIonBeam_Electrons_61.h5...
Writing negativeIonBeam_ElectronsDensity_61.vsh5.

==================================
Analysis completed successfully
```

Analyze: **NO RUN** Open and run a simulation

Fig. 6.44: The Analyze Window at the end of execution.

The resulting data can be visualized as “ElectronsDensity” under the Scalar Data menu in the *Visualize* Tab. A plot of this data is shown below in Fig. 6.45. The density of H2plus, Hminus or Hneutral can also be calculated if those species names are used in place of “Electrons” and the analyzer is re-run. If you have previously navigated to the *Visualize* Tab, you will need to press the *Reload Data* button at the bottom of the *Visualize* Tab to view the data.



Data View: Data Overview

☑ ElectronsDensity

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

Dump: 39

Visualize: READY Visualization available

Fig. 6.45: Plot of the electron density at dump 64.

Visualizing the Results

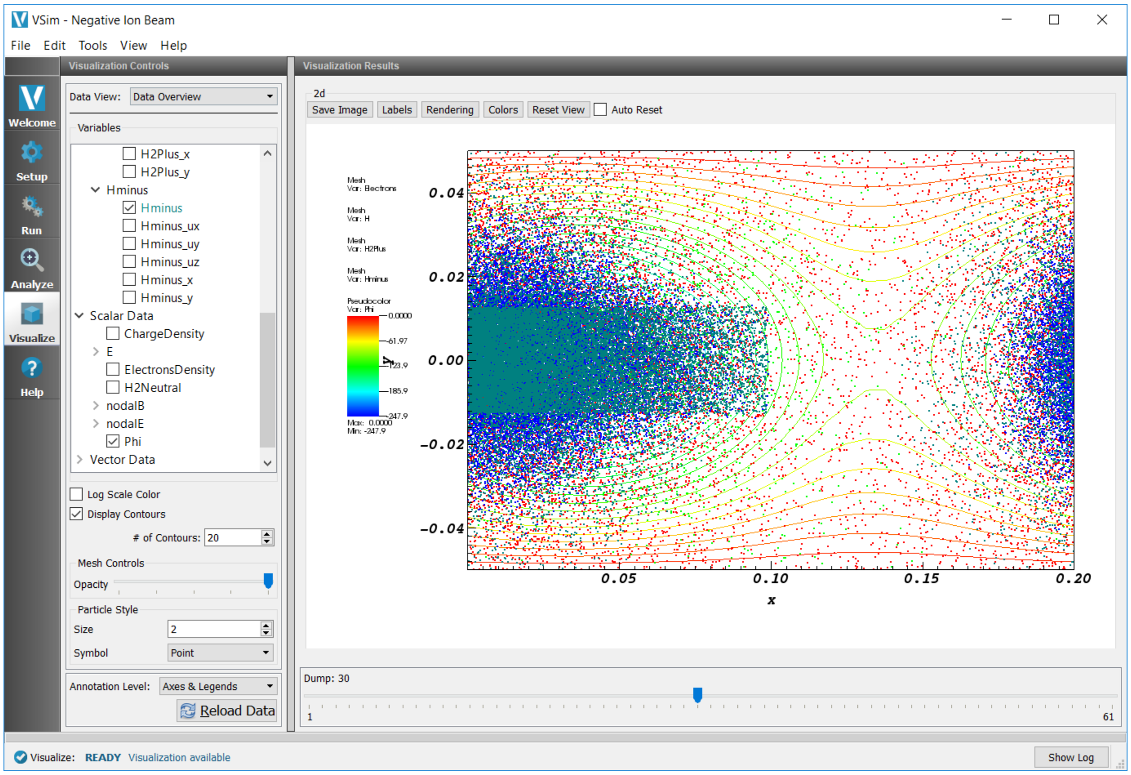
After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.
- Expand “Particle Data” and select “Electrons,” “H,” “H2Plus,” and “Hminus.
- Then expand “Scalar Data” and select “Phi.”
- Check the *Display Contours” box, which is below the *Variables* box in the *Visualization Controls* pane.
- Set the “# of Contours” to 20. The scroll through the dumps to produce the image in Fig. 6.46.

Further Experiments

The background gas pressure is higher than one would typically see in an accelerator in this example so that the example will produce results quickly. Decreasing the pressure will give the same results, but over longer time scales.

Since this beam is negatively charged, it repulses electrons from the region near the beam. Decreasing the beam current will produce more neutralizing H2+ near the beam as the electrons can more effectively ionize the background H2 gas in that area.



Data View: Data Overview

☑ Hminus

☑ Phi

□ Log Scale Color

☑ Display Contours

# of Contours: 20

Opacity

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

Dump: 30

Visualize: READY Visualization available

Fig. 6.46: Visualization of particle densities as a color contour plot, overlaid with a scatter plot of the particle positions.

6.5.3 Neutral Heat Transport (neutralHeatTransport.sdf)

Keywords:

`heat transport, DSMC, elastic collisions, reactions`

Problem description

VSim may be used to model the heat flux through a neutral gas confined between two plates of different temperatures. This problem is a common benchmark for DSMC simulations, and is described by Bird in “Molecular gas dynamics and the direct simulation of molecular gas flows” (1994) on page 280. In this example, we model the heat transport between cold (250K) and hot (1000K) plates separated by a meter. Between the plates is a volume of neutral Argon gas that transports the heat through either free-molecular motion (in the case of lower pressure) or through collisional transport via elastic collisions (in the case of higher pressure). The simulated heat flux can then be compared to the analytic result, validating the reactions framework in VSim.

This simulation can be performed with a VSimPD license.

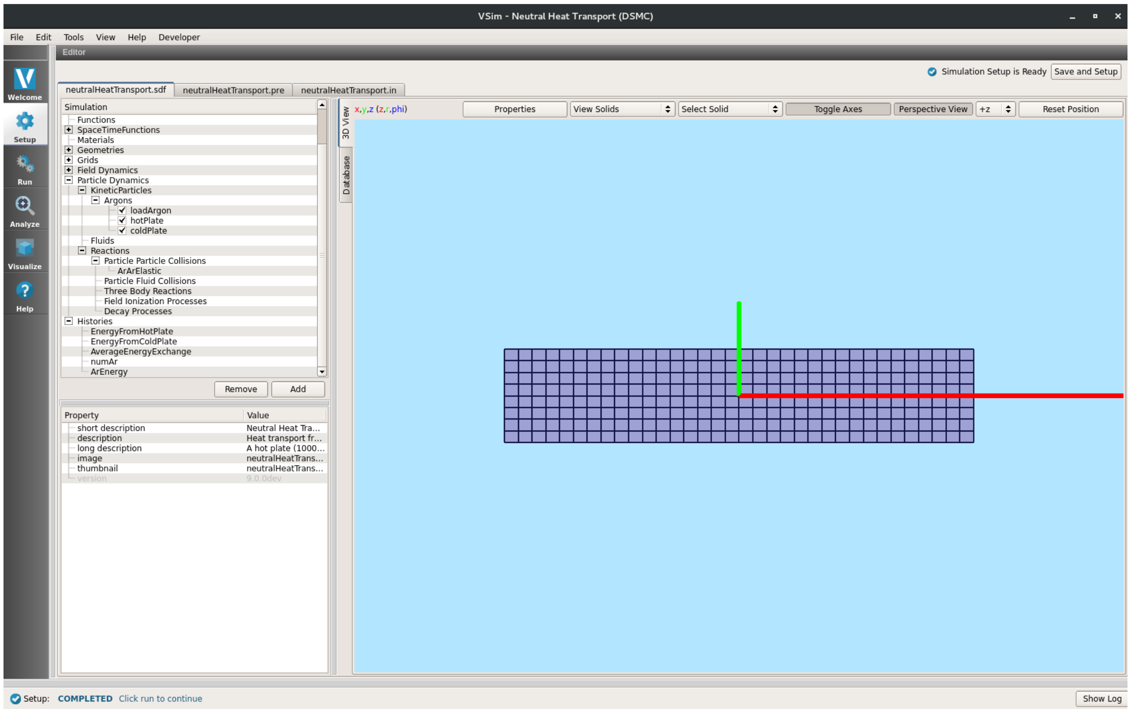
Opening the Simulation

The Neutral Heat Transport example is accessed from within VSimComposer by the following actions:

- Select the *New –> From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Processes* option.

- Select "Neutral Heat Transport" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* is shown with all the implemented physics and geometries in Fig. 6.47.



Simulation Setup is Ready

Simulation
- Functions
- ⊞ SpaceTimeFunctions
- Materials
- ⊞ Geometries
- ⊞ Grids
- ⊞ Field Dynamics
- ⊟ Particle Dynamics
  - ⊟ KineticParticles
    - ⊟ Argons
      - ☑ loadArgon
      - ☑ hotPlate
      - ☑ coldPlate
  - Fluids
  - ⊟ Reactions
    - ⊟ Particle Particle Collisions
      - ArArElastic
    - Particle Fluid Collisions
    - Three Body Reactions
    - Field Ionization Processes
    - Decay Processes
- ⊟ Histories
  - EnergyFromHotPlate
  - EnergyFromColdPlate
  - AverageEnergyExchange
  - numAr
  - ArEnergy

Property	Value
short description	Neutral Heat Tra...
description	Heat transport fr...
long description	A hot plate (1000...
image	neutralHeatTrans...
thumbnail	neutralHeatTrans...
version	9.0.0dev

Setup: COMPLETED Click run to continue

Fig. 6.47: Setup Window for the Neutral Heat Transport example.

Simulation Properties

This input file contains one kinetic species of neutral Argon, the required thermalizing boundary conditions for the hot and cold plates, and the Ar-Ar elastic collisions. The constants and parameters are set up so that the Argon pressure (ARPRES) in Pa can be changed, and the simulation grid will adjust resolution to ensure that the mean-free path is always resolved. This means that multiple simulations can be run to match the analytic result for a variety of pressures/collisionality.

Running the Simulation

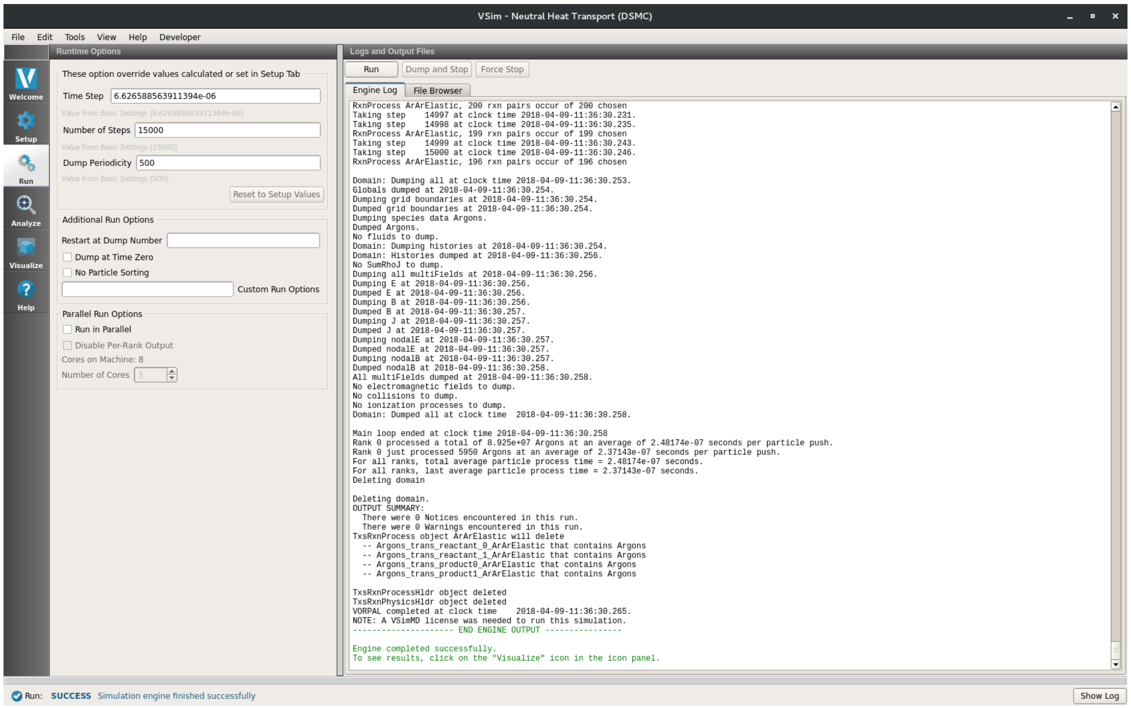
After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in this pane. The run has completed when you see the output, "Engine completed successfully." This is shown in the window below.

Visualizing the Results

After run completion, continue as follows:

- Proceed to the *Visualize* Tab by pressing the Visualize button in the left column of buttons.



These option override values calculated or set in Setup Tab

Time Step: 6.626588563911394e-06
Value from Basic Settings (6.626588563911394e-06)

Number of Steps: 15000
Value from Basic Settings (15000)

Dump Periodicity: 500
Value from Basic Settings (500)

Additional Run Options

Restart at Dump Number:
☐ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options:

Parallel Run Options

☐ Run in Parallel
☐ Disable Per-Rank Output
Cores on Machine: 8
Number of Cores: 3

```
RxnProcess ArArElastic, 200 rxn pairs occur of 200 chosen
Taking step    14997 at clock time 2018-04-09-11:36:30.231.
Taking step    14998 at clock time 2018-04-09-11:36:30.235.
RxnProcess ArArElastic, 199 rxn pairs occur of 199 chosen
Taking step    14999 at clock time 2018-04-09-11:36:30.243.
Taking step    15000 at clock time 2018-04-09-11:36:30.246.
RxnProcess ArArElastic, 196 rxn pairs occur of 196 chosen

Domain: Dumping all at clock time 2018-04-09-11:36:30.253.
Globals dumped at 2018-04-09-11:36:30.254.
Dumping grid boundaries at 2018-04-09-11:36:30.254.
Dumped grid boundaries at 2018-04-09-11:36:30.254.
Dumping species data Argons.
Dumped Argons.
No fluids to dump.
Domain: Dumping histories at 2018-04-09-11:36:30.254.
Domain: Histories dumped at 2018-04-09-11:36:30.256.
No SumRhoJ to dump.
Dumping all multiFields at 2018-04-09-11:36:30.256.
Dumping E at 2018-04-09-11:36:30.256.
Dumped E at 2018-04-09-11:36:30.256.
Dumping B at 2018-04-09-11:36:30.256.
Dumped B at 2018-04-09-11:36:30.257.
Dumping J at 2018-04-09-11:36:30.257.
Dumped J at 2018-04-09-11:36:30.257.
Dumping nodalE at 2018-04-09-11:36:30.257.
Dumped nodalE at 2018-04-09-11:36:30.257.
Dumping nodalB at 2018-04-09-11:36:30.257.
Dumped nodalB at 2018-04-09-11:36:30.258.
All multiFields dumped at 2018-04-09-11:36:30.258.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-04-09-11:36:30.258.

Main loop ended at clock time 2018-04-09-11:36:30.258
Rank 0 processed a total of 8.925e+07 Argons at an average of 2.48174e-07 seconds per particle push.
Rank 0 just processed 5950 Argons at an average of 2.37143e-07 seconds per particle push.
For all ranks, total average particle process time = 2.48174e-07 seconds.
For all ranks, last average particle process time = 2.37143e-07 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
TxsRxnProcess object ArArElastic will delete
  -- Argons_trans_reactant_0_ArArElastic that contains Argons
  -- Argons_trans_reactant_1_ArArElastic that contains Argons
  -- Argons_trans_product0_ArArElastic that contains Argons
  -- Argons_trans_product1_ArArElastic that contains Argons

TxsRxnProcessHldr object deleted
TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-04-09-11:36:30.265.
NOTE: A VSimMD license was needed to run this simulation.
--------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 6.48: The Run Window at the end of execution.

- Select "History" from the *Data View* drop down menu, which is located in the upper right corner the window.

Two graphs will be shown in the resulting window (see Fig. 6.5.3). The first graph, ArEnergy, shows the total kinetic energy of the argon species. The second, AverageEnergyExchange, shows the average energy transferred between the particles and the plates as a function of time. The AverageEnergyExchange plot divided by the cross-sectional area of a plate gives the average heat flux. A python script, validation.py, is provided to calculate this heat flux from the simulation data, and plot the heat flux versus the analytic heat flux. To run this script, go to the examples directory and run python from the command line (using the command "python validation.py"). The first plot is the same histories seen in the VSim Composer visualization. The second plot is the validation.

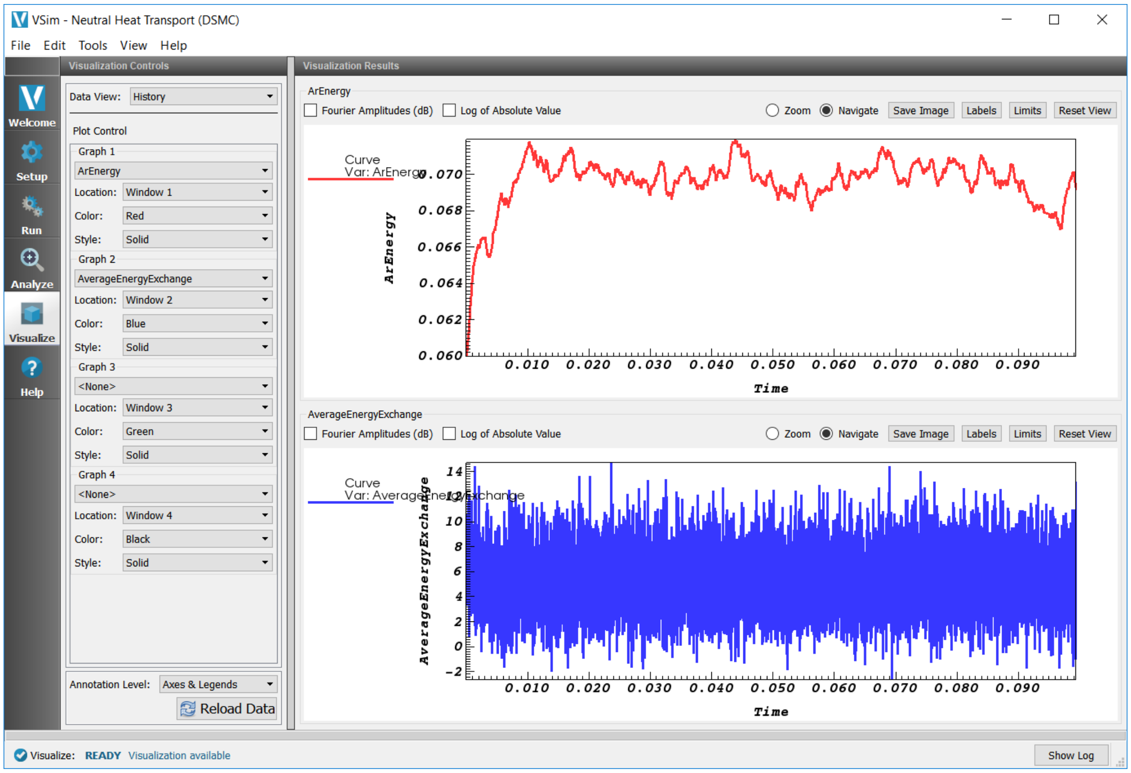
Further Experiments

As stated in the simulation properties section, simulations can be run with varying pressures (maintaining all else constant) and the resulting heat fluxes plotted against the analytic result, as shown in Fig. 6.5.3. The provided python script will only plot one simulation result at a time, but it can be modified easily to overplot multiple simulations. Each simulation should lie on the analytic green line. It is important to ensure that the statistics of the collisions are good enough, so when moving to lower collisionality (pressure) the number of macro particles per cell should be increased. Additionally, it is useful to switch the kinetic particle type so that it is variable weight with managed weights. This allows an isotropic macroparticle density while accounting for a variable physical particle density. Alternatively, the temperature of the plates, distance between them, species of neutral gas, etc. can all be modified to test the generality of the model and collisions.

6.5.4 Proton Beam (protonBeam.sdf)

Keywords:

electromagnetic, particle in cell, material boundary, reactions, particle emitter



VSim - Neutral Heat Transport (DSMC)

File Edit Tools View Help

Visualization Controls

Data View: History

Plot Control

Graph 1
ArEnergy
Location: Window 1
Color: Red
Style: Solid

Graph 2
AverageEnergyExchange
Location: Window 2
Color: Blue
Style: Solid

Graph 3
<None>
Location: Window 3
Color: Green
Style: Solid

Graph 4
<None>
Location: Window 4
Color: Black
Style: Solid

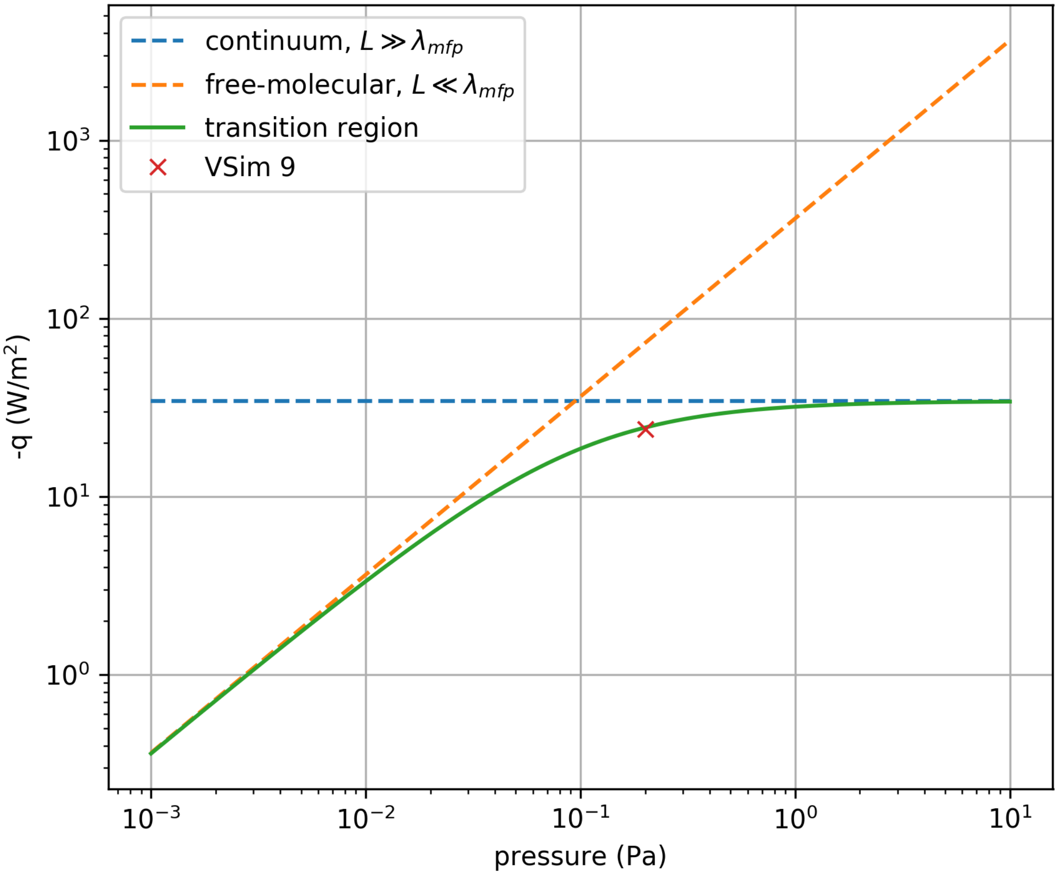
Annotation Level: Axes & Legends

Visualization Results

ArEnergy
□ Fourier Amplitudes (dB) □ Log of Absolute Value ○ Zoom ● Navigate

AverageEnergyExchange
□ Fourier Amplitudes (dB) □ Log of Absolute Value ○ Zoom ● Navigate

Visualize: READY Visualization available



Problem description

This example injects a proton beam into a column of neutral H2 gas. The geometry is setup like an electron column in an accelerator beamline (ie. external solenoidal B-field and negative electrodes on either end for electron confinement). Upon entering the neutral gas multiple reactions begin to occur including ionization, charge exchange, dissociation, H3+ formation, and others. The beam leaves the column, leaving behind a combination of ions, electrons, and neutrals that are either confined or ejected by the background electrode potential.

In this simulation, a beam of H+ ions propagates through a background H2 gas. Collisions between the beam ions and the background gas produce electrons, H2+, neutral H, and H3+ through the following reactions:

$H^+ + H_2 \rightarrow H^+ + H_2^+ + e^-$ (ion impact ionization)

$e^- + H_2 \rightarrow H_2^+ + 2e^-$ (electron impact ionization)

$H^+ + H_2 \rightarrow H^+ + H_2$ (elastic)

$H^+ + H_2 \rightarrow H + H_2^+$ (charge exchange)

$H_2^+ + H_2 \rightarrow H_3^+ + H$ (H3+ formation)

$H_2^+ + H_2 \rightarrow H_2^+ + H_2$ (charge exchange)

$H_2 + e^- \rightarrow H^+ + H + 2e^-$ (dissociative ionization)

$H_3^+ + H_2 \rightarrow H_3^+ + H_2$ (elastic)

Fig. 6.49 shows the cross sections for the above reactions as a function of incident energy.

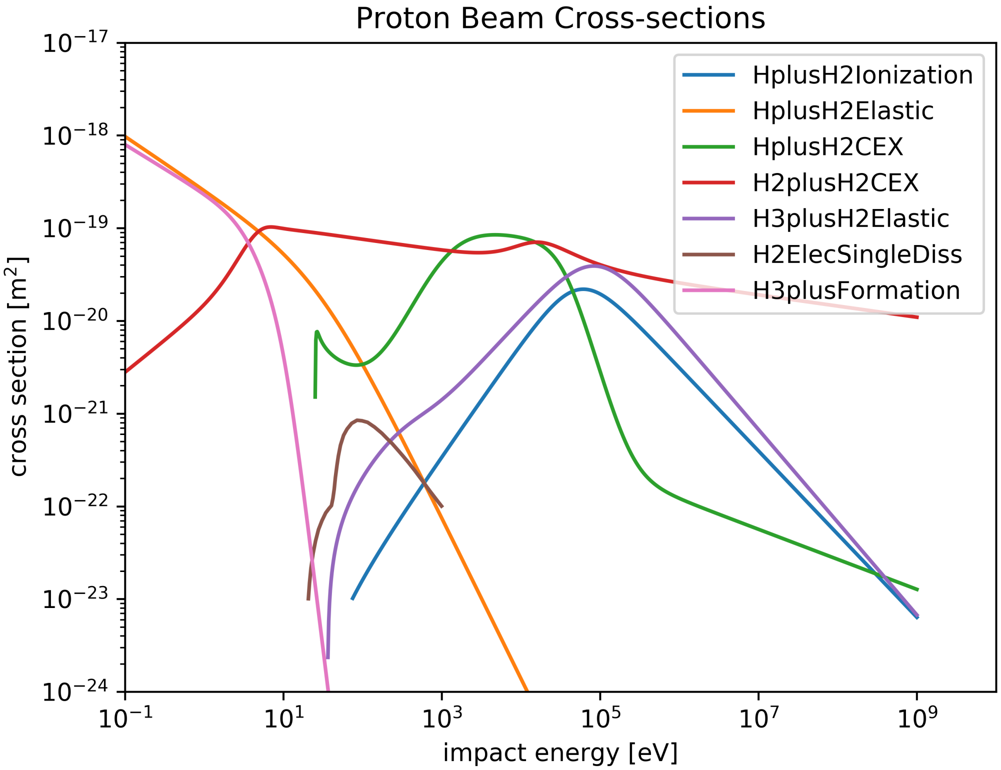


Fig. 6.49: Cross sections for the collisions included in this example.

This simulation can be performed with a VSimPD license.

Opening the Simulation

The Proton Beam example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Processes* option.
- Select “Proton Beam” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 6.50. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to Grid.

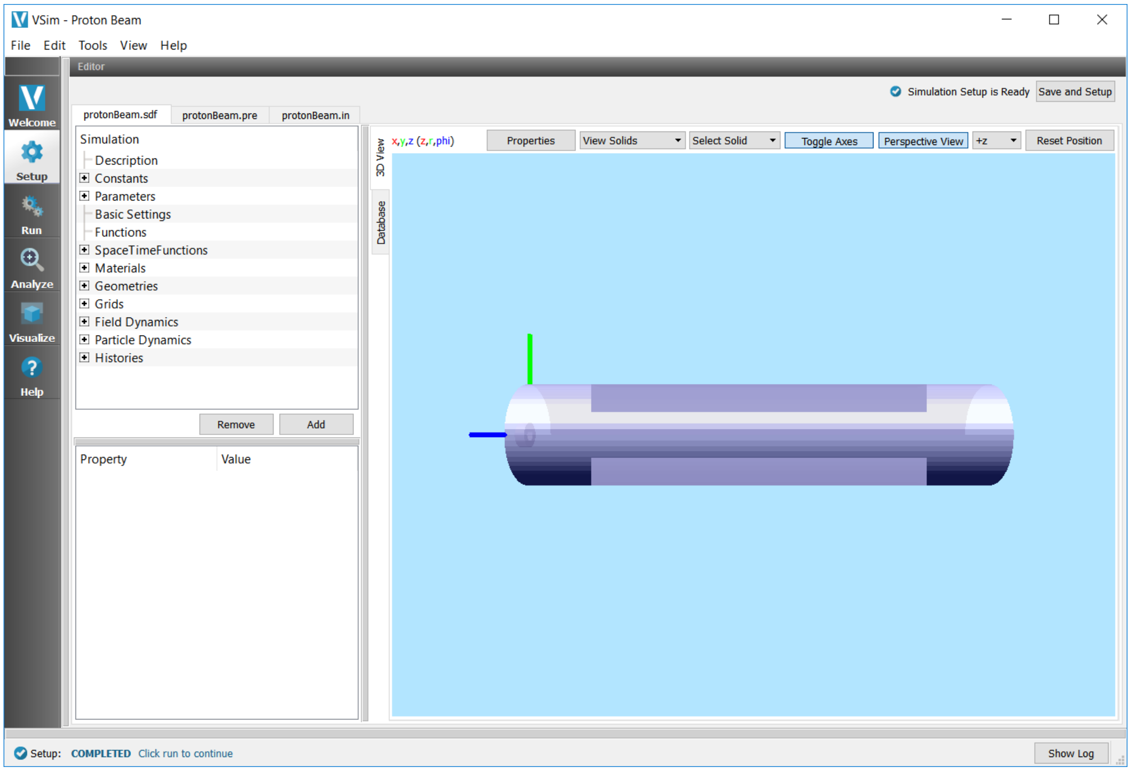


Fig. 6.50: Setup Window for the Proton Beam example.

Simulation Properties

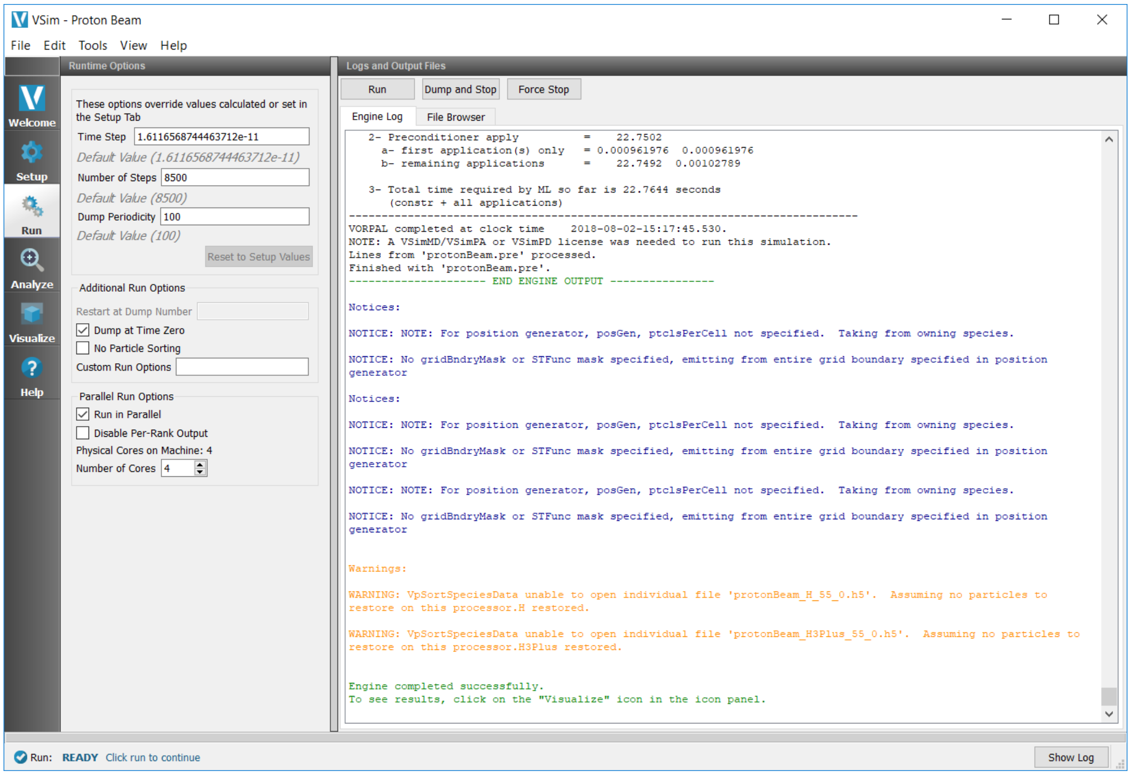
Constants are set up to allow setting the proton beam energy and current, the background H2 pressure and temperature, and the cross-sectional size of the beam emission.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.

- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” A snapshot of the simulation run completion is shown in Fig. 6.51.



These options override values calculated or set in the Setup Tab

Time Step 1.6116568744463712e-11

Default Value (1.6116568744463712e-11)

Number of Steps 8500

Default Value (8500)

Dump Periodicity 100

Default Value (100)

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 4

Number of Cores 4

```
   2- Preconditioner apply          =    22.7502
     a- first application(s) only   = 0.000961976  0.000961976
     b- remaining applications      =    22.7492  0.00102789

   3- Total time required by ML so far is 22.7644 seconds
      (constr + all applications)
------------------------------------------------------------------------------
VORPAL completed at clock time    2018-08-02-15:17:45.530.
NOTE: A VSimMD/VSimPA or VSimPD license was needed to run this simulation.
Lines from 'protonBeam.pre' processed.
Finished with 'protonBeam.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: NOTE: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.

NOTICE: No gridBndryMask or STFunc mask specified, emitting from entire grid boundary specified in position
generator

Notices:

NOTICE: NOTE: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.

NOTICE: No gridBndryMask or STFunc mask specified, emitting from entire grid boundary specified in position
generator

NOTICE: NOTE: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.

NOTICE: No gridBndryMask or STFunc mask specified, emitting from entire grid boundary specified in position
generator


Warnings:

WARNING: VpSortSpeciesData unable to open individual file 'protonBeam_H_55_0.h5'.  Assuming no particles to
restore on this processor.H restored.

WARNING: VpSortSpeciesData unable to open individual file 'protonBeam_H3Plus_55_0.h5'.  Assuming no particles to
restore on this processor.H3Plus restored.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

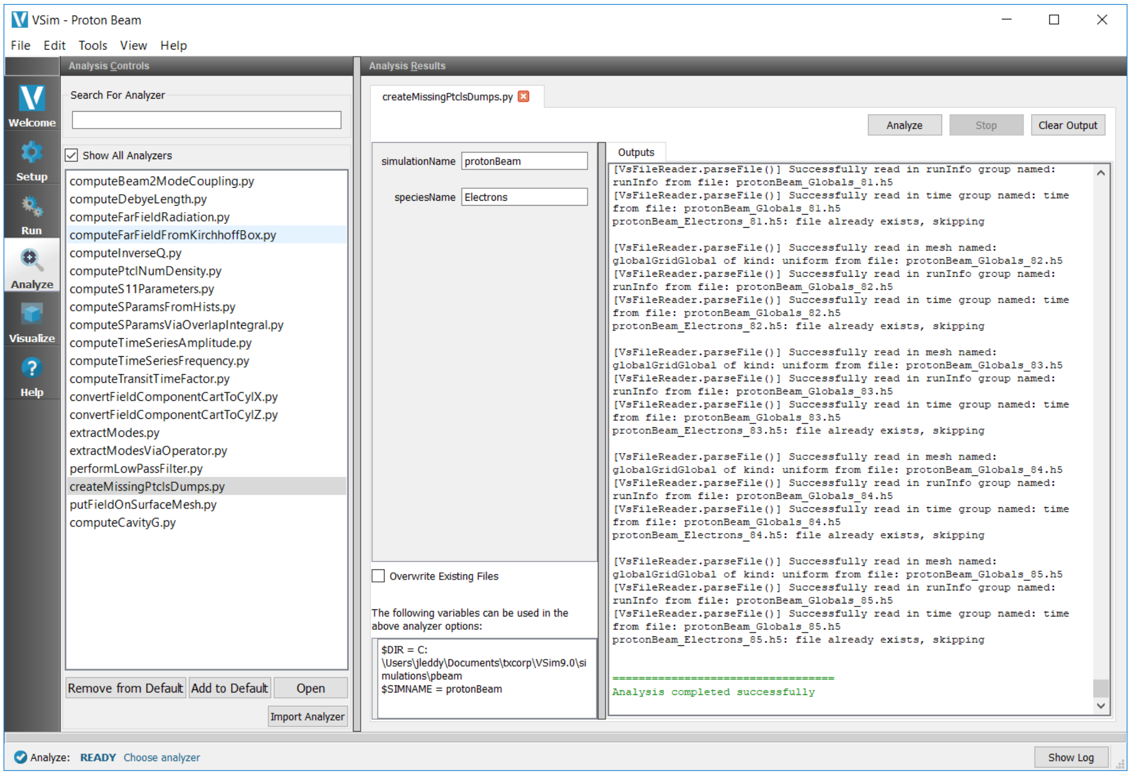
Run: READY Click run to continue

Fig. 6.51: The Run Window at the end of execution.

Analyzing the Results

The reactions do not occur at the beginning of the simulation because the protons must propagate to the neutral gas. This means that some species, such as the electrons, do not exist in the simulation until a later time. For the time slider to be consistent across species in the visualization, we must generate empty time slices for these particles for early in time. This can be quickly and easily accomplished in the Analyzers tab.

- First click on the *Analyze* Tab.
- Click *Show All Analyzers* and choose *createMissingPtclsDumps.py*. Then click *Open* at the bottom of the *Analysis Controls* pane.
- Ensure that the “simulationName” field is “protonBeam” and enter “Electrons” in the “speciesName” field.
- Press the *Analyze* button on in the upper right corner of the window to run the analysis. Below, the Analyze Tab is shown at the end of a successful run.
- Repeat this for the H2Plus and any other species that are generated (be default, only these two)


VSim - Proton Beam
File
Edit
Tools
View
Help
Analysis Controls
Analysis Results
Welcome
Setup
Run
Analyze
Visualize
Help
Search For Analyzer
☑ Show All Analyzers
computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
createMissingPtclsDumps.py
putFieldOnSurfaceMesh.py
computeCavityG.py
Remove from Default
Add to Default
Open
Import Analyzer
createMissingPtclsDumps.py
Analyze
Stop
Clear Output
Outputs
Show Log
Analyze: READY Choose analyzer

simulationName protonBeam

speciesName Electrons

□ Overwrite Existing Files

The following variables can be used in the above analyzer options:

$DIR = C:\Users\jleddy\Documents\txcorp\VSim9.0\simulations\pbeam
$SIMNAME = protonBeam

```
[VsFileReader.parseFile()] Successfully read in runInfo group named:
runInfo from file: protonBeam_Globals_81.h5
[VsFileReader.parseFile()] Successfully read in time group named: time
from file: protonBeam_Globals_81.h5
protonBeam_Electrons_81.h5: file already exists, skipping

[VsFileReader.parseFile()] Successfully read in mesh named:
globalGridGlobal of kind: uniform from file: protonBeam_Globals_82.h5
[VsFileReader.parseFile()] Successfully read in runInfo group named:
runInfo from file: protonBeam_Globals_82.h5
[VsFileReader.parseFile()] Successfully read in time group named: time
from file: protonBeam_Globals_82.h5
protonBeam_Electrons_82.h5: file already exists, skipping

[VsFileReader.parseFile()] Successfully read in mesh named:
globalGridGlobal of kind: uniform from file: protonBeam_Globals_83.h5
[VsFileReader.parseFile()] Successfully read in runInfo group named:
runInfo from file: protonBeam_Globals_83.h5
[VsFileReader.parseFile()] Successfully read in time group named: time
from file: protonBeam_Globals_83.h5
protonBeam_Electrons_83.h5: file already exists, skipping

[VsFileReader.parseFile()] Successfully read in mesh named:
globalGridGlobal of kind: uniform from file: protonBeam_Globals_84.h5
[VsFileReader.parseFile()] Successfully read in runInfo group named:
runInfo from file: protonBeam_Globals_84.h5
[VsFileReader.parseFile()] Successfully read in time group named: time
from file: protonBeam_Globals_84.h5
protonBeam_Electrons_84.h5: file already exists, skipping

[VsFileReader.parseFile()] Successfully read in mesh named:
globalGridGlobal of kind: uniform from file: protonBeam_Globals_85.h5
[VsFileReader.parseFile()] Successfully read in runInfo group named:
runInfo from file: protonBeam_Globals_85.h5
[VsFileReader.parseFile()] Successfully read in time group named: time
from file: protonBeam_Globals_85.h5
protonBeam_Electrons_85.h5: file already exists, skipping


==================================
Analysis completed successfully
```

Fig. 6.52: The Analyze Window at the end of execution.

Visualizing the Results

We can now visualize all of the particles at a particular time slice. To do this:

- Proceed to the Visualize Window by pressing the *Visualize* button in the left column of buttons
- Expand *Particle Data* and select Electrons, H2Plus, and Hplus.
- Slide the time slider to advance the simulation in time (step 50 is shown in Fig. 6.53)

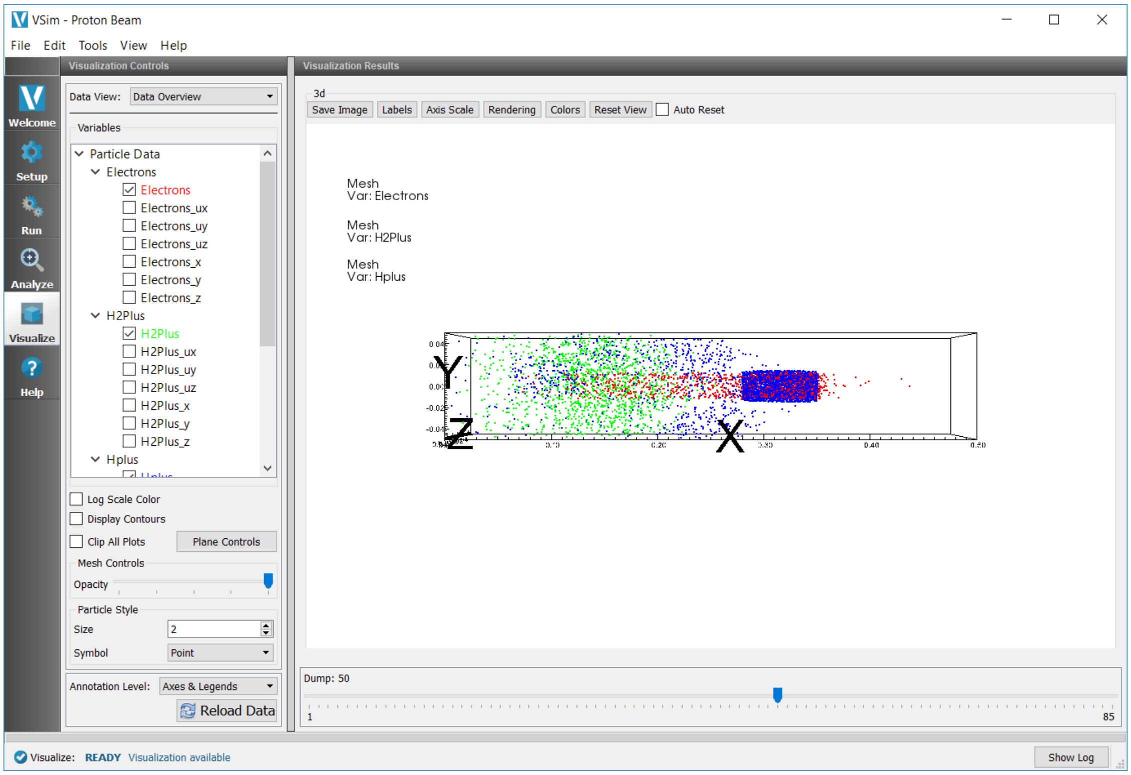
Next we can visualize the potential due to the particles and the electrodes:

- Unselect the particle data (Electrons, H2Plus, and Hplus).
- Expand *Scalar Data* and select *Phi*.
- Check the *Clip All Plots* box and scroll through the dumps.

The potential shown in Fig. 6.54 is the total potential, that is, the potential due to the static electrodes, the proton beam, and other charged species resulting from the reactions.

Further Experiments

Try changing the neutral gas pressure (which in turn will modify its density). At higher densities more reactions will occur and the proton beam will not be able to traverse the column intact. For lower densities, which are more in line with experiment, the proton beam will cause small amounts of ionization in the background gas, generating an electron cloud that is confined by the electrodes that can provide space-charge compensation for the beam. Lowering the beam energy will allow some lower energy reactions, such as H3+ formation, to occur.



Data View: Data Overview

☑ Electrons

☑ H2Plus

□ Log Scale Color

□ Display Contours

□ Clip All Plots

Size: 2

Symbol: Point

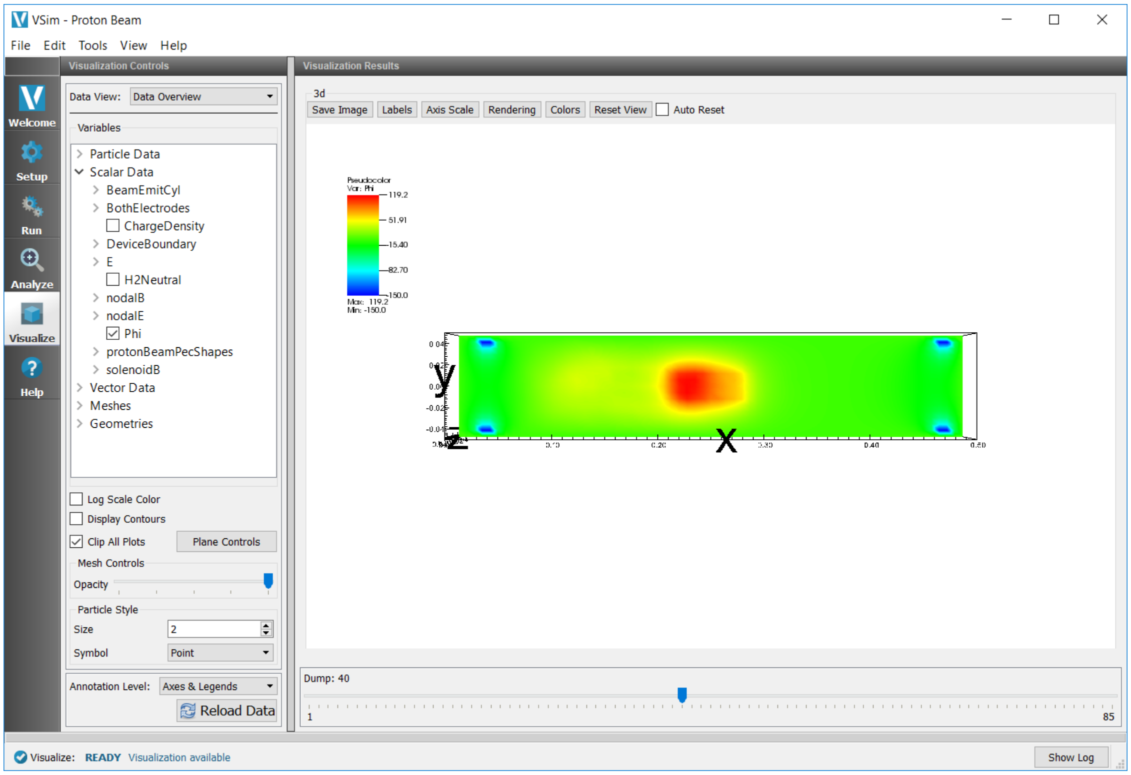
Annotation Level: Axes & Legends

□ Auto Reset

Dump: 50

Visualize: READY Visualization available

Fig. 6.53: Plot of all the particles at timestep 50. Notice that the electrons are confined by the magnetic field to the inner radius of the device. Some will also be confined by the electrodes to oscillate along the device.



Data View: Data Overview

☑ Phi

□ Log Scale Color

□ Display Contours

☑ Clip All Plots

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

□ Auto Reset

Dump: 40

Visualize: READY Visualization available

Fig. 6.54: The electrostatic potential

6.5.5 Single Particle Circular Motion (singleParticleCircularMotion.sdf)

Keywords:

single particle, circular motion, finite difference effects

Problem Description

This example shows how to simulate the uniform circular motion of a single electron in a constant, uniform magnetic field in VSim. The electron is loaded inside a cylindrical capacitor with grounded walls to eliminate any stray electric fields. The electron is loaded far from the walls to reduce any effects from image charges. The magnetic field points down the positive z-axis.

Due to the finite difference algorithm utilized by VSim, two corrections must be made in order to get the electron to take a true circular trajectory. The first correction is to the cyclotron frequency. In the finite difference world of VSim, the electron does not move along a circular arc from time step to time step, instead it moves along a straight line. To correct for this we need to set our $\omega_{cyclo_{FD}} = \frac{2}{\Delta T}\arctan\left(\frac{\omega_{cyclo}\Delta T}{2}\right)$ [1] (see chapter 4 section 3).

The next correction is to account for the implementation of the Boris Method [1], the algorithm used in VSim to push particles. In the Boris Method, the position of the particle, $\vec{x}(t)$, is defined at full time steps, while the velocity, $\vec{v}(t)$, is defined at half time steps. This scheme of 'well-centered' derivatives means that VSim is automatically accurate to second order, but it means we have to be careful about our initial conditions for the electron's velocity. The initial velocity is set under Particle Dynamics → Kinetic Particle → electrons0 → particleLoader0 then velocity distribution. VSim will assume that this is the particle's velocity a **ONE HALF** time step before the start of the simulation, so we must load the particle with the velocity it would have a half time step before the start of the simulation.

This simulation can be performed with a VSimBase license.

Opening the Simulation

The Single Particle Circular Motion example is accessed from within VSimComposer by the following actions:

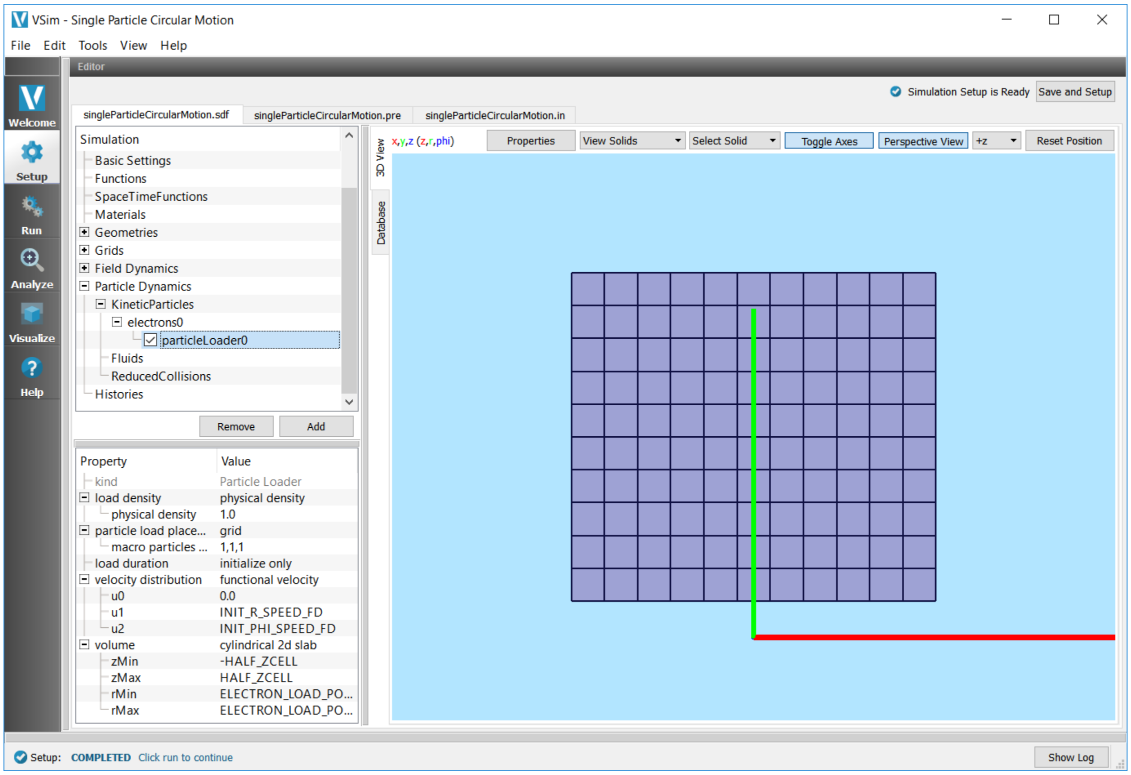
- Select the *New* → *From Example...* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Processes* option.
- Select *Single Particle Circular Motion* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 6.55. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.

Simulation Properties

The Single Particle Circular Motion example includes some constants for easy adjustment of simulation properties:

- B0: The magnitude of the magnetic field
- VOLTAGE_OUTER and VOLTAGE_INNER: sets the value of the radial electric field experienced by electron (default value for both is zero)



Property	Value
kind	Particle Loader
load density	physical density
physical density	1.0
particle load place...	grid
macro particles ...	1,1,1
load duration	initialize only
velocity distribution	functional velocity
u0	0.0
u1	INIT_R_SPEED_FD
u2	INIT_PHI_SPEED_FD
volume	cylindrical 2d slab
zMin	-HALF_ZCELL
zMax	HALF_ZCELL
rMin	ELECTRON_LOAD_PO...
rMax	ELECTRON_LOAD_PO...

Setup: COMPLETED Click run to continue

Fig. 6.55: Setup Window for the Single Particle Circular Motion example.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- Here you can set run parameters. The default is to run in serial.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in this pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in Fig. 6.56

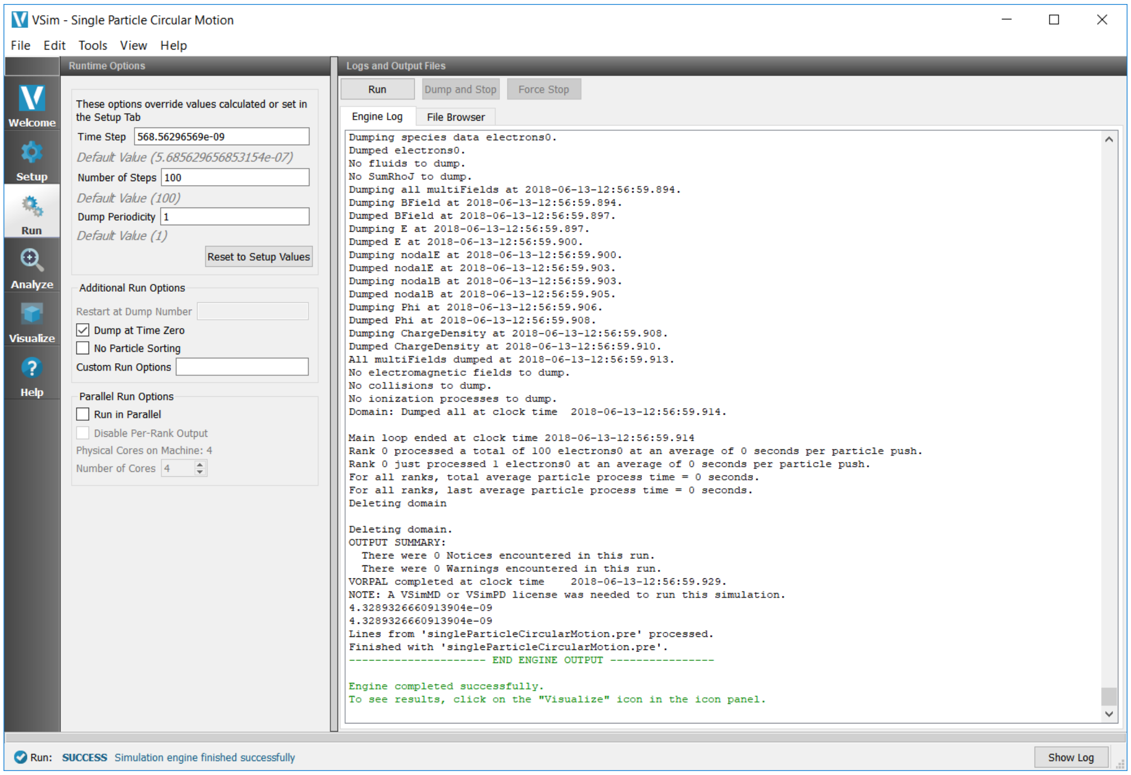
In serial, this simulation only takes seconds to run.

Visualizing the results

After performing the above actions, continue as follows:

Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

- In the Visualize Window, expand ‘Particle Data’ then ‘electrons0’ and check the box next to the red ‘electrons0.’ This will plot our single electron.
- Expand ‘Meshes’ then ‘globalGridGlobal’ and check the box next to ‘globalGridGlobal (ChargeDensity)’ as shown in Fig. 6.57.
- Scroll through the dump slider (found below the plot), the electron will be stationary because the axial coordinate (phi) has been compressed. This means the electron remains at the same r and z coordinate (this is a 2D simulation).



These options override values calculated or set in the Setup Tab

Time Step 568.56296569e-09

Default Value (5.685629656853154e-07)

Number of Steps 100

Default Value (100)

Dump Periodicity 1

Default Value (1)

Reset to Setup Values

Additional Run Options

Restart at Dump Number

☑ Dump at Time Zero

□ No Particle Sorting

Custom Run Options

Parallel Run Options

□ Run in Parallel

□ Disable Per-Rank Output

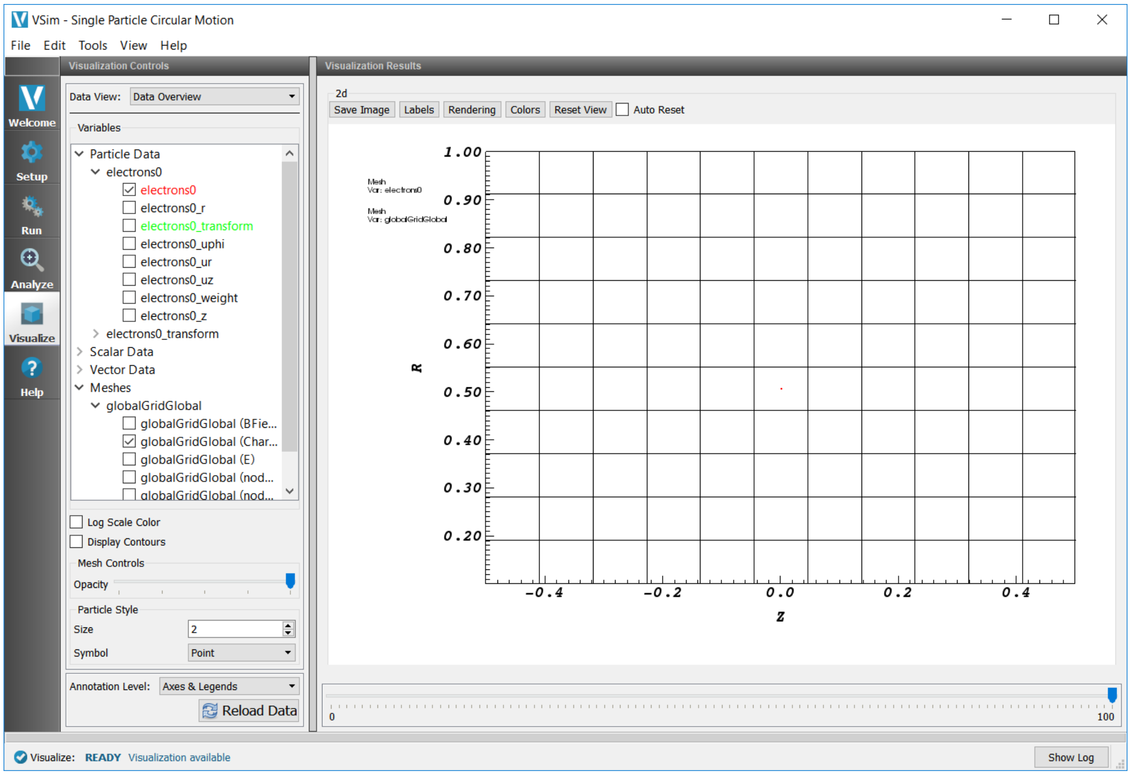
Physical Cores on Machine: 4

Number of Cores 4

```
Dumping species data electrons0.
Dumped electrons0.
No fluids to dump.
No SumRhoJ to dump.
Dumping all multiFields at 2018-06-13-12:56:59.894.
Dumping BField at 2018-06-13-12:56:59.894.
Dumped BField at 2018-06-13-12:56:59.897.
Dumping E at 2018-06-13-12:56:59.897.
Dumped E at 2018-06-13-12:56:59.900.
Dumping nodalE at 2018-06-13-12:56:59.900.
Dumped nodalE at 2018-06-13-12:56:59.903.
Dumping nodalB at 2018-06-13-12:56:59.903.
Dumped nodalB at 2018-06-13-12:56:59.905.
Dumping Phi at 2018-06-13-12:56:59.906.
Dumped Phi at 2018-06-13-12:56:59.908.
Dumping ChargeDensity at 2018-06-13-12:56:59.908.
Dumped ChargeDensity at 2018-06-13-12:56:59.910.
All multiFields dumped at 2018-06-13-12:56:59.913.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-13-12:56:59.914.

Main loop ended at clock time 2018-06-13-12:56:59.914
Rank 0 processed a total of 100 electrons0 at an average of 0 seconds per particle push.
Rank 0 just processed 1 electrons0 at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
VORPAL completed at clock time    2018-06-13-12:56:59.929.
NOTE: A VSimMD or VSimPD license was needed to run this simulation.
4.3289326660913904e-09
4.3289326660913904e-09
Lines from 'singleParticleCircularMotion.pre' processed.
Finished with 'singleParticleCircularMotion.pre'.
-------------------- END ENGINE OUTPUT ----------------

Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 6.56: The Run Window at the end of execution in serial.



Data View: Data Overview

Variables

- Particle Data
  - electrons0
    - ☑ electrons0
    - □ electrons0_r
    - □ electrons0_transform
    - □ electrons0_uphi
    - □ electrons0_ur
    - □ electrons0_uz
    - □ electrons0_weight
    - □ electrons0_z
  - electrons0_transform
- Scalar Data
- Vector Data
- Meshes
  - globalGridGlobal
    - □ globalGridGlobal (BFie...
    - ☑ globalGridGlobal (Char...
    - □ globalGridGlobal (E)
    - □ globalGridGlobal (nod...
    - □ globalGridGlobal (nod...

□ Log Scale Color

□ Display Contours

Mesh Controls

Opacity

Particle Style

Size 2

Symbol Point

Annotation Level: Axes & Legends

□ Auto Reset

Visualize: **READY** Visualization available

Fig. 6.57: Visualization of Single Particle Circular Motion at dump 100.

Further Experiments

Simulations are correct only to some accuracy. The corrections we made to the cyclotron frequency and the initial velocity make this simulation correct to second order. By looking at the phase space plot, we can explore the second order accuracy of this simulation. Navigate to the Visualize Window, select 'Phase Space' from the 'Data View' drop down menu, and plot 'electrons0_r' vs 'electrons0_ur.'

As you scroll through the dumps for the first time (with the 'Auto Reset' box UN-checked), the axes will adjust. The particle is taking an elliptical path in phase space. In a perfect simulation, the electron would remain at the same position in phase space with constant radius and zero radial velocity. Instead, the electron oscillates between the positions $r = 0.50500$ m and $r = 0.50542$ m for a $\Delta r = 0.00042$ m. Cut the time step in half and double the number of timesteps taken (so that the simulation runs through the same amount of time). Now look at the phase space plot again. By approximately what factor did Δr drop? Since the simulation is correct to second order, dropping the time step by a factor of 2 should drop the error by a factor of 4.

Other things you can play around with:

- Reset the electron speed, electron loading position, or the cyclotron frequency, OMEGA, back to the uncorrected versions and redo the error analysis described above.
- Change the values for VOLTAGE_INNER and VOLTAGE_OUTER to see the effects of a radial electric field on the single electron.

References

[1] Birdsall, C. K., & Langdon, A. B. (1985). Plasma Physics via Computer Simulation. New York: McGraw-Hill.

6.6 Processes (text-based setup)

6.6.1 Negative Ion Beam (negativeIonBeamT.pre)

Keywords:

`negativeIonBeamT, beam transport, monte carlo, electrostatic`

Problem description

VSim may be used to model ion beam transport and particle dynamics where the beam is represented by kinetic simulation particles. Low density background gasses can cause instabilities in the beams due to collisions between the beam particles and the background gas.

In this simulation, a beam of H- ions propagates through a background H2 gas. Collisions between the beam ions and the background gas produce electrons, H2+, and neutral H through the following reactions:

$H^- + H_2 \rightarrow H_- + H_2^+ + e^-$ (ion impact ionization)

$e^- + H_2 \rightarrow H_2^+ + 2e^-$ (electron impact ionization)

$H^- + H_2 \rightarrow H + H_2 + e^-$ (detachment)

$H^- + e^- \rightarrow H + 2e^-$ (stripping)

There are other reactions that are not included in this tutorial simulation. Typically these reactions have low cross sections. Fig. 6.58 shows the cross sections for the above reactions as a function of incident energy.

This simulation can be performed with a VSimPD license.

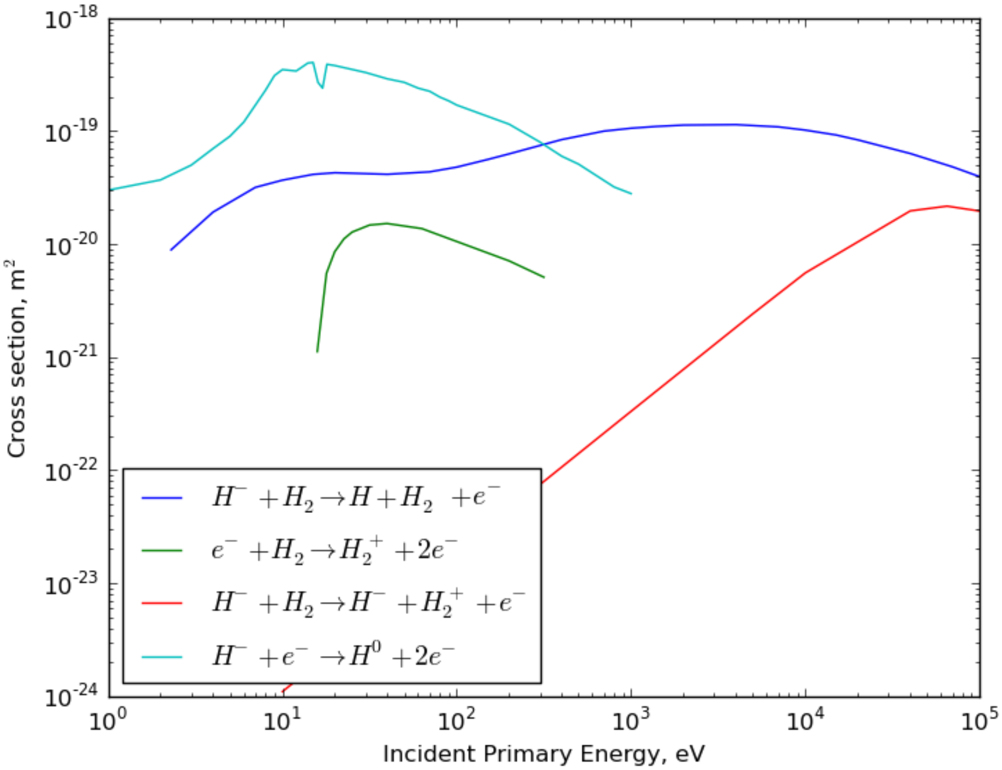


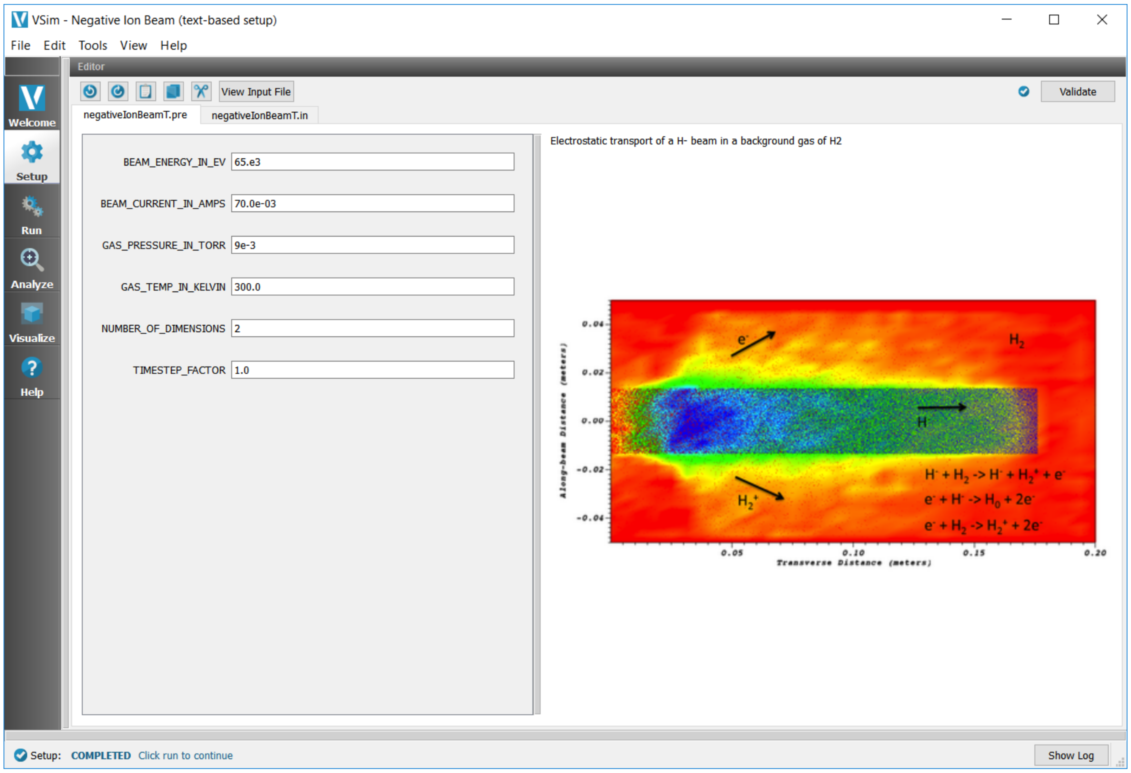
Fig. 6.58: Cross sections for the four collision reactions included in this example.

Opening the Simulation

The Kinetic Collisions example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Processes (text-based setup)* option.
- Select "Negative Ion Beam" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 6.59.


VSim - Negative Ion Beam (text-based setup)
File Edit Tools View Help
Editor
View Input File
Validate
negativeIonBeamT.pre
negativeIonBeamT.in
Welcome
Setup
Run
Analyze
Visualize
Help
0.04
0.02
0.00
-0.02
-0.04
Along-beam Distance (meters)
0.05
0.10
0.15
0.20
Transverse Distance (meters)
e^-
H_2
H^-
H_2^+
$H^- + H_2 \rightarrow H^- + H_2^+ + e^-$
$e^- + H^- \rightarrow H_0 + 2e^-$
$e^- + H_2 \rightarrow H_2^+ + 2e^-$
Show Log

BEAM_ENERGY_IN_EV 65.e3

BEAM_CURRENT_IN_AMPS 70.0e-03

GAS_PRESSURE_IN_TORR 9e-3

GAS_TEMP_IN_KELVIN 300.0

NUMBER_OF_DIMENSIONS 2

TIMESTEP_FACTOR 1.0

Electrostatic transport of a H- beam in a background gas of H2

Setup: COMPLETED Click run to continue

Fig. 6.59: Setup Window for the Kinetic Collisions example.

Input File Features

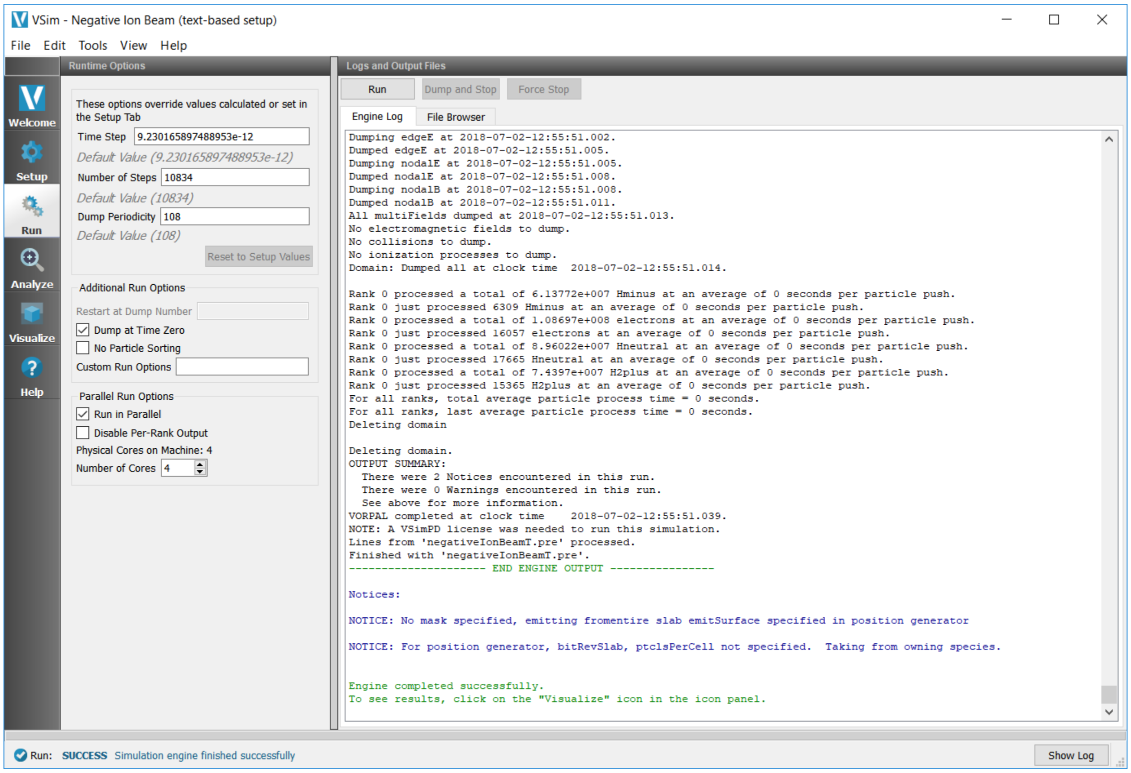
This input file contains a number of different kinetic species as well as a background fluid description of a gas. Ionization collisions between kinetic particles and the background gas are described by Monte Carlo interaction blocks of kind impactIonization, and detachment of electrons due to a collision with the background gas are of kind negativeIonDetachment. Collisions between kinetic particles and other kinetic particles are described in the input file by an interaction of kind binaryIonization.

The fields are electrostatically solved for at each time step, including the fields due to all charged particles, subject to the boundary conditions specified in the input file. There are a number of histories that record the number of particles for different species, their energies, as well as currents absorbed at the boundaries.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the window. You will see the output of the run in the right pane. The run has completed when you see the output, “Engine completed successfully.” This is shown in the window below.



These options override values calculated or set in the Setup Tab

Time Step 9.230165897488953e-12
Default Value (9.230165897488953e-12)
Number of Steps 10834
Default Value (10834)
Dump Periodicity 108
Default Value (108)

Restart at Dump Number
☑ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

☑ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
Dumping edgeE at 2018-07-02-12:55:51.002.
Dumped edgeE at 2018-07-02-12:55:51.005.
Dumping nodalE at 2018-07-02-12:55:51.005.
Dumped nodalE at 2018-07-02-12:55:51.008.
Dumping nodalB at 2018-07-02-12:55:51.008.
Dumped nodalB at 2018-07-02-12:55:51.011.
All multiFields dumped at 2018-07-02-12:55:51.013.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-02-12:55:51.014.

Rank 0 processed a total of 6.13772e+007 Hminus at an average of 0 seconds per particle push.
Rank 0 just processed 6309 Hminus at an average of 0 seconds per particle push.
Rank 0 processed a total of 1.08697e+008 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 16057 electrons at an average of 0 seconds per particle push.
Rank 0 processed a total of 8.96022e+007 Hneutral at an average of 0 seconds per particle push.
Rank 0 just processed 17665 Hneutral at an average of 0 seconds per particle push.
Rank 0 processed a total of 7.4397e+007 H2plus at an average of 0 seconds per particle push.
Rank 0 just processed 15365 H2plus at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 2 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-07-02-12:55:51.039.
NOTE: A VSimPD license was needed to run this simulation.
Lines from 'negativeIonBeamT.pre' processed.
Finished with 'negativeIonBeamT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: No mask specified, emitting fromentire slab emitSurface specified in position generator

NOTICE: For position generator, bitRevSlab, ptclsPerCell not specified.  Taking from owning species.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 6.60: The Run Window at the end of execution.

Analyzing the Results

If it is desired to calculate the density of the electrons the analysis script *computePtclNumDensity.py* must be used. First click on the *Analyze* Tab. Then select *computePtclNumDensity.py* From the drop down menu in the *Control* Pane. This script accepts the simulationName of the simulation (Name of the input file) and speciesName to be calculated (species of particles). In this case the simulationName is negativeIonBeamT and the speciesName is electrons. Then click on the *Run* button. The resulting data will be visualizable as “electronsDensity” under the Scalar Data menu in the *Visualize* Tab. The density of H2plus, Hminus or Hneutral can also be calculated if those species names are used in place of electrons.

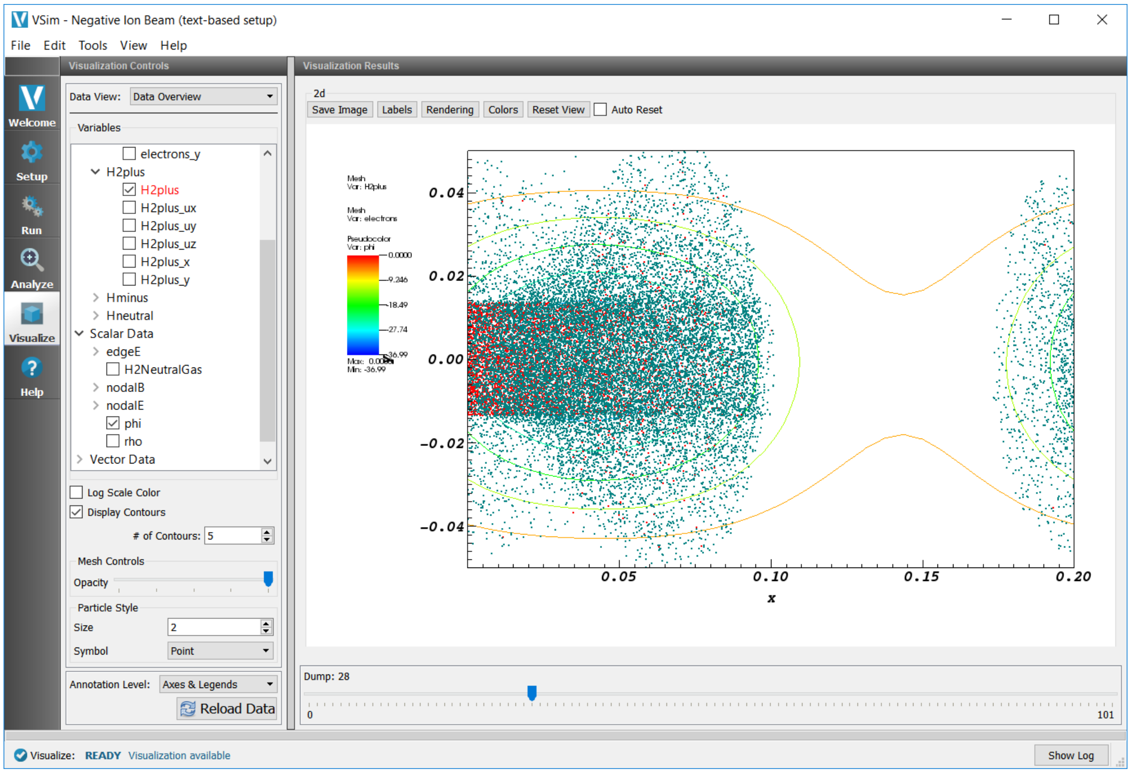
Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

Under *Particle Data* select H2plus and electrons to view the beam traversing the simulation space.

You can also add electric potential contours by expanding *Scalar Data* and selecting *phi*, and clicking *Display Contours* in the bottom half of the *Visualization Controls* pane. The visualization of the particles and contours is shown in Fig. 6.61.



Data View: Data Overview

☑ H2plus

☑ phi

□ Log Scale Color

☑ Display Contours

# of Contours: 5

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

Dump: 28

Visualize: READY Visualization available

Fig. 6.61: Visualization of particles and potential field.

Further Experiments

The background gas pressure is higher than one would typically see in an accelerator in this example so that the example will produce results quickly. Decreasing the pressure will give the same results, but over longer time scales.

Since this beam is negatively charged, it repulses electrons from the region near the beam. Decreasing the beam current will produce more neutralizing H2+ near the beam as the electrons can more effectively ionize the background H2 gas in that area.

6.7 Spacecraft

6.7.1 Coupon Array Charging (couponArrayCharging.sdf)

Keywords:

`solar wind, electrostatics, surface charging`

Problem description

In orbit, insulating outer surfaces of satellites will develop a surface charge due to the impinging solar wind. If enough surface charge accumulates electric breakdown can occur across or through the satellite and damage the craft.

This simulation models the accumulation of solar wind particles on an array of solar cells (coupons). The array includes 6 coupons, a kapton backing, and 6 metal busbars. Using post-simulation analysis, the component of the electric field normal to the surface of the satellite is calculated.

With additional data specific a particular spacecraft and materials, this simulation can indicate locations where breakdown is likely to occur.

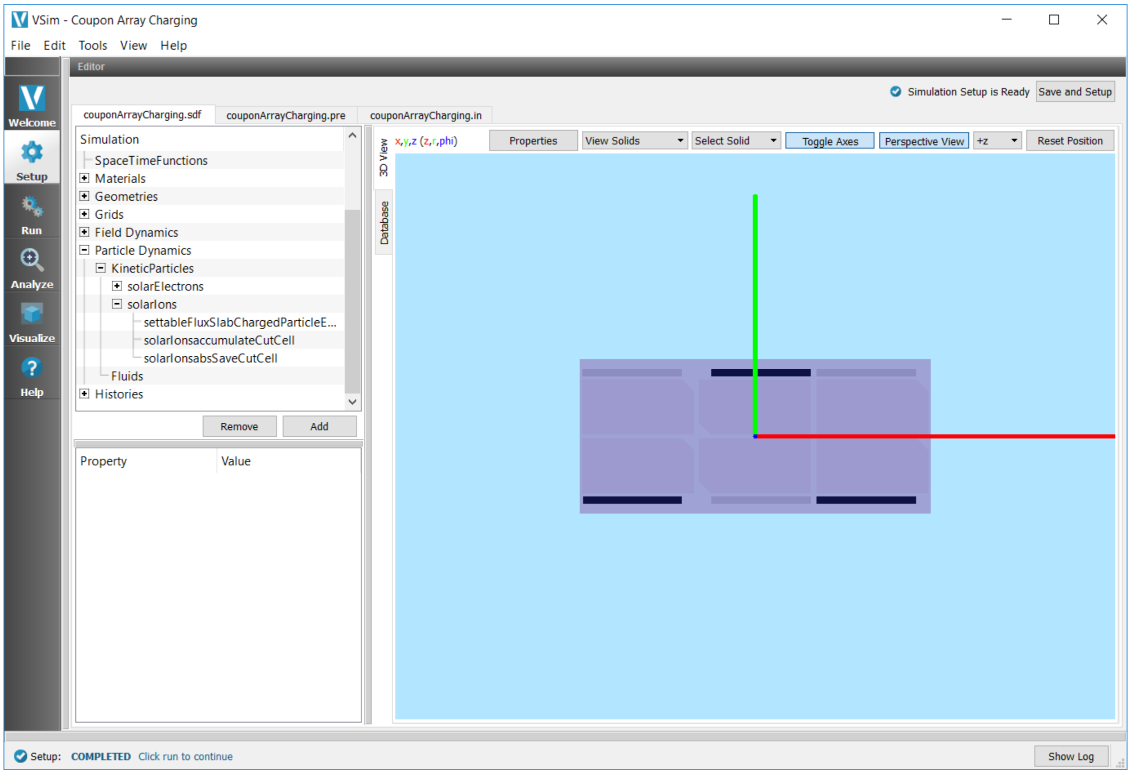
This simulation can be performed with a VSimPD license.

Opening the Simulation

The Electron Drifting example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Spacecraft* option.
- Select "Coupon Array Charging" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The *Setup Window* as seen after opening the example is shown in Fig. 6.62.



Simulation Setup is Ready

Setup: COMPLETED Click run to continue

Fig. 6.62: Setup Window for the Coupon Array Charging example.

Simulation Properties

The geometry for the busbars and coupon array are imported from stl files. Material properties are set on the geometries: perfect electrical conductor (PEC) for the busbars, and absorbium, a insulating particle absorbing material, on the array of cells.

A voltage of 5 volts is set on the busbars. The upper z boundary is set as the $V = 0$ point, a Neumann boundary condition is set on the lower z boundary of the simulation grid, which enforces that the gradient of the electric field normal to this surface is zero. Periodic boundary conditions (for particles and fields) are set on all other simulation boundaries.

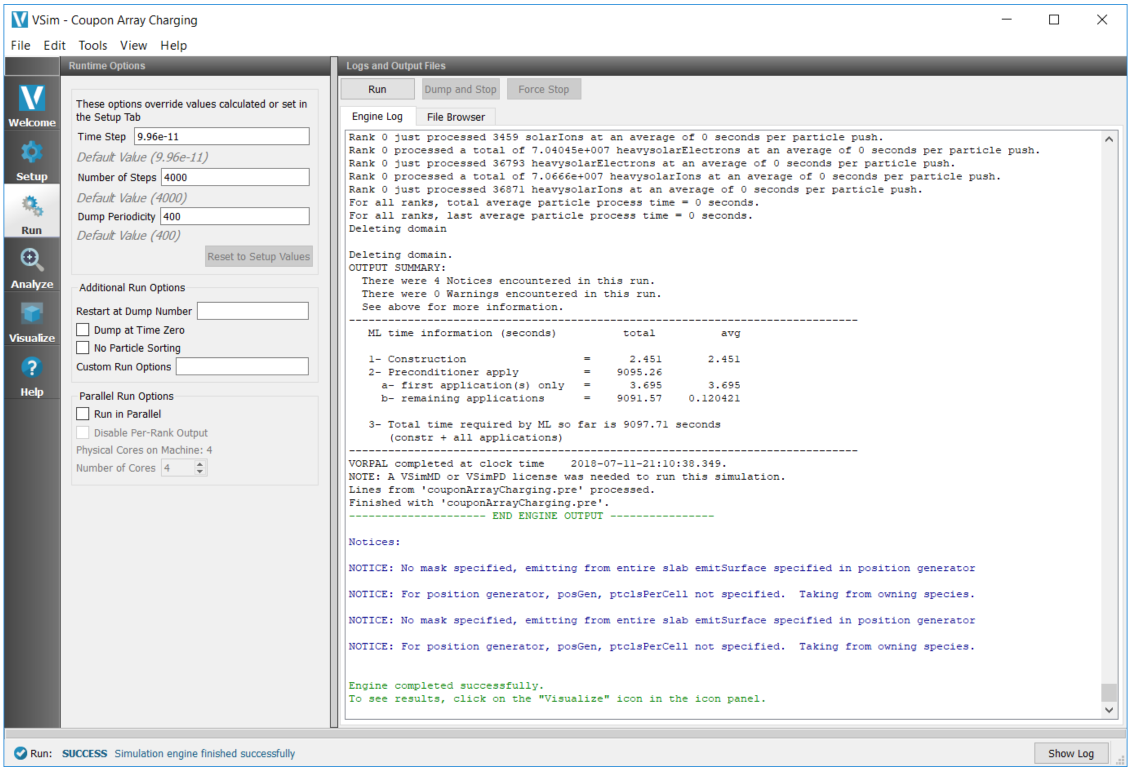
The solar wind is emitted off the upper z boundary of the simulation domain with a number density of 1.e7 particles per meter cubed. The masses of the ions are artificially set to 100x the mass of the electrons. Particle accumulation boundary conditions are set on the insulating surface of the coupons, and a particle absorbing boundary condition is set on the metal busbars.

Histories save the absorbed particle energy deposited onto the satellite surface.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in this pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 6.63 below.



These options override values calculated or set in the Setup Tab

Time Step 9.96e-11
Default Value (9.96e-11)
Number of Steps 4000
Default Value (4000)
Dump Periodicity 400
Default Value (400)

Additional Run Options
Restart at Dump Number
□ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options
□ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
Rank 0 just processed 3459 solarIons at an average of 0 seconds per particle push.
Rank 0 processed a total of 7.04045e+007 heavysolarElectrons at an average of 0 seconds per particle push.
Rank 0 just processed 36793 heavysolarElectrons at an average of 0 seconds per particle push.
Rank 0 processed a total of 7.0666e+007 heavysolarIons at an average of 0 seconds per particle push.
Rank 0 just processed 36871 heavysolarIons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 4 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
  See above for more information.
------------------------------------------------------------------------------
   ML time information (seconds)          total          avg

   1- Construction                  =      2.451       2.451
   2- Preconditioner apply          =    9095.26
     a- first application(s) only   =      3.695       3.695
     b- remaining applications      =    9091.57    0.120421

   3- Total time required by ML so far is 9097.71 seconds
      (constr + all applications)
------------------------------------------------------------------------------
VORPAL completed at clock time    2018-07-11-21:10:38.349.
NOTE: A VSimMD or VSimPD license was needed to run this simulation.
Lines from 'couponArrayCharging.pre' processed.
Finished with 'couponArrayCharging.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: No mask specified, emitting from entire slab emitSurface specified in position generator

NOTICE: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.

NOTICE: No mask specified, emitting from entire slab emitSurface specified in position generator

NOTICE: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

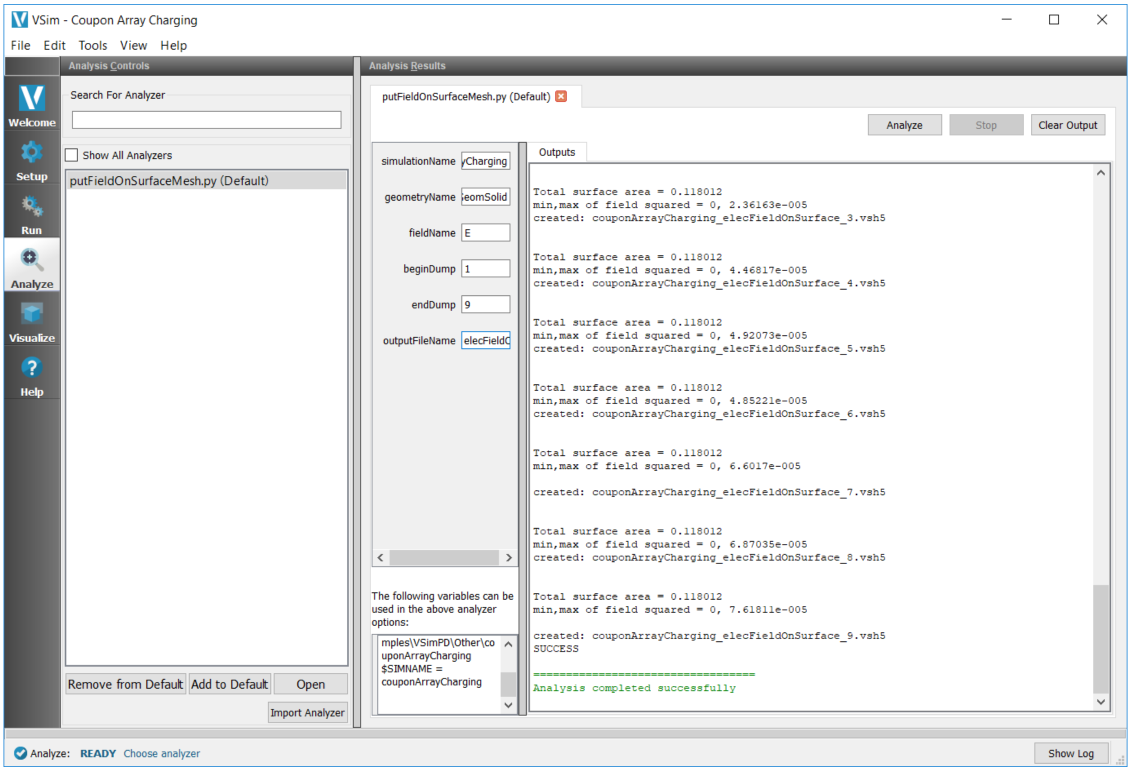
Fig. 6.63: The Run Window at the end of execution.

Analyzing the Results

The physics engine, vorpal, inside VSim only calculates field values on edges, nodes, or faces of grid cells. The *putFieldOnSurfaceMesh.py* analyzer can interpolate the values calculated on the grid to the surface of a geometry in the simulation.

To calculate the normal component of the electric field on the surface of the array, proceed to the *Analyze* Tab. The *putFieldOnSurfaceMesh.py* analyzer is included by default to this simulation. Click on the text "putFieldOnSurfaceMesh.py (Default)" to highlight it, then click the "Open" button at the bottom of the *Analysis Controls* pane. Ensure the following is entered into each field:

- **`simulationName`:** "couponArrayCharging"
- **`geometryName`:** "satelliteSurfaceGeomSolid"
- **`fieldName`:** "E"
- **`beginDump`:** "1"
- **`endDump`:** "9"
- **`outputFileName`:** "elecFieldOnSurface"
- Click *Analyze* in upper right corner of the window. When the analysis is finished, you should see a window similar to Fig. 6.64.



simulationName yCharging

geometryName ;eomSolid

fieldName E

beginDump 1

endDump 9

outputFileName elecFieldC

The following variables can be used in the above analyzer options:

```
mples\VSimPD\Other\co
uponArrayCharging
$SIMNAME =
couponArrayCharging
```

```
Total surface area = 0.118012
min,max of field squared = 0, 2.36163e-005
created: couponArrayCharging_elecFieldOnSurface_3.vsh5


Total surface area = 0.118012
min,max of field squared = 0, 4.46817e-005
created: couponArrayCharging_elecFieldOnSurface_4.vsh5


Total surface area = 0.118012
min,max of field squared = 0, 4.92073e-005
created: couponArrayCharging_elecFieldOnSurface_5.vsh5


Total surface area = 0.118012
min,max of field squared = 0, 4.85221e-005
created: couponArrayCharging_elecFieldOnSurface_6.vsh5


Total surface area = 0.118012
min,max of field squared = 0, 6.6017e-005

created: couponArrayCharging_elecFieldOnSurface_7.vsh5


Total surface area = 0.118012
min,max of field squared = 0, 6.87035e-005
created: couponArrayCharging_elecFieldOnSurface_8.vsh5


Total surface area = 0.118012
min,max of field squared = 0, 7.61811e-005

created: couponArrayCharging_elecFieldOnSurface_9.vsh5
SUCCESS

===================================
Analysis completed successfully
```

Analyze: READY Choose analyzer

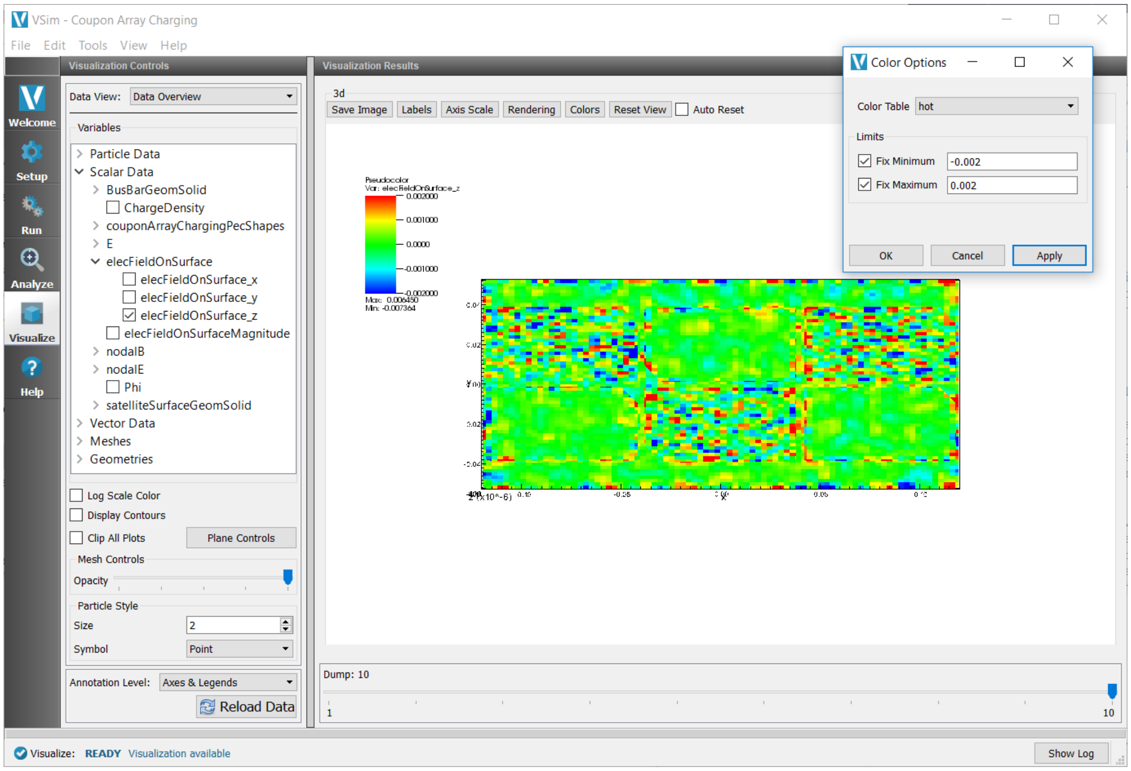
Fig. 6.64: The Analyze Window at the end of execution.

Visualizing the Results

After run completion, continue as follows:

Proceed to the Visualize Tab by pressing the Visualize button in the left column of buttons. To view the normal component of the electric field on the surface of the array follow the following steps.

- If you have previously switched to the Visualize Tab, you will have to click the *Reload Data* button at the bottom of the *Visualization Controls* pane.
- Expand “Scalar Data” then expand “elecFieldOnSurface” and check the box for “elecFieldOnSurface_z” to plot the component of the electric field normal to the surface of the coupon array. It is also possible to plot the two tangential components of the field, as well as the magnitude.
- To visualize the field better, click the “Colors” button in the row of buttons in the upper right of the *Visualization Results* pane. The *Color Options* window will open.
- In the *Color Options* window, check the boxes for “Fix Minimum” and “Fix Maximum” and set the values to “-0.002” and “0.002,” respectively. Press the “Apply” Button to apply the color range.
- The resulting visualization is shown in Fig. 6.7.1.



VSim - Coupon Array Charging

Data View: Data Overview

Variables

- Particle Data
- Scalar Data
  - BusBarGeomSolid
  - ☐ ChargeDensity
  - couponArrayChargingPecShapes
  - E
  - elecFieldOnSurface
    - ☐ elecFieldOnSurface_x
    - ☐ elecFieldOnSurface_y
    - ☑ elecFieldOnSurface_z
  - ☐ elecFieldOnSurfaceMagnitude
  - nodalB
  - nodalE
  - ☐ Phi
  - satelliteSurfaceGeomSolid
- Vector Data
- Meshes
- Geometries

☐ Log Scale Color

☐ Display Contours

☐ Clip All Plots

Particle Style: Size 2, Symbol Point

Annotation Level: Axes & Legends

Pseudocolor
Var: elecFieldOnSurface_z

Max: 0.006450
Min: -0.007364

Color Options

Color Table: hot

Limits

☑ Fix Minimum: -0.002

☑ Fix Maximum: 0.002

Dump: 10

Visualize: READY Visualization available

Further Experiments

Perform the same analysis done above for the electric potential, “Phi,” and charge density, “ChargeDensity,” to create plots of those fields on the surface of the satellite geometry.

Import your own geometry and reset the materials, grid size, and particle absorbers as necessary.

Increase the grid resolution to get finer data on the electric field that develops on the surface of the spacecraft.

Change the number density and speeds of the incident particles to values for orbits at different altitudes.

6.7.2 Cylindrical Hall Thruster (cylHallThruster.sdf)

Keywords:

electric propulsion, Hall thruster channel, erosion models.

Problem description

Electric Hall thrusters are used for in-space propulsion and satellite station-keeping needs. The discharge plasma inside the Hall thruster channel is produced by the ionization of electrons with a neutral propellant gas such as xenon. The electrons are emitted from the neutralizer cathode placed at the exit of the Hall thruster (cathode end). The neutral gas is fed into the channel from the anode end of the Hall thruster channel. The electrons are confined inside the Hall thruster channel by the radial magnetic field applied through the solenoidal magnetic fields. Plasma xenon ions are accelerated out of the channel at high velocity, which produces the thrust necessary for space propulsion. Recently these thrusters are being designed to support long life time, high-power and high-thrust operations. The channel wall erosion occurring inside of the Hall thruster is one of the main limitations to these design needs. It becomes important to understand the plasma discharge processes occurring inside the Hall thruster channel and predict the lifetime of the Hall thruster based on the calculations of sputtered material from the Hall thruster channel.

This example demonstrates elements of the full cylindrical Hall thruster text-based example. Please refer to that documentation for a detailed description of the simulation geometry, and physical properties of the full model. This visual-setup model employs the same physical geometry (2D, cylindrical coordinates) as the text-based setup model. As with the text-based setup example, there is an electron source and a background xenon gas that is ionized by kinetic electrons to produce kinetic xenon ion particles. Particle sinks, and the physical extent of the background gas are the same as in the text-based example.

The primary difference is that there are no dielectric materials in this example, and so the electromagnetic fields are different. This leads to a different pattern for electrons exiting the simulation than in the text-based example. Also, there is no neutral particle sputtering of wall material included in this example. This example does demonstrate ionization of neutral gas in cylindrical geometry.

This simulation can be performed with a VSimPD license.

Opening the Simulation

The Cylindrical Hall Thruster example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Spacecraft* option.
- Select *Cylindrical Hall Thruster* and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

All of the properties and values that create the simulation are now available in the Setup Window as shown in Fig. 6.65. You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. To show or hide the grid, expand the Grid element and select or deselect the box next to `Grid`.

Simulation Properties

This example contains many user defined *Constants* which help simplify the setup and make it easy to modify. These include constants such as:

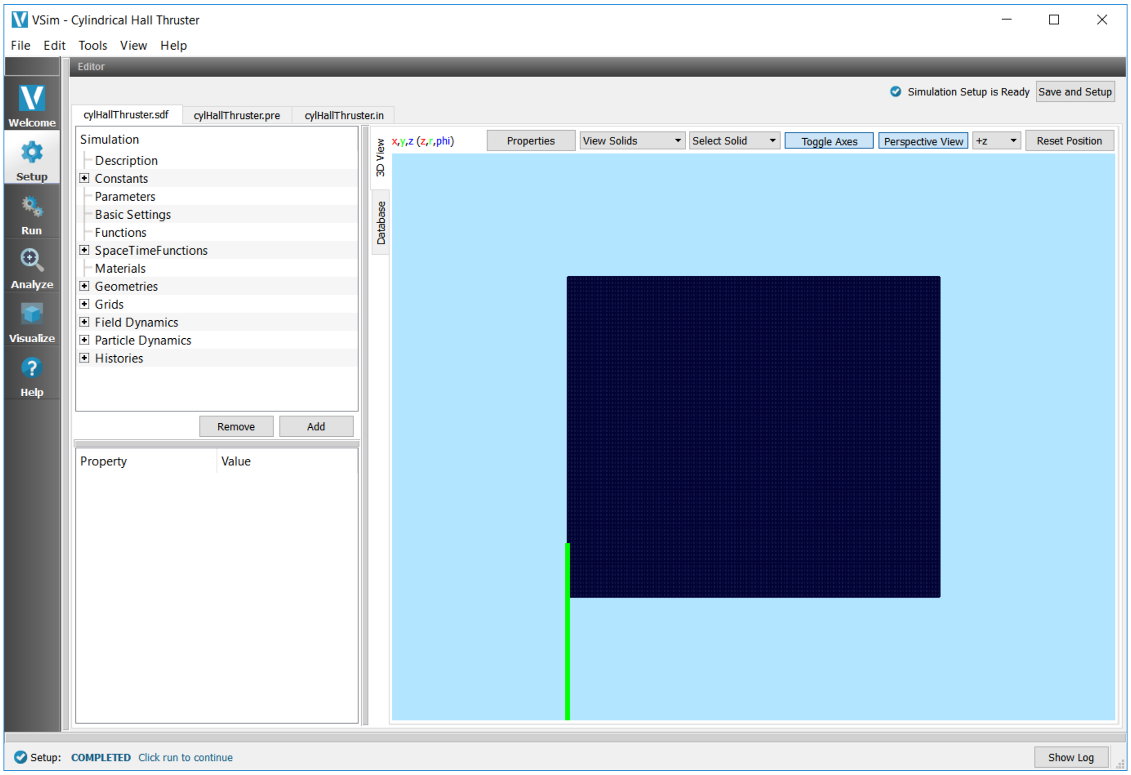


Fig. 6.65: Setup Window for the Cylindrical Hall Thruster Channel example.

- B0: The amplitude of the background magnetic field
- anodeV: the anode voltage
- innerRad and outerRad: inner and outer cylinder radius
- xeMaxDensity: the maximum density of the background Xe gas

There are also several *SpaceTimeFunctions* that are used to define spatially and/or temporally varying inputs to other properties. These include:

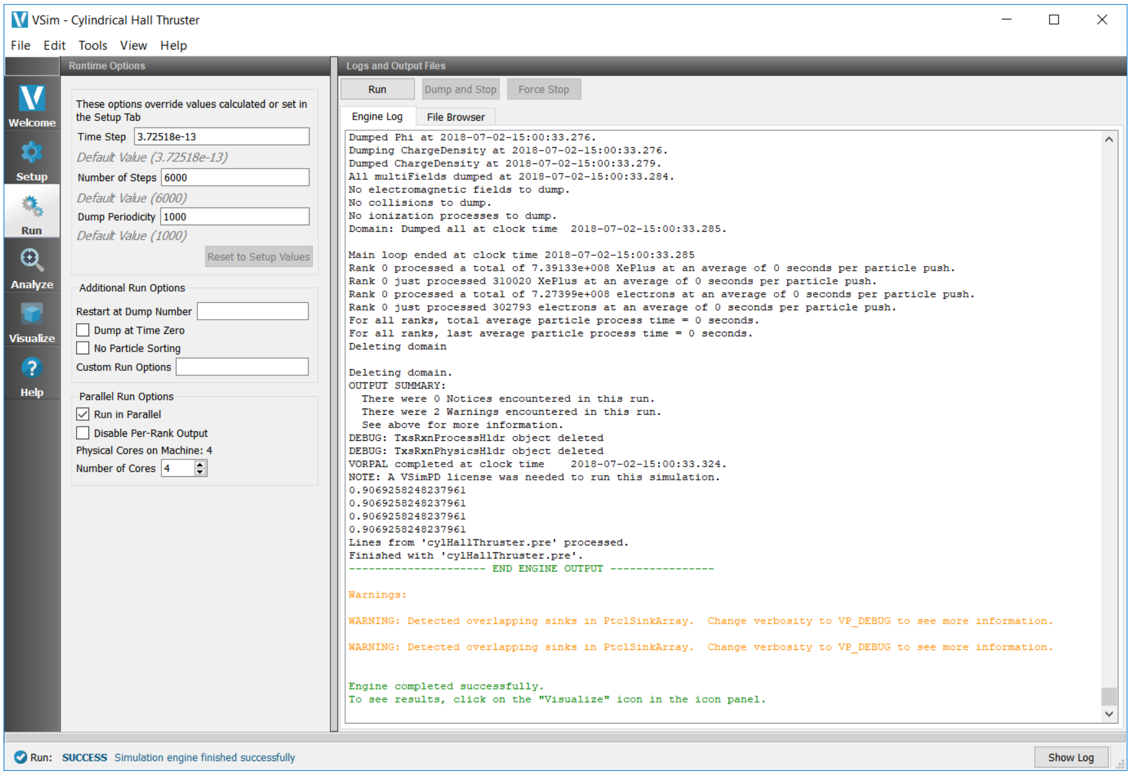
- By: the magnetic field profile
- initialGasDensity: the profile for the background gas density

The self-consistent electric field is solved from Poisson's equation by the electrostatic solver in a cylindrical coordinate system. The simulation is performed in axisymmetric 2-D fashion. The plasma is represented by macro-particles which are moved using the Boris pusher in cylindrical coordinate system. Various types of elastic and inelastic collisions of the particles are also taken into account.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the *Logs and Output Files* pane. You will see the output of the run in this pane. The run has completed when you see the output, "Engine completed successfully." A snapshot of the simulation run completion is shown in Fig. 6.66.



These options override values calculated or set in the Setup Tab

Time Step 3.72518e-13
Default Value (3.72518e-13)
Number of Steps 6000
Default Value (6000)
Dump Periodicity 1000
Default Value (1000)

Additional Run Options

Restart at Dump Number
☐ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
☐ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
Dumped Phi at 2018-07-02-15:00:33.276.
Dumping ChargeDensity at 2018-07-02-15:00:33.276.
Dumped ChargeDensity at 2018-07-02-15:00:33.279.
All multiFields dumped at 2018-07-02-15:00:33.284.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-02-15:00:33.285.

Main loop ended at clock time 2018-07-02-15:00:33.285
Rank 0 processed a total of 7.39133e+008 XePlus at an average of 0 seconds per particle push.
Rank 0 just processed 310020 XePlus at an average of 0 seconds per particle push.
Rank 0 processed a total of 7.27399e+008 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 302793 electrons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 0 Notices encountered in this run.
  There were 2 Warnings encountered in this run.
  See above for more information.
DEBUG: TxsRxnProcessHldr object deleted
DEBUG: TxsRxnPhysicsHldr object deleted
VORPAL completed at clock time    2018-07-02-15:00:33.324.
NOTE: A VSimPD license was needed to run this simulation.
0.9069258248237961
0.9069258248237961
0.9069258248237961
0.9069258248237961
Lines from 'cylHallThruster.pre' processed.
Finished with 'cylHallThruster.pre'.
--------------------- END ENGINE OUTPUT ----------------

Warnings:

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 6.66: The Run Window at the end of execution.

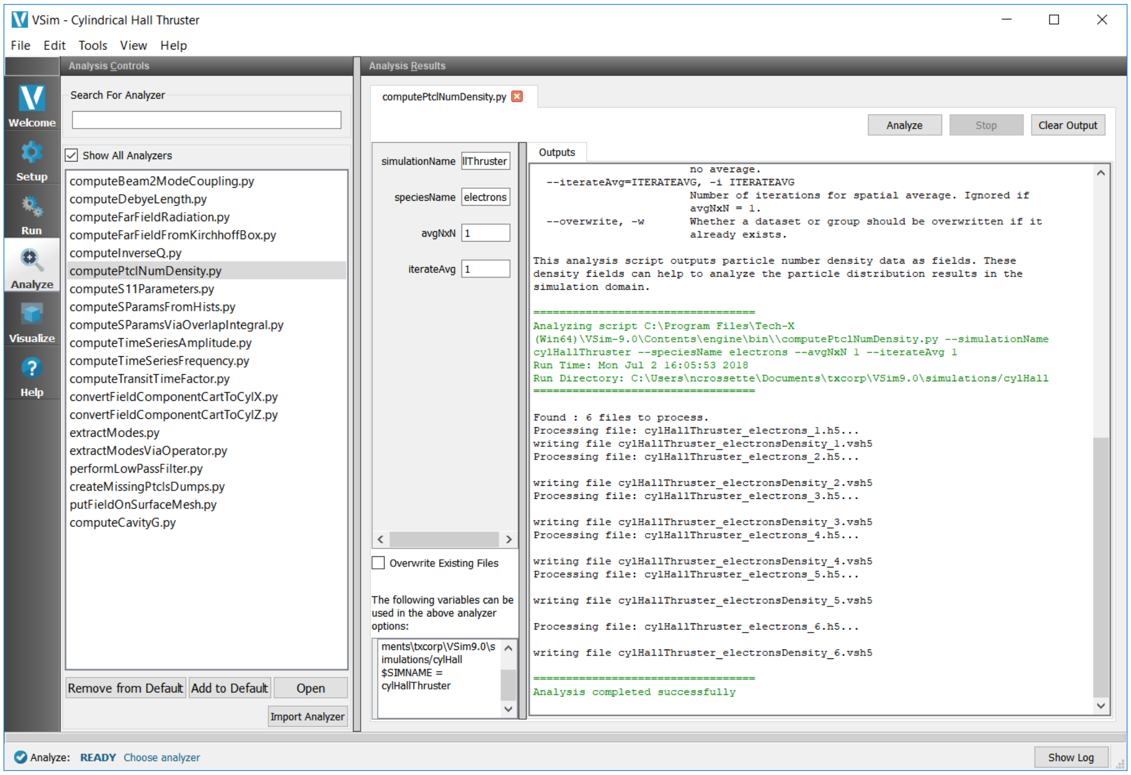
On an 8 core I7-6000 CPU, it takes about 40 minutes to run for 6,000 time steps. To reach steady state, about 100,000 time steps are required.

Analyzing the Results

If it is desired to calculate the density of the electrons or ions the analysis script *computePtclNumDensity.py* must be used.

- First click on the *Analyze* Tab.
- Click *Show All Analyzers* and choose *computePtclNumDensity.py*. Then click *Open*.
- This script accepts the *simulationName* (Name of the input file) and *speciesName* to be calculated (species of particles).
- To calculate the density of the electrons, set the simulationName to "cylHallThruster" and the speciesName to "electrons".
- Click on the *Analyze* button at the top right of the *Analysis Results* pane.
- A snapshot of the simulation run completion is shown in Fig. 6.67.

The resulting data will be visualizable as *electronsDensity* under the *Scalar Data* menu in the *Visualize* Tab. The density of other particles such as heavyIons, or Xeplus can also be calculated if those species names are used in place of electrons.


VSim - Cylindrical Hall Thruster
File Edit Tools View Help
Analysis Controls
Welcome
Setup
Run
Analyze
Visualize
Help
Search For Analyzer
☑ Show All Analyzers
computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
createMissingPtclsDumps.py
putFieldOnSurfaceMesh.py
computeCavityG.py
Remove from Default
Add to Default
Open
Import Analyzer
Analysis Results
computePtclNumDensity.py
Analyze
Stop
Clear Output
Outputs
Analyze: READY Choose analyzer
Show Log

simulationName llThruster

speciesName electrons

avgNxN 1

iterateAvg 1

□ Overwrite Existing Files

The following variables can be used in the above analyzer options:

ments\txcorp\VSim9.0\simulations/cylHall
$SIMNAME =
cylHallThruster

```
                       no average.
  --iterateAvg=ITERATEAVG, -i ITERATEAVG
                       Number of iterations for spatial average. Ignored if
                       avgNxN = 1.
  --overwrite, -w      Whether a dataset or group should be overwritten if it
                       already exists.

This analysis script outputs particle number density data as fields. These
density fields can help to analyze the particle distribution results in the
simulation domain.

==================================
Analyzing script C:\Program Files\Tech-X
(Win64)\VSim-9.0\Contents\engine\bin\\computePtclNumDensity.py --simulationName
cylHallThruster --speciesName electrons --avgNxN 1 --iterateAvg 1
Run Time: Mon Jul 2 16:05:53 2018
Run Directory: C:\Users\ncrossette\Documents\txcorp\VSim9.0\simulations/cylHall
==================================

Found : 6 files to process.
Processing file: cylHallThruster_electrons_1.h5...
writing file cylHallThruster_electronsDensity_1.vsh5
Processing file: cylHallThruster_electrons_2.h5...

writing file cylHallThruster_electronsDensity_2.vsh5
Processing file: cylHallThruster_electrons_3.h5...

writing file cylHallThruster_electronsDensity_3.vsh5
Processing file: cylHallThruster_electrons_4.h5...

writing file cylHallThruster_electronsDensity_4.vsh5
Processing file: cylHallThruster_electrons_5.h5...

writing file cylHallThruster_electronsDensity_5.vsh5

Processing file: cylHallThruster_electrons_6.h5...

writing file cylHallThruster_electronsDensity_6.vsh5

==================================
Analysis completed successfully
```

Fig. 6.67: The Analyze Window at the end of execution.

Visualizing the Results

To visualize the results, continue as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

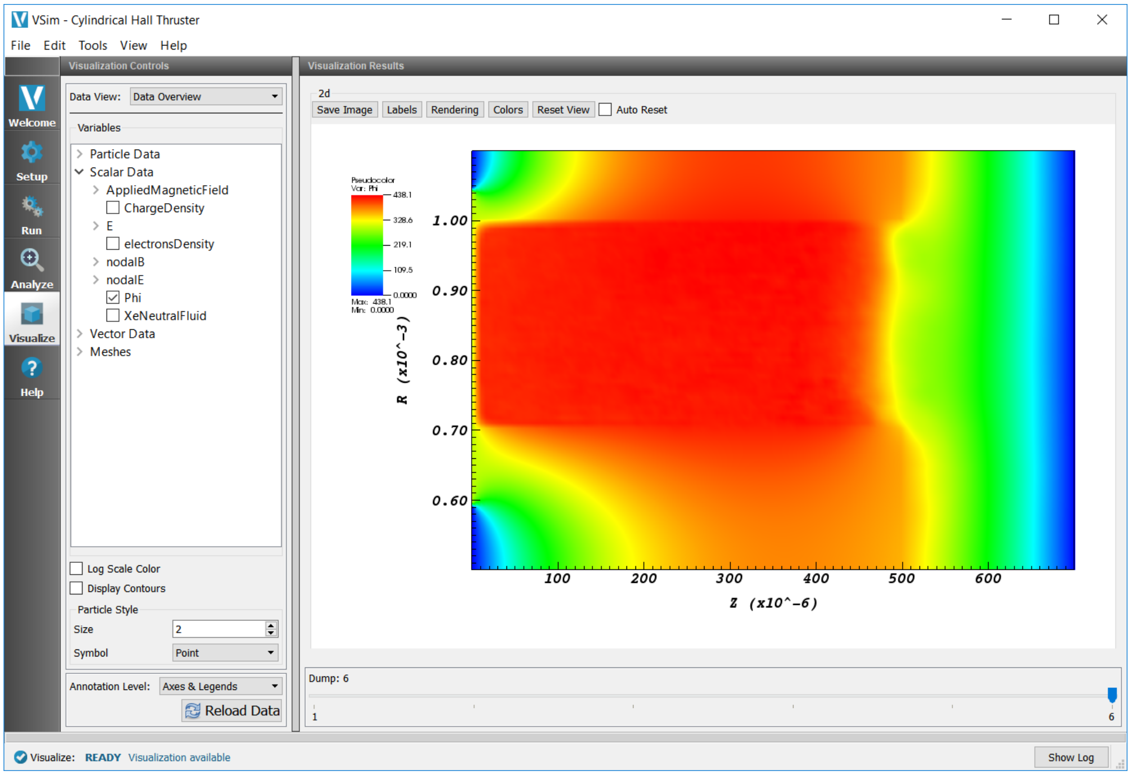
There are many different fields, particles, and histories that can be visualized in this example. The horizontal axis represents Z direction and the vertical axis represents R direction.

To view the electric potential, switch to the *Data View* to *Data Overview* in the Controls pane. Expand *Scalar Data* and choose *Phi*. Move the *Dump* slider to the right most position. Fig. 6.68 shows the visualization seen for the electric potential of the cylindrical Hall thruster channel and in the exit region.

In the Hall thruster channel plasma, the electrons injected from the right end (i.e., exit of the channel) are accelerated towards the anode biased wall at the left end. To plot the particles, expand *Particle Data* and select "electrons" and "XePlus" check boxes The figure below, Fig. 6.69, shows the distribution of positively charged xenon ions (red dots) and electrons (blue dots).

The static radial magnetic field distribution considered for the SPT-100 Hall thruster channel set up is shown in Fig. 6.70. To reproduce this plot, expand *Scalar Data* then *nodalB* and select *nodalB_r*. The magnetic field is strong near the inner cylinder and has a Gaussian bell-shaped field distribution both inside and at the exit of the channel.

The background xenon neutral gas density distribution (plottable as *XeNeutralFluid* under *Scalar Data*) used in the simulation set up is shown in Fig. 6.71. The maximum neutral gas density is taken at the left end of the channel near the anode wall. A linearly varying neutral gas density is assumed.



VSim - Cylindrical Hall Thruster

Data View: Data Overview

Variables

- Particle Data
- Scalar Data
  - AppliedMagneticField
  - □ ChargeDensity
  - E
  - □ electronsDensity
  - nodalB
  - nodalE
  - ☑ Phi
  - □ XeNeutralFluid
- Vector Data
- Meshes

□ Log Scale Color

□ Display Contours

Particle Style

Size: 2

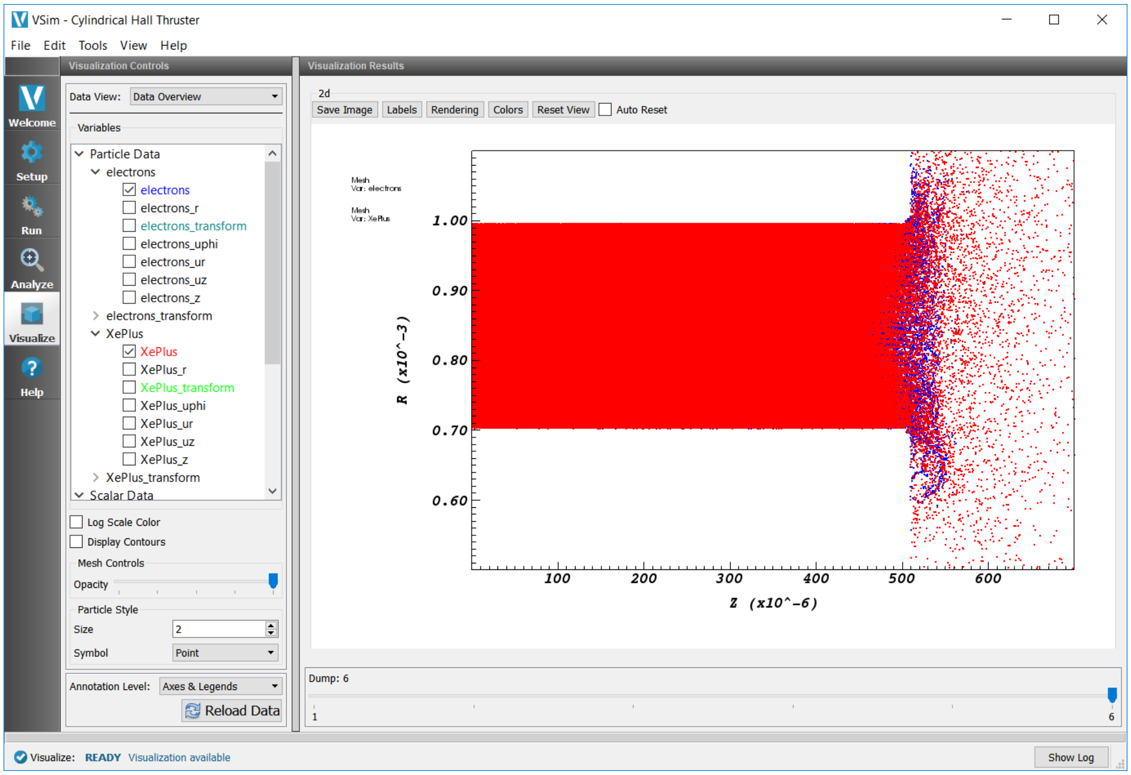
Symbol: Point

Annotation Level: Axes & Legends

Dump: 6

Visualize: READY Visualization available

Fig. 6.68: Visualization of Cylindrical Hall thruster channel electric potential results.



VSim - Cylindrical Hall Thruster

Data View: Data Overview

Variables

- Particle Data
  - electrons
    - ☑ electrons
    - □ electrons_r
    - □ electrons_transform
    - □ electrons_uphi
    - □ electrons_ur
    - □ electrons_uz
    - □ electrons_z
  - electrons_transform
  - XePlus
    - ☑ XePlus
    - □ XePlus_r
    - □ XePlus_transform
    - □ XePlus_uphi
    - □ XePlus_ur
    - □ XePlus_uz
    - □ XePlus_z
  - XePlus_transform
- Scalar Data

□ Log Scale Color

□ Display Contours

Mesh Controls

Opacity

Particle Style

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

Dump: 6

Visualize: READY Visualization available

Fig. 6.69: Visualization of Cylindrical Hall thruster channel particle phase-space distribution results.

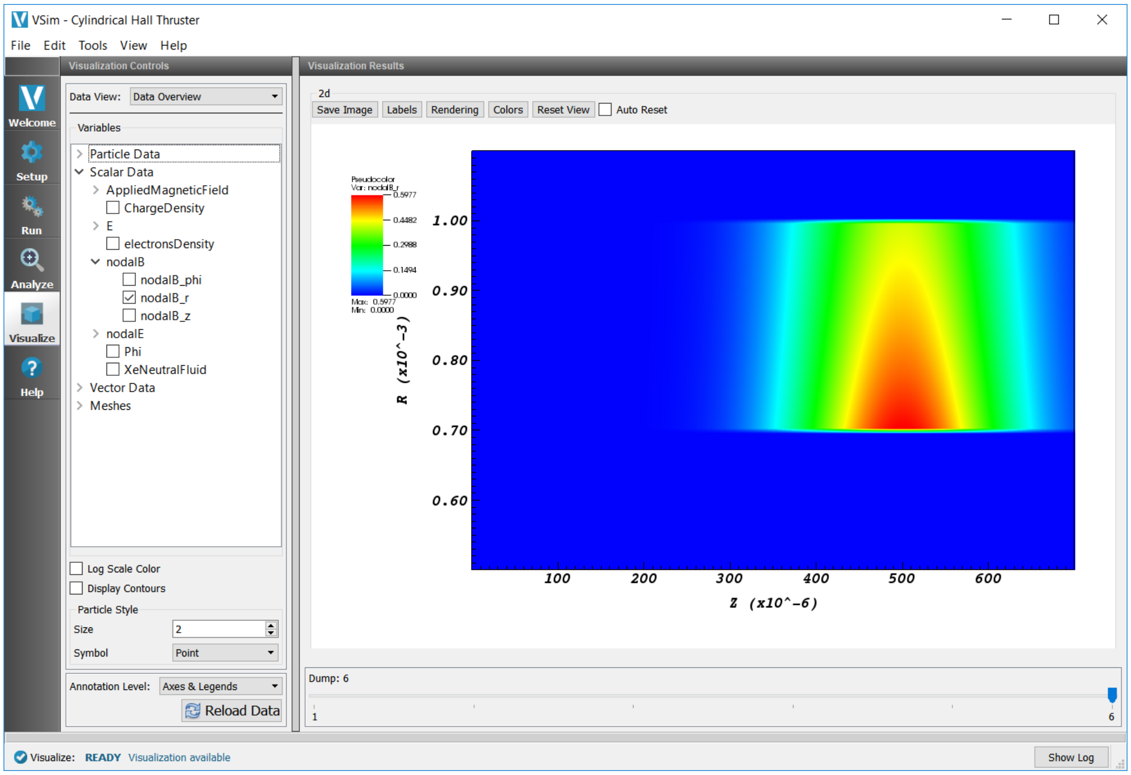


Fig. 6.70: Visualization of radial magnetic field in the Cylindrical Hall thruster channel.

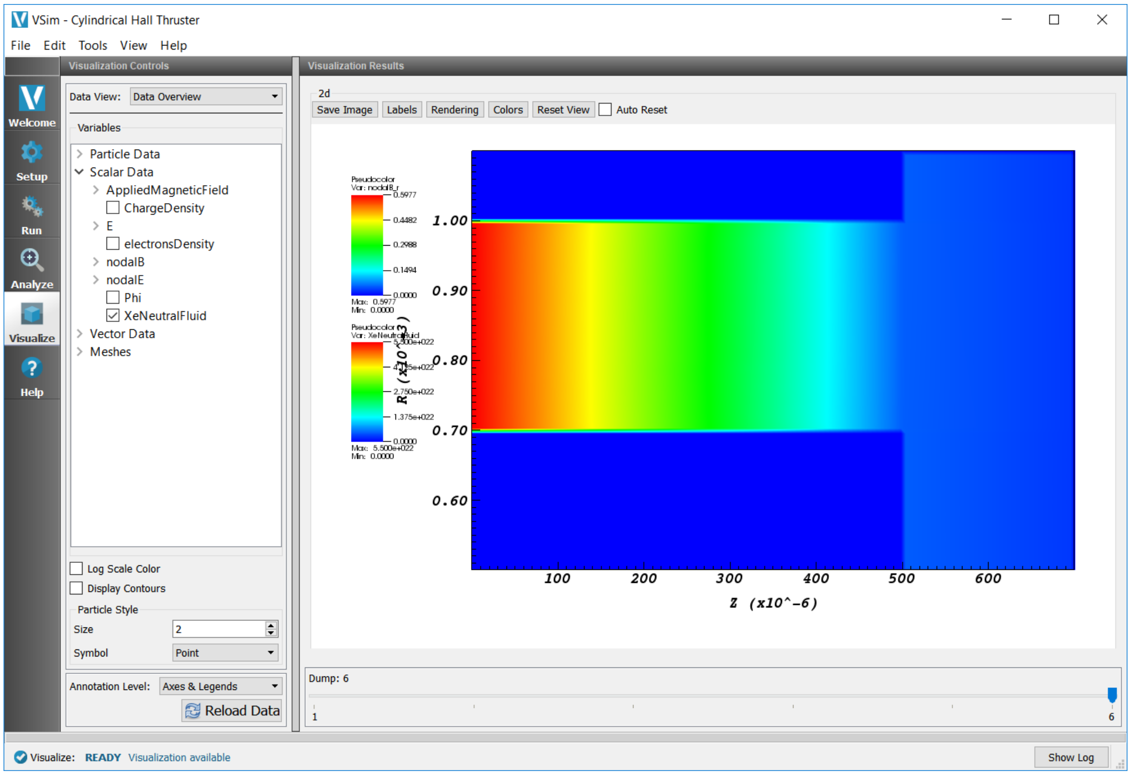


Fig. 6.71: Visualization of xenon neutral gas density in the Cylindrical Hall thruster channel.

Further Experiments

This example can be modified to test different design parameters such as varying anode voltages, varying background neutral gas densities and varying electron emission currents. This will allow users to study high-to-low power and high-to-low throttle levels.

Also the background gas type can be changed to investigate other neutral gas kinds in this simulation set up.

6.8 Spacecraft (text-based setup)

6.8.1 Cylindrical Hall Thruster (cylHallThrusterT.pre)

Keywords:

`electric propulsion, Hall thruster channel, erosion models.`

Problem description

Electric Hall thrusters are used for in-space propulsion and satellite station-keeping needs. The discharge plasma inside the Hall thruster channel is produced by the ionization of electrons with a neutral propellant gas such as xenon. The electrons are emitted from the neutralizer cathode placed at the exit of the Hall thruster (cathode end). The neutral gas is fed into the channel from the anode end of the Hall thruster channel. The electrons are confined inside the Hall thruster channel by the radial magnetic field applied through the solenoidal magnetic fields. Plasma xenon ions are accelerated out of the channel at high velocity, which produces the thrust necessary for space propulsion. Recently these thrusters are being designed to support long lifetime, high-power and high-thrust operations. The channel wall erosion occurring inside of the Hall thruster is one of the main limitations to these design needs. It is important to understand the plasma discharge processes occurring inside the Hall thruster channel to predict the lifetime of the Hall thruster based on the calculations of sputtered material from the Hall thruster channel.

This example demonstrates the xenon discharge plasma processes of a Stationary Plasma Thruster (SPT-100) channel. The outer cylinder has a radius of 5 cm and the inner cylinder has a radius of 3.5 cm. The radial channel gap is 1.5 cm. The channel length is 2.5 cm and the simulation domain extends 1 cm outside the channel region (to simulate both the channel and the plume plasma). The anode on the left wall is biased to 300 V. Both inner and outer cylinders are modeled as dielectric cylinders (with hexagonal boron-nitride dielectric material coating) with a dielectric permittivity ratio of 4.6. The exit boundary is set to 0 V. An electron source is placed at the channel exit to simulate electron emission from the neutralizer cathode and the cathode emission current is set to 4.5 A. Neutral xenon gas is modeled as a static fluid background. The neutral gas density is set to be linearly decreasing with a maximum gas density at the anode end of the channel. The simulation is initiated from a uniform plasma with both electrons and xenon ions. Self-similar scaling system laws for the simulation of Hall thruster channel are enabled as described by Figure 1 in *[MCL+11]*. This is based on earlier work by Taccogna *[TLCS04][TLCS05]*. The scale factor is set to 1/50, i.e., the thruster dimensions are scaled by 1/50. This scaling is followed so that the kinetic Hall thruster channel plasma simulations can be performed in a reasonable run time. The scaling affects the physical dimensions and external potentials.

This simulation can be performed with a VSimPD license.

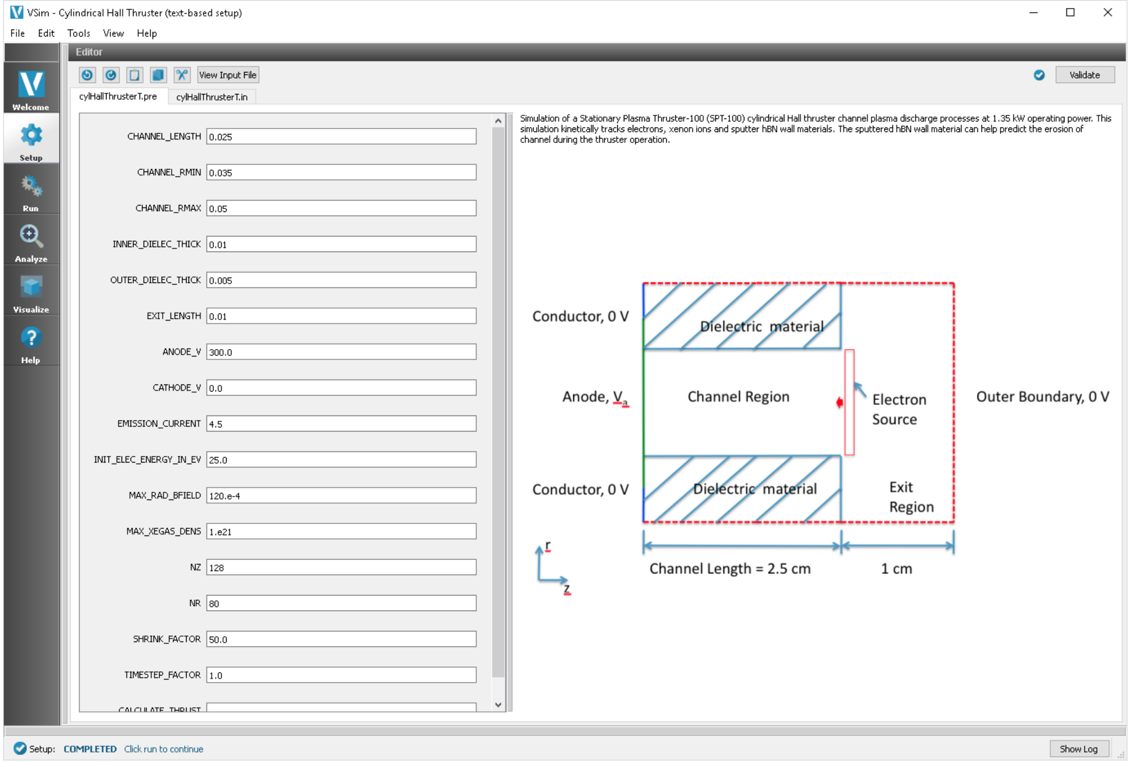
Opening the Simulation

The cylindrical Hall thruster example is accessed from within VSimComposer by the following actions:

- Select the *New → From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.

- Expand the *Spacecraft (text-based setup)* option.
- Select "Cylindrical Hall Thruster (text-based setup)" and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup* pane, as shown in Fig. 6.72.



CHANNEL_LENGTH 0.025
CHANNEL_RMIN 0.035
CHANNEL_RMAX 0.05
INNER_DIELEC_THICK 0.01
OUTER_DIELEC_THICK 0.005
EXIT_LENGTH 0.01
ANODE_V 300.0
CATHODE_V 0.0
EMISSION_CURRENT 4.5
INIT_ELEC_ENERGY_IN_EV 25.0
MAX_RAD_BFIELD 120.e-4
MAX_XEGAS_DENS 1.e21
NZ 128
NR 80
SHRINK_FACTOR 50.0
TIMESTEP_FACTOR 1.0
CALCULATE_THRUST

Simulation of a Stationary Plasma Thruster-100 (SPT-100) cylindrical Hall thruster channel plasma discharge processes at 1.35 kW operating power. This simulation kinetically tracks electrons, xenon ions and sputter hBN wall materials. The sputtered hBN wall material can help predict the erosion of channel during the thruster operation.

Setup: COMPLETED Click run to continue

Fig. 6.72: Setup Window for the Cylindrical Hall Thruster Channel example.

Input File Features

The input file allows one to choose the Hall thruster channel dimensions (length, inner and outer cylinder radius, inner cylinder dielectric wall thickness, outer cylinder dielectric wall thickness, channel exit region length) and physical quantities (anode voltage, cathode voltage, emission current, emitted electron energy, radial magnetic field strength, neutral number density), the resolution (number of cells in each direction) and *SHRINK_FACTOR*, the factor by which the size of the thruster is reduced in each dimension in the simulation.

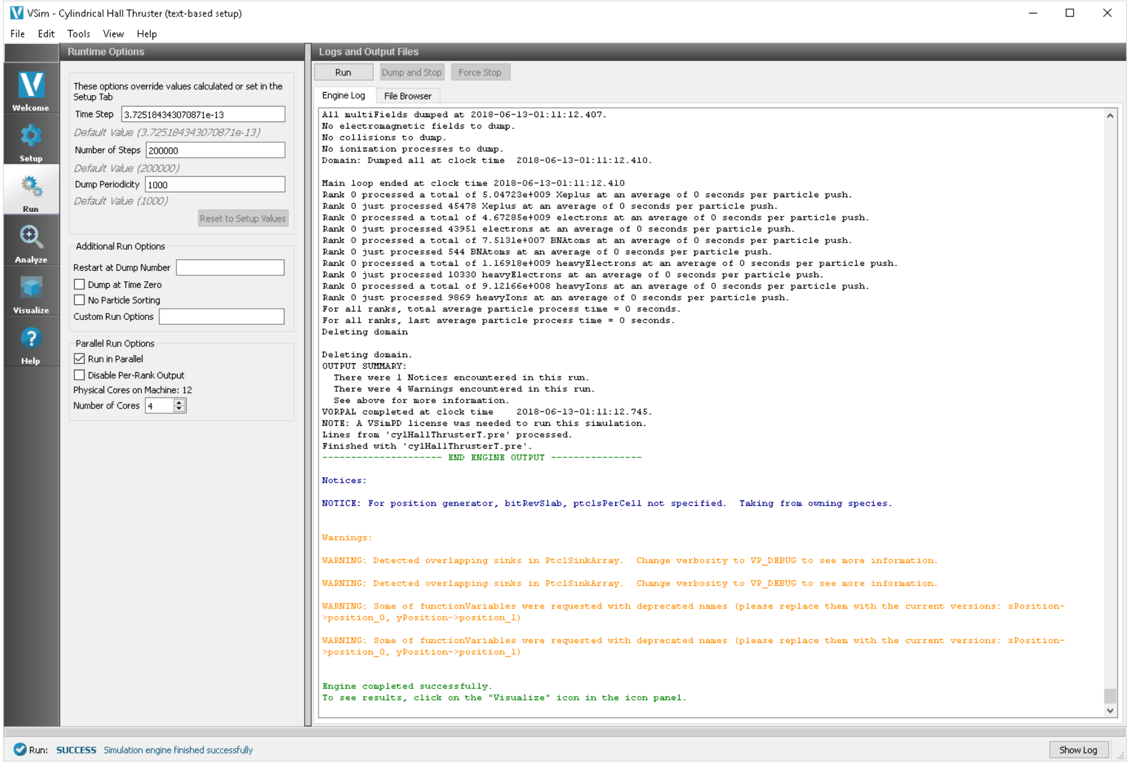
The self-consistent electric field is solved from Poisson's equation by the electrostatic solver in a cylindrical coordinate system. Dielectric coaxial cylinders are considered in this simulation. The simulation is performed in axisymmetric 2-D fashion.

The plasma is represented by macro-particles which are moved using the Boris scheme in a cylindrical coordinate system. Various types of elastic and inelastic collisions of the particles are calculated with the Vorpal engine's Monte Carlo package.

Running the simulation

After performing the above actions, continue as follows:

- Proceed to the run window by pressing the **Run** button in the left column of buttons.
- This simulation requires approximately 34 hours when running on four MPI cores on a modern CPU to simulate 200,000 time-steps. To view only the initial transient conditions, set *Number of Steps* to 1000 and *Dump Periodicity* to 10. Then the simulation can either be reset and rerun with the default run parameters, or by entering in the last dump into *Restart at Dump Number* (in this case, 100) the simulation can be restart from the chosen data dump.
- To run the file, click on the *Run* button in the upper right corner of the window. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully" as shown in Fig. 6.73.



These options override values calculated or set in the Setup Tab

Time Step 3.725184343070871e-13
Default Value (3.725184343070871e-13)
Number of Steps 200000
Default Value (200000)
Dump Periodicity 1000
Default Value (1000)

Restart at Dump Number
☐ Dump at Time Zero
☐ No Particle Sorting
Custom Run Options

☑ Run in Parallel
☐ Disable Per-Rank Output
Physical Cores on Machine: 12
Number of Cores 4

```
All multiFields dumped at 2018-06-13-01:11:12.407.
No electromagnetic fields to dump.
No collisions to dump.
No ionization processes to dump.
Domain: Dumped all at clock time  2018-06-13-01:11:12.410.

Main loop ended at clock time 2018-06-13-01:11:12.410
Rank 0 processed a total of 5.04723e+009 Xeplus at an average of 0 seconds per particle push.
Rank 0 just processed 45478 Xeplus at an average of 0 seconds per particle push.
Rank 0 processed a total of 4.67285e+009 electrons at an average of 0 seconds per particle push.
Rank 0 just processed 43951 electrons at an average of 0 seconds per particle push.
Rank 0 processed a total of 7.5131e+007 BNAtoms at an average of 0 seconds per particle push.
Rank 0 just processed 544 BNAtoms at an average of 0 seconds per particle push.
Rank 0 processed a total of 1.16918e+009 heavyElectrons at an average of 0 seconds per particle push.
Rank 0 just processed 10330 heavyElectrons at an average of 0 seconds per particle push.
Rank 0 processed a total of 9.12166e+008 heavyIons at an average of 0 seconds per particle push.
Rank 0 just processed 9869 heavyIons at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 1 Notices encountered in this run.
  There were 4 Warnings encountered in this run.
  See above for more information.
VORPAL completed at clock time    2018-06-13-01:11:12.745.
NOTE: A VSimPD license was needed to run this simulation.
Lines from 'cylHallThrusterT.pre' processed.
Finished with 'cylHallThrusterT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: For position generator, bitRevSlab, ptclsPerCell not specified.  Taking from owning species.


Warnings:

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.

WARNING: Some of functionVariables were requested with deprecated names (please replace them with the current versions: xPosition-
>position_0, yPosition->position_1)

WARNING: Some of functionVariables were requested with deprecated names (please replace them with the current versions: xPosition-
>position_0, yPosition->position_1)


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: SUCCESS Simulation engine finished successfully

Fig. 6.73: The Run Window at the end of execution.

Analyzing the Results

If the electron density is desired the analysis script *computePtclNumDensity.py* may be used.

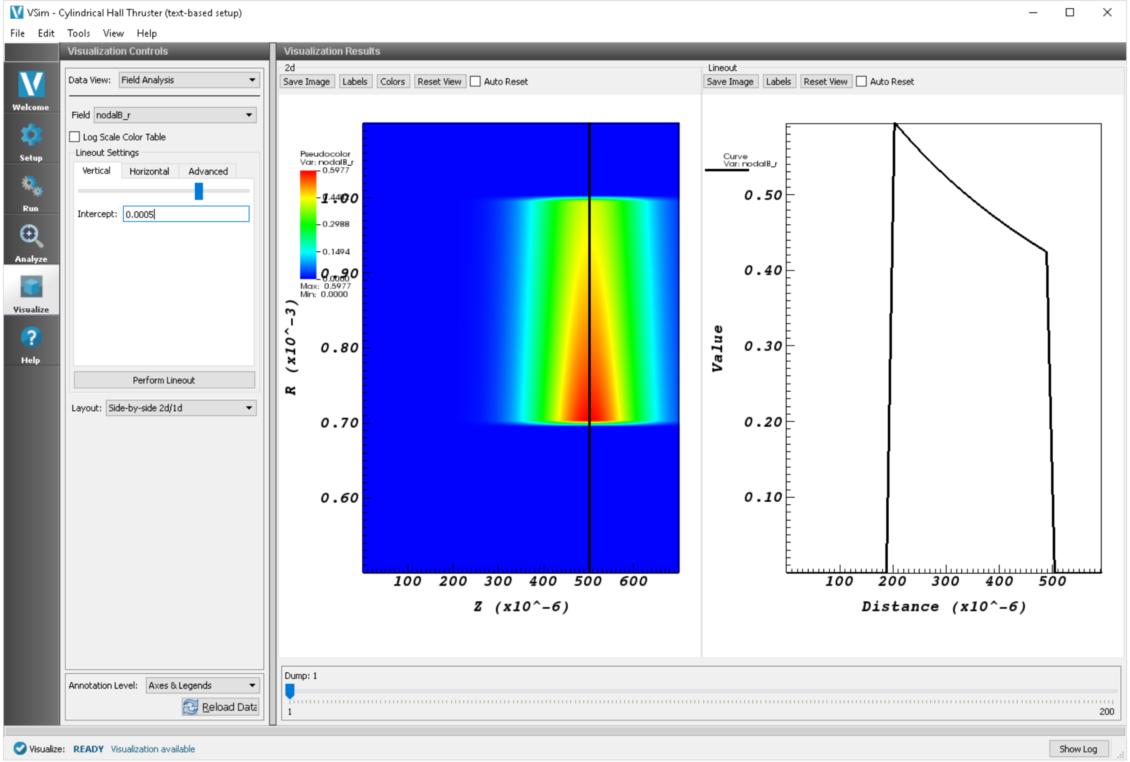
- In the leftmost panel, click the **Analyze** button. Select *Show All Analyzers* and then select *computePtclNumDensity.py* from the list of analyzers, then click *Open* at the bottom of the Analysis Controls pane.
- Enter "electrons" into the *speciesName* field.
- Click the *Analyze* button near the upper right of the Analysis Results pane.
- Repeat with other particle species if desired ("Xeplus", "XeNeutralsGas")

The analysis results are now viewable in the Visualize window, as shown in the following section.

Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize window by clicking the
- *Visualize** button in the leftmost panel.
- In the top of the *Visualization Controls* pane, switch the *Data View* dropdown menu to *Field Analysis*.
- In the *Field* dropdown menu, select *nodalB_r* to view the radial component of the magnetic field (the magnetic field is only radial). A pseudocolor plot of the potential with a radial lineout performed should be displayed as shown in Fig. 6.68. The magnetic field is strongest near the inner cylinder and falls off towards the outer cylinder.
- To plot the radial profile of the magnetic field at its strongest point, in the *Lineout Settings* section, set *Intercept* to "0.0005" and click *Perform Lineout*. If desired, select the *Advanced* tab to choose arbitrary start and end points for the lineout.


VSim - Cylindrical Hall Thruster (text-based setup)

Data View: Field Analysis

Field nodalB_r

□ Log Scale Color Table

Lineout Settings

Intercept: 0.0005

Layout: Side-by-side 2d/1d

Annotation Level: Axes & Legends

Dump: 1

Visualize: READY Visualization available

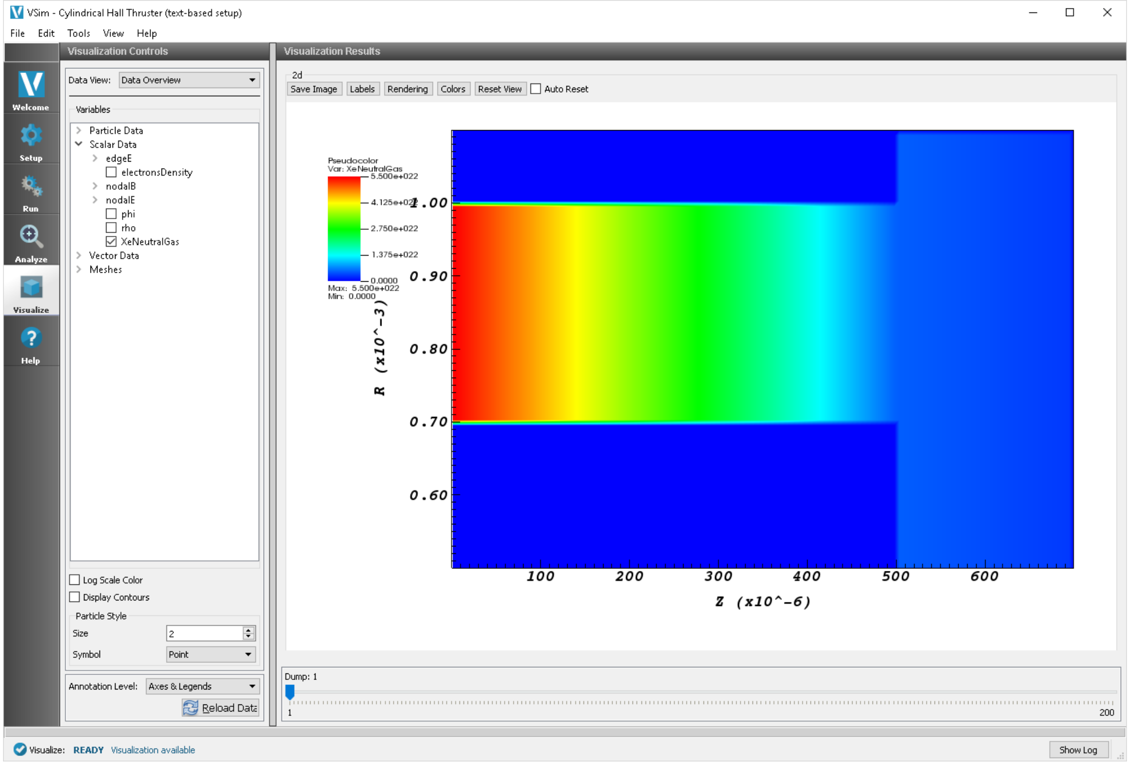
Fig. 6.74: Visualization of radial magnetic field in the cylindrical Hall thruster channel.

- To view the The background xenon neutral gas density, in the top of the *Visualization Controls* pane, switch the *Data View* dropdown menu to *Data Overview*.
- In the *Variables* section, expand *Scalar Data*.
- Select *XeNeutralGas*. A plot of the electron number density distribution will be displayed as shown in Fig. 6.75.

The maximum neutral gas density is set near the anode wall at the left end of the channel and decreases linearly until a minimum at channel opening at the right end of the channel.

To analyze the plasma sheath, plot the electric potential and particle positions by following the steps below:

- In the top of the *Visualization Controls* pane, switch the *Data View* dropdown menu to *Field Analysis*.



Data View: Data Overview

Variables
- Particle Data
- Scalar Data
  - edgeE
  - □ electronsDensity
  - nodalB
  - nodalE
  - □ phi
  - □ rho
  - ☑ XeNeutralGas
- Vector Data
- Meshes

□ Log Scale Color
□ Display Contours

Particle Style
Size: 2
Symbol: Point

Annotation Level: Axes & Legends

Dump: 1 (1 – 200)

Visualize: READY Visualization available

Fig. 6.75: Visualization of xenon neutral gas density in the cylindrical Hall thruster channel.

- In the *Field* dropdown menu, select *phi* to view the electric potential. A pseudocolor plot of the potential with a radial lineout performed should be displayed as shown in Fig. 6.76. The voltage is highest down the center of the channel.
- To plot the axial profile of the potential as shown in Fig. 6.76, in the *Lineout Settings* section, select the *Horizontal* tab, set *Intercept* to “0.00085” and click *Perform Lineout*. If desired, select the *Advanced* tab to choose arbitrary start and end points for the lineout.

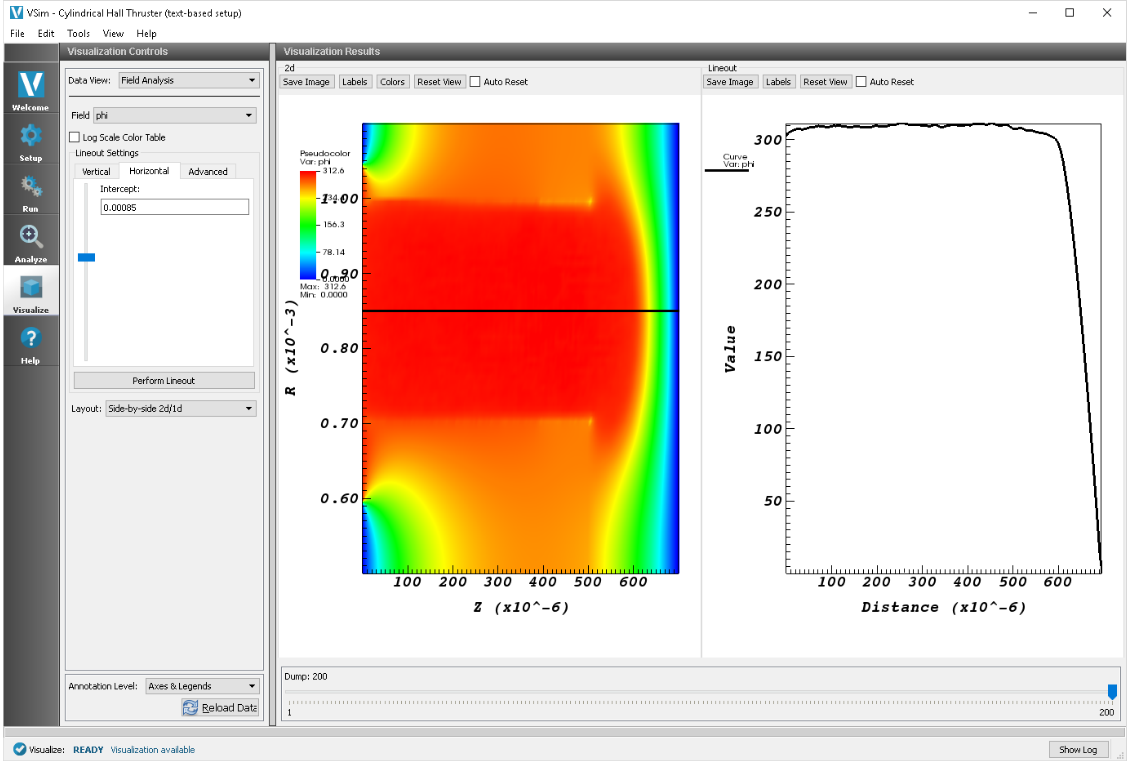
In the lineout, it can be seen that the potential of the plasma bulk is approximately 310 volts. The anode voltage is set to exactly 300 volts, so the potential drop at the anode sheath is approximately 10 volts, while the potential drop at the exit sheath is the same as the bulk plasma potential.

- To view the macroparticle locations, return to the *Data View* dropdown menu to *Data Overview*.
- In the Variables section, expand *Scalar Data* and select *phi*
- Expand *Particle Data*, then expand *Xeplus* and select *Xeplus_uz* to view the locations of xenon ion macroparticles colored by their axial (z) velocity.
- In the Particle Style section, set *Size* to 6 and *Symbol* to “Sphere”.
- Still under *Particle Data*, expand *electrons* and select *electrons* to view electron macroparticle positions monochromatically.

Plots of the electron and ion macroparticle positions overlayed on the electric potential should now be visible as shown in Fig. 6.77

- To plot the sputter material macroparticle locations, deselect the electrons and xenon ions, then expand *BNAtoms*, and select *BNAtoms*, as shown in Fig. 6.78

The sputtered material from the Hall Thruster channel can be shown by plotting the BNAtoms (red).



VSim - Cylindrical Hall Thruster (text-based setup)

Data View: Field Analysis

Field phi

□ Log Scale Color Table

Lineout Settings: Horizontal

Intercept: 0.00085

Layout: Side-by-side 2d/1d

Annotation Level: Axes & Legends

Dump: 200

Visualize: READY Visualization available

Fig. 6.76: Visualization of the cylindrical Hall thruster electric potential with a line-out showing the axial sheath structure.

Further Experiments

This input file can be modified to test different design parameters such as varying anode voltages, varying background neutral gas densities and varying electron emission currents. This will allow users to study high-to-low power and high-to-low throttle levels.

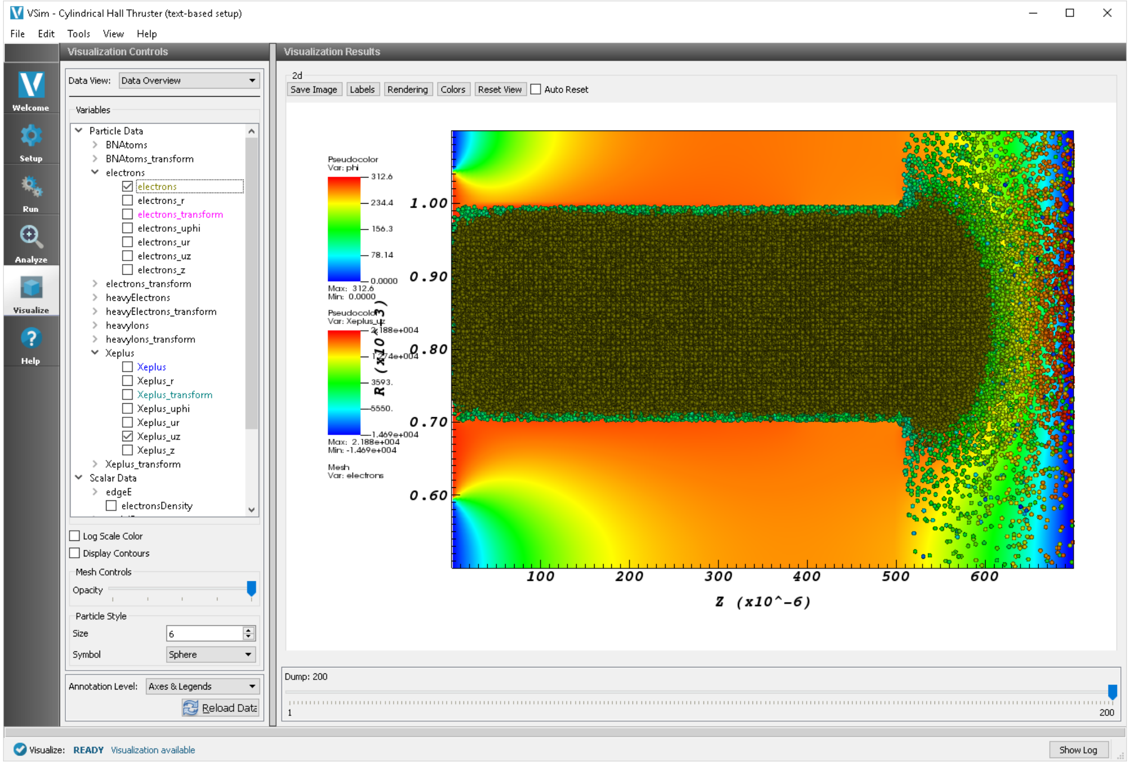
Also the background gas type can be changed to investigate other neutral gas kinds in this simulation set up.

A more substantial further step for a user would be to take the following steps to calculate the thrust from the device. This will require adding the history (see the code block below) to collect the appropriate data, writing a post-processing analysis script to calculate the thrust, then importing the analyzer into VSim.

- Return to the Setup window by clicking the **Setup** button in the leftmost panel
- Near the top of the Editor pane, click *View Input File*
- Scroll to the bottom of the input file and paste the following code:

```
<History absorbedIonVelocity>
  kind = speciesAbsPtclData2
  species = [ Xeplus ]
  ptclAbsorbers = [ topOuterIonAbsorber rightIonAbsorber botOuterIonAbsorber ]
  ptclAttributes = [ xVelocity numPtclsInMacro kineticEnergy xPosition yPosition ]
  collectMethod = statsForEachStep
</History>
```

the *collectMethod* = *statsForEachStep* provides sums of the desired quantities in the history file. The calculation thrust would be performed in the analysis tab. Other post-run analyses can be performed in order to see a sum of where the


VSim - Cylindrical Hall Thruster (text-based setup)

Data View: Data Overview

- Particle Data
  - BNAtoms
  - BNAtoms_transform
  - electrons
    - ☑ electrons
    - □ electrons_r
    - □ electrons_transform
    - □ electrons_uphi
    - □ electrons_ur
    - □ electrons_uz
    - □ electrons_z
  - electrons_transform
  - heavyElectrons
  - heavyElectrons_transform
  - heavyIons
  - heavyIons_transform
  - Xeplus
    - □ Xeplus
    - □ Xeplus_r
    - □ Xeplus_transform
    - □ Xeplus_uphi
    - □ Xeplus_ur
    - ☑ Xeplus_uz
    - □ Xeplus_z
  - Xeplus_transform
- Scalar Data
  - edgeE
    - □ electronsDensity

□ Log Scale Color

□ Display Contours

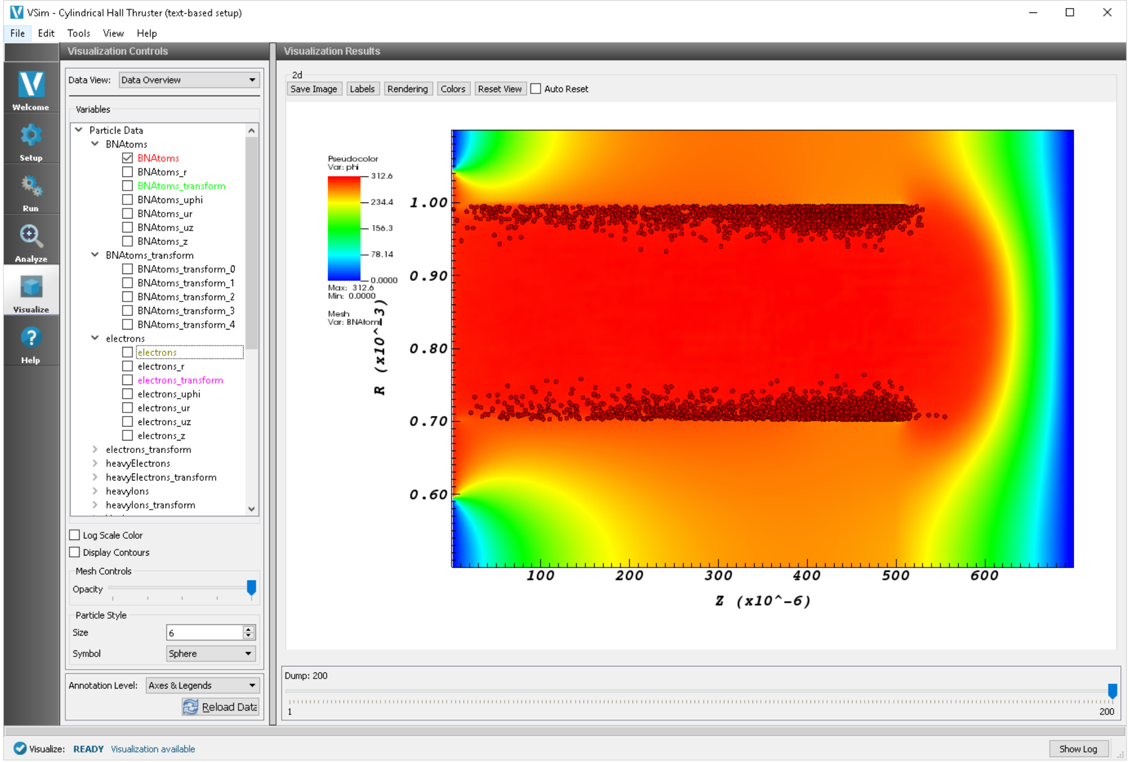
Particle Style: Size 6, Symbol Sphere

Annotation Level: Axes & Legends

Dump: 200

Visualize: READY Visualization available

Fig. 6.77: Visualization of the cylindrical Hall thruster sheath via ion and electron macroparticle positions with electric potential and ions colored according to their axial velocity.


VSim - Cylindrical Hall Thruster (text-based setup)

Data View: Data Overview

- Particle Data
  - BNAtoms
    - ☑ BNAtoms
    - □ BNAtoms_r
    - □ BNAtoms_transform
    - □ BNAtoms_uphi
    - □ BNAtoms_ur
    - □ BNAtoms_uz
    - □ BNAtoms_z
  - BNAtoms_transform
    - □ BNAtoms_transform_0
    - □ BNAtoms_transform_1
    - □ BNAtoms_transform_2
    - □ BNAtoms_transform_3
    - □ BNAtoms_transform_4
  - electrons
    - □ electrons
    - □ electrons_r
    - □ electrons_transform
    - □ electrons_uphi
    - □ electrons_ur
    - □ electrons_uz
    - □ electrons_z
  - electrons_transform
  - heavyElectrons
  - heavyElectrons_transform
  - heavyIons
  - heavyIons_transform

□ Log Scale Color

□ Display Contours

Particle Style: Size 6, Symbol Sphere

Annotation Level: Axes & Legends

Dump: 200

Visualize: READY Visualization available

Fig. 6.78: Visualization of the sputtered material from the channel walls.

particles are arriving, their velocity, and the creation of other figures of merit to determine the performance of the the thruster.

6.8.2 Ion Thruster (ionThrusterT.pre)

Keywords:

`electric ion thruster discharge chamber plasma processes in 2D cylindrical system.`

Problem Description

Ion thrusters are electric propulsion devices used for in-space propulsion and satellite station-keeping. In this device, a propellant gas (xenon in this example) is ionized into a plasma state inside a cylindrical discharge chamber. The plasma ions are accelerated out of the chamber through an electrostatic grid optics system to produce thrust. The device consists of the following components: an anode-biased discharge chamber, a discharge hollow cathode assembly, permanent magnet rings, a neutral propellant feed system, grid optics (screen and accelerator plates), and a neutralising hollow cathode. The discharge hollow cathode is placed in the center of the discharge chamber and emits energetic electrons (primary electrons) into the system. Primary electrons undergo ionizing collisions with the neutral propellant gas inside the chamber to produce plasma ions and secondary electrons. An energetic electron impacting a xenon atom that is already singly ionized may cause another electron to detach, resulting in a doubly ionized xenon atom. The permanent magnetic rings within the discharge chamber confine the electrons, increasing their time of flight in the chamber, and thus their chances of ionizing neutrals before collection at the anode-biased discharge chamber walls. The plasma ions produced in the chamber leave primarily through the screen grid plate with some losses to the cathode biased walls. To ensure long discharge cathode lifetimes, a protective enclosure called a cathode keeper (generally kept between 3 and 5 volts above the discharge cathode voltage) is used to shield the cathode plate from plasma ion collisions. The bombardment of singly charged and doubly charged ions during thruster operation will over time erode the face of the cathode keeper and expose the discharge cathode to energetic ions. Thus, it becomes important to model the ion flux around the cathodes. Recently ion thrusters have been designed to meet high-power and high-thrust-to-power space propulsion requirements. Numerical discharge chamber plasma simulations provide a detailed understanding of the plasma processes that go on inside a discharge chamber and help with the calculation of electron discharge currents, ion beam currents, and ion current losses to the chamber walls.

This example demonstrates the xenon discharge plasma processes inside of a cylindrical discharge chamber with a three-ring magnetic circuit arrangement. One magnetic ring is mounted on the forward wall (seen as the left wall in the geometry of the example) and two magnetic rings are mounted to the exterior wall of the cylindrical discharge chamber (seen as the top wall in the example setup). The radius of the cylindrical chamber is 20 cm and it is 18 cm long. The screen grid plate has a radius of 18 cm and is placed at the aft end of the discharge chamber (far right in the example setup). The discharge hollow cathode assembly is placed at the center of the discharge chamber. The radius of the cathode keeper assembly is taken to be 0.75 cm and its orifice protrudes out 7 cm from the forward wall (from the left wall in the example diagram). An electron particle source is implanted next to the cathode keeper orifice to model the electron emission of the discharge cathode. In this simulation the cathode emission current is taken as 10 A. The same cathode emission source location is also used for modelling the neutral propellant flow from the discharge cathode. The main xenon neutral propellant source is modelled along the exterior wall (top wall in the example diagram). We have taken neutral propellant flow rates of 4.5 sccm and 43.5 sccm for the discharge cathode neutral source and main neutral source respectively. The anode biased discharge chamber walls are kept at 25 V. The discharge cathode keeper is biased at 5 V and the screen grid plate is kept at 0 V. Finally, we enable a self-similar scaling system for the simulation of discharge chamber plasma described by figure 1 in *[MCL+11]*. This is based on earlier work by Taccogna *[TLCS04][TLCS05]*. By default the shrink scale factor is 200, i.e., the thruster dimensions are scaled by 1/200. This scaling ensures that simulations can be performed in a reasonable run time but it requires use of an inflated permittivity scale factor, i.e. the permittivity of free space is artificially inflated so that numerical parameters like grid spacing and time step values satisfy the smaller plasma frequency and Debye length.

The simulation is initiated with the chamber pre-filled with xenon neutrals. This is because the neutrals are heavy and slow, and it would take a great many time steps at the start of every run to populate an empty chamber. To view the initial distribution of neutrals, the input file can be run with particle sources turned-off. To do this, switch the TURN_THRUSTER_OFF parameter to 1 and run with a time step of ~10^-6 sec (timestep is 1 ps by default).

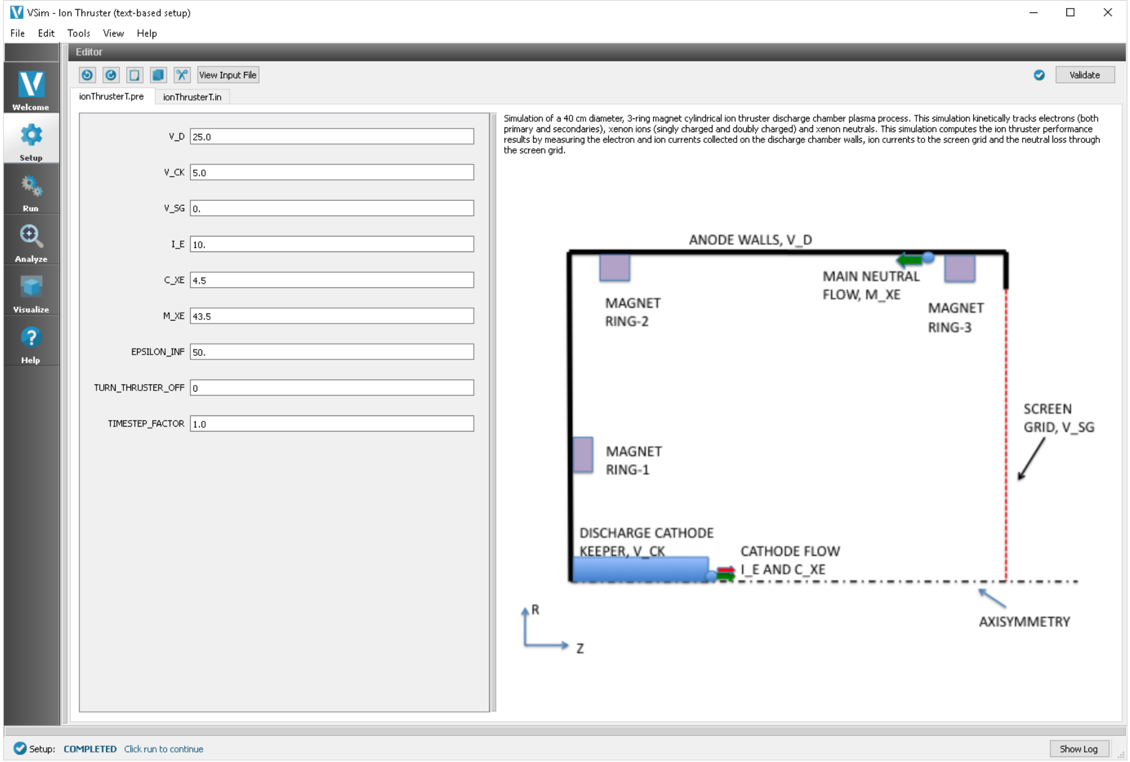
This simulation can be performed with a VSimPD license.

Opening the Simulation

The ion thruster example is accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item from the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Spacecraft (text-based setup)* option.
- Select *Ion Thruster (text-based setup)* and click the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the *Setup Window*, as shown in Fig. 6.79.



V_D 25.0

V_CK 5.0

V_SG 0.

I_E 10.

C_XE 4.5

M_XE 43.5

EPSILON_INF 50.

TURN_THRUSTER_OFF 0

TIMESTEP_FACTOR 1.0

Simulation of a 40 cm diameter, 3-ring magnet cylindrical ion thruster discharge chamber plasma process. This simulation kinetically tracks electrons (both primary and secondaries), xenon ions (singly charged and doubly charged) and xenon neutrals. This simulation computes the ion thruster performance results by measuring the electron and ion currents collected on the discharge chamber walls, ion currents to the screen grid and the neutral loss through the screen grid.

Setup: COMPLETED Click run to continue

Fig. 6.79: Setup Window for the Ion Thruster example.

Input File Features

The input file allows the user a choice of ion thruster operating parameters such as discharge voltage, cathode keeper voltage, screen grid voltage, discharge cathode electron emission current, cathode neutral flow rate, and main neutral flow rate. Also it gives the user an option to specify the inflated permittivity scale factor by which the real permittivity of free space is scaled.

The self-consistent electric field is solved from Poisson's equation by the electrostatic solver in cylindrical coordinates. The simulation is performed in an axisymmetric 2-D domain. The actual thruster dimensions are reduced by the `SHRINK_FACTOR` variable in the input file (default 200). Correspondingly the physical parameters such as electric fields, magnetic fields, and particle densities are scaled by the shrink factor to maintain consistent physical effects (e.g. Larmor radius, Knudsen number).

The plasma is represented by macro-particles which are moved via the Boris pusher in cylindrical coordinates. Various types of elastic and inelastic particle collisions are calculated with the computational engine's Monte Carlo package. In this simulation the propellant xenon neutrals are tracked as kinetic particles and undergo collisions with electrons. The simulation employs variable-weight particle splitting and self-combination via `NullInteraction` blocks to help maintain good particle resolution over orders-of-magnitude variations in density across the domain.

This input file contains an imported magnetic field. The external magnetic field file is in units of Gauss, and is converted into Teslas when imported by VSim.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the Run Window by pressing the **Run** button in the left column of buttons.
- Check the box labeled "Dump at Time Zero" so that the initial electric potential may be plotted.
- To run the file, click on the *Run* button in the upper right corner of the window. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." A snapshot of the simulation run completion is shown in Fig. 6.80.

The default number of time-steps for this simulation is 5,000, but to approach steady-state, approximately 500,000 time-steps are required. The *Visualizing the Results* section provides a review of the results at 5,000 time-steps, while the *Further Experiments* section is a review of the results after 500,000 time-steps.

Analyzing the Results

If the electron density is desired the analysis script *computePtclNumDensity.py* may be used.

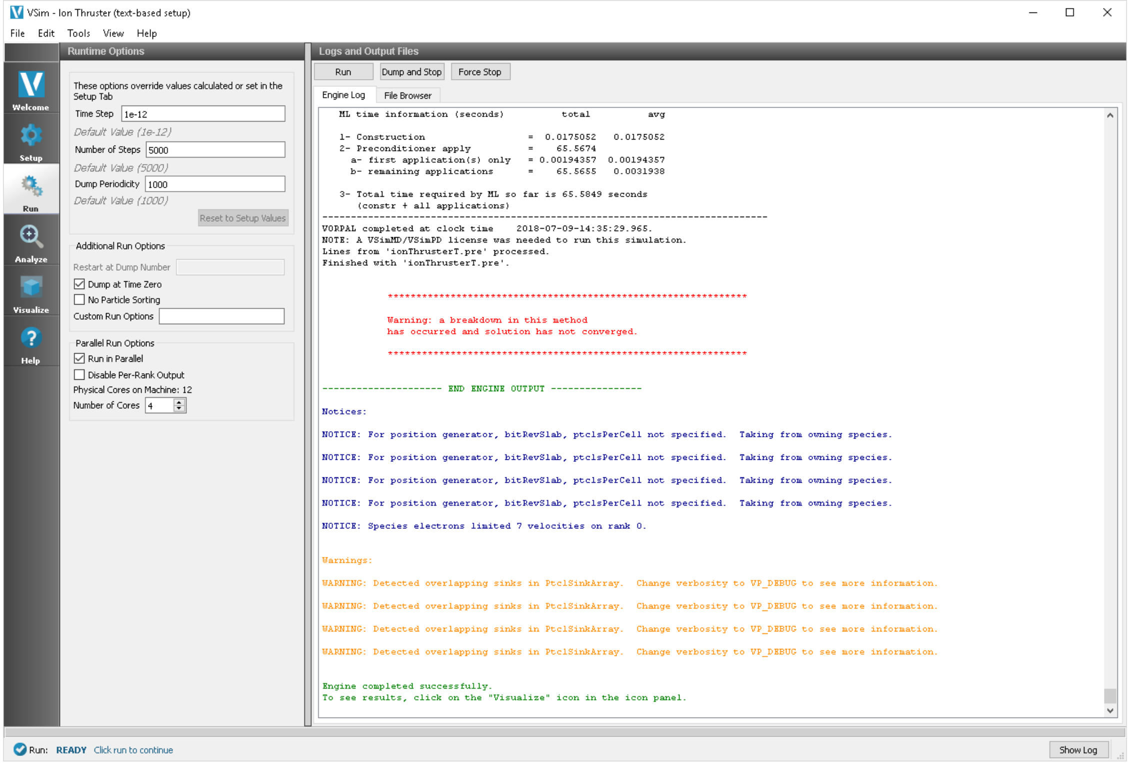
- In the leftmost panel, click the **Analyze** button. Select *Show All Analyzers* and then select *computePtclNumDensity.py* from the list of analyzers, then click *Open* at the bottom of the Analysis Controls pane.
- Enter "electrons" into the *speciesName* field.
- Click the *Analyze* button near the upper right of the Analysis Results pane.
- Repeat with other particle species if desired ("XeIons", "XeNeutrals")

The analysis results are now viewable in the Visualize window, as shown in the following section.

Visualizing the Results

After performing the above actions, continue as follows:

- Proceed to the Visualize window by clicking the **Visualize** button in the leftmost panel.
- In the top of the *Visualization Controls* pane, switch the *Data View* dropdown menu to *Field Analysis*.
- In the *Field* dropdown menu, select *phi*. A pseudocolor plot of the potential with a radial lineout performed should be displayed as shown in Fig. 6.81.



These options override values calculated or set in the Setup Tab

Time Step: 1e-12
Default Value (1e-12)

Number of Steps: 5000
Default Value (5000)

Dump Periodicity: 1000
Default Value (1000)

Additional Run Options

Restart at Dump Number:

☑ Dump at Time Zero

☐ No Particle Sorting

Custom Run Options:

Parallel Run Options

☑ Run in Parallel

☐ Disable Per-Rank Output

Physical Cores on Machine: 12

Number of Cores: 4

```
   ML time information (seconds)          total          avg

   1- Construction                  =  0.0175052   0.0175052
   2- Preconditioner apply          =    65.5674
     a- first application(s) only   = 0.00194357  0.00194357
     b- remaining applications      =    65.5655   0.0031938

   3- Total time required by ML so far is 65.5849 seconds
      (constr + all applications)
------------------------------------------------------------------------------
VORPAL completed at clock time    2018-07-09-14:35:29.965.
NOTE: A VSimMD/VSimPD license was needed to run this simulation.
Lines from 'ionThrusterT.pre' processed.
Finished with 'ionThrusterT.pre'.


           **************************************************************


           Warning: a breakdown in this method
           has occurred and solution has not converged.

           **************************************************************


-------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: For position generator, bitRevSlab, ptclsPerCell not specified.  Taking from owning species.

NOTICE: For position generator, bitRevSlab, ptclsPerCell not specified.  Taking from owning species.

NOTICE: For position generator, bitRevSlab, ptclsPerCell not specified.  Taking from owning species.

NOTICE: For position generator, bitRevSlab, ptclsPerCell not specified.  Taking from owning species.

NOTICE: Species electrons limited 7 velocities on rank 0.


Warnings:

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

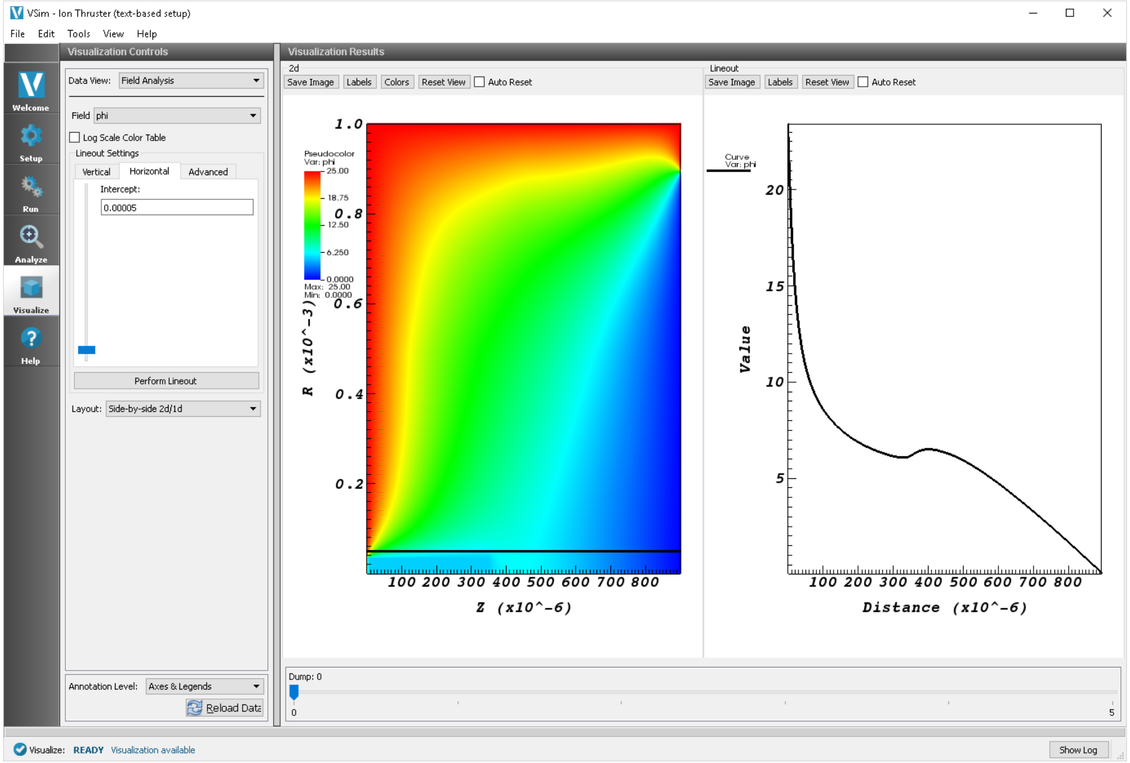
Run: READY Click run to continue

Fig. 6.80: The Run Window at the end of execution.

- To plot the axial potential profile, in the *Lineout Settings* section, select the *Horizontal* tab, change the intercept to `0.00005`, and click *Perform Lineout* to plot the axial accelerating potential as shown in Fig. 6.81. If desired, select the *Advanced* tab to choose arbitrary start and end points for the lineout.
- In the top of the *Visualization Controls* pane, switch the *Data View* dropdown menu to *Phase Space*.
- In the *Base Variable* dropdown menu, select *electrons*.
- To maintain the same z-r convention as the previous electric potential plot, in the *X-axis* dropdown menu select *electrons_z* and in the *Y-axis* dropdown menu select *electrons_r*.
- Near the bottom of the *Visualization Controls* pane click *DRAW* and at the bottom of the *Visualization Results* pane move the *Dump* slider to the right to dump 5. The z-r phase space should be visible as shown in Fig. 6.82.

Recall that the electron number density distribution was calculated in *Analyzing the Results*. Plot the results of this analyzer as follows:

- In the top of the *Visualization Controls* pane, switch the *Data View* dropdown menu to *Data Overview*.
- In the *Variables* section, expand *Scalar Data*.
- Select *electronDensity*. A plot of the electron number density distribution should be displayed, though due to the large variation in densities, only a small portion of the domain will appear to be non-zero
- At the bottom of the variables section of the Visualization Controls pane, select the *Log Scale Color* checkbox.
- At the top of the Visualization Results pane click the *Colors* button, and in the resulting dialog set the limits to a minimum of 1e16 and maximum of 1e22, or experiment with limits as desired. The electron density on a logarithmic color scale should now displayed as shown in Fig. 6.83.



VSim - Ion Thruster (text-based setup)

Data View: Field Analysis

Field: phi

□ Log Scale Color Table

Lineout Settings

Intercept: 0.00005

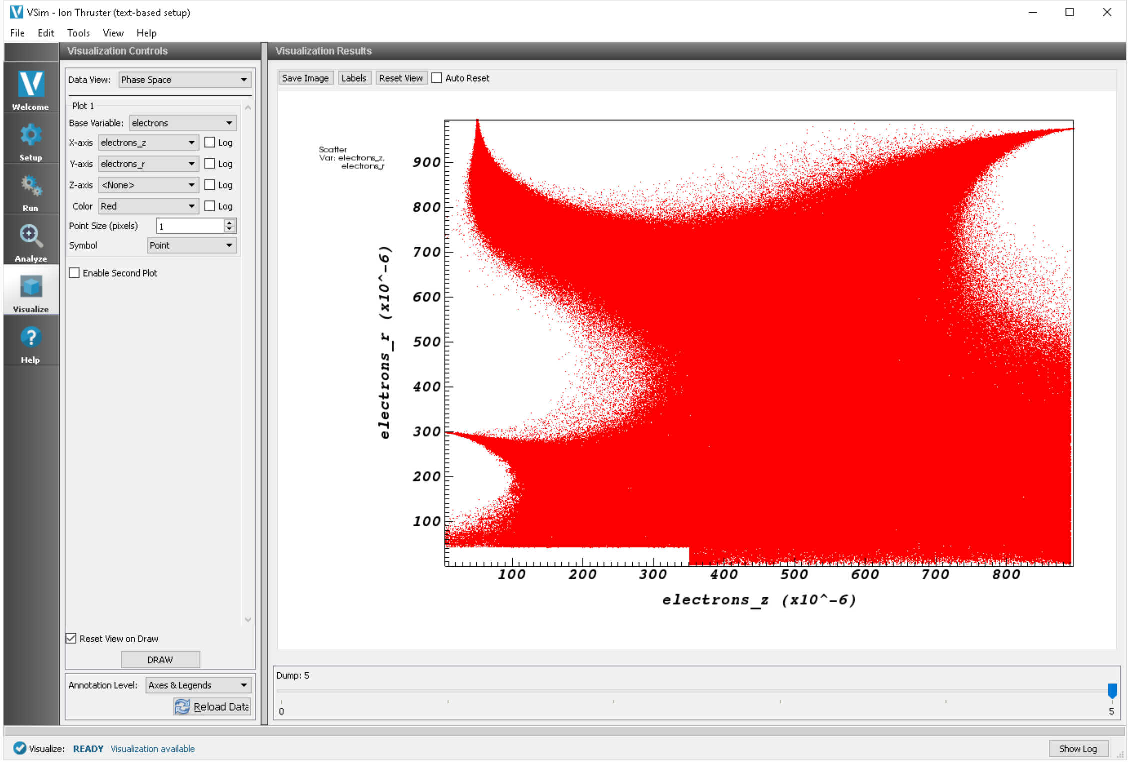
Layout: Side-by-side 2d/1d

Annotation Level: Axes & Legends

Dump: 0

Visualize: READY Visualization available

Fig. 6.81: Visualization of the *Field Analysis* result for the electric potential inside the ion thruster discharge chamber after 5,000 steps.



VSim - Ion Thruster (text-based setup)

Data View: Phase Space

Plot 1

Base Variable: electrons

X-axis electrons_z □ Log

Y-axis electrons_r □ Log

Z-axis <None> □ Log

Color: Red □ Log

Point Size (pixels): 1

Symbol: Point

□ Enable Second Plot

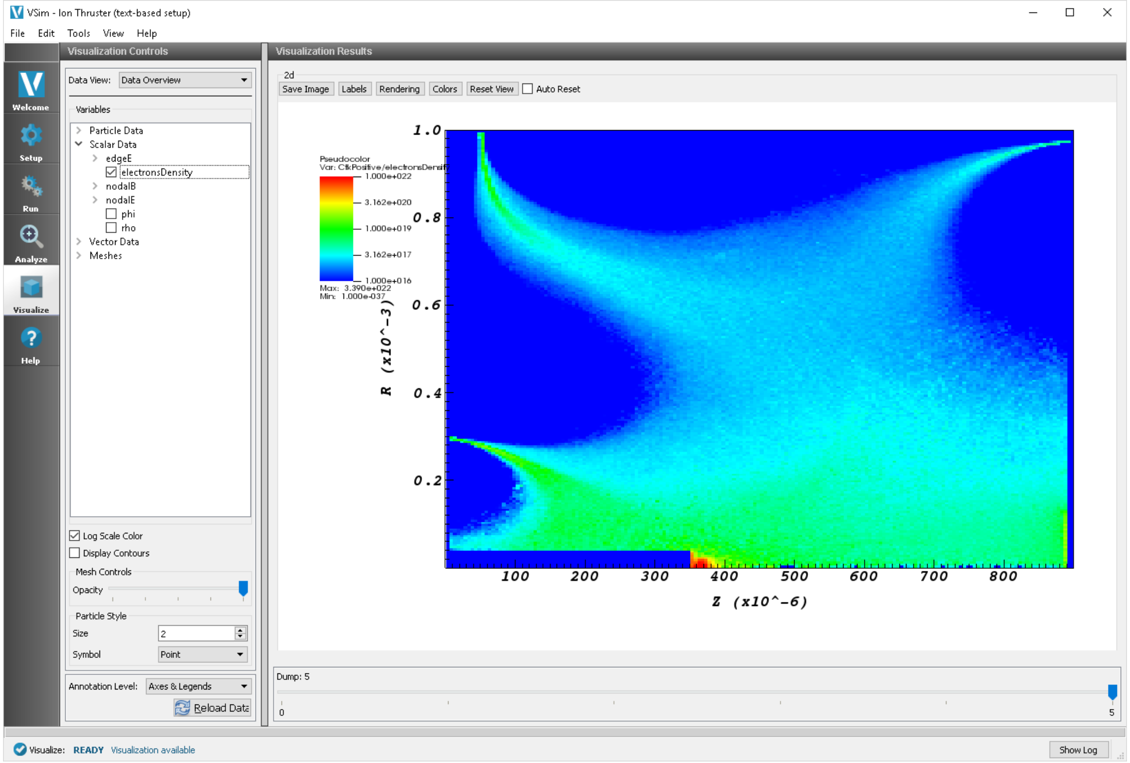
☑ Reset View on Draw

Annotation Level: Axes & Legends

Dump: 5

Visualize: READY Visualization available

Fig. 6.82: Electron phase-space distribution results after 5,000 steps.



Data View: Data Overview

☑ electronsDensity

☑ Log Scale Color

□ Display Contours

Size: 2

Symbol: Point

Annotation Level: Axes & Legends

Dump: 5

Visualize: READY Visualization available

Fig. 6.83: Electron number density distribution results inside the discharge chamber after 5,000 steps.

Further Experiments

Return to the Run window by clicking on the **Run** button in the leftmost panel, change *Number of Steps* to 500,000, and then click the *Run* button at the top of the Logs and Output Files pane. A high-performance computing cluster is recommended for this run, which will require approximately 6 days running on 64 cores. When the run has completed, take the following steps.

- Plot the potential at the final data dump similar to the steps taken in *Visualizing the Results*.
- In the Visualization Results pane, in the *2d* section, click the *Colors* button
- Set the minimum to 0 and the maximum to 25 (Volts). The resulting plot is shown in Fig. 6.84

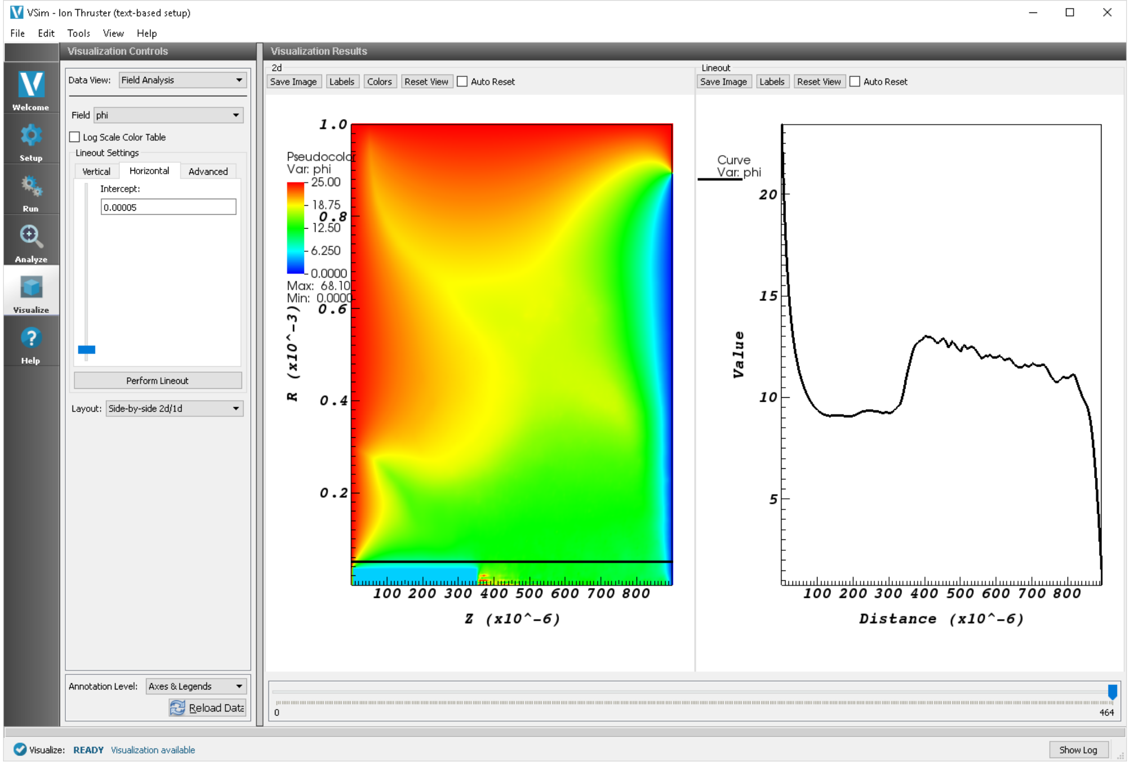
It can be seen that the ions experience most of their acceleration in the sheath near the right-side boundary of the plasma chamber. Plot the electron and ion densities by taking the following steps:

- Following once again the steps taken in *Visualizing the Results*, run the *computePtclNumDensity.py* analyzer on both *electrons* and *XeIons*.
- Plot the electrons density using the color log scale and the same limits as previous, as shown in Fig. 6.85
- Plot the ion density using the color log scale with a minimum of 1e18 and a maximum of 1e23 to get the image shown in Fig. 6.86 or experiment with the limits as desired.

It can be seen from Fig. 6.85 and Fig. 6.86 that the electrons are confined by the magnetic field lines while the much heavier ions are not, allowing a more uniform acceleration of ions out the right side of the chamber, resulting in thrust.

Plot the electron and ion macroparticle positions with the following steps:

- In the top of the *Visualization Controls* pane, switch the *Data View* dropdown menu to *Phase Space*.
- Under *Plot 1* click the *Base Variable* drop-down menu and select *electrons*



Data View: Field Analysis

Field: phi

☐ Log Scale Color Table

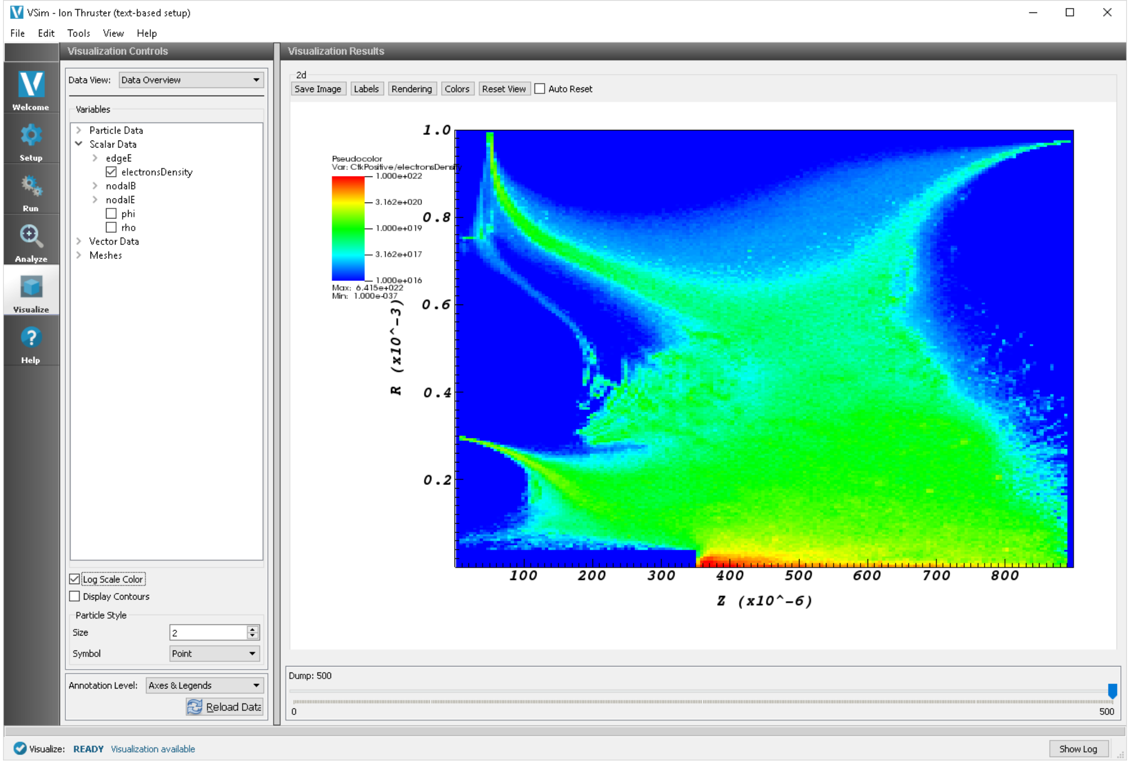
Intercept: 0.00005

Layout: Side-by-side 2d/1d

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 6.84: Electric potential of the plasma inside the ion thruster discharge chamber after 500,000 time-steps.



Data View: Data Overview

☑ electronsDensity

☐ phi

☐ rho

☑ Log Scale Color

☐ Display Contours

Size: 2

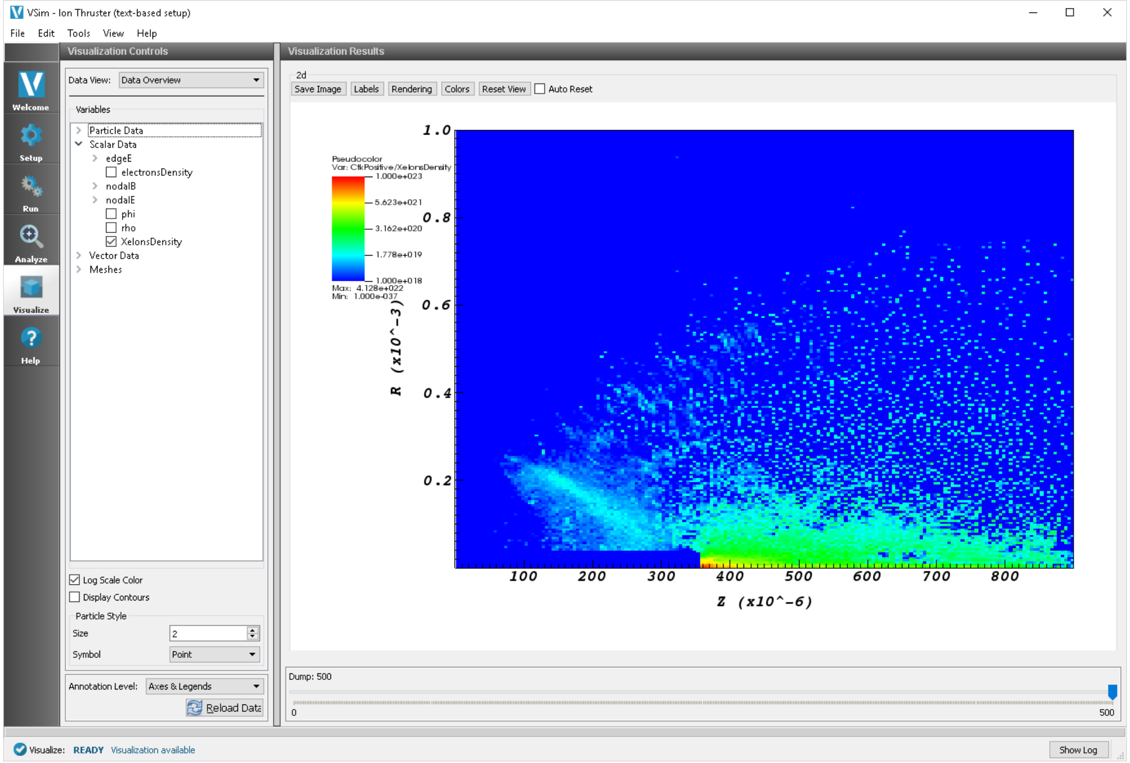
Symbol: Point

Annotation Level: Axes & Legends

Dump: 500

Visualize: READY Visualization available

Fig. 6.85: Electron number density distribution results inside the discharge chamber after 500,000 steps.



Data View: Data Overview

Variables
- Particle Data
- Scalar Data
  - edgeE
  - ☐ electronsDensity
  - nodalB
  - nodalE
  - ☐ phi
  - ☐ rho
  - ☑ XeIonsDensity
- Vector Data
- Meshes

☑ Log Scale Color
☐ Display Contours

Particle Style
Size: 2
Symbol: Point

Annotation Level: Axes & Legends

☐ Auto Reset

Dump: 500
0 500

Visualize: READY Visualization available

Fig. 6.86: Ion number density distribution results inside the discharge chamber after 500,000 steps.

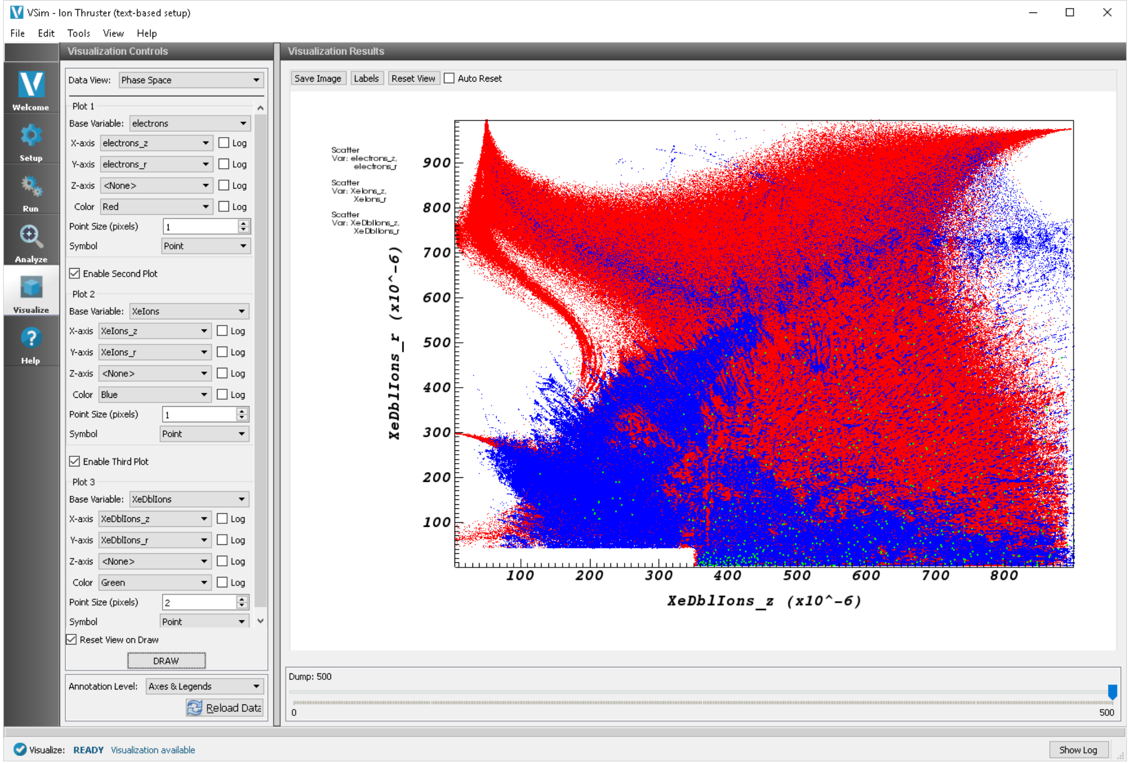
- Change *X-axis* to *electrons_z* and *Y-axis* to *electrons_r*, change *Point Size* to `1`, and at the bottom of the Visualization Controls pane, click *DRAW* to see the electron macro-particle positions.
- Check the *Enable Second Plot* button.
- Under *Plot 2* change *Base Variable* to *XeIons*, change *X-axis* to *XeIons_z* and *Y-axis* to *XeIons_r*, change *Point Size* to `1`, and click *DRAW* once again to see the electron and singly-ionized xenon macro-particle positions.
- Check the *Enable Third Plot* button.
- Under *Plot 3* change *Base Variable* to *XeDblIons*, change *X-axis* to *XeDblIons_z* and *Y-axis* to *XeDblIons_r*, leave *Point Size* at `2`, and click *DRAW* once again. The ion and electron positions should be displayed as shown in Fig. 6.87.

The electrons appear well confined by the 3-ring magnetic circuit arrangement, and move along the magnetic cusp regions formed between the magnets. Most of the electrons are lost to the discharge chamber walls through the magnetic cusps and are absorbed at the walls in 3 small areas. Only a few electrons are able to cross the strong magnetic field lines and reach the top wall between the cusps.

Singly and doubly ionized xenon are generated inside the discharge chamber through ionizing collisions of electrons with xenon neutrals. Only electrons with energies above the ionization thresholds (12.1 eV for the first ionization level and 21.25 eV for the second) can ionize neutrals.

This input file can be modified to test different design parameters covering a range of anode voltages, xenon flow rates, and electron emission currents, to allow study of high-to-low power and high-to-low throttle levels.

A more substantial further step for a user would be to take the following steps to calculate the thrust from the device by recording the velocities of exiting ions. This will require adding a `History` block (see code below) to collect the appropriate data, writing a post-processing analysis script to calculate the thrust, then importing the analyzer into VSim. Details of how to write your own analyzer are provided at VSim Customization: Custom Analyzers.



Data View: Phase Space

Plot 1
Base Variable: electrons
X-axis electrons_z □ Log
Y-axis electrons_r □ Log
Z-axis <None> □ Log
Color Red □ Log
Point Size (pixels) 1
Symbol Point

☑ Enable Second Plot

Plot 2
Base Variable: XeIons
X-axis XeIons_z □ Log
Y-axis XeIons_r □ Log
Z-axis <None> □ Log
Color Blue □ Log
Point Size (pixels) 1
Symbol Point

☑ Enable Third Plot

Plot 3
Base Variable: XeDblIons
X-axis XeDblIons_z □ Log
Y-axis XeDblIons_r □ Log
Z-axis <None> □ Log
Color Green □ Log
Point Size (pixels) 2
Symbol Point

☑ Reset View on Draw

Annotation Level: Axes & Legends

Dump: 500

0 500

Visualize: READY Visualization available

Fig. 6.87: Electron and ion phase-space distribution results after 500,000 steps.

- Return to the Setup window by clicking the **Setup** button in the leftmost panel
- Near the top of the Editor pane, click *View Input File*
- Scroll to the bottom of the input file and paste the following code:

```
<History absorbedIonVelocity>
  kind = speciesAbsPtclData2
  species = [Xeplus]
  ptclAbsorbers = [topOuterIonAbsorber rightIonAbsorber botOuterIonAbsorber]
  ptclAttributes = [xVelocity numPtclsInMacro kineticEnergy xPosition yPosition]
  collectMethod = statsForEachStep
</History>
```

the *collectMethod = statsForEachStep* provides sums of the desired quantities in the history file. The calculation thrust would be performed in the analysis tab. Other post-run analyses can be performed in order to see a sum of where the particles are arriving, their velocity, and the creation of other figures of merit to determine the performance of the the thruster.

6.8.3 Satellite Surface Charging (satelliteSurfaceChargeT.pre)

Keywords:

`electrostatics, surface charges`

Problem description

Satellites and other spacecraft operating in the space environment often suffer arcing and breakdown problems due to surface charging. Charged particles build up on the spacecraft surfaces (such as solar panel arrays and other components) leading to localized arcing/breakdown discharges that can critically fail a component or the entire unit. This problem is made worse as the demand for high power space missions in both satellite and deep-space applications rises. These high-power spacecraft are outfitted with high-voltage solar panels. These panels minimize the overall payload requirements and offer other advantages over more massive, low-voltage arrays. However, they are also more vulnerable to surface charge related arcing. It therefore becomes important to predict the surface charge build-up on spacecraft bodies operating in different space environments, where the ion sources may be natural solar wind or human-made space plasma resulting from electric thruster plasma plumes.

This example demonstrates a satellite body operating in the solar wind environment where the space plasma consists of ions and electrons. The simulation box is set up with dimensions of 15 m x 30 m x 15 m. The satellite system is placed in the middle of the domain. It has a 3 m radius x 5 m long cylindrical central unit connected to solar panels at either end. Each solar panel has a total span length of 7.8 m and a width of 5 m. The satellite central unit has a 5-volt equipotential circular body with radius 2 m and length 3 m. The satellite system is treated as a conductor floating in free space. The system domain boundaries are assumed to have zero perpendicular electric field, i.e. Neumann boundary conditions. The solar wind plasma is introduced in the simulation domain from the positive z direction. The solar wind density is set to $1 \times 10^7 \mathrm{m}^{-3}$ with a temperature of 10 eV. The number of physical particles per macroparticle is set to 5000. Both electrons and ions are introduced from the source based on the solar wind density and temperature. To maintain plasma uniformity within finite bounds, the electron source rate is inflated slightly because electrons are lighter and leave the system more quickly than do ions. At the same time, the positive ions are imbued with a lighter mass value to speed up the simulation. All simulation boundaries are set up to absorb particles. The charges collected in the satellite system are counted by emitting a heavy electron or heavy ion at the point where an electron or ion was absorbed. The heavy electron/ion is not a physical concept, it is a computational trick whereby any charged particle striking the satellite gets converted to a new species, one with equivalent charge but drastically swollen mass (1 kg in this case) and suppressed energy (suppressed by a factor of 10 billion). In this way, the heavy particles do not propagate from their point of origin, effectively sticking to the satellite surface. To limit the number of macro heavy particles tracked we apply a particle combining algorithm which limits the number of macro particle per cell to one. The collected electron and ion currents on the satellite surfaces are otput as histories.

This simulation can be performed with a VSimPD license.

Opening the Simulation

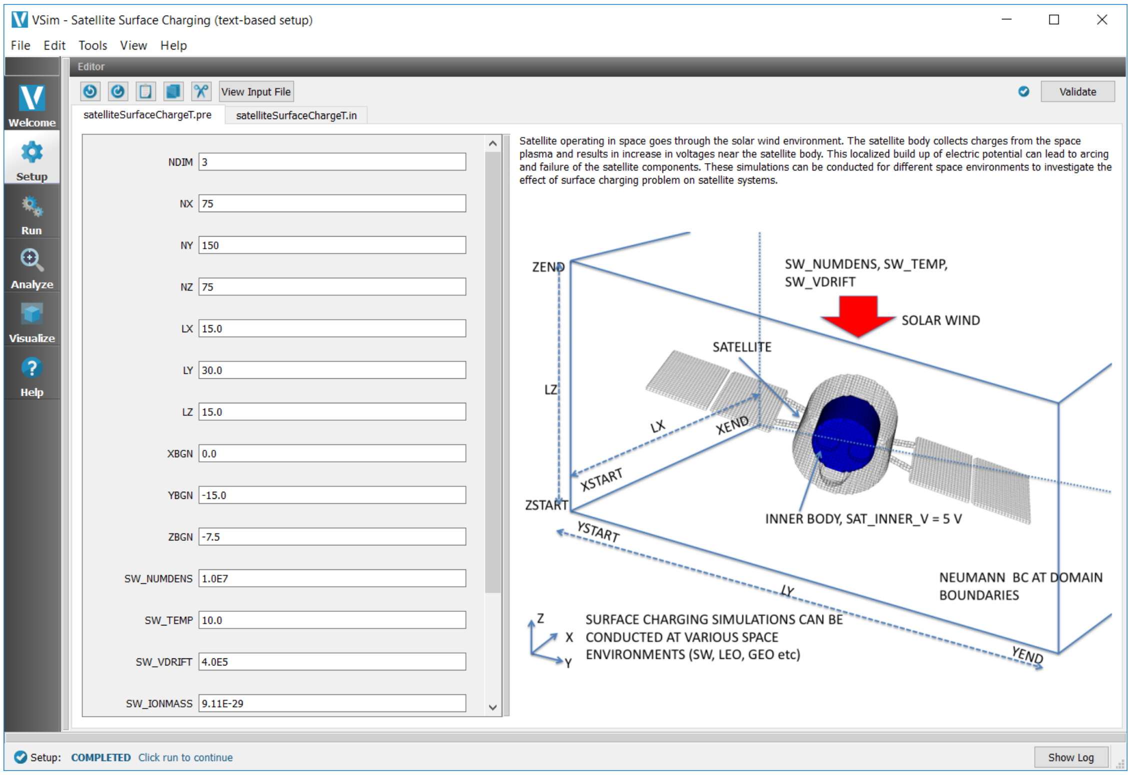
The satellite surface charging example is accessed from within VSimComposer by the following actions:

- Select the *New → From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Spacecraft (text-based setup)* option.
- Select “Satellite Surface Charging (text-based setup)” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

The basic variables of this problem should now be alterable via the text boxes in the left pane of the Setup Window, as shown in Fig. 6.88.

Input File Features

The input file allows a choice of space environment parameters (number density, plasma temperature, drift speed), satellite body voltage, simulation domain size, and resolution (number of cells in each direction).



NDIM 3

NX 75

NY 150

NZ 75

LX 15.0

LY 30.0

LZ 15.0

XBGN 0.0

YBGN -15.0

ZBGN -7.5

SW_NUMDENS 1.0E7

SW_TEMP 10.0

SW_VDRIFT 4.0E5

SW_IONMASS 9.11E-29

Satellite operating in space goes through the solar wind environment. The satellite body collects charges from the space plasma and results in increase in voltages near the satellite body. This localized build up of electric potential can lead to arcing and failure of the satellite components. These simulations can be conducted for different space environments to investigate the effect of surface charging problem on satellite systems.

SURFACE CHARGING SIMULATIONS CAN BE CONDUCTED AT VARIOUS SPACE ENVIRONMENTS (SW, LEO, GEO etc)

Setup: COMPLETED Click run to continue

Fig. 6.88: Setup Window for the Satellite Surface Charging example.

The self-consistent electric field is solved from Poisson's equation by the electrostatic solver. The far-field space boundaries are handled with Neumann boundary conditions. The satellite inner body is set up with an equipotential boundary. The surface charges collected on the satellite system make the satellite body float at a slightly higher voltage than the space plasma.

This is a large domain, 3-D problem, and its resolution is aided by several numerical methods. The plasma is represented by macro-particles which are moved according to the Boris pusher. Variable weight particle treatment is employed on all simulated species, reducing the overall number of macro-particles in the computation. Additionally, null interactions are considered as part of the Monte Carlo analysis to limit the number of macro-particles per cell; macro-particles are eliminated in overcrowded cells by means of inelastic combination.

Running the simulation

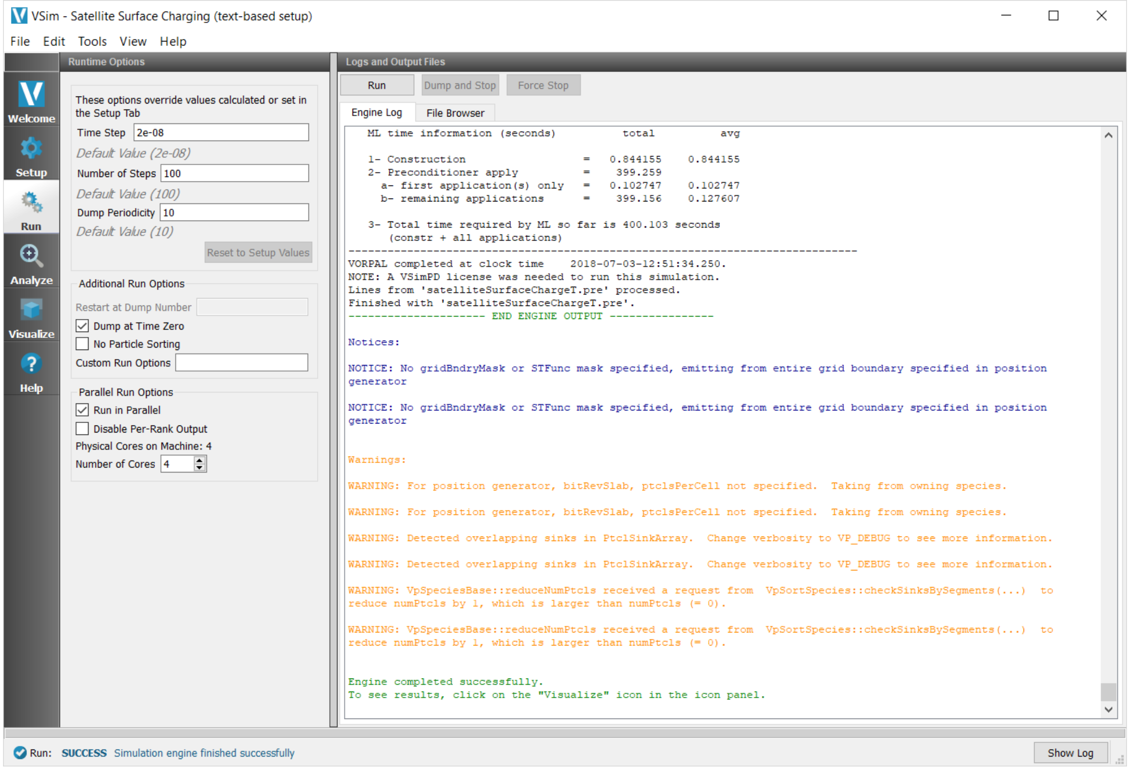
After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- To run the file, click on the *Run* button in the upper left corner of the right panel. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 6.89.

Analyzing the Results

If the electron density is desired, then proceed as follows:

- Press the Analyze button in the left column of buttons.



These options override values calculated or set in the Setup Tab

Time Step 2e-08
Default Value (2e-08)
Number of Steps 100
Default Value (100)
Dump Periodicity 10
Default Value (10)

Additional Run Options
Restart at Dump Number
☑ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options
☑ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
  ML time information (seconds)          total          avg

   1- Construction                  =   0.844155    0.844155
   2- Preconditioner apply          =    399.259
     a- first application(s) only   =   0.102747    0.102747
     b- remaining applications      =    399.156    0.127607

   3- Total time required by ML so far is 400.103 seconds
      (constr + all applications)
-----------------------------------------------------------------------------
VORPAL completed at clock time    2018-07-03-12:51:34.250.
NOTE: A VSimPD license was needed to run this simulation.
Lines from 'satelliteSurfaceChargeT.pre' processed.
Finished with 'satelliteSurfaceChargeT.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: No gridBndryMask or STFunc mask specified, emitting from entire grid boundary specified in position
generator

NOTICE: No gridBndryMask or STFunc mask specified, emitting from entire grid boundary specified in position
generator


Warnings:

WARNING: For position generator, bitRevSlab, ptclsPerCell not specified.  Taking from owning species.

WARNING: For position generator, bitRevSlab, ptclsPerCell not specified.  Taking from owning species.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.

WARNING: Detected overlapping sinks in PtclSinkArray.  Change verbosity to VP_DEBUG to see more information.

WARNING: VpSpeciesBase::reduceNumPtcls received a request from  VpSortSpecies::checkSinksBySegments(...)  to
reduce numPtcls by 1, which is larger than numPtcls (= 0).

WARNING: VpSpeciesBase::reduceNumPtcls received a request from  VpSortSpecies::checkSinksBySegments(...)  to
reduce numPtcls by 1, which is larger than numPtcls (= 0).


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: **SUCCESS** Simulation engine finished successfully

Fig. 6.89: The Run Window at the end of execution.

- Check the *Show All Analyzers* box.
- Select *computePtclNumDensity.py*.
- Click *Open*.
- Enter the following parameters in the appropriate fields:
  - simulationName = satelliteSurfaceChargeT
  - speciesName = solarElectrons
  - avgNxN = 1
  - iterateAvg = 1
- Click the *Analyze* button in the upper right corner of the window.

See Fig. 6.90.

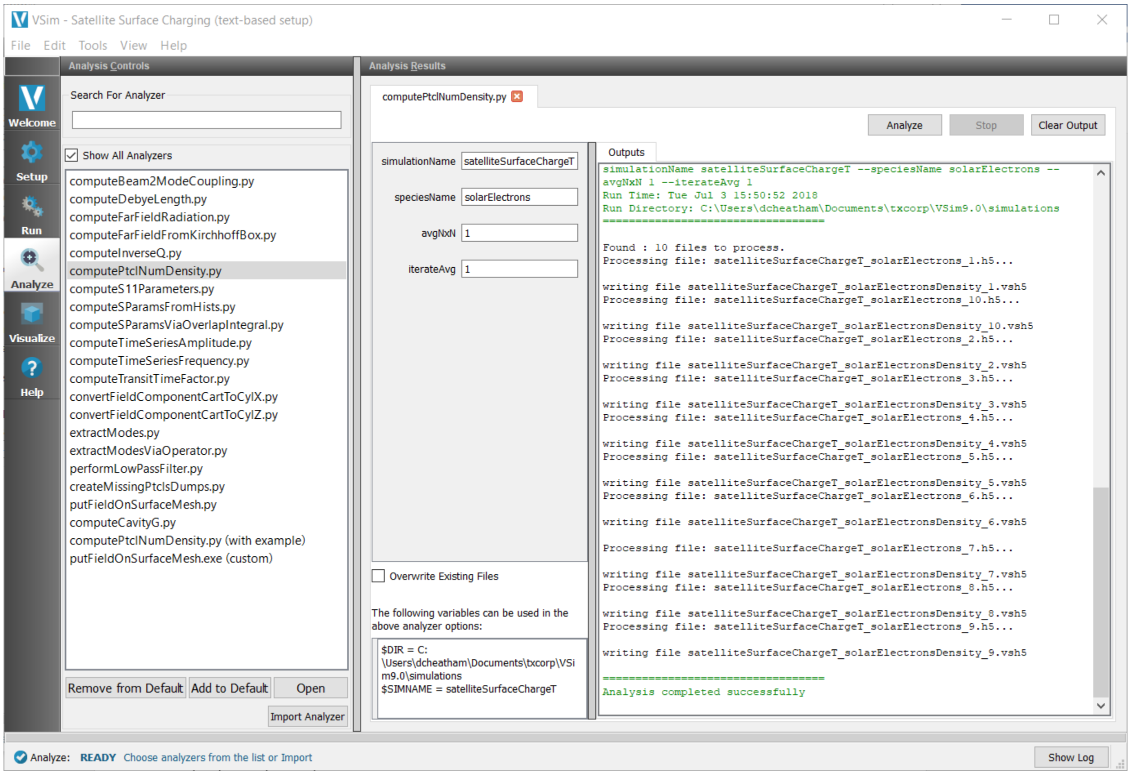
The resulting data will be visualizable as *solarElectronsDensity* under the *Scalar Data* menu in the *Visualize* Tab. The density of solarIons can be calculated in the same way by substituting that species name in place of *solarElectrons*.

Visualizing the results

After performing the above actions, proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.

To visualize the satellite geometry with electrons Fig. 6.91, proceed as follows:

- Expand *Particle Data*.


VSim - Satellite Surface Charging (text-based setup)
File Edit Tools View Help
Analysis Controls
Welcome
Setup
Run
Analyze
Visualize
Help
Search For Analyzer
☑ Show All Analyzers
computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
createMissingPtclsDumps.py
putFieldOnSurfaceMesh.py
computeCavityG.py
computePtclNumDensity.py (with example)
putFieldOnSurfaceMesh.exe (custom)
Remove from Default
Add to Default
Open
Import Analyzer
Analysis Results
computePtclNumDensity.py
Analyze
Stop
Clear Output
Outputs
Show Log

simulationName satelliteSurfaceChargeT

speciesName solarElectrons

avgNxN 1

iterateAvg 1

□ Overwrite Existing Files

The following variables can be used in the above analyzer options:

$DIR = C:\Users\dcheatham\Documents\txcorp\VSim9.0\simulations
$SIMNAME = satelliteSurfaceChargeT

```
simulationName satelliteSurfaceChargeT --speciesName solarElectrons --
avgNxN 1 --iterateAvg 1
Run Time: Tue Jul 3 15:50:52 2018
Run Directory: C:\Users\dcheatham\Documents\txcorp\VSim9.0\simulations
==================================

Found : 10 files to process.
Processing file: satelliteSurfaceChargeT_solarElectrons_1.h5...

writing file satelliteSurfaceChargeT_solarElectronsDensity_1.vsh5
Processing file: satelliteSurfaceChargeT_solarElectrons_10.h5...

writing file satelliteSurfaceChargeT_solarElectronsDensity_10.vsh5
Processing file: satelliteSurfaceChargeT_solarElectrons_2.h5...

writing file satelliteSurfaceChargeT_solarElectronsDensity_2.vsh5
Processing file: satelliteSurfaceChargeT_solarElectrons_3.h5...

writing file satelliteSurfaceChargeT_solarElectronsDensity_3.vsh5
Processing file: satelliteSurfaceChargeT_solarElectrons_4.h5...

writing file satelliteSurfaceChargeT_solarElectronsDensity_4.vsh5
Processing file: satelliteSurfaceChargeT_solarElectrons_5.h5...

writing file satelliteSurfaceChargeT_solarElectronsDensity_5.vsh5
Processing file: satelliteSurfaceChargeT_solarElectrons_6.h5...

writing file satelliteSurfaceChargeT_solarElectronsDensity_6.vsh5

Processing file: satelliteSurfaceChargeT_solarElectrons_7.h5...

writing file satelliteSurfaceChargeT_solarElectronsDensity_7.vsh5
Processing file: satelliteSurfaceChargeT_solarElectrons_8.h5...

writing file satelliteSurfaceChargeT_solarElectronsDensity_8.vsh5
Processing file: satelliteSurfaceChargeT_solarElectrons_9.h5...

writing file satelliteSurfaceChargeT_solarElectronsDensity_9.vsh5

==================================
Analysis completed successfully
```

Analyze: READY Choose analyzers from the list or Import

Fig. 6.90: The Run Window at the end of execution.

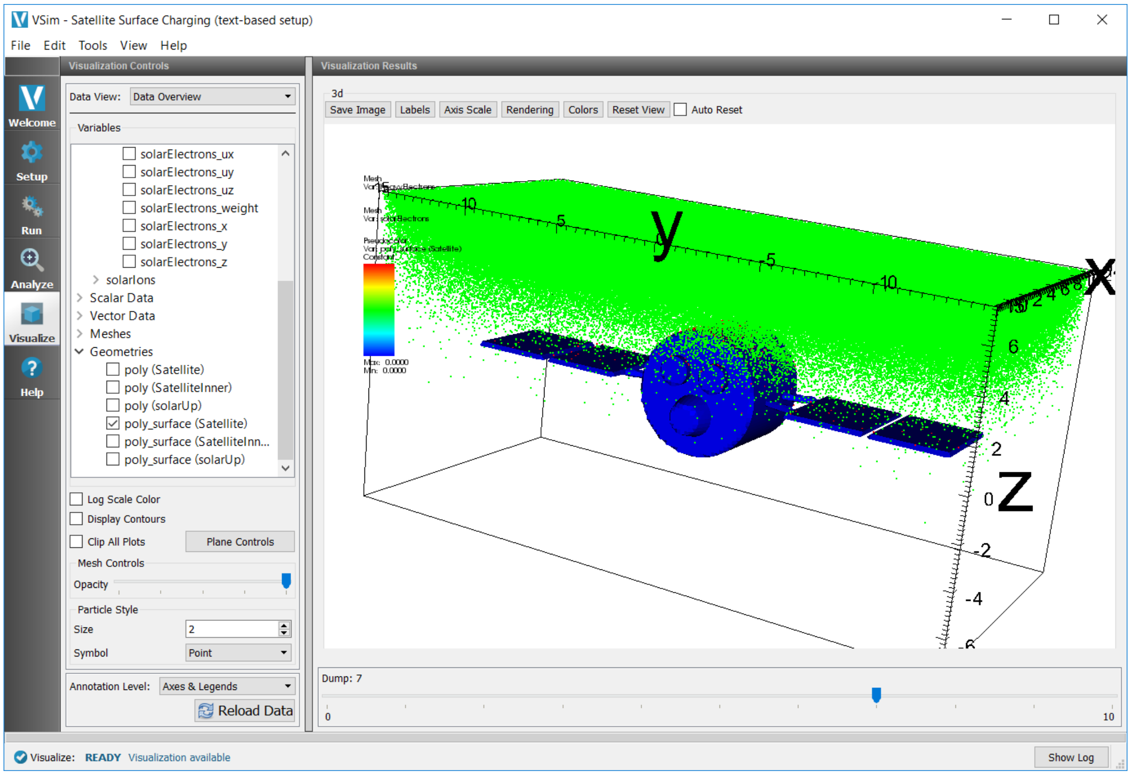
- Expand *heavyElectrons*.
- Select "heavyElectrons" in red.
- Expand *solarElectrons*.
- Select "solarElectrons" in green.
- Expand *Geometries*.
- Select "poly_surface (Satellite)".
- Move the Dump slider to dump 7.

Here are some things to try:

- Under *Data Overview* you can access plots of the electric field, charge density (rho), and electric potential (phi). Select the *Display Contours* check box for viewing these.
- To view the phase space distribution for the electrons and ions, click on the *Data View* drop down menu and select *Phase Space*. Click the *Draw* button to generate a plot.
- Also from the *Data View* menu select *History* to observe the satellite currents and the time history results for the number of macro-particles broken down by species.

To generate Fig. 6.92, that shows the satellite system with the inner equipotential cylindrical body, proceed as follows:

- In the *Data View* pane on the left side select "Data Overview" from the drop-down menu.
- Expand *Geometries*
- Select "poly (Satellite)"
- Select "poly_surface (SatelliteInner)".



Data View: Data Overview

- ☐ solarElectrons_ux
- ☐ solarElectrons_uy
- ☐ solarElectrons_uz
- ☐ solarElectrons_weight
- ☐ solarElectrons_x
- ☐ solarElectrons_y
- ☐ solarElectrons_z
- ☐ poly (Satellite)
- ☐ poly (SatelliteInner)
- ☐ poly (solarUp)
- ☑ poly_surface (Satellite)
- ☐ poly_surface (SatelliteInn...
- ☐ poly_surface (solarUp)

☐ Log Scale Color
☐ Display Contours
☐ Clip All Plots

Opacity
Size: 2
Symbol: Point

Annotation Level: Axes & Legends

Dump: 7

Visualize: READY Visualization available

Fig. 6.91: Visualization plot of satellite system with solar wind electrons in green and the electrons that stick to the satellite surface in red.

The phase-space distribution of the positive ions (solarIons species) surrounding the satellite system is shown in Fig. 6.93 which is obtained after running the simulation for 500 time steps. Solar wind plasma enters into the simulation system from the top z boundary, i.e. above the satellite body.

Surface charge accumulation on the satellite body after 93,000 time steps is shown in Fig. 6.94. The red dots indicate electrons and the green dots ions. The surface charges on the satellite body can be viewed in VSimComposer by turning on Particle Data –> heavyElectrons and ParticleData -> heavyIons under the Data Overview pane.

The charge density built-up on the satellite system is shown in Fig. 6.95 after running for 93,000 time steps. To view the charge density in the simulation domain, turn on the Scalar Data -> rho field in the *Data Overview* pane. In this figure the satellite body is also included by turning on the Geometries -> poly_surface(Satellite) option in the *Data Overview* pane. The charge density appears net positive in most regions of the solar panels.

To view the electrostatic potential, turn on phi under the Scalar Data in the *Data Overview* pane. The electrostatic potential of the satellite system simulated is shown in Fig. 6.96 after running for 93,000 time steps. The electrostatic potential is plotted in X-Y-Z with domain clipping in the X and Z directions. The bulk of the plasma potential in the space region is close to 0 volts (blue contours). The surface charge built-up on the solar panels raises the surface potential by up to 4 to 5 volts above the bulk space plasma.

The magnitude of the electric field distribution on the satellite surface after 93,000 time steps is shown in Fig. 6.97. The peak of the distribution coincides with regions on the solar array where there is net positive charge build-up. The magnitude of the electric field was computed using the *Expressions* function in the Visit interface. Should you wish to get to those calculations, right-click on the plot and select *Open GUI*. (You must have the *Enable VisIt context menu* box check-marked in Vizualization Options. Go to Tools -> Settings -> Visualization Options, to enable this.) This will launch the VisIt control panel. From there, go to Controls -> Expressions. You can select any plottable variable to view its mathematical definition.

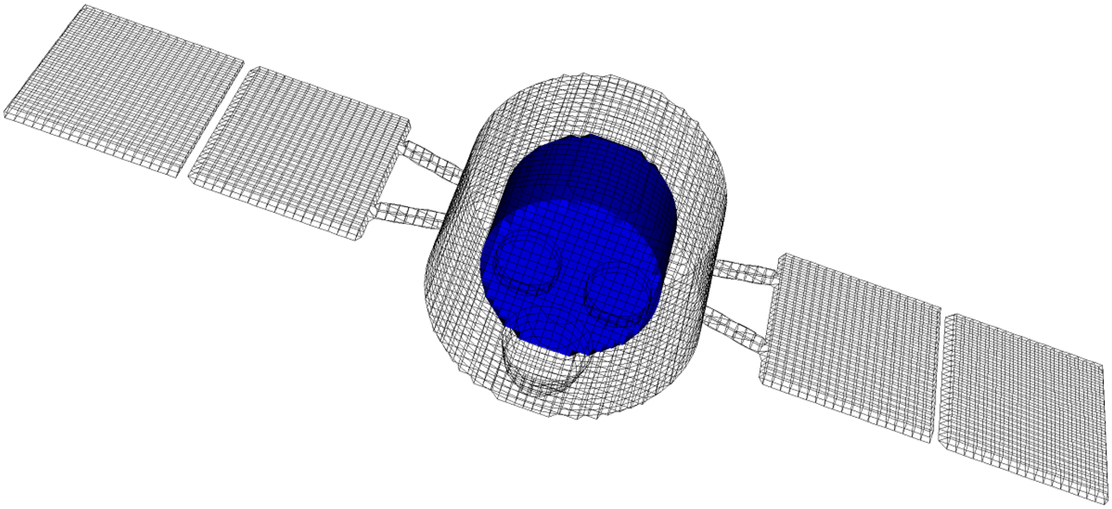


Fig. 6.92: Visualization of the inner body inside the satellite system.

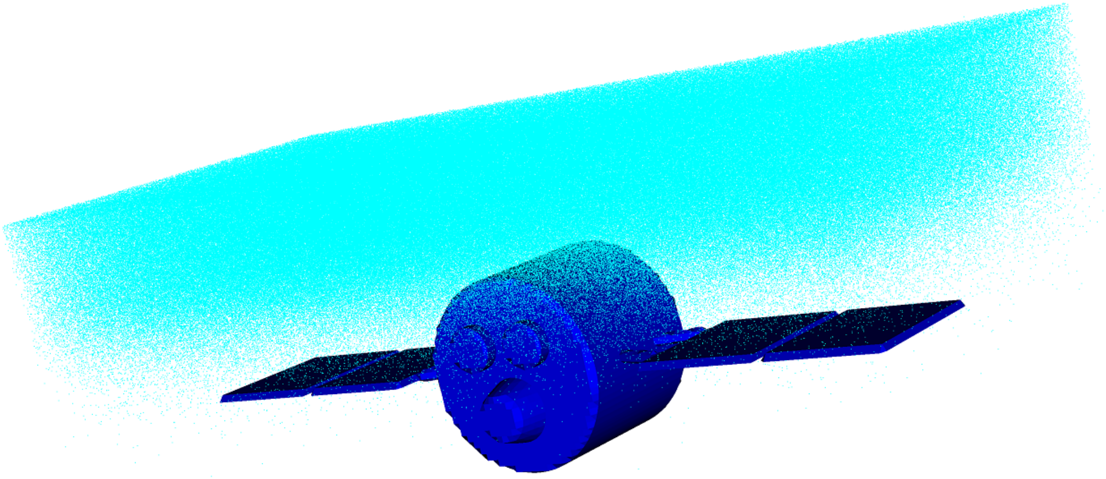


Fig. 6.93: Visualization of the satellite system with solar ions.

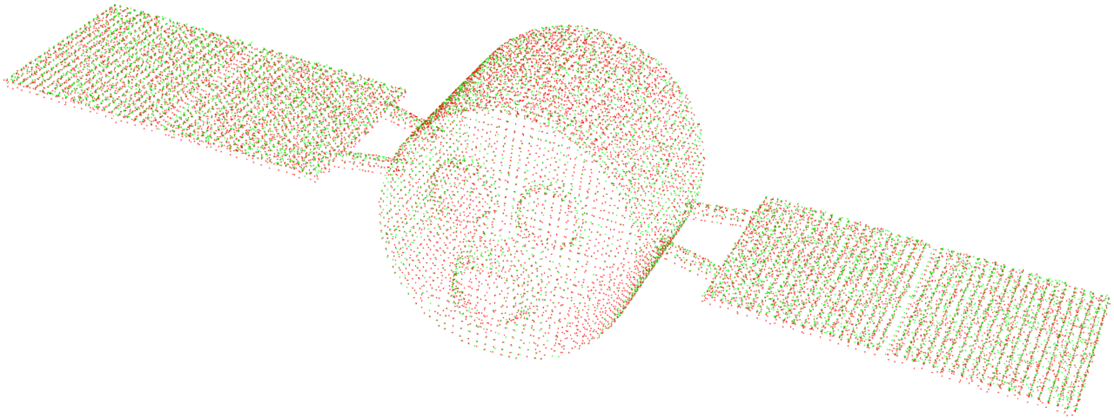


Fig. 6.94: Visualization of surface charge build-up on the satellite system after 93,000 time steps.

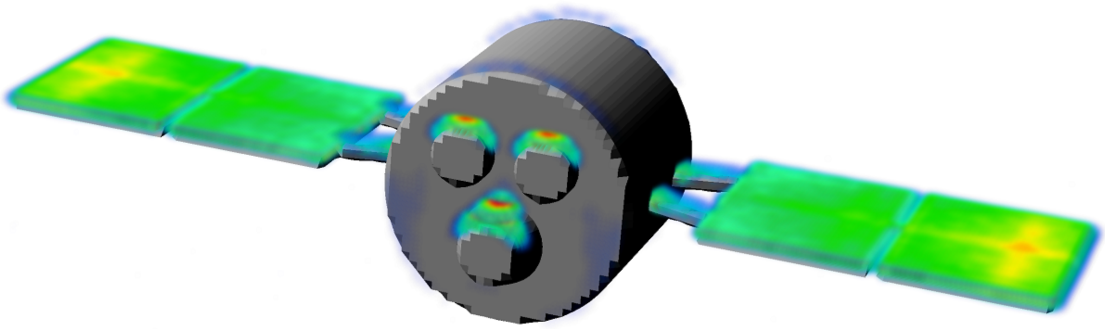


Fig. 6.95: Visualization of the charge density on the satellite system.

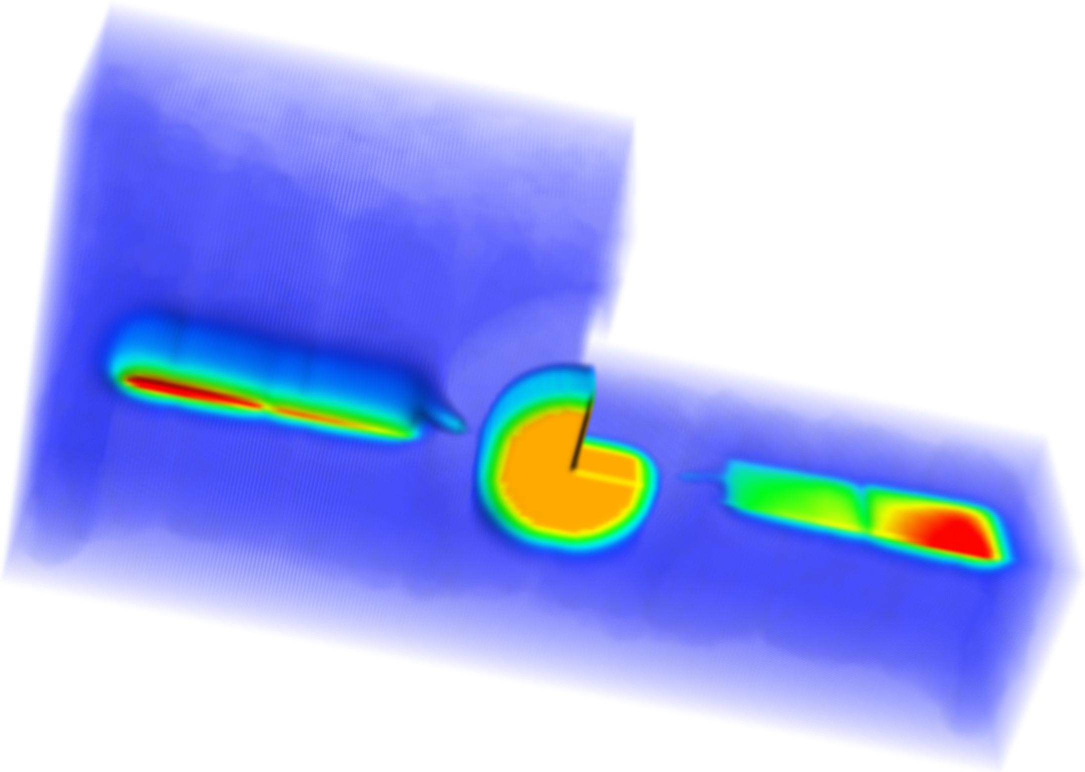


Fig. 6.96: Visualization of the electric potential surrounding the satellite system.

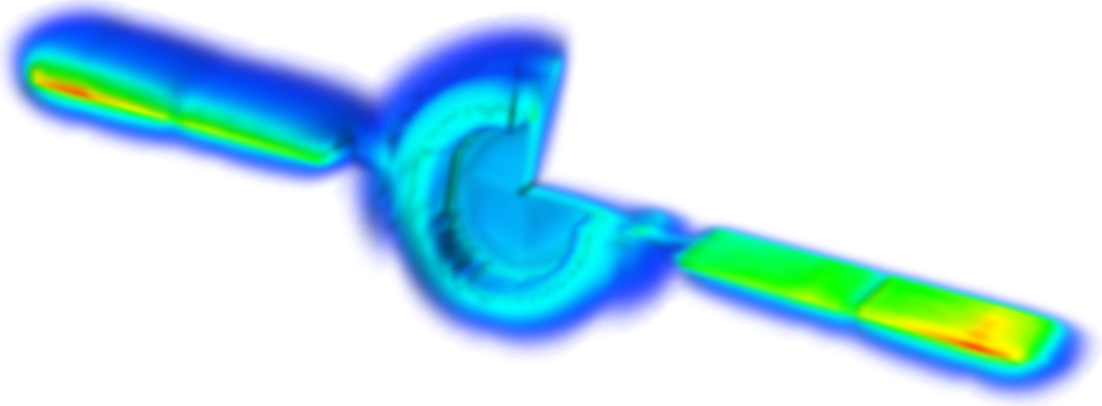


Fig. 6.97: Visualization of the magnitude of electric field surrounding the satellite system.

Further Experiments

The geometry and background space plasma parameters of this input file can be modified to test satellite inner body voltages and satellite surface charge collection in a variety of different space environments.

VSim allows the use of “open” boundary conditions to represent the far-field boundaries in the space environment.

6.9 Sputtering

6.9.1 Ion Beam Sputtering (ionBeamSputtering.sdf)

Keywords:

`sputtering, ion beam, electrostatics`

Problem Description

In this simulation, a 450 eV beam of positively charged argon ions strikes a copper plate (cathode) at -25 volts with respect to the walls on the upper and lower x boundaries of the simulation. A Neumann boundary condition (the slope of the electric potential is zero) is placed on the upper and lower y walls of the simulation.

The argon ion beam travels through a vacuum and sputter off neutral copper atoms from the cathode.

This demonstrates how to set up a sputtering interaction through the Visual Setup in VSimComposer.

This simulation can be performed with a VSimPD license.

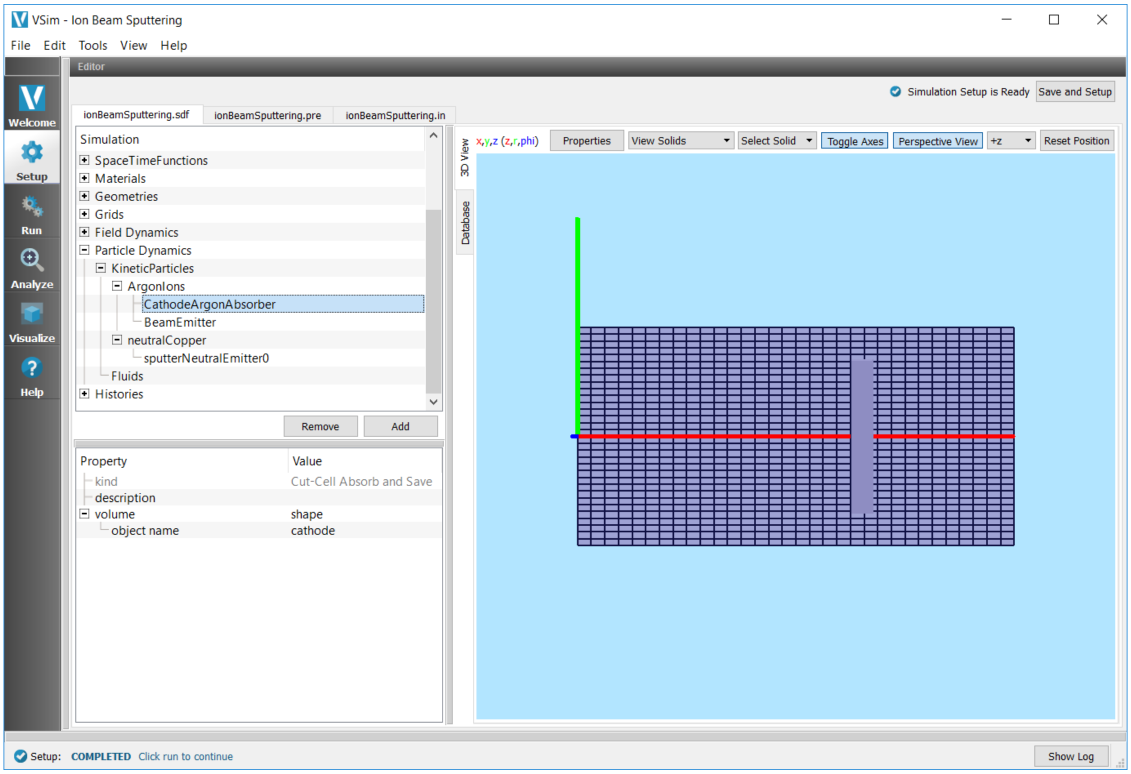
Opening the Simulation

The Ion Beam Sputtering example can be accessed from within VSimComposer by the following actions:

- Select the *New* → *From Example…* menu item in the *File* menu.
- In the resulting *Examples* window expand the *VSim for Plasma Discharges* option.
- Expand the *Sputtering* option.

- Select “Ion Beam Sputtering” and press the *Choose* button.
- In the resulting dialog, create a New Folder if desired, and press the *Save* button to create a copy of this example.

You can expand the tree elements and navigate through the various properties, making any changes you desire. The right pane shows a 3D view of the geometry, if any, as well as the grid, if actively shown. The setup window, with the *Particle Dynamics* → *Kinetic Particles* → *ArgonIons*, and *neutralCopper* elements expanded is shown in Fig. 6.98.



Simulation
- SpaceTimeFunctions
- Materials
- Geometries
- Grids
- Field Dynamics
- Particle Dynamics
  - KineticParticles
    - ArgonIons
      - CathodeArgonAbsorber
      - BeamEmitter
    - neutralCopper
      - sputterNeutralEmitter0
  - Fluids
- Histories

Property	Value
kind	Cut-Cell Absorb and Save
description	
volume	shape
object name	cathode

Setup: COMPLETED Click run to continue

Fig. 6.98: Setup window for the Ion Beam Sputtering example.

Simulation Properties

The Ion Beam Sputtering example includes some constants for easy adjustment of simulation properties:

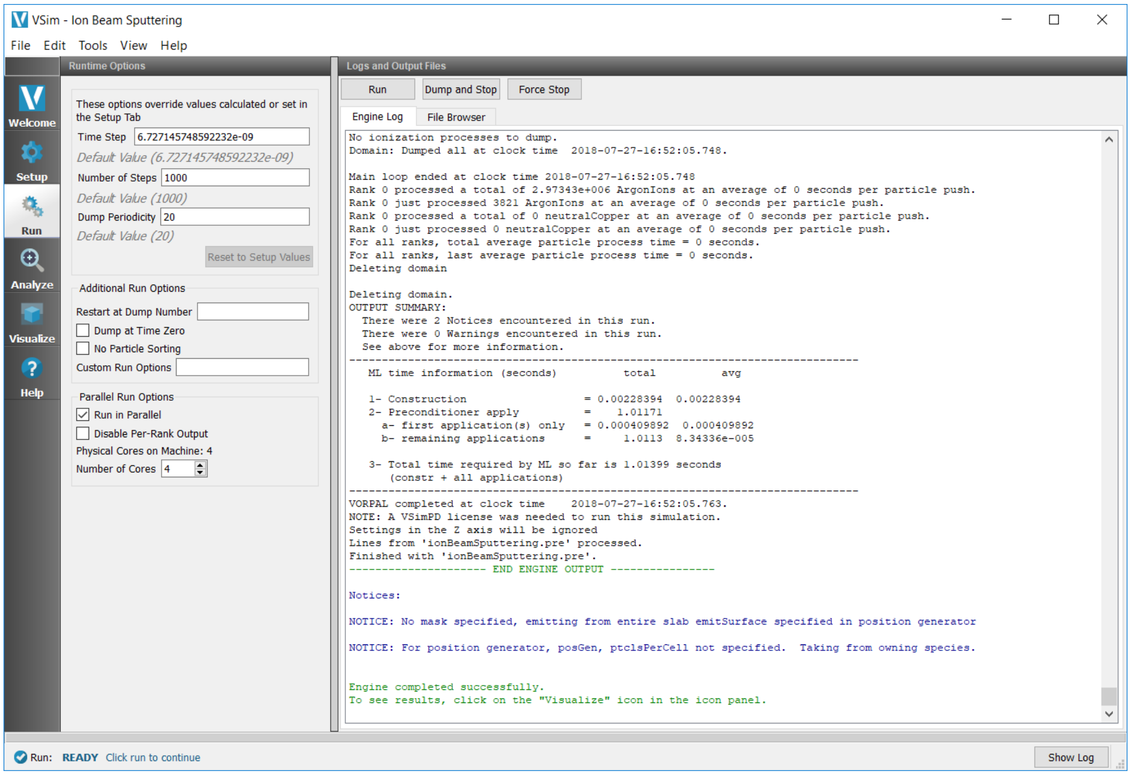
- **BEAM_RADIUS**: sets the radius of the argon ion beam
- **BEAM_ENERGY**: sets the energy (and speed) of the ion beam
- **BEAM_CURRENT**: sets the emitted current. *Note*: the total current emitted will be less than this value if a mask is applied to the particle emitter (which is the case in this simulation).
- **CATHODE_VOLTAGE**: the negative bias for the cathode.

Running the Simulation

After performing the above actions, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- Here you can set run parameters, including how many cores to run with (under the MPI tab).

- When you are finished setting run parameters, click on the *Run* button in the upper right corner. You will see the output of the run in the right pane. The run has completed when you see the output, "Engine completed successfully." This is shown in Fig. 6.99.



These options override values calculated or set in the Setup Tab

Time Step 6.727145748592232e-09
Default Value (6.727145748592232e-09)
Number of Steps 1000
Default Value (1000)
Dump Periodicity 20
Default Value (20)

Additional Run Options

Restart at Dump Number
□ Dump at Time Zero
□ No Particle Sorting
Custom Run Options

Parallel Run Options

☑ Run in Parallel
□ Disable Per-Rank Output
Physical Cores on Machine: 4
Number of Cores 4

```
No ionization processes to dump.
Domain: Dumped all at clock time  2018-07-27-16:52:05.748.

Main loop ended at clock time 2018-07-27-16:52:05.748
Rank 0 processed a total of 2.97343e+006 ArgonIons at an average of 0 seconds per particle push.
Rank 0 just processed 3821 ArgonIons at an average of 0 seconds per particle push.
Rank 0 processed a total of 0 neutralCopper at an average of 0 seconds per particle push.
Rank 0 just processed 0 neutralCopper at an average of 0 seconds per particle push.
For all ranks, total average particle process time = 0 seconds.
For all ranks, last average particle process time = 0 seconds.
Deleting domain

Deleting domain.
OUTPUT SUMMARY:
  There were 2 Notices encountered in this run.
  There were 0 Warnings encountered in this run.
  See above for more information.
--------------------------------------------------------------------------------
   ML time information (seconds)          total          avg

   1- Construction                  = 0.00228394  0.00228394
   2- Preconditioner apply          =    1.01171
     a- first application(s) only   = 0.000409892  0.000409892
     b- remaining applications      =     1.0113  8.34336e-005

   3- Total time required by ML so far is 1.01399 seconds
      (constr + all applications)
--------------------------------------------------------------------------------
VORPAL completed at clock time    2018-07-27-16:52:05.763.
NOTE: A VSimPD license was needed to run this simulation.
Settings in the Z axis will be ignored
Lines from 'ionBeamSputtering.pre' processed.
Finished with 'ionBeamSputtering.pre'.
--------------------- END ENGINE OUTPUT ----------------

Notices:

NOTICE: No mask specified, emitting from entire slab emitSurface specified in position generator

NOTICE: For position generator, posGen, ptclsPerCell not specified.  Taking from owning species.


Engine completed successfully.
To see results, click on the "Visualize" icon in the icon panel.
```

Run: READY Click run to continue

Fig. 6.99: The Run window at the end of execution.

Analyzing the data

This simulation includes particle species which enter the simulation at different times. When this is the case, the *createMissingPtclDumps.py* analyzer must be run. If no particles of a particular species exist in a simulation when data is written (dumped), then there will be no dump files for that particular species. This will cause particle data to not be lined up to the same dump in the *Visualize Tab*.

To fill in these missing files, navigate to the *Analyze* Tab and check the box "Show All Analyzers". Select *createMissingPtclDumps.py* from the list and click "Open". Ensure the following is entered into each field:

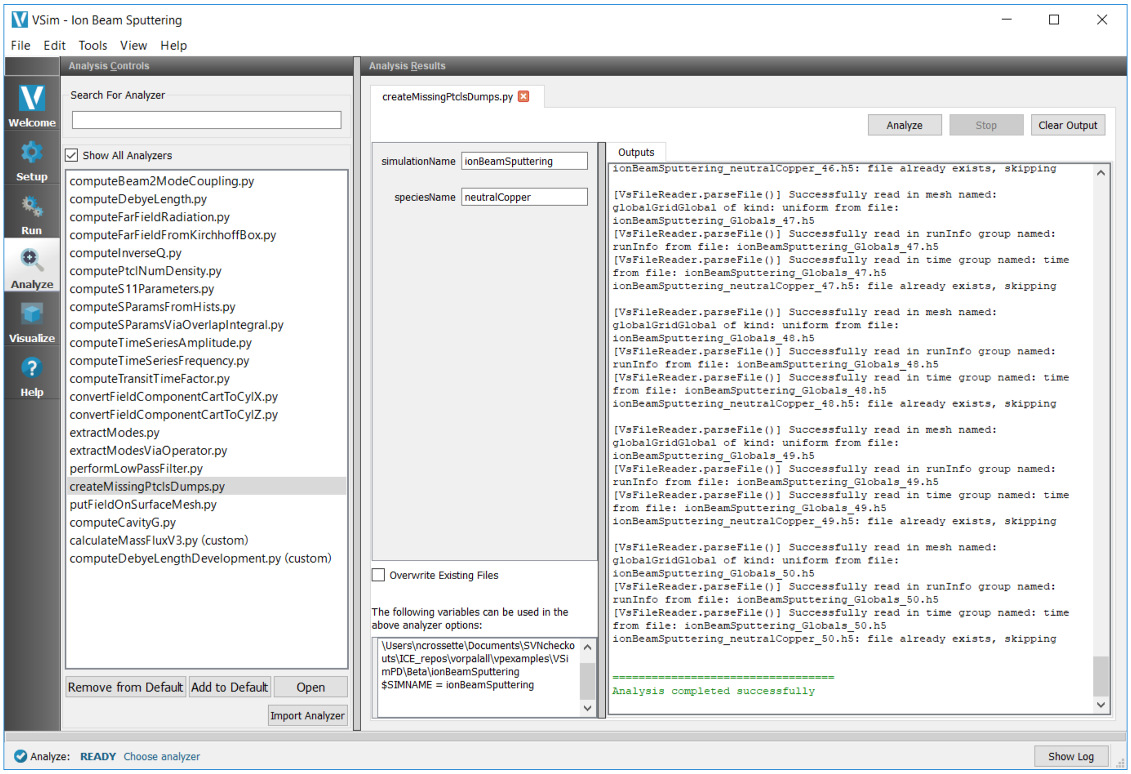
- **`simulationName`:** "ionBeamSputtering"
- **`speciesName`:** "neutralCopper"

Click *Analyze* in upper right corner of the window. When the analysis is finished, you should see a window similar to Fig. 6.9.1.

Visualizing the results

After performing the above actions, continue as follows:

1. Proceed to the *Visualize* window by pressing the Visualize button in the left column of buttons. Be sure to press the "Reload Data" button at the bottom of the window if you have previously navigated to the *Visualize* window.



☑ Show All Analyzers

computeBeam2ModeCoupling.py
computeDebyeLength.py
computeFarFieldRadiation.py
computeFarFieldFromKirchhoffBox.py
computeInverseQ.py
computePtclNumDensity.py
computeS11Parameters.py
computeSParamsFromHists.py
computeSParamsViaOverlapIntegral.py
computeTimeSeriesAmplitude.py
computeTimeSeriesFrequency.py
computeTransitTimeFactor.py
convertFieldComponentCartToCylX.py
convertFieldComponentCartToCylZ.py
extractModes.py
extractModesViaOperator.py
performLowPassFilter.py
createMissingPtclsDumps.py
putFieldOnSurfaceMesh.py
computeCavityG.py
calculateMassFluxV3.py (custom)
computeDebyeLengthDevelopment.py (custom)

createMissingPtclsDumps.py

simulationName: ionBeamSputtering
speciesName: neutralCopper

☐ Overwrite Existing Files

The following variables can be used in the above analyzer options:

```
\Users\ncrossette\Documents\SVNcheckouts\ICE_repos\vorpalall\vpexamples\VSimPD\Beta\ionBeamSputtering
$SIMNAME = ionBeamSputtering
```

```
ionBeamSputtering_neutralCopper_46.h5: file already exists, skipping

[VsFileReader.parseFile()] Successfully read in mesh named:
globalGridGlobal of kind: uniform from file:
ionBeamSputtering_Globals_47.h5
[VsFileReader.parseFile()] Successfully read in runInfo group named:
runInfo from file: ionBeamSputtering_Globals_47.h5
[VsFileReader.parseFile()] Successfully read in time group named: time
from file: ionBeamSputtering_Globals_47.h5
ionBeamSputtering_neutralCopper_47.h5: file already exists, skipping

[VsFileReader.parseFile()] Successfully read in mesh named:
globalGridGlobal of kind: uniform from file:
ionBeamSputtering_Globals_48.h5
[VsFileReader.parseFile()] Successfully read in runInfo group named:
runInfo from file: ionBeamSputtering_Globals_48.h5
[VsFileReader.parseFile()] Successfully read in time group named: time
from file: ionBeamSputtering_Globals_48.h5
ionBeamSputtering_neutralCopper_48.h5: file already exists, skipping

[VsFileReader.parseFile()] Successfully read in mesh named:
globalGridGlobal of kind: uniform from file:
ionBeamSputtering_Globals_49.h5
[VsFileReader.parseFile()] Successfully read in runInfo group named:
runInfo from file: ionBeamSputtering_Globals_49.h5
[VsFileReader.parseFile()] Successfully read in time group named: time
from file: ionBeamSputtering_Globals_49.h5
ionBeamSputtering_neutralCopper_49.h5: file already exists, skipping

[VsFileReader.parseFile()] Successfully read in mesh named:
globalGridGlobal of kind: uniform from file:
ionBeamSputtering_Globals_50.h5
[VsFileReader.parseFile()] Successfully read in runInfo group named:
runInfo from file: ionBeamSputtering_Globals_50.h5
[VsFileReader.parseFile()] Successfully read in time group named: time
from file: ionBeamSputtering_Globals_50.h5
ionBeamSputtering_neutralCopper_50.h5: file already exists, skipping



==================================
Analysis completed successfully
```

Analyze: READY Choose analyzer

2. Expand “Particle Data” then expand “ArgonIons” and check the red “ArgonIons” box.
3. Expand “neutralCopper” and check the green “neutralCopper” box.
4. Expand “Scalar Data” and check “Phi”. Then check the “Display Contours” box.
5. Expand “Geometries” and select poly (cathode).

Scrolling through the dumps, you should see the argon beam expand as it travels towards the cathode, as in Fig. 6.100. Because we ran the *createMissingPtclDumps.py* analyzer, copper atoms won’t appear until after the argon beam strikes the cathode.

Next, in the *Data View* drop down menu at the top left of the window, switch from “Data Overview” to “History”. In *Graph 2*, change from “emittedArgonCurrent” to “<None>” to see the plots shown in Fig. 6.101

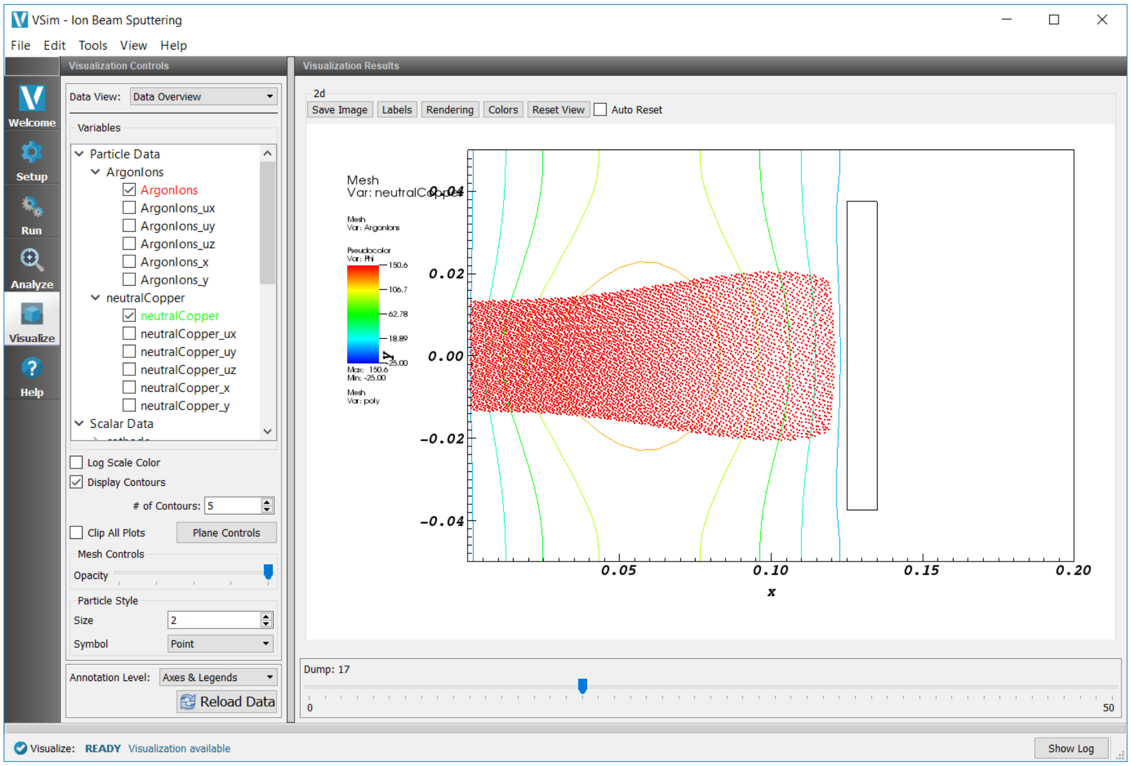
These plots show the energy deposited onto the cathode from the argon ion species as well as the number of neutral copper macro particles. From these plots its clear that the ion beam strikes the cathode after about 2.5 microseconds. A history for the total number of physical copper atoms is also available to be added to the simulation.

Further Experiments

1. Vary the *BEAM_ENERGY* constant, the species of ions in the beam, and the target material to see how the sputter yield changes in response (see the “numMacroCopper” history available in the *Visualize* Tab).

   To change the material of the target to see the affect on the sputtering yield, expand the *Particle Dynamics* element, then *neutralCopper*. Select the *sputterNeutralEmitter* and choose a new material from the “material properties” dropdown menu.

   Note: Savvy users may notice that the material of the cathode is also set in the *Geometries* element under *CSG* → *cathode*. This assignment of “PEC” sets the _electromagnetic_ properties of the geometry, not the particle



VSim - Ion Beam Sputtering

Data View: Data Overview

☑ ArgonIons

☑ neutralCopper

□ Log Scale Color

☑ Display Contours

# of Contours: 5

□ Clip All Plots

Size: 2

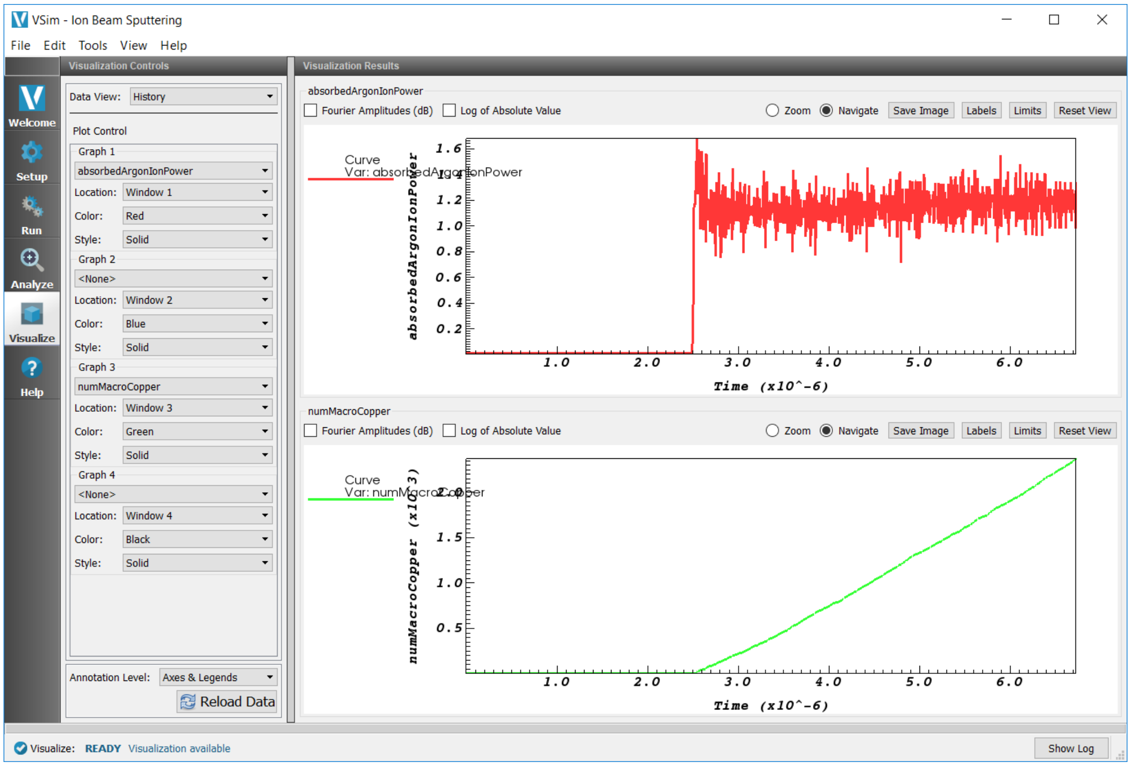
Symbol: Point

Annotation Level: Axes & Legends

Dump: 17

Visualize: READY Visualization available

Fig. 6.100: Ion Beam Sputtering a few dumps after the argon beam strikes the wall. The first few sputtered copper atoms can be seen in green.



VSim - Ion Beam Sputtering

Data View: History

Graph 1: absorbedArgonIonPower, Location: Window 1, Color: Red, Style: Solid

Graph 2: <None>, Location: Window 2, Color: Blue, Style: Solid

Graph 3: numMacroCopper, Location: Window 3, Color: Green, Style: Solid

Graph 4: <None>, Location: Window 4, Color: Black, Style: Solid

Annotation Level: Axes & Legends

Visualize: READY Visualization available

Fig. 6.101: Plots of the absorbed power and neutralCopper macro particle count.

properties.

2. Add electrons, secondary electron emission, an RF oscillating frequency on the cathode, and possibly an external magnetic field. Add a copperIons species and an electron impact ionization process to create copperIons from neutralCoppers. Add a second copper sputter emitter to the cathode that creates copper neutrals from copperIons to simulate self-sputtering.
3. Modify the beam current to account for the loss in current due to the mask. See the Negative Ion Beam example (*Negative Ion Beam (negativeIonBeam.sdf)*) for an example of how this is accomplished.

CHAPTER

SEVEN

TRADEMARKS AND LICENSING

For VSim™ licensing details please email sales@txcorp.com. All trademarks are the property of their respective owners. Redistribution of any VSim™ input files from the VSim™ installation or the VSim™ document set, including *VSim Installation*, *VSim Examples*, *VSim User Guide*, *VSim Reference*, and *VSim Customization*, is allowed provided that this Copyright statement is also included with the redistribution.

INDEX