# **XSim Examples**

Release 1.0.0rc3

**Tech-X Corporation** 

Apr 01, 2024

# CONTENTS

1	Over	view		1
2	XSim	n for Ele	ctromagnetics Examples	3
	2.1	Antenn	as	3
		2.1.1	2.4 GHz Yagi Uda Antenna (YagiUda2p4.sdf)	3
		2.1.2	Antenna Array 2D (antennaArray2D.sdf)	8
		2.1.3	Antenna on Human Hand with Dielectric (antennaOnHand.sdf)	17
		2.1.4	Antenna on Predator Drone (predatorDrone.sdf)	22
		2.1.5	Dipole Above Conducting Plane (dipoleOnConductingPlane.sdf)	28
		2.1.6	Dipole Antenna (dipoleAntenna.sdf)	31
		2.1.7	Dish Antenna (dishAntenna.sdf)	35
		2.1.8	Half-Wave Dipole in Free Space (halfWaveDipoleAntenna.sdf)	38
		2.1.9	Horn Antenna (hornAntenna.sdf)	43
		2.1.10	Loop Antenna from a Coaxial Cable (coaxialLoopAntenna.sdf)	47
		2.1.11	Patch Antenna Far Field (patchAntennaFarField.sdf)	50
	2.2	Cavitie	s and Waveguides	54
		2.2.1	Cylindrical Waveguide (cylindricalWaveguide.sdf)	54
		2.2.2	Circular Metal Waveguide Dispersion (circMetalWaveguideDisp.sdf)	58
		2.2.3	Pillbox Cavity (pillboxCavity.sdf)	70
		2.2.4	Rectangular Waveguide (rectangular Waveguide.sdf)	78
		2.2.5	Rectangular Metal Waveguide Dispersion (rectMetalWaveguideDisp.sdf)	80
		2.2.6	S-Matrix of Box Cavity (sMatrix.sdf)	92
	2.3	Photon	ics	97
		2.3.1	Dielectric Waveguide Mode Calculation (dielectricWaveguideModeCalc.sdf)	97
		2.3.2	Microring Resonator with Gaussian Launcher (ringResonatorGaussianLaunch.sdf)	106
		2.3.3	Dielectric Waveguide With Roughness (dielectric Waveguide Mode Calc Rough.sdf)	111
		2.3.4	Microring Resonator with Mode Launcher (ringResonatorModeLaunch.sdf)	118
		2.3.5	Y Splitter (ySplitter.sdf)	124
		2.3.6	Three Material Waveguide (three Material Waveguide.sdf)	129
	2.4	Scatteri	ing	135
		2.4.1	Scattering off a Metal Sphere (metalSphere.sdf)	135
		2.4.2	Scattering off a Metal Sphere with a Dielectric Coating (dielecCoatedMetalSphere.sdf)	140
		2.4.3	Scattering off Multiple Objects (dielecPlusMetalObjs.sdf)	144
In	dex			149

### Index

## CHAPTER

# ONE

# **OVERVIEW**

These are examples for illustrating the capabilities of XSim.

#### CHAPTER

TWO

# **XSIM FOR ELECTROMAGNETICS EXAMPLES**

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

These examples can be run with any license.

# 2.1 Antennas

### 2.1.1 2.4 GHz Yagi Uda Antenna (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 parasitic . Directionality is achieved by requiring that there be one longer element adjacent to the source element, which is referred to as the reflector. The rest of the elements being adjacent to the source but opposite to the reflector, and shorter than the source element, are referred to as 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.

#### **Opening the Simulation**

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

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the XSim 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. 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 Grids element and select or deselect the box next to Grid.



Fig. 2.1: 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.
- Check that you are using these run parameters:
  - Time Step: 9.532874347655025e-13
  - Number of Steps: 6000
  - Dump Periodicity: 200
  - Dump at Time Zero: Checked
- 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 result is shown in Fig. 2.2.



Fig. 2.2: 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 computeFarFieldFromKirchhoffBox.py from the list and select "Open" (Fig. 2.3)
- Input values for the analyzer parameters. The analyzer may be run multiple times, allowing the user to experiment with different values.
  - simulationName yagiUda2p4
  - fieldLabel E
  - farFieldRadius 1024.0
  - backgroundEpsRel 1.0
  - precision double
  - numPeriods 0.25
  - numFarFieldTimes 2
  - frequency 2.4e9
  - numTheta 45
  - numPhi 60
  - zeroThetaDirection (0,1,0)
  - zeroPhiDirection (0,0,1)
  - incidentWaveAmplitude blank
  - incidentWaveDirection (0,0,0)
  - varyingMeshMaxRadius 1024.0

- principalPlanesOnly checked
- Click "Analyze"
- Depending on the values of numTheta, numPhi, and numFarFieldTimes, the script may need to run for several minutes.

🔀 X5im - 2	.4 GHz Yagi Uda Antenna		- 0 X
File Edit	Tools View Help Window		
	Analysis & Controls	Analysis & Results	
X	Search: computeFarFieldFromKirchhoffBox	computeFarFieldFromKirchhoffBox.py 🔽	
		Annh To: Primary Run *	
Welcome	Available Analyzers: Open	During the primary start	Analyze Stop Clear Output
*	computeFarFieldFromKirchhoffBox.py		
- <del>**</del>		similationName Vani Ha2nd	
Setup		engeneral leftereth.	<u> </u>
:-		fektabel F	
13			
Prepare		farFieldRadks 1024 Writing Power and Phase on Principal Planes	
-			
- <del>1</del> 0		Use Variable: Dataset: PdBiThetaCompTheta90Plane written.	
Run		backgroundEpsRel 1 Dataset: PdBiThetaCompPhi0Plane written.	
=		Use Variable:	
		Databet: FusilitetationprintsGrade witchen.	
Design		harren loope	
0		numPeriods 0.25 Dataget: PdBiPhiCompTheta90Plane written.	
4		Lies Marihin V	
Analyze		Dataset: PdBiPhiComphiDPlane written.	
		numFarFieldTimes 2 Dataset: PdBiPhiCompPhi90Plane written.	
A Course Press		Use Variable	
visualize			
2		frequency 2.489 Dataset: PdBiTotalTheta90Plane written.	
Help		Use Variable: Dataset: PdBiTotalPhiOPlane written.	
		numTheta 45	
		Dataset: FasilotalPhisoPlane Writen Done writing file: Yagildarb PrincipalPlanes.vah5.	
		Use Variable:	
		numPhi 60 Dataset: PhasePhiCompTheta90Plane written.	
		Dataset: PhasePhiCompPhiDPlane written.	
		zeroThetaDirection (0,1,0) Dataset: PhasePhiCompPhi90Plane written.	
		Done writing file: YagiUda2p4_PwrAndPhsePrincipalPlanes.vsh5.	
		2eroPhiDirection ((J,U,1)	
		Inclantification Armola da	
		Datast: PhaseThetaComoPhiOPlane written.	
		Use Variable:	
		Dataset: PhaseThetaCompPhi90Plane written.	
		The following variables can be used in the above analyzer options:	
		[computeFarFieldFromKirchhoffBox] Not writing general 1D dataset	s.
	1	Analysis completed successfully	
	Delete Import Custom Analyzer		<b>•</b>
		u <u> </u>	
😋 ánahz	e: ANALYZER SUCCESS Analyzer finished successful	b.	Chrow Lon

Fig. 2.3: The Analysis Window.

#### Visualizing the results

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

To view the near field pattern, do the following:

- Expand Scalar Data
- Expand E
- Select *E\_x*
- Check the Set Minimum box and set the value to -0.1
- Check the Set Maximum box and set the value to 0.1
- Check the *Clip Plot* box
- Expand Geometries
- Select YagiUda2p4PecShapesTriangles\_surface
- 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. Uncheck the  $E_x$  dataset and check the farE\_magnitude box under *farE*.



Fig. 2.4: The electric field near-field pattern.



Fig. 2.5: The electric field manifestation of the far field pattern.

#### **Further Experiments**

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

# 2.1.2 Antenna Array 2D (antennaArray2D.sdf)

Keywords:

antennaArray2D, far field, radiation, s-parameters

#### **Problem Description**

This set of 2-D XSimEM simulations shows how to obtain the far fields, S11 parameter, gain, and phase shift of a one-element antenna as well as the far fields, gain, S parameters, and phase shift of a multiple-element antenna array with one excited element. These simulations can be used as a basis for measuring coupling in phased array antennas. The analyzer compute2DantennaGainAndPhase.py is set up to calculate the S parameter for the excited element and any other reference element defined by the constant *S\_PARAM\_ELEM*.

#### **Opening the Simulation**

The Antenna Array 2D example is accessed from within XSimComposer by the following actions:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting Examples window expand the XSim for Electromagnetics option.
- Expand the Antennas option.
- Select Antenna Array 2D and press the Choose button.
- In the resulting dialog, create a New Folder if desired, then press the Save button to create a copy of this example.

The resulting Setup Window is shown Fig. 2.6.

#### **Simulation Properties**

The antennas are waveguide apertures excited with a frequency of 1 GHz and the aperture width is  $0.1\lambda$  (see the parameter GAP in the element tree). The distance between the gaps is  $0.4\lambda$ .

A different array of geometries can be created using input parameters such as number of elements in the array (N\_ELEM) and the distance between the elements in each direction. To recreate a different antenna array, expand *Geometries*, expand CSG, right-click on  $gap \rightarrow$  Create Array. In the *Array Description* window, select the "Union elements" checkbox, type in the number of elements to the value under N\_ELEM, and the distance between elements to the value under DIST\_ELEM. Then select the CSG "metal", hold down Ctrl and select *gapElemUnion* located at the end of the gap array elements  $\rightarrow$  Boolean Operation  $\rightarrow$  select *metal\_gapElemUnion*. Rename accordingly and assign the material PEC to the newly created geometry.



Fig. 2.6: Setup Window for the Antenna Array 2D example.

#### **Running the Simulation**

Once finished with the setup, continue as follows:

- Proceed to the Run Window by pressing the Run button in the navigation column out left.
- 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 successfully when you see the output, "Engine completed successfully."
- First run settings (default):
  - Number of Steps: 6000
  - Dump Periodicity: 3000
  - Dump at Time Zero: box checked

After the first run completes, proceed as follows:

- Second run settings:
  - Number of Steps: 1800
  - Dump Periodicity: 45 (Value taken from the parameter DUMP\_PER\_SECOND\_RUN)
  - Set Restart at Dump Number to 2

Note: If the grid properties change, these values will have to be adjusted.

The end of the second run is shown in Fig. 2.7.

V

File Edit	File Edit Tools View Help Window			
	Runtime Options	Logs and Output Files		
X	Parameters Run Mode	Run Dump and Stop Force Stop Clear Log		
	Values entered here will override values from the simulation file.	Engine Log File Browser		
Welcome	Time Step	All multiFields dumped at 2024-03-21-15:09:22.069.		
<b>1</b>	Value: 5.5929042895358e-12	No electromagnetic fields to dump.		
	Use Variable:	No collisions to aump.		
Setup		Domain: Dump 43 completed at 2024-03-21-15:09:22.070.		
	Number of Steps	Deleting downin		
Decement	Value: 1000 Deraut value (0000)			
Prepare	Use variable:	Deleting downin.		
2.	Dump Periodicity	There were 0 Notices encountered in this run.		
Dum	Value:  45 Detault Value (3000)	There were 1 Warnings encountered in this run.		
Run	Use Variable:	VORPAL completed at clock time 2024-03-21-15:09:27.909.		
	Restart at Dump Number	NOTE: This simulation can be run if all of the packages ' GPU XSimBase XSimEN' are enabled. Other package combinations may be possible as well.		
Design	Value: 2	Please contact support@txcorp.com to investigate package combinations to meet your needs.		
Design	Use Variable:	Importing antennakray2D from D:\Users qar]bocuments\Tech-X\XSim-1.0\simulations\antennakray2D EM.		
	Dump at Time Zero	Importing verbosity from C:\Program Files\Tech-XXSim-1.0.0dev/Contents\engine\share\mmcros.		
Analuza	No Particle Sorting	Importing mathphys from C: Program Files/lech=Alsim=1.0.0dev(contents)tengine(share)matchmatchmatchs.		
Fullalyze	Disable Per-Rank Output	Importing VSim from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
	Custom Run Options	Importing deviceselect from C:\Program Files\Tech-XXSim-1.0.0dev/Contents\engine\share\macros.		
Visualize		Importing logicals from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
TISCICLEC	Reset to Setup Values	Importing units from C:/Program Files/Tech-X/XSim-1.0.0dev/Contents/engine/share/macros.		
2		Importing vertilies from Cirrogram Files)tech=ALSIM=L0.00ev(Ontents)engine)smatelmacros.		
Heln		Importing VSimEm from C:\Program Files\Tech-XXSim-1.0.0dev\Contents\engine\share\macros.		
		Importing mands from C:\Program Files\Tech=X\xsim=1.0.0mev\contents\engine\snare\macros.		
		Importing delayeval from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
		Importing matrices from C:\Program Files\Tech-X\XSim-1.0.0dev\Concents\engine\share\macros.		
		Importing emarting from Cifriguen Files/Tech-XiSin-10.04ev/Contents/engine/Smarton.		
		Importing electrostatics from C:\Program Files\Tech-XXXSim-1.0.0dev/(ontents)engine\Share\macros.		
		Importing implicite ectromagnetics from civrogram rikes/iece=/xikim=1.0.0mev(cntents)engine/share/marcos.		
		Importing grids from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
		Importing timing from C:)Program FilesTech-XXSim-1.0.0dev(Contents)engine)share)macros.		
		Importing shapes from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
		Importing multifields from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
		Importing stitutes from C:program Files)tech=XiX5h=1.0.0dev(Contents)engine/smartermacros.		
		Settings in the Z axis will be ignored		
		Lines from 'antennakray20.pre' processed. Finished with 'antennakray20.pre'.		
		END ENGINE OUTPUT		
		Francisco construction and an anticipation of the second		
		To see results, click on the "Visualize" icon in the icon panel.		
		Total simulation wall-clock time was 1142.54 seconds.		
		<u>1</u>		
💙 Run:	SUCCESS Simulation engine finished successfully	Show Lor		

Fig. 2.7: The Run Window at the end of the second run execution.

#### **Visualizing the Results**

After performing the above actions, the results can be visualized as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the navigation column
- Expand Scalar Data in the Visualization Controls pane
- Expand E
- Select *E\_x*
- Check the box for Set Minimum and set it to -100
- Check the box for *Set Maximum* and set it to 100
- Select the dump slider and move it to higher dump numbers to see the evolution of the electric field in time.

The resulting visualization is shown in Fig. 2.8.

Figure Fig. 2.8 shows the near and far electric fields at the end of the simulation run. The dispersion of the electric field through the non-excited waveguides can also be seen.



Fig. 2.8: The near and far electric fields in the x-direction at the end of the simulation.

#### Single Element Antenna

- Expand Constants
- Change constants N\_ELEM to 1
- Change N\_EXCITED\_ELEM to 1
- Expand Geometries
- Expand CSG
- Deactivate array
- Deactivate gapArray
- Select *metal*, hold Ctrl, select *gap*, right click  $\rightarrow$  *Boolean Operation*  $\rightarrow$  select *metal gap*
- Select metalMinusgap
- For material select PEC from the drop-down menu

You can now assign any name of your choice to the *metalMinusgap* geometry (e.g., aperture). Save and proceed to the Run tab. Follow the same run steps as described above in the section *Running the Simulation*.

Second vizualization results are shown in Fig. 2.9.



Fig. 2.9: The near and far electric fields in the x-direction for a 1-element antenna.

#### **Calibration Runs**

For both the multiple-element and single-element antenna simulations, calibration runs are needed for the analyzer. For the original multiple-element array setup, proceed as follows:

- Proceed to the Setup Window
- In the top left corner, select  $File \rightarrow Save Simulation As \dots$
- Rename the simulation to antennaArray2DCalibration.sdf

Note: If your simulation has a different name, add the word Calibration before .sdf

- Click Save
- Expand Geometries
- Expand CSG
- Deactivate array
- Deactivate gapArray
- Select gap
- Change the height to HEIGHT\_METAL\_CALIB
- Change the x position setting to XBGN\_EXCITED\_GAP
- Select metal
- Change the height to HEIGHT\_METAL\_CALIB

- Click on *metal*, hold down Ctrl button and select gap right click  $\rightarrow$  Boolean Operation
- Select metal\_gap
- Select metalMinusgap
- Select PEC under material from the drop-down menu.

You can now assign any name of your choice to the metalMinusgap geometry (e.g., myWaveguide).

- Expand Field Dynamics
- Expand FieldBoundaryConditions
- Remove *malUpperY*
- Right-click *FieldBoundaryConditions*  $\rightarrow$  Add FIeldBoundaryCondition  $\rightarrow$  select *Port*
- Select *upper y* for the boundary surface from the drop-down menu
- Save and proceed to the Run tab.
- Change *Number of Steps* to 7800

**Note:** The calibration number of steps must equal the total number of steps that the simulation ran for during the regular run.

Repeat the same steps for the single-element antenna simulation setup.

#### **Analyzing the Results**

After performing the above actions, continue as follows:

- Proceed to the Analysis Window by pressing the *Analyze* button in the navigation column.
- Open the compute2DantennaGainAndPhase.py analyzer by selecting it and selecting "open".
- The default analyzer fields are the following:
  - simulationName: antennaArray2D
  - dumpNr: 30
  - nlambda: 15.0
  - gapWidth: 0.03
  - center: 0.0,-4.4969
  - dt: 5.59290428954e-12
  - freq: 100000000.0
- The overwrite box should be checked
- Click *Analyze* in the top right corner.
- The analysis is completed when you see the output shown in Fig. 2.10.

The S-parameters for the excited element as well as the reference element associated with the constant S\_PARAM\_ELEM in the simulation setup are shown at the end of the analyzer run.

This analyzer creates a text file with 5 columns. The first column is the theta direction in degrees, the second column is the analytical gain of the ISOLATED excited element in dB, the third column is the gain measured by XSim in dB, the

City Call	n sam Pancharan ang du				
Pile Edit	tools view Help window				
	Analysis & Controis	Analysis & Results			
X	Search: compute2DantennaGainAndPhase	compute2DantennaGainAndPhase.py 🛛			
Welcome	Available Analyzers: Open	Apply To: Primary Run 💌	Analyze Stop Clear Output		
1	compute2DantennaGainAndPhase.py	I Run Automatically After Engine. Order: J.			
Sotup		simulationName antennaArray2D	Outputs		
i Sactop		dumpNir 30	-Jordwitte, - Whether a dataset or group should be overwritten if it already exists.		
Prepare		nlambda 15.0	This script prints out the S parameter of the excited element of the antenna		
Ø.,		aval4idta 0.02	array, as well as the S paramter of any of the non-excited elements (the non- excited element for this calculation is set in the simulation SETUP. Must be a		
Pup		gaprona i p.c.s	2D simulaton in the x-y plane, see VSimEM example Antenna Array 2D. This script creates a text file with 5 columns. The first column is the theta direction in		
		center 0.0,~4.4969	degrees, the second column is the analytical gain of the ISOLATED excited		
Design		dt 5.59290428954e-12	column is the analytical phase of the ISOLATED excited element in degrees, and the fith column is the phase measured by VSim in degrees. The name of the text		
0		freq 100000000.0	The is Simulationwant gainandrinsepata.txt. Ine default values of this analyzer are set to match the default setup of the Antenna Array 2D simulation and its		
Analyze		overwrite 🗆	documentation.		
~			Executing analyzer:		
			C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\bin\compute2DantennaGainkndPhase.py simulationName="antennakrray2D"dumpNr="30"nlambda="15.0"gapWidth="0.03"		
Visualize			center="0.0,-4.4969"dt="5.59290428954e-12"freq="1000000000.0" Run Start Time: Thu Mar 28 11:15:21 2024		
2			Run Directory: C:\Users\qar\Documents\Tech-X\XSim-1.0\simulations\antenna&rray2D_EM		
Help					
			Python variables defined by command line arguments for this analyzer:		
			dumpNr :: 30 [string]		
			nlambda :: 15.0 [string] gapWidth :: 0.03 [string]		
			center :: 0.0,-4.4969 [string]		
			freq:: 100000000.0 [string]		
			overwrite :: False [bool]		
			S_excitedElem = -2.66 + (7.34)i		
			S_excitedElem  = 7.81		
		The following variables can be used in the above analyzer options:	S_referenceElem = 0.43 + (-0.14)i  S referenceElem] = 0.45		
			File writtem: antennakrrav2D gainAndPhaseData.txt		
	1		Analysis completed successfully		
	Delete Import Custom Analyzer	<u>   </u>	<u>.</u>		
🕑 Analyz	Analyze: AVALYZER SUCCESS Analyzer finished successfully.     Show Log				

Fig. 2.10: The S-parameters for the excited element as well as the reference element associated with the constant  $S_{PARAM}$  ELEM in the simulation setup are shown at the end of the analyzer run.

fourth column is the analytical phase of the ISOLATED excited element in degrees, and the fith column is the phase measured by XSim in degrees. The name of the text file is SIMULATIONNAME\_gainAndPhaseData.txt.

For the default simulation settings (i.e., the center element of a 25-element array is excited while the other elements are turned off), plotting the second and third columns (analytical and measured gains) against the first column (as a function of theta) will give the results shown in Fig. 2.11.

Plotting the fourth and third columns (analytical and measured field phases) against the first column (as a function of theta) will give the results shown in Fig. 2.12.

#### **Further Experiments**

A different array of geometries can be created changing input parameters such as number of elements in the array (N\_ELEM) and the distance between the elements in each direction (DIST\_ELEM). After changing these *Constants*, to create a different antenna array, proceed as follows:

- Expand Geometries
- Expand CSG
- Right-click on  $gap \rightarrow Create Array$

In the *Array Description* window, select the "Union elements" checkbox, type in the number of elements to the value under N\_ELEM, and the distance between elements to the value under DIST\_ELEM. Then select the CSG "metal", hold down Ctrl and select *gapElemUnion* located at the end of the gap array elements  $\rightarrow$  Boolean Operation  $\rightarrow$  select *metal\_gapElemUnion*. Rename accordingly and assign the material PEC to the newly created geometry.

Repeating the analysis steps for a 1-element antenna ( $N\_ELEM = 1$  in the simulation setup) will give the results shown in Fig. 2.13 and Fig. 2.14.

A different element can be excited by changing input parameter N\_EXCITED\_ELEM.



Fig. 2.11: The gain pattern of a 25-element array with the center excited element.



Fig. 2.12: The phase pattern of a 25-element array with the center excited element.



Fig. 2.13: The gain pattern of a 1-element array.



Fig. 2.14: The phase pattern of a 1-element array.

Repeating the analysis steps for a 25-element antenna with the edge element excited (N\_EXCITED\_ELEM = 25 in the simulation setup) will give the results shown in Fig. 2.15 and Fig. 2.16.



Fig. 2.15: The gain pattern of a 25-element array with the edge excited element.

### 2.1.3 Antenna on Human Hand with Dielectric (antennaOnHand.sdf)

Keywords:

antennaOnHand, far field, radiation



Fig. 2.16: The phase pattern of a 25-element array with the edge excited element.

#### **Problem Description**

This problem calculates the far-field radiation pattern of a small wifi antenna. The fields interact with the human hand for which the bone structure was approximated by long thin cylinders. The antenna frequency can be either 2.4 or 5 GHz, the two most common wifi bands.

#### **Opening the Simulation**

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

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting Examples window expand the XSim 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. 2.17.



Fig. 2.17: Setup Window for the Antenna on Human Hand with Dielectric example, with Grid and farFieldBox History hidden.

#### **Simulation Properties**

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

#### **Running the Simulation**

- Proceed to the run window by pressing the Run button in the left column of buttons.
- Check that you are using these run parameters:
  - Time Step: 3.659083082938294e-12
  - Number of Steps: 3000
  - Dump Periodicity: 300
  - Dump at Time Zero: Checked
- 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 result is shown in Fig. 2.18.



Fig. 2.18: 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 from the list and select "Open" (Fig. 2.19)
- Input values for the analyzer parameters. The analyzer may be run multiple times, allowing the user to experiment with different values.
  - simulationName: antennaOnHand
  - fieldLabel: E
  - farFieldRadius: 1024.0
  - backgroundEpsRel: 1
  - precision: double
  - numPeriods: 0.25
  - numFarFieldTimes: 2
  - frequency: 5e9
  - numTheta: 45
  - numPhi: 60
  - *zeroThetaDirection*: (1,0,0)
  - *zeroPhiDirection*: (0,0,1)
  - incidentWaveAmplitude: blank

- *incidentWaveDirection*: (0,0,0)
- varyingMeshMaxRadius: 0.05
- principalPlanesOnly: checked
- Click "Analyze"
- Depending on the values of numTheta, numPhi, and numFarFieldTimes, the script may need to run for several minutes or longer.

🔀 XSim - J	XSm - Anterna on Human Hand with Dielectric – O X			
File Edit	Tools View Help Window			
	Analysis & Controls	Analysis & Results		
X	Search: computeFarFieldFromKirchhoffBox	computeFarFieldFromKrathoffBox.py		
Welcome	Available Analyzers: Open	Apprilo: Initiality Additional Content in Analyze Stop Clear Output		
Setup	,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,	simulationName [anternaCnHand		
睛		field.abol E Writing Power and Phase on Principal Planes		
Prepare		farFieloRadus 1024 Dataset: PdBiThetaCompTheta90Plane written.		
\$0		Use Variable: Dataset: PdBiThetaCompPhiOPlane written.		
Run		Dackytufuzpowej 1 Use Variable: Dataset: PdB:TheteCompPhi90Plane written. Done writing file: antenneOnHand_PwrAndPhsePrincipalPlanes.vsh5.		
Design		precision double Dataset: PdBiPhiCompTheta90Plane written.		
Q		numPeriods 0.25 Dataset: PdB1Ph1CompPh10Plane written.		
Analyze		Use Variable: Dataset: PdBiPhiCompPhi90Plane written. pumEwrEwHTimme 2		
Visualize		Use Variable: Dataset: PdBiTotalTheta90Plane written.		
<b>?</b>		frequency 569 Dataset: PdBiTotalPhiOPlane written.		
Help		Use Variable: Dataset: FdB3TotalPhi90Plane written. numTheka  45		
		Use Variable: Dataset: PhasePhiCompTheta90Plane written.		
		numPhi [60		
		zeroThetaOrection (10.00) Detaset: PhasePhiCompPhiGOPlane written. Done writing file: antennaOnHand_PwrAndPhsePrincipalPlanes.vsh5.		
		20r0PhDirection (0,0,1) Dataset: PhaseThetaCompThetagOPlane written.		
		ncidentWawAmpihudo Dataset: PhaseThetaCompPhiOPlane written.		
		Decaset: PhaseThetaCompPh:SOPlane witten. Done writing file: antenasOnHand PwrAndFhsePrincipalPlanes.vsh5.		
		[computeFarFieldFromKirchhoffBox] Not writing general 1D datasets.		
		imalygis completed successfully		
	Delete Import Custom Analyzer			
Analia	e: ANALYZER SUCCESS Analyzer finished successfully	Show Loo		

Fig. 2.19: The Analyze panel after running computeFarFieldFromKirchhoffBox.py.

#### Visualizing the Results

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.
- Expand Scalar Data
- Expand *farE*,
- Select *farE\_Magnitude*.
- Expand Geometries
- Check HandGeomSolidTriangles\_surface



Fig. 2.20: 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- translation values may be needed.

### 2.1.4 Antenna on Predator Drone (predatorDrone.sdf)

#### Keywords:

predatorDrone, far field, radiation

#### **Problem Description**

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

#### **Opening the Simulation**

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

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting Examples window expand the XSim 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. 2.21.



Fig. 2.21: Setup Window for the Predator Drone example.

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.

#### **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 that you are using these run parameters:
  - *Time Step*: 1.3068153867636764e-11
  - Number of Steps: 2500
  - Dump Periodicity: 100
  - Dump at Time Zero: checked
- 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 result is shown in Fig. 2.22.

Note the first with the first wit	Xim - Antenna on Predator Drone - O X			
Nume from         Log and number         Log and number           Number         Number         Number         Number         Number           Number         Number         Number         Number         Number         Number           Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Number         Num	File Edit	Tools View Help Window		
Restore         In standing         <		Runtime Options	Logs and Output Files	
<ul> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card for a tea card to the standard</li> <li>A tea card to the standard</li></ul>	M	Parameters Run Mode	Run Dump and Step Force Step Clear Log	
Way 100 marks 100	X	Values entered here will override values from the simulation file.	Engine Log File Browsor	
<pre>k w w w w w w w w w w w w w w w w w w w</pre>	Welcome	Time Step	Dumping invFaceFracs, invFaceFracs at 2024-03-19-13:09:15.148.	
Mark 12.82.179.212.000-11     Mark 12.83.179.212.000-11     Mark 12.84.179.212.000-11     Mark 12.84.179.212.000	~	Default Value (1.5246179512242887e-11)	Dumped invFaceFrace.invFaceFrace at 2024-03-19-13:09:15.162.	
	- <del>22</del>	Value: 1.5246179512242887e-11	Dumped invFaceFracs at 2024-03-19-13:09:15.162.	
	Setun	Use Variable:	All multiFields dumped at 2024-03-19-13:09:15.163.	
	octup	b) underst of Observ	No electromagnetic fields to dump.	
		Number of Steps	No collisions to dump.	
Write wind wind wind wind wind wind wind wind	100	Value: [2500 Default Value (2500)	Domain Duma 25 completed at 2024-03-19-13:09:15.163.	
	Prepare	Use Variable:		
• The second	-04	Durse Pariodicity	Main loop ended at clock time 2024-03-19-13:09:15.163	
	20	Uniter 100	Deleting domain	
		Value: 100 Denator value (100)		
With the second test and the second test and test	Run	Use Variable:	Deleting dowin.	
Wint wind wind wind wind wind wind wind wind		- Restart at Dumn Number	There were 0 Motices encountered in this run	
Note:         Note: <td< th=""><th></th><th>Value</th><th>There were 0 Warnings encountered in this run.</th></td<>		Value	There were 0 Warnings encountered in this run.	
• Works         • Works           • Works         <	Design	Tube.	VORPAL completed at clock time 2024-03-19-13:09:15.389.	
Res       R		Use variable:	NOTE: This simulation can be run if all of the packages ' XSimBase XSimEN' are enabled. Other package combinations may be possible as well. Please contact support@txcorp.com	
Particle Strage	• • •	Dump at Time Zero	to investigate package combinations to meet your needs.	
Picture       Image: Pictu		The Destine Costing	No expand symbols file. Will not limit expansion.	
Procession       Procession         Procession       P	Analyze	Find Fandle Sol ang	Importing predatorprome from D1()ders(dat)documents) leta-x1,x51m-1.003 mulations) predatorprome_an.	
Image: Section 2010 Control 100	~	Uisable Per-Rank Output	Importing values for a control fuel (Tech-S) SSID-1.0.0dev/Contents/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/engine/and/	
Amage         Amage <th< th=""><th></th><th>Custom Run Options</th><th>Importing mathphys from C:\Program Files\Tech-X\ZSim-1.0.0dev\Contents\engine\share\macros.</th></th<>		Custom Run Options	Importing mathphys from C:\Program Files\Tech-X\ZSim-1.0.0dev\Contents\engine\share\macros.	
Ment Status Yuku       The status Status Yuku         We       The status Status Yuku       The status Status Yuku Yuku Yuku Yuku Yuku Yuku Yuku Yu	Secoline	1	Importing VSim from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
Phy       Image: Internet	TRUCINEC	Reset to Setup Values	Importing deviceselect from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
The second se			Importing listUtilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
Where         The processing will like a free of Chrogone File Proc. Size + 1.0. dev/Construction register likes ress.           The processing will like a free of Chrogone File Proc. Size + 1.0. dev/Construction register likes ress.           The processing will like a free of Chrogone File Proc. Size + 1.0. dev/Construction register likes ress.           The processing will like a free of Chrogone File Proc. Size + 1.0. dev/Construction register likes ress.           The processing will like a free of Chrogone File Proc. Size + 1.0. dev/Construction register likes ress.           The processing will like a free of Chrogone File Proc. Size + 0.0. dev/Construction register likes ress.           The processing will ress.           The procestrip will ress. <th></th> <th></th> <th>Importing logicals from C:)Frogram Files/Tech-X(XSim-1.0.00eV)contents/engine/share/macros.</th>			Importing logicals from C:)Frogram Files/Tech-X(XSim-1.0.00eV)contents/engine/share/macros.	
The states much series and the state and the	Help		Importing whiles from circlegementates include the content of the content and	
Partial states and the construction of the state is a state is			Importing vputilities from C:\Program Files\Tech-X\ZSim-1.0.0dev\Contents\engine\share\macros.	
Phase to the citype File/Neb-323b-1.0.60v/Content/english hards/metro.           Importing meter from citype File/Neb-323b-1.0.8v/Content/english hards/metro.           Importing metrics from Citype File/Neb-323b-1.0.8v/Content/english hards/metro.           Importing file/Neb-323b-1.0.8v/Content/english ha			Importing VSimEm from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
The sectors in solution in the form of the sectors in the interpret of the sectors in the interpret of the sectors in the sect			Importing embcs from C:\Program Files\Tech-X\X3im-1.0.0dev\Contents\engine\share\macros.	
The sectors Sender was finded accessful.			Importing plasmaDielectric from C:\Program Files)Tech-XXXIm-1.0.0dev\Contents\engine\share\macros.	
The sectors makes the sectors and the sectors			Importing delayeval from CilyProgram FileSTRedm-AXS1m-1.0.0dev(Contents)engine(share)macros	
The sectors in the sector of t			Importing delayeral from C: Program Files/Tech-X/XSim-1.0. dev/Contents/engine/macros.	
Phr. sectors         Smaller			Importing esmatrix from C:\Program Files\Tech-X\Z3im-1.0.0dev\Contents\engine\share\macros.	
Par. MIXINS Sender more finded accessful:			Importing electrostatics from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
The sectors in the sectors in the sectors in the sectors and t			Importing implicitelectromagnetics from C:\Program Files\Tech-X\XSim-1.0.Odev\Contents\engine\share\macros.	
The sectors is subtrace that in the sectors in the			Importing transverse from C:\Program FileNTech-XXSim-1.0.0dev/Contents\engine\shareimerros.	
Part 1947 Studies and high studies and the stu			Importing grind from C: Program Files/Hon-A.A.SIM-I.U.U.GOVICONCENTS/Engline/Shake:ImScTOS.	
Derive stattes Smukton ergen fnied successfel			Importing that is from C: Program Files/Tech-XXSim-1.0.0dev/Contents/antenimetimetimetimetimetimetimetimetimetimet	
Phone starts         Substrate         Substrate         Substrate			Importing shapes from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
Importing strutes from (1)Program PilesTech-33301-1.0.049/Construct sequely hase/baseros. Importing strutes from (1)Program PilesTech-33501-1.0.049/Construct sequely hase/baseros. In Pilester Structure S			Importing multifields from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
Part 19 welling willing from C/Nerger File. The All Int. All Net All Int. All Net All Int. All Net All Net All Int. All			Importing stfuncs from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
In the Start of personal second in the Start of Star			Importing emfilters from Ct\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
In this is the second sec			Lines from 'predatorDrone.pre' processed.	
Tujike completed successful, Tujike completed successful, Tujike completed successful, Tujike completed successful, O'An: SKLISS Smukkon ergre funded successful O'An: SKLISS Smukkon ergre funded successful			Finished with "predactivities pre-	
Impute completed successfully.           To see results, clicko and the "Virualize" icon in the icon pasel.           Total similation will-clock time was 174,15 seconds.				
Image: To see results, click on the "Virmalize" icon in the icon panel.           Image: Total simulation wall-clock time was 274.15 seconds.           Image: Strukton ergine finited successfult			Engine completed successfully.	
Total similation wall-clock time was 174.15 seconds.			To see results, click on the "Visualize" icon in the icon panel.	
Image: specific section and specific and			Total simulation wall-clock time was 274.15 seconds.	
Arr. strass snuktor ege friede scorely			<u> </u>	
Run: SUCCESS Simulation engine finishind successfully Show Log			м —	
	Run:	Run: SUCCESS Simulation engine finished successfully Show Log		

Fig. 2.22: The Run Window at the end of execution.

#### **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 *computeFarFieldFromKirchhoffBox.py*, then click on the *Open* button.

If you want, you can grab the dividing 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.

After performing the above actions continue as follows to compute the far field radiation pattern:

- In the resulting list, select computeFarFieldFromKirchhoffBox and press Open
- The analyzer fields should be filled as below:
  - simulationName: predatorDrone
  - fieldLabel: E
  - farFieldRadius: 1024.0
  - backgroundEpsRel: 1
  - numPeriods: 0.25
  - numFarFieldTimes: 2
  - frequency: 1.0e9
  - numTheta: 45
  - numPhi: 60
  - *zeroThetaDirection*: (0,0,1)
  - *zeroPhiDirection*: (1,0,0)
  - incidentWaveAmplitude blank
  - incidentWaveDirection (0,0,0)
  - varyingMeshMaxRadius 1024.0
  - principalPlanesOnly checked
- Click *Analyze* in the top right corner.
- The analysis is completed when you see the output shown in Fig. 2.23.

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

• Press the Analyze button in the top left of the window.

At completion, you will see Fig. 2.23. The far field data is written to vsh5 files in the simulation directory.



Fig. 2.23: 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
- Check Clip Plot
- Check Display Contours and set the number of contours to 10
- Set minimum to -75 and maximum to 60
- Click on *Plane Controls*, change only Plane3 and set Z to -1.
- Expand Geometries
- Check predatorDronePecShapesTriangles\_surface
- Move the Dump slider to dump 15 to see the same far field as Fig. 2.24.

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 Results* can also be displayed. Remove the previous image. Then check the *PdBiTotal* box under Scalar Data, and move the dump slider to the beginning dump. You will see a 3D radiation surface, representing the Far Field radiation power level at each angle that was processed. Colors and radius are in units of dBi, decibels relative to isotropic. A notable peak in the radiation pattern is evident in the forward, upward, and downward directions, as seen in Fig. 2.25.



Fig. 2.24: The radiation pattern in real space



Fig. 2.25: 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.

### 2.1.5 Dipole Above Conducting Plane (dipoleOnConductingPlane.sdf)

Keywords:

#### dipoleOnConductingPlane, far field, radiation

#### **Problem Description**

This problem illustrates how to obtain far fields within XSim 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 XSimBase, but modified with functionality available as part of the XSimEM 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.

#### **Opening the Simulation**

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

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the XSim 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. 2.26. 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. 2.26: 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.
- Check that you are using these run parameters:
  - Time Step: 1.7380644643956894e-11
  - Number of Steps: 500
  - Dump Periodicity: 50
  - Dump at Time Zero: Checked
- 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 result is shown in Fig. 2.27.



Fig. 2.27: 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 1024.0
  - numPeriods 0.25
  - numFarFieldTimes 2
  - frequency 3.0e9
  - 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)
  - incidentWaveAmplitude blank
  - incidentWaveDirection (0,0,0)
  - varyingMeshMaxRadius 1024.0
  - principalPlanesOnly checked
- Click the *Analyze* button near the top right of the window.



Fig. 2.28: 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. Expand farE and then check the farE\_magnitude box. You may need to rotate the view and check the *Clip Plot* box to hide the virtual far field pattern under the conducting plane.

#### **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 analysis.

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 achieve 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 infinitesimal dipole is placed the same distance from the conducting "plane" in order to achieve the result of having an infinite electric conductor.

#### 2.1.6 Dipole Antenna (dipoleAntenna.sdf)

Keywords:

antenna, electromagnetics, current source



Fig. 2.29: The far field radiation pattern

#### **Problem Description**

Dipole antennas are the simplest and most widely used type of antenna. In the most basic setup, a dipole antenna is composed of an oscillating current/voltage source in between two electrodes. The frequency of the source will determine the wavelength of the electromagnetic radiation emitted from the antenna according to the dispersion relation

$$\lambda = \frac{c}{f}.$$

Most commonly, the electrodes will be 1/4 of the emitted wavelength. In this example, the antenna will be driven with a current oscillating with a frequency of 1 GHz. Therefore, the emitted wavelength will be roughly 30 cm, meaning we will make each of the electrodes 7.5 cm. This will make the total length of the antenna 15 cm, which is why dipole antennas are sometimes called half-wave antennas. It is easiest to drive the antenna when the electrodes are a quarter wavelength.

For more background information on dipole antennas, visit the Wikipedia page: https://en.wikipedia.org/wiki/Dipole\_antenna

#### **Opening the Simulation**

The Dipole Antenna example is accessed from within XSimComposer by the following actions:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting Examples window expand the XSim for Electromagnetics option.
- Expand the Antennas option.
- Select Dipole Antenna and press the Choose button.
- In the resulting dialog box, create a *New Folder* if desired, then press the *Save* button to create a copy of this example.

The resulting Setup Window is shown Fig. 2.30.


Fig. 2.30: Setup Window for the Dipole Antenna example.

## **Simulation Properties**

In this simulation, we will excite the antenna and watch the dipole electromagnetic radiation emanate from the antenna. A *distributed current* source is used to apply the driving current. A volume for the current source and the functional form of the current is set under *Field Dynamics*  $\rightarrow$  *CurrentDistributions*  $\rightarrow$  *driveCurrent*. The user has the ability to set all three components of the current within the volume. In this example, we set the x-component of the current using the *driveCurrent* spacetime function. The *driveCurrent* function is a sine wave oscillating at 1 GHz to which a smooth turn on profile has been applied.

There are open boundaries on the walls of 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.
- Check that you are using these run parameters:
  - *Time Step*: 4.573853853672867e-12
  - Number of Steps: 1000
  - Dump Periodicity: 100
  - Dump at Time Zero: Checked
- 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 result is shown in Fig. 2.31.



Fig. 2.31: The Run window at the end of execution.

## **Visualizing the Results**

After performing the above actions, the results can be visualized as follows:

- Proceed to the Visualize Window by pressing the Visualize button in the navigation column.
- 1. Expand Scalar Data
- 2. Expand E
- 3. Select  $E_y$
- 4. Check Clip Plot
- 5. Set Set Minimum to -0.1, and Set Maximum to 0.1.
- 6. Scroll through the dumps to see how the y-component of the electric field changes with time. The last dump is shown in Fig. 2.32.

## **Further Experiments**

- 1. Add an RCS Box around the antenna to measure the far field radiation pattern at a location of your choosing.
- 2. Modify the driving frequency or the dimensions of the electrodes.

🔀 XSim - U	Jipole Antenna	- 0 X
File Edit	Tools View Help Window	
	Add a Data View 💌 Choose a Run: PrimaryRun	Unicad Data     Rebad Data
X	Data Overview1 🛛	
Welcome	Visualization Controls	Visualization Results
	·	
\$	Variables	.30 Swe Labols Limits Axis Scale Rendering  +z → Resol Valw   IF Preserve Aspect Ratio F Auto Reset
Setup	Scalar Data	
国	antennaLeftinOut     antennaRightInOut	
Droparo	B destation for formation	
riepare	R- F	Psychocolor
Ф <sub>0</sub> –	E_magnitude	
Run	E.y	-00000
	- Ez	
	⊕ invFaceFracs	
Decim	8-1	
Design	SeparableCurrentDistributionTCross      Vector Data	
	Meshes	-0.1000
	1 0 C	Marc 0/344 Min - 0/310 6 6 -
Analyze	The second secon	
	Clin Plot Plane Controls	
Visualize	Log Scale Color	
	🗆 Display Contours 5	
	Opacity	
Help		
	Color Table hot_desaturated	
	Limits	
	Set Minimum -0.1	
	Provide Sector International Provide Sector I	
	Reset minimum	
	Set Maximum 0.1	
	Reset Maximum	
		412
	Reset to Default	
		0 0 (2.50 0.10 0.10 0.10 0.10 0.10 0.10 0.10 0
	, I	Dump: 6
	Annotation Level: Axes & Legends	0 10
🖸 Visuali	ize: READY Visualization available	Store Log

Fig. 2.32: The y-component of the Electric Field shows the expected 4-lobe pattern.

## 2.1.7 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.

## **Opening the Simulation**

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

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the XSim 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. 2.33. 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. 2.33: 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.
- Check that you are using these run parameters:
  - Time Step: 9.147707707345734e-12
  - Number of Steps: 1000
  - Dump Periodicity: 50
  - Dump at Time Zero: Checked
- 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 result is shown in Fig. 2.34.



Fig. 2.34: 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. 2.35, do the following:

- Expand Scalar Data
- Expand *E*
- Select *E\_x*
- Check Clip Plot
- Expand Geometries
- Select dishAntennaPecShapesTriangles

It is easier to see the fields if you change the color scale minimum and maximum. To do so, check the *Set Minimum* and *Set Maximum* boxes, 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.



Fig. 2.35: 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.

## 2.1.8 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 XSim simulation data. The simulation itself consists of a half-wavelength long current source in free space.

## **Opening the Simulation**

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

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the XSim 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. 2.36. 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. 2.36: Setup Window for the Half Wave Dipole Antenna example.

## **Simulation Properties**

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

- AMPLITUDE: The amplitude of the signal
- 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.
- Check that you are using these run parameters:
  - Time Step: 9.147707707345834e-12
  - Number of Steps: 4000
  - Dump Periodicity: 100
  - Dump at Time Zero: Checked
- 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 result is shown in Fig. 2.37.

• NOTE: the correct elements will not appear in the visualization step if the analysis step has not been performed first.

Eile Edit	Anni Tharmar Ugazi Unice and C				
File Edit	tools view Help window		_		
	Runtime Options	Logs and Output Files			
X	Parameters Run Mode	Run Dump and Step Force Step	ar Log		
	Values entered here will override values from the simulation file.	Engine Log File Provener			
Welcome	Time Step				
	Default Value (4.573853853672867e-12)	Dumped 3.3 at 2024-03-19-14:21:10.797	-		
23	Value: 4.573853853672867e-12	Pumptu v de destru-12-12-11:0:12:1:0:05			
	The Marinhia	No electromagnetic fielde to duro.			
Setup	Ose variable.]	No collisions to dump.			
:	Number of Steps	No ionization processes to dump.			
13	Value: 4000 Default Value (4000)	Domain: Dump 40 completed at 2024-03-19-14:21:10.853.			
0	the theight				
Prepare	ose variable:	Nain loop ended at clock time 2024-03-19-14:21:10.853			
100	Dump Periodicity	Deleting dozeain			
~ 8	Value: 100 Default Value (100)	An landow downlo			
Pun		Destring domain.			
Run	Use variable:	There were 0 Notices encountered in this run.			
	Restart at Dump Number	There were 1 Warnings encountered in this run.			
	Value:	See above for more information.			
Design	TUND.	VORPAL completed at clock time 2024-03-19-14:21:10.879.			
_	Use Variable:	NOTE: This simulation can be run if all of the packages ' GPU XSimBase' are enabled. Other package combinations may be possible as well. Please contact supportStxcorp.com to			
	Duran at Timo Toro	investigate package combinations to meet your needs.			
~	V Dump ac nine zero	No expand symbols file. Will not limit expansion.			
Analyze	No Particle Sorting	Importing halfWaveDipoleAntenna from Di\Users\qar\Documents\Tech-X\XSim-1.0\simulations\halfWaveDipoleAntenna_EM.			
~	Disable Per-Rank Output	Importing Verboarty Trom C:\Program Files\Tech=X:XSIm=1.0.0dev(contents)engine\share=hmacros.			
	Outon Run Ontione	Importing enums from C: Program FileS / ech-Alasim-1.0. Oudevicents/engine same: macros.			
	custom reports j	Importing Machines itom cirreduced Files itom Alasim 1.0.0000/concerts/engine same immetos.			
Visualize	Reset to Setup Values	Importing devices elect from Cifrogram Files Tech-XXSim-10.0dev(Contents)engine/share/macros.			
		Importing listUtilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing logicals from C:\Program Files\Tech-X\XSim-1.0.Odev\Contents\engine\share\macros.			
		Importing units from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
Help		Importing utilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing vputilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing VSimEm from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing embcs from C:\Program Files\Tech-X\XSim-1.0.Odev\Contents\engine\share\macros.			
		Importing plasmaDielectric from C:\Program Files\Tech-X\XSim-1.0.Odev\Contents\engine\share\macros.			
		Importing delayeval from Ct)Program Files\Tech-XtXSim-1.0.0dev\Contents\engine\share\macros.			
		Importing matrices itom civrogram Files/ied=Axia3m=1.0.0000/contents/engine/smare/matros.			
		Important de la construction de			
		Importing constraint into the composition files (controls) and the state (matter) and the state (matter) and the state (matter) and the state of the			
		Importing enclosed to four cirtuly and rises represented and represented and the share harron.			
		Importing tfsfhoxxd from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing grids from C:\Program Files\Tech-X\ZSim-1.0.0dev\Contents\engine\share\macros.			
		Importing timing from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing histories from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing shapes from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing multifields from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing stfuncs from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing emfilters from C:\Program Files\Tech=X\XSim=1.0.0dev\Contents\engine\share\macros.			
		Lines from 'half@aveDipoleAntenna.pre' processed.			
		Finished with 'halfWaveDipoleAntenna.pre'.			
		END ENGINE OUTPUT			
		Engine completed successfully.			
		To see tesuits, click on the "visualite" icon in the icon panel.			
		Inder STRATEGINE ASTLATON ATHE ASD GOODAG SCONNO.	크		
📿 Run:	SUCCESS Simulation engine finished successfully	Sho	w Log		

Fig. 2.37: 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:
  - simulationName halfWaveDipoleAntenna (name of the input file)
  - fieldLabel E (name of the electromagnetic field)
  - farFieldRadius 10.0 (radius of the far sphere, i.e., distance to the far zone)
  - numPeriods 0.25
  - numFarFieldTimes 2
  - frequency 3.0e9
  - numTheta 45 (number of points in the theta direction)
  - numPhi 90 (number of points in the phi direction)
  - zeroThetaDirection (0,0,1) (determines orientation of far field coordinate system)
  - zeroPhiDirection (1,0,0) (determines orientation of far field coordinate system
  - incidentWaveDirection (0,0,0)
  - incidentWaveAmplitude blank
  - varyingMeshMaxRadius 10.0
  - principalPlanesOnly checked
- Click the Analyze button in the upper right corner.

🔀 XSim -	talf-Wave Dipole in Free Space		- • ×
File Edit	Tools View Help Window		
	Analysis & Controls	Analysis & Results	
X	Search: computeFarFieldFromKirchhoffBox	computeFarFieldFromKirchhoffBox.py 🔯	
	Available Analyzers: Onen	Apply To: Primary Run 💌	
weacome		Run Automatically After Engine.     Order:	
\$	computeFarFieldFromKirchhoffBox.py	sim lationName halfWaveDinoleAnterna	Outputs
Setup			areaTotal = 12.5664, E2total = 5.43779e+14, sin2kvg = 0.666531
I.		fieldLabel  E	
Prepare		farFieldRadius 10	Writing Power and Phase on Principal Planes
Ø.,		Use Variable:	Dataget: PdBiTbetaComeTbeta90Plane written.
Run		backgroundEpsRel 1	Darager: P/8(ThetaCounDh(OP)ane written
=		Use Variable:	Darager: D/B(Thata/wwnD)(GOD)ang wr(fran
Design		precision double	Done writing file: halfWaveDipoleAntenna_PwrAndPhsePrincipalPlanes.vsh5.
0		numPeriods 0.25	hataast, BdBiDtiCompState00Diana unittee
Anakan		I top Variable	Pacades, Publich and Philophine and
Analyze			bacaset: Pasivnicompynioplane written.
		the Unitida	Dataset: FusiFricomprisoriane written. Done writing file: halfWaveDipoleAntenna_PwrAndPhsePrincipalPlanes.vsh5.
Visualize			
(2)		frequency (3.0e9	Dataset: PdBiTotalTheta90Plane written.
Help		Use Variable:	Dataset: PdBiTotalPhiOPlane written.
		numTheta  45	Dataset: PdBiTotalPhi90Plane written. Done writing file: halfWaveDipoleAntenna PwrAndPhsePrincipalPlanes.vsh5.
		Use Variable:	
		numPhi  90	Dataset: PhasePhiCompTheta90Plane written.
		Use Variable:	Dataset: PhasePhiCompPhiOPlane written.
		zeroThetaDirection (0,0,1)	Dataset: PhasePhiCompPhisoPhane written.
		zeroPhiDirection (1,0,0)	
		incidentWaveAmplitude	Dataset: PhaseThetaCompTheta90Plane written.
		Lise Varishie:	Dataset: PhaseThetaCompPhiOPlane written.
			Dataset: PhaseThetaCompPhi90Plane written.
		The following unrighter are becaused to the above real-set entropy	Done writing file: halfWaveDipoleAntenna_PwrAndPhsePrincipalPlanes.vsh5.
		The numming variables can be used in the above all the above all all all all all all all all all al	LannauraPacPicldPacePicldPacePicabbaddPacel Max melting general 15 december
			loomputeratriciumittomotibos) not witting general in unionets.
			Analysis completed successfully
	Delete Import Custom Analyzer		<b>X</b>
		U	
📿 Anah	e: ANALYZER SUCCESS Analyzer finished successfully		Show Log

Fig. 2.38: 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



Fig. 2.39: The far field radiation pattern

## **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 too small, the results will be distorted as the entire near field must be within the simulation domain in order to achieve a proper transformation to the far field.

## 2.1.9 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.

## **Opening the Simulation**

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

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the XSim 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. 2.40. 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.



Fig. 2.40: Setup Window for the Horn Antenna example.

### **Simulation Properties**

The antenna geometry in this example has been set up 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.

There are two scales that we need to resolve in this simulation. One is the wavelength and one is the smallest geometric scale to resolve (i.e. in this simulation it is the wall width). So, NX, NY, and NZ have been set up to resolve both scales.

## **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 that you are using these run parameters:
  - *Time Step*: 8.928466081792827e-12
  - Number of Steps: 1300
  - Dump Periodicity: 100
  - Dump at Time Zero: Checked
- 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 result is shown in Fig. 2.41.

🔀 XSim - I	Xiin-Hon Artena – O X				
File Edit	Tools View Help Window				
	Runtime Options	Logs and Output Files			
	Parameters Run Mode	Run Dump and Stop Force Stop	Clear Log		
	Values entered here will override values from the simulation file.	Engine Log File Browser			
Welcome	Time Step	Dumped invFaceFracs.invFaceFracs at 2024-03-19-14:24:28.814.	-		
A	Default Value (8.9284660317928276-12)	Dumped invFaceFracs at 2024-03-19-14:24:28.814.			
~	Value: 10.9204000017920270-12	All multiFields dumped at 2024-03-19-14:24:28.815.			
Setup	Use variable:	No collisions to dump.			
:=	Number of Steps	No ionization processes to dump.			
13	Value: 1300 Default Value (1300)	Domain: Dump 13 completed at 2024-03-19-14:24:28.816.			
Prepare	Use Variable:	Main loop ended at clock time 2024-03-19-14:24:28.816			
22	Dump Periodicity	Deleting domain			
	Value: 100 Default Value (100)	Deleting domain.			
Run	Use Variable:	OUTPUT SUMMARY:			
-	Restart at Dump Number	There were 0 Notices encountered in this run. There were 1 Meringe encountered in this run			
-	Value:	See above for more information.			
Design	I ke Varishle	VORPAL completed at clock time 2024-03-19-14:24:28.905.			
		NOTE: This simulation can be tun if all of the packages 'GPU Ximbase Ximba' are enabled. Other package combinations may be possible as well. Please contact supportExpropr.com to investigate package combines to meet your meeds.			
	Dump at Time Zero	No expand symbols file. Will not limit expansion.			
Analyze	No Particle Sorting	Importing hornAntenna from DiVersen quribournets/Itech-X/XSim-1.0) simulations/hornAntenna EN.			
~	Disable Per-Rank Output	Importing verification of the state of the s			
	Custom Run Options	Importing mathphys from C:\Programs Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
Visualize	Reset to Setun Values	Importing VSim from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\equive\share\manros.			
		Importing devices rectified on the other states and the states and			
		Importing logicals from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
нер		Importing units from C:\Program Files\Tech=X:XSim=1.0.0dev\Contents\=engine\Bare\macros.			
		Importing vputilities from C:\Program Files\Tech-XXSin-1.0.0dev\Contentslengine\share\mascros.			
		Importing VSimEm from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing embeds from C:  Program Files/Tech-XiXish-1.0.0499(contents)engine)share\macros			
		Importing delayeval from C:/Program Files/Tech-X/XSim-1.0.0dev/Contents/engine/share/bare/onc			
		Importing matrices from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing delayeval from CiVrogram FileS)rech_Xixim-1.0.000V/contents/engine/share/imacros.			
		Importing electrostatics from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing implicitelectromagnetics from C:)Program Files/Tech-X)XSim-1.0.0dev/Contents/engine/share/macros.			
		Importing transved from C:)Frogram FileStech-XiXSim-1.0.000V(Contents)engine(share)macros. Importing grids from C:)Frogram FileStech-XiXSim-1.0.000V(Contents)engine(share)macros.			
		Importing timing from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing histories from C:\Program Files\Tech-Z\ZSim-1.0.0dev\Contents\engine\share\macros.			
		Importing snapes from C:\Program Files\Pech-XXSIm-1.0.0dev\Contents\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engles\engl			
		Importing stfuncs from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Importing emfilters from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.			
		Lines from 'hornAntenna, pre' processed.			
		END ENGINE OUTPUT			
		To see results, cluck on the "Visualize" icon in the icon panel.			
		Total simulation wall-clock time was \$4.596 seconds.			
			<u> </u>		

Fig. 2.41: 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.
- In the resulting dialog, select computeFarFieldFromKirchhoffBox.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 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)
  - numPeriods 0.25
  - numFarFieldTimes 2
  - frequency 2.0e9
  - numTheta 45 (number of theta points in the far field, 18 for a quick calculation, 45 for finer resolution)
  - numPhi 90 (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
  - incidentWaveAmplitude blank
  - incidentWaveDirection (0,0,0)
  - varyingMeshMaxRadius 10.0
  - principalPlanesOnly checked
- 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  $E_{magnitude}$ . To slice inside the horn, select *Clip Plot* in the lower left hand corner. Click on *Plane Controls* and change the cut-plane normal to lie along Y instead of Z. 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. Try adjusting the min and max until the signal is well resolved (see Fig. 2.43).

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*.



Fig. 2.42: The Analyze panel after running computeFarFieldFromKirhhoffBox.py.



Fig. 2.43: The  $E_{magnitude}$  field propagating out of the horn at dump 3. The color scale has been log scaled and the min and max have been fixed to 0.005 and 6.5, respectively. The optimal min/max values will depend on the dump selected. The view has been rotated to show the x-z plane.



Fig. 2.44: The far field radiation pattern.

### **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 resolutions 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.

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.

## 2.1.10 Loop Antenna from a Coaxial Cable (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.

## **Opening the Simulation**

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

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting Examples window expand the XSim for Electromagnetics option.
- Expand the Antennas option.
- Select "Loop Antenna 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. 2.45.



Fig. 2.45: Setup Window for the Coaxial Loop Antenna example.

## **Simulation Properties**

This simulation makes use of the new coaxial waveguide Field Boundary Condition in XSim 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 exists from at least 1 cell outside of the simulation boundary 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.

### **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 that you are using these run parameters:
  - *Time Step*: 1.829541541469147e-12
  - Number of Steps: 400
  - Dump Periodicity: 20
  - Dump at Time Zero: Checked
- Click on the *Run* button in the upper left 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."

🔀 XSim - I	🕅 XSm- Loop Anterna from a Cosmil Cable — — 🔿 X					
File Edit	Tools View Help Window					
	Runtime Options	Logs and Output Files	٩.			
	Parameters Run Mode	Run Dump and Stop Force Stop Clear Log	3			
	Values entered here will override values from the simulation file.	Engine Log File Browser				
Welcome	Time Step	All multiFields dumped at 2024-03-22-09:48:15.743.	1			
<b>1</b>	Value (1.829541541469147e-12)	No electromagnetic fields to dump.	1			
	Use Variable:	No collisions to dump.				
Setup	At unkers of Chees	Domain: Dump 20 completed at 2024-03-22-09:48:15.743.				
	Value: 400 Default Value (400)	Main loop ended at clock time 2024-03-22-09:48:15.743				
Drenare	Use Variable:	Deleting domain				
- Charles	Dump Periodicity	Deleting domain.				
88	Value: 20 Default Value (20)	OUTPUT SUMMARY:				
Run	Use Variable:	There were 0 Notlees encountered in this run. There were 1 Warnings encountered in this run.				
-		See above for more information.				
	Value	VORPAL completed at clock time 2024-03-22-09:48:15.838. NOTE: This simulation can be run if all of the packages ' OPU XSimBase XSimEH' are enabled. Other package combinations may be possible as well.				
Design	Lise Variable	Please contact support@txcorp.com to investigate package combinations to meet your needs.				
		No expand symbols file. Will not limit expansion. Importing coaxialLoopAntenna from D:Versigaribocuments/Tech-X/XSim-1.0/simulations/coaxialLoopAntenna EM.				
	V Dump at Time Zero	Importing Verbosity from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.				
Analyze	No Parocle Sorong     Disable Par-Park Output	Importing enume from C:)Program Files/Tech-X:XSim-1.0.0dev/ContentS)engine(share/macros. Importing mathahys from C:)Program Files/Tech-X:XSim-1.0.0dev/ContentS)engine(share/macros.				
		Importing VSim from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.				
	Custom kun Options j	Importing deviceselect from C:\Program Files\Tech-X\XSim-1.0.0deviContentS\engine\share\macros. Importing listUtilities from C:\Program Files\Tech-X\XSim-1.0.0deviContentS\engine\share\macros.				
Visualize	Reset to Setup Values	Importing logicals from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.				
		Importing units from c://rogram fileS/lech-X/xSim=1.0.0dev/contentS/englme/share/macros.				
Help		Importing vputilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.				
пар		Importing VSimEm from C:)Programs Files/Tech-X/XSim-1.0.Odev/ContentS/engine/share/macros. Importing embcs from C:)Programs Files/Tech-X/XSim-1.0.Odev/ContentS/engine/share/macros.				
		Importing plasmaDielectric from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.				
		Importing delayeval from C:\Program Files\Tech-XXSim-1.0.0dev\Contents\engine\share\macros. Importing matrices from C:\Program Files\Tech-XXSim-1.0.0dev\Contents\engine\share\macros.				
		Importing delayeval from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.				
		Importing emmatrix from C:\Program Files\Tech-XXSim=1.0.0dev\Contents\engine\share\matrix. Importing electrostatics from C:\Program Files\Tech-XXSim=1.0.0dev\Contents\engine\share\matrix.				
		Importing implicitelectromagnetics from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.				
		Importing tfsfboxxd from Ci)Frogram Files/Tech-XXSim-1.0.0dev/Contents/engine/share/macros.				
		Importing timing from C:\Program Files\Tech-X\XX in-1.0.0dev\Contents\engine\share\macros.				
		Importing histories from C:\Program Files\Tech-XXSim-1.0.0dev\Contents\engine\share\macros.				
		Importing multifields from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\Smarehmacros.				
		Importing struncs from C:\Program Files\Tech-X)XSim-1.0.0dev\Contents\engine\share\macros.				
		Lines from 'coaxiallopantenna, pre' processed.				
		Finished with 'coaxialLoopAntenna, pre'.				
		Engine completed successfully.				
		Total simulation wall-clock time was 37.206 seconds.	1			
		1 IIII	1			
		u				
📿 Run:	Run: SUCCESS Simulation engine finished successfully Show Log					

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

#### **Visualizing the Results**

After the run completes, the field may be visualized:

- Proceed to the Visualize Window by pressing the Visualize button in the left column of buttons.
- Expand Scalar Data
- Expand E,
- Select *E\_z*.
- check Clip Plot in the lower left hand corner,
- Click on Plane Controls and change the clip plane normal to Y instead of Z.

- Adjust the origin of the normal vector from Y = 0 to Y = 0.05.
- Check the Set Minimum and Set Maximum boxes
- Set the minimum to -10 and the maximum to 10.
- Drag the *Dump* slider to the far right for dump 20.
- Finally, click and drag the visualization to rotate it so that you can see the field.

🔀 XSim -	Loop Antenna from a Coaxial Cable		- 0 ×
File Edit	Tools View Help Window		
	Add a Data View 💌 Choose a Run: PrimaryRu		Unload Data Reload Data
X	Data Overview1		
Welcome	Visualization Controls	Visualization Results	
		21	
<b>.</b>	Variables	Save Labels Limits Axis Scale Rendering + Reset View Preserve Aspect Ratio Auto Reset	
Setup	🖻 Scalar Data 🔺		
三	B-B		
:====	B-E		
Prepare	E_magnitude	Vor Ez 1000	
₽¢.	E.y M E z		
Run	edgeFracs     imeExtenses	-5.000	
=			
	B- loopAntenna1inOut	-5.000	
Design	<ul> <li>IoopAntenna2InOut</li> <li>waveLauncherHorizontalPlusCross</li> </ul>		
	waveLauncherVerticalPlusCrossDev	Nor - 102 # 10.00	
Analyze			
-	Clip Plot Plane Controls		
	Log Scale Color		
visualize	E Display Contours 5	<b>4</b>	
2	Opacity		
Help			
	Color Table  hot_desaturated _		
	Limits		
	Peret Minimum		
	Set Maximum 10		
	Reset Maximum		
	Reset to Default		
			1
		Lump: 20	
	Annotation Level: Axes & Legends 💌		20
		A	
🕑 Visua	alize: READY Visualization available		Show Log

Fig. 2.47: The  $E_z$  field propagating off of the loop antenna.

## 2.1.11 Patch Antenna Far Field (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.

## **Opening the Simulation**

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

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting Examples window expand the XSim for Electromagnetics option.
- 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. 2.48.



Fig. 2.48: Setup Window for the Patch Antenna 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.
- Check that you are using these run parameters:
  - *Time Step*: 1.877687371507808e-12
  - Number of Steps: 2343
  - Dump Periodicity: 100

- Dump at Time Zero: Checked
- 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 result is shown in Fig. 2.49.



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

### Analyzing the Results

After performing the above actions continue as follows to compute the far field radiation pattern:

- Proceed to the Analysis Window by pressing the Analyze button in the navigation column.
- In the resulting list, select computeFarFieldFromKirchhoffBox and press Open
- The analyzer fields should be filled as below:
  - simulationName: patchAntennaFarField
  - *fieldLabel*: E
  - farFieldRadius: 1024.0
  - backgroundEpsRel: 1
  - precision: double
  - numPeriods: 0.25
  - numFarFieldTimes: 2
  - frequency: 5.5e9
  - numTheta: 45

- numPhi: 60
- *zeroThetaDirection*: (0,0,1)
- *zeroPhiDirection*: (1,0,0)
- *incidentWaveAmplitude*: blank
- *incidentWaveDirection*: (0,0,0)
- varyingMeshMaxRadius: 1024.0
- principalPlanesOnly: checked
- Click *Analyze* in the top right corner.
- The analysis is completed when you see the output shown in Fig. 2.50.

🔀 XSim - Patch Antenna Far Field			- • ×
File Edit Tools View Help Window			
Analysis & Controls	Analysis & Results		
Search: computeFarFieldFromKirchhoffBox	computeFarFieldFromKirchhoffBox.py		
Unisonna Available Analyzers: Onen	Apply To: Primary Run 💌		testes Con Class Coned
compute Suffield From Kirchhoff Poy nu	Run Automatically After Engine. Order:	1 🔆	Analyze Sup Clear Output
Q2	simulationName patchAntennaFarField	Outputs	
Setup			-
15.	fieldLabel  E	Writing Power and Phase on Principal Planes	
Prepare	farFieldRadius 1024.0		
Ø.,	Use Variable:	pataset: publiketa.ompineta.oppineta.	
Run	backgroundEpsRel	Dataset: PdBiThetaCompPhiOPlane written.	
		Dataset: PdBiThetaCompPhi90Plane written.	
E	Use Variable:	pone writing file: patchantennararrield_rwrandrhserrincipalrianes.vsh5.	
Design	precision double	Dataset: PdBiPhiCompTheta90Plane written.	
0	numPeriods 0.25	Dataset: PdBiPhiCompPhiOPlane written.	
Analyze	Use Variable:	Dataset: PdBiPhiCompPhi90Plane written.	
*	numFarFieldTimes	Done writing file: patchAntennaFarField_PwrAndPhsePrincipalPlanes.vsh5.	
	Use Variable:		
Visualize	frammery 5 5a0	Pacasec: Pasificalinecasoriane written.	
	Line Versiehlen	pataset: PdBiTotalPhiOpiane written.	
Help		Dataset: PdBiTotalPhi90Plane written. Done writing file: patch&ntennaFarField_Pwr&ndPhsePrincipalPlanes.vsh5.	
	numineta  45	_	
	Use Variable:	Dataset: PhasePhiCompTheta90Plane written.	
	numPhi  60 —	Dataset: PhasePhiCompPhiOPlane written.	
	Use Variable:	Dataset: PhasePhiCompPhi90Plane written.	
	zeroThetaDirection (0,0,1)	Done writing file: patchAntennaFarField_PwrAndPhsePrincipalPlanes.vsh5.	
	zeroPhDirection (1.0.0)	Dataset: PhaseThetaCompTheta9OPlane written.	
		Dataset: PhaseThetaComoPhiOPlane written.	
	incidentWaveAmplitude	Depart Dear There Comp De (ODD) ave un (tran	
		Done writing file: patchAntennaFarField_PwrAndPhsePrincipalPlanes.vsh5.	
	The solowing variables can be used in the above analyzer options:		
		[computerarrieidfromKirchhoffBox] Not writing general 1D datasets.	
		Analysis completed successfully	
Delete Import Custom Analyzer		· · · · · · · · · · · · · · · · · · ·	·
	U ~ .		
Analyze: ANALYZER SUCCESS Analyzer finished successful	y		Show Log

Fig. 2.50: Add the computeFarFieldFromKirhhoffBox.py script to your simulation.

## **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*.



Fig. 2.51: The Far Field Radiation Pattern

## **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 expanding 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.

# 2.2 Cavities and Waveguides

## 2.2.1 Cylindrical Waveguide (cylindricalWaveguide.sdf)

### Keywords:

electromagnetics, waveguide, dispersion

### **Problem Description**

This XSimEM example illustrates how to find the modes of a cylindrical waveguide.

### **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 XSimComposer by the following actions:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting Examples window expand the XSim for Electromagnetics 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. 2.52. 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.

The sinc hat function is used to excite this example. This function has a Fourier spectrum that is fairly flat for  $f_l < f_h$  and falls off rapidly over a frequency width of  $\delta_f$ , so that it is nearly zero for  $f < f_l - \delta_f$  or  $f < f_h + \delta_f$ .  $\delta_f$  is automatically calculated by the sinc hat function based on the suppression factor and frequency gap factor. 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 button in the left column of buttons.
- Check that you are using these run parameters:
  - *Time Step*: 5.168041416871939e-12
  - Number of Steps: 20000
  - Dump Periodicity: 2000
  - Dump at Time Zero: Checked
- Click on the *Run* button in the upper left corner of the right pane.



Fig. 2.52: Initial Setup Window for the Cylindrical Waveguide example.

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 result is shown in Fig. 2.53.

## **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. The first History tab will have eMid\_0 and eMid\_1 plotted. Open a second History tab in order to plot eMid\_2 as well.
- Then for each plot select the Fourier Amplitude (dB) 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. 2.54.

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.



Fig. 2.53: Run Window for the Cylindrical Waveguide example after the initial run.



Fig. 2.54: Spectrum for the Cylindrical Waveguide example after the initial run.

### **Computing More Accurate Modes**

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. 2.55.



Fig. 2.55: 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. 2.56.

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. 2.57. The well obtained mode occupies dumps 1-16.

## 2.2.2 Circular Metal Waveguide Dispersion (circMetalWaveguideDisp.sdf)

Keywords:

Waveguide, Dispersion Relation, Fourier Transform, Phase Shifting Boundary Conditions

🔀 XSim - (	Cylindrical Waveguide		– • ×
File Edit	Tools View Help Window		
	Analysis & Controls	Analysis & Results	
X	Search: extractModes	extractModes.py 🛛	
Welcome	Available Analyzers: Open	Apply To: Primary Run 💌	ánalize Ston Clear Outruit
	antractModer nu	Run Automatically After Engine.     Order:	
<b>\$</b>	extractModesViaOperator.py	simulationhama evindricatiVavaguida	Outputs
Setup		annaada marne juyn na uannarogane	already exists.
15		field E	This analyzer outputs three columns of data with the titles from (Hz) (aigapende
:===		hashift was 140	frequency), invQ (inverse quality factor), and SVD (singular value
prepare		Beginbump [10	decomposition), and creates new eigenmode fields that may be viewed in the VSimComposer Visualize tab. For example, a simulation with a magnetic field
. Ф <sub>о</sub>		Use Variable:	faceB would have the scalar fields faceB_0/1/2 (eigenmode) in addition to the
Run		endDump 16	magnetic fields faces_0/1/2 (faces).
-		Use Variable:	Executing analyzer:
		nModes 2	C:\Programs Files\Tech-X\XSim-1.0.0dev\Contents\engine\bin\extractModes.py
Design			numberUniformPts="10"numberRandomPts="100"randomSampleconstructoverwrite
O,		Use Variable:	Run Start Time: Thu Mar 21 11:38:14 2024 Run Directory: D:\Users\qar\Documents\Tech-X\XSim-1.0\simulations/cylindricalWaveguide EM
Analyze		numberUniformPts 10	
~		Use Variable:	
		numberRandomPts 100	Python variables defined by command line arguments for this analyzer: simulationName :: cylindricalWaveguide [string]
Visualize		Ise Variable	field :: E [VsHdf5.Field]
2		redexCambo II	endDump :: 16 [int]
Help		randomsample IV	nModes :: 2 [int] numberUniformPts :: 10 [int]
		construct 🔽	numberRandomPts :: 100 [int]
		overwrite 🔽	construct :: True [bool]
		Refresh Variable Lists	overwrite :: True (bool)
			freq (Hz) invQ SVD Amplitude
			(1) 1.49277e+09 6.411e-05 11463 2.082e-01
			[FilterDiagonalizationMethod.py] Retaining positive mode by convention for mode: 0
			[FilterDiagonalizationMethod.py] Wrote file cylindricalWaveguide_EigenE_0.vsh5
		The following variables can be used in the above analyzer options:	[FilterDiagonalizationMethod.py] Choosing positive mode by convention for mode: 1
			(FilterDiagonalizationMethod.pv) Wrote file cylindricalWaveguide EigenE 1.vsh5
	1		Analysis completed successfully
	Delete Import Custom Analyzer		
		4	
Anaka Anaka	TO: ANALYZER CHECKER Analyzer finished automotivily		Show Log

Fig. 2.56: Analysis window for the Cylindrical Waveguide example for mode extraction.



Fig. 2.57: Analysis window for the Cylindrical Waveguide example for mode extraction.

## **Problem description**

This XSimEM example demonstrates several unique capabilities of XSim that can be used to efficiently model waveguides. One unique capability is the use of phase shifting periodic boundary conditions. These work by adding a phase to a wave at a boundary to mimic a much longer physical dimension. We also demonstrate how to use an analyzer called extractModesViaOperator.py to accurately compute the excited modes in the waveguide, even if some of the modes are much weaker than the dominant mode. In this example, we use extractModesViaOperator.py to compute the dispersion relation ( $\omega$  vs k) and compare the numerically determined dispersion relation with the theoretical dispersion relation using standard waveguide theory.

This example involves five steps. In the first step, the waveguide is "pinged" with a short pulse in the current density that excites a range of modes. The Fourier transform then shows the range of frequencies of the modes. In the second step, the waveguide is excited using a sinc pulse function multiplied by a Gaussian envelope to excite a flat band of frequencies with sharp cutoffs at either end. The excitation current density is in the transverse directions (z and y) which excites an electric field primarily in the transverse direction. In the third step, the data is restarted from the end of the second run and saved at shorter time intervals in order to resolve the frequencies of interest. The output from the third step is used by the extractModesViaOperator.py analyzer to compute the eigenmodes in step 4. Finally, in step 5, the dispersion relation is computed by varying the wavelength which is resolved in the simulation.

## **Opening the Simulation**

The circular metal waveguide dispersion example is accessed from within XSimComposer by the following actions:

- Go to  $File \rightarrow New \rightarrow From Example...$
- In the resulting *Examples* window expand the XSim for Electromagnetics option.
- Expand the Cavities and Waveguides option.
- Select "Circular Metal Waveguide 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.

The properties and values that create the simulation are accessible in the left pane when the Setup Window is selected as shown in Fig. 2.58. The right pane shows a 3D view of the selected geometry components, grids and current distributions.

The geometry can be visualized by expanding "Geometries" in the left pane. The hollow circular waveguide can be seen more clearly by de-selecting the "Grid" option. The length in the direction of propagation (x) is a small fraction of either transverse direction. This is possible because we are using "phase shifting periodic" boundary conditions in the *x*-direction. The phase shifting BCs are selected under "Basic Settings".

A sinc hat function is used to excite the waveguide.

## Phase Shifting Boundary Conditions and Phase Shift

Before discussing phase-shifting periodic boundary conditions, we first review ordinary periodic BCs. In this discussion, periodic BC's are applied in the x-direction. With normal periodic BC's, there are two criteria in resolving a wave (or mode) of interest. (1) There must be enough grid points along the wavelength to resolve the spatial profile. Typically we sample at least 20 cells along a wavelength ( $\lambda_x$ ), or  $DX = \lambda_x/20$ , and (2) The length of the simulation domain, represented by  $L_x$ , must be one wavelength long,  $L_x = \lambda_x$ . Periodic BC's in the x-direction means that  $F(0) = F(L_x)$ , where F is any field quantity. Now introduce a grid that extends from 0,1,...,nx and let x = 0 at grid point 0 and  $x = L_x$  at grid point nx. Periodicity on the grid implies F(0) = F(nx). Finally, let's assume that  $F(x) \sim \sin(k_x x)$ . Then, applying the condition for periodicity,  $\sin(0) = \sin(k_x L_x)$ , which is exactly met if  $L_x = \lambda_x$ .

With phase-shifting periodic BC's, we are no longer required to meet the second criteria discussed in the preceding paragraph. To see this, let's suppose that  $L_x < \lambda_x$ . Then the periodicity condition can still be met by setting  $k_x L_x - \phi_0 = 0$ ,



Fig. 2.58: Initial Setup Window for the Waveguide Dispersion example. The block in the middle is the region in which the current density is driven.

where  $\phi_0$  is the phase shift equal to  $k_x L_x$ , which is the exact phase shift we have chosen to apply in XSim for this example. The numerical implementation is more challenging than the conceptual picture we just discussed. To implement phase-shifting periodic BC's, we need to treat the fields as complex numbers and set  $F(L_x) = \exp(i\phi_0)F(0)$ . For the grid, we pick 2 cells such that  $L_x = 2DX$ .

By using phase-shifting periodic BC's, we can simulate different physical lengths without changing the simulation length  $L_x$  through the phase shift  $\phi_0 = k_x L_x$ . Because  $k_x = 2\pi/L_x$ , we can solve for  $\omega(k_x)$  without requiring a simulation of length  $L_x$ .

### **Running the Simulation and Analyzing Results**

We now walk through the five steps discussed in the Introduction. In the first step, we test to determine the minimum frequency that will propagate through the waveguide. For the circular waveguide in this example, we know the analytical result which is the lowest cutoff frequency. However, we still do this step to demonstrate how to determine the lowest frequency that will propagate for cases that are not analytic. In the second step, the current density "rings up" to a maximum value, then rings down to 0. Because we are exciting modes at resonant frequencies of the cavity, the *E*- and *B*-fields continue to oscillate after the excitation is turned off. We wish to determine the resonant modes using Eigen mode analysis using data saved in Step 3 when the externally driven source is turned off and the cavity is "ringing" at its natural frequencies. In step 4, extractModesViaOperator.py is used to compute the excitation frequencies in the cavity at a given mode. Finally, in step 5, you will change the value of "mode", which is defined under "Constants" to compute the dominant frequencies a traditional periodic simulation is 0.2 m and denote this as *lambdaMax*. We then define the parameter  $k_x = \frac{mode}{12} \frac{2\pi}{lambdaMax}$  and run 25 simulations with *mode* =0,1,2,...,24. Finally, we set the phase shift to  $k_x \times L_x$  to ensure the correct phase shift is applied for the phase shifting periodic BC's. In this way, we are able to compare the simulation with waveguide theory without explicitly changing the length of the simulation domain in the *x*-direction.

### Step 1: Determining the lowest cutoff frequency - The "Ping" Run

The Ping run is used to determine the lowest propagation frequency in the waveguide. In this simulation, we can analytically compute the lowest propagation frequency at a given k based on the allowable modes in a circular wave guide. We can then compare the computed lowest propagation frequency with theory. However, for arbitrarily-shaped waveguides, this step is necessary since no theory is available to compute the lowest propagation frequency. We have defined a PINGON constant (with values of 0 or 1) in order to easily transition from the "Ping" run to the "Excitation" run in Step 2. The default is PINGON=1 to do the "Ping" run first. For Step 1, perform the following steps

- Click on the Run Window.
- This simulation may be accelerated by changing the Run Mode to Parallel
- Click the Run button on the upper left corner of the Logs and Output Files pane

The run has completed when you see the output, "Engine completed successfully." A snapshot of the simulation run completion is shown in Fig. 2.59.

XSim - 0	Circular Metal Waveguide Dispersion	- 0 X
File Edit	Tools View Help Window	
	Runtime Options	Logs and Output Files
X	Parameters Run Mode	Run Dump and Stop Force Stop Clear Log
	Values entered here will override values from the simulation file.	Engine Log   File Browser
weicome	Time Step Default Value (1.9804685343689165e-12)	111 multiFields dumped at 2024-03-21-11:49:04.453.
<b>\$</b>	Value: 1.9804685343689165e-12	No collisions to dump.
Setup	Use Variable:	No ionization processes to dump.
:=	-Number of Steps	Pomain: Pump s completed at 2024-03-21-11:49:04.453.
13	Value: 5000 Default Value (5000)	Nain loop ended at clock time 2024-03-21-11:49:04.453 Delative downin
Prepare	Use Variable:	e contra
2.	Dump Periodicity	Deleting domain. OUTPUT SUMMARY:
	Value: 1000 Default Value()	There were 0 Notices encountered in this run.
Run	Use Variable:	inere were i warnings encountered in this run. See above for more information.
	Restart at Dump Number	VORPAL completed at clock time 2024-03-21-11:49:04.491.
Design	Value:	Nois, nue sinatation chin de tan in all la ching parages ord asimate asima all empirita vinet parage constinctions may be possible as well. Please contact supportExcoord incords in a contracting and a contact support of the contact support support of the contact support of the contact support of t
	use variable.j	No expand symbols file. Will not limit expansion. Importing circRetalWavenuideDism from DivUsers/carbocuments/Tech-X/XSim-1.0/simulations/circRetalWavenuideDism EM.
e	Dump at Time Zero	Importing verbosity from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.
Analyze	Disable Per-Bank Output	Importing enume from C: Program Files/Tech-X(XXIn-1.0.0dev)ContentS/engine(share)smacros. Importing mathphys from C: Program Files/Tech-X(XXIn-1.0.0dev)ContentS/engine(share)smacros.
1	Custom Run Ontione	Importing VSim from C: ) Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.
Visualizo		Importing devices/etcot from Criving and Files/Tech-ASX Sim-1.0.0dev(Contents)tengine)share(macros.
TISCOLLC	Reset to Setup Values	Importing logicals from CiProgram FileSTech-XXSim-1.0.0dev/Contents)engine/share/macros.
		Importing utilities from C:\Program Files\Tech-Z\XSIm-1.0.0dev\Contents\engine share\macros.
Help		Importing Vputilities from C:\Program Files\Tech-X\X3im-1.0.0dev\Contents\engine\share\macros. Importing VSimEm Fom C:\Program Files\Tech-X\X3im-1.0.0dev\Contents\engine\share\macros.
		Importing embcs from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.
		Importing plasmaDielectric from C:/Program Files/Tech-X:XSim-1.0.0dev/Contents/engine!share/mmarcos. Importing delayeval from C:/Program Files/Tech-X:XSim-1.0.0dev/Contents/engine!share/mmarcos.
		Importing matrices from Ci\Program Files/Tech-XXSim-1.0.0dev/Contents)engine/share/macros.
		Importing densative for CiFrogram FileSiteCim-ALSIMP.LO.Odev(Contents)englime(SmaltemBottoS.
		Importing electrostatics from C:   Forgram Files Tech-X XSim-1.0.0dev/Contents engine share macros.
		Importing if firstelections includes the state of the sta
		Importing grids from C:)Program Files/Tech-X/3Sim-1.0.0dev/Contents/engine/share/macros. Importing timing from C:)Program Files/Tech-X/3Sim-1.0.0dev/Contents/engine/share/macros.
		Importing histories from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.
		Importing shapes from C:1Program Files/Tech-XXSim-1.0.0dev/Contents/engine/share/imacros. Importing multifields from C:1Programs Files/Tech-XXSim-1.0.0dev/Contents/engine/share/imacros.
		Importing stfuncs from C:\Program Files\Tech-X\XSim-1.0.Odev\Contents\engine\share\macros.
		Importing emfilters from C:\Program Files\Tech-XXSim-1.0.0dev\Contents\engine\share\macros. Lines from 'circHealWavequideDisp.prc' processed.
		Finished with 'circMetalWaveguideDisp.pre'.
		antipation and an antipation and antipation antipati
		Engine completed successfully.
		Total simulation wall-clock time was 21.062 seconds.
		1 <u> </u>
Run:	SUCCESS Simulation engine finished successfully	Show Loo

Fig. 2.59: The Run window at the end of a Step 1.

After the run has completed, click on the *Visualize* tab on the left side of the visualization window. After the data has loaded, click on *Add a Data View* and then select *History*. Perform the following steps to analyze the history:

- Graph 1 select *jMid\_1*
- Graph 2 select *eMid\_1*

In Graph 1, to see the data, click on the "Limits" option above the plot. Set the X-axis upper limit to 2e-10. You should see a step function in the current density beginning at t=0 and ending abruptly after about 20 time steps. This is the "ping" that we impose on the simulation. To determine the lowest frequency mode that we excite, click the "Fourier Amplitudes (dB)" box in Graph 2. Again, click on the "Limits" button above and to the right of the graph. Set the X-axis upper limit to 4e9. You will see that the first peak occurs at a frequency of about  $1.75 \times 10^9$  Hz, which is the lowest cutoff frequency in the simulation and is what we find analytically using waveguide theory. Therefore, we have confirmed that XSim reproduces linear theory. You can confidently use XSim to numerically determine the

lowest cutoff frequency for non-analytically solvable shapes. The visualization window after the above steps have been performed is shown in Fig. 2.60



Fig. 2.60: History plots showing the lowest cutoff frequency at  $1.75 \times 10^9$  Hz as the result of pinging the waveguide with an abruptly applied current density.

### **Step 2: Excitation**

From Step 1, we know that the lowest propagation frequency is  $\sim 1.75 \times 10^9$  Hz. We can now check that the lowest cutoff frequency *(frequency low in SincHat)* is correct. The parameter *FREQ\_MIN* is passed into *frequency low* in the **SincHat** function and is indeed  $\sim 1.75 \times 10^9$  Hz. We can now proceed to excite the waveguide with confidence knowing that lowest excitation frequency is correct. To excite the waveguide, perform the following steps:

- Go to the Setup Window and expand the Constants option
- Set PINGON to 0
- This will change the applied current to the SincHat function.
- Click on *Save and Setup* in the upper right corner of the visualization window
- Go to the Run Window by pressing the Run button in the left column of buttons.
- Set Number of Steps to 21000. This ensures that the simulation runs long enough for the current density to "ring up" then "ring down" to 0.
- Leave the "Dump Periodity" at 1000.
- This simulation may be accelerated by running on multiple MPI ranks. The parallel options are in the *Run Mode* 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." A snapshot of the simulation run completion is shown in Fig. 2.61.



Fig. 2.61: The Run window at the end of a Step 2.

### Step 3: Evolving the excited cavity

The purpose of Step 3 is to continue the simulation with only the *E*- and *B*- fields excited due to the externally imposed current density from Step 2. We will also save more data which we can analyze using Eigenmode analysis for Step 4.

- Go to the Setup Window and expand the Basic Settings option.
- Change "number of steps" to "10000".
- Change "steps between dumps" to "100".
- Change "dump in groups of" to "3".
- Click on Save and Setup in the upper right corner of the visualization window.
- Go to the Run Window and click the "Reset to Setup Values" button in the lower right corner of the "Run Options" section.
- In the Run Options section, uncheck the Dump at Time Zero box and set the Restart at Dump Number to 21.
- NOTE: the "Dump Periodicity" must be blank for the "dump in groups of 3" setting to work. In this case, the simulation will revert to the values defined in "Basic Settings" which are to dump every 100 time steps in groups of 3 (21000,21001,21002, ..., 21100,21101,21102,..., etc). If you fill in an integer into Dump Periodicity, the default gets overwritten and the data are not dumped in groups of 3, which is required for the extractModesVia-Operator.py analyzer. So, there should be 300 new dumps making a total of 321 dumps.
- 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. 2.62. When this run is finished, the last step should be step 31000.

File Edit	Tool: View Haln Window		~ ~
Pile Eult	Puntime Ontione	Love and Output Elize	_
	Natione Options	Logis and Output thes	
V	Parameters Run Mode	Run Dump and Stop Force Stop	Clear Log
	Values entered here will override values from the simulation	Engine Log File Browser	
Welcome	file.	111 milt fields dumad at 2024_03_28_15:11:46.060	
	Time Step	No electromagnetic fields to dump.	
- <del>V</del>	Default Value (1.9004000343009100e-12)	No collisions to dump.	
Setup	Value: 1.96040653430691038-12	No ionization processes to dump.	
	Use variable:	Powarn. Pump 321 Completed at 2021-03-20-13.11.10.000.	
13	Number of Steps	Main loop ended at clock time 2024-03-28-15:11:46.060	
Prenare	Value: 10000 Default Value (10000)	Deleting domain	
Trepare	Use Variable:	Deleting downin.	
20.	- Dumo Parindicity	CUTPUT SUMMARY:	
	Value: Default Value1	There were 0 Notices encountered in this run.	
Run	Use Use Idea and a second and a second	There were 1 Warnings encountered in this run.	
	Use variable:	VORPAL completed at clock time 2024-03-28-15:11:46.098.	
	Restart at Dump Number	NOTE: This simulation can be run if all of the packages ' GPU XSimEM' are enabled. Other package combinations may be possible as well. Please	
Design	Value: 21	contact support@txcorp.com to investigate package combinations to meet your meeds.	
	Use Variable:	No expand symmols file. will not limit expansion. Importing circMetalWayenuideDism from D:Wasen\oarDocuments\Tech-X\XSim-1.0\simulations\circMetalWayenuideDism EM.	
	E Duran et Time Terr	Importing verbosity from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
Analyze	bump at time zero	Importing enums from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
~	E Diskle Der Derle Grant	Importing mathphys from C:)Program Files)Tech-XXX31m-1.0.00eV(Contents)engine(Share)macros.	
	Disable Per-Kank Output	Importing deviceselect from C:)Program FileS/Tech-X,35m-1.0.0dev[Contents]englie[share]macros.	
Visualize	Custom Run Options	Importing listUtilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
-	Reset to Seturi Values	Importing logicals from C:\Program Files\Tech-XXSim-1.0.0dev\Contents\engine\share\macros.	
2		Importing utilities from C: Program Files/Tech-XISIm=1.0.00dev(Contents/engine/smattemacros.	
Links		Importing vputilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
нер		Importing VSimEm from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing champed from C: Program FileS) icon-xisim-1.0.0000/Contents/engine/smartementos.	
		Importing delayeval from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing matrices from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing delayeval from C:\Program Files\Tech-X XSim-1.0.0dev\Contents\engine\share\matrix	
		Importing electrostatics from C:\Program Files/Tech-XXSim-1.0.0dev\Contents/tengine\share\macros.	
		Importing implicitelectromagnetics from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing ffsboxd from C:\Program Files\Tech-X:XSim-1.0.0dev/Contents)engine\share\macros.	
		Importing grins from C: Program Files Tech-Alaim-1.0.0dev/Contents/engine/smart/mactos.	
		Importing histories from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing shapes from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing multifields from C: (Program Files)Tech-AlxSim-1.0.0dev)Contents(engine)Share(macros.	
		Importing emfilters from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Lines from 'circHetalWaveguideDisp.pre' processed.	
		Finished with 'circMetalWaveguideDisp.pre'.	
		END ENGINE 001P01	
		Engine completed successfully.	
		To see results, click on the "Visualize" icon in the icon panel.	
		local simulation wall-clock time was 61.935 seconds.	
			·
-			_
💙 Run:	SUCCESS Simulation engine finished successfully		Show Log

Fig. 2.62: The Run window at the end of Step 3.

### Step 4: Computing the eigenmodes

- Go to the analyzer window by selecting Analyze in the left column.
- Select *extractModesViaOperator.py* from the list of available analyzers. Then click "Open" on the top right of the *Analysis Controls* pane.
- Compute the electric field eigenfunctions. After the analyzer loads, ensure the following parameters are entered:
  - simulationName: "circMetalWaveguideDisp"
  - outputSimName: leave blank
  - realFields: "E"
  - imagFields: leave blank
  - secondFieldFactor: leave blank
  - operator: "d2dt2"
  - dumpRange: "21:321"
  - cellSamples: ":,5:55:5,5:55:5"
  - cutoff: "1e-12"
  - maxNumModes: "-1"
  - initialModeNumber: "0"
  - normalizeModes: checked
  - testing: unchecked
  - compMajorC: unchecked

### - overwrite: checked

The dump range runs only over the data saved in step 3 and we are only sampling every 5 cells in the *z*- and *y*- directions which is adequate to compute the eigen modes. Double-check your entries against what is shown in Fig. 2.63. After you run the analyzer, you will need to scroll up to find the computed frequencies. The screenshot shown in Fig. 2.63 is from a visualization window that is scrolled up so that the computed frequencies can be compared with what you ran. Figure Fig. 2.63 shows that there are 6 unique modes. One indication that you are using extractModesViaOperator.py correctly is that the imaginary part of the frequency (which represents attenuation of the wave) is nearly 0 or at least much smaller than the real part. The lowest excited mode is  $\sim 1.76 \times 10^9$  Hz, which is discussed further below in the context of standard waveguide theory.



Fig. 2.63: Computing the electric field eigenfunctions and frequencies using the extractModesViaOperator.py analyzer.

## Step 5: Computing the Dispersion Relation

Once the Eigenmode analysis is complete in Step 4, it is necessary to record the three lowest-order modes. The preloaded value of *mode* is 1, which means we are simulating  $k_x = \frac{1}{24} \frac{2\pi}{\lambda}$ , where  $\lambda = 0.2$  m. Also, the waveguide dispersion relation is given by  $\omega^2 = k_x^2 c^2 + \omega_{mn}^2$ , where  $\omega_{mn}^2 = \frac{c^2}{R^2} (x'_{mn})^2$ , where *m* and *n* are integers and  $x'_{mn}$  is the nth root of  $J'_m(x) = 0$  (the derivative of the cylindrical Bessel function of the first kind). The first three allowable modes are  $x'_{11} \approx 1.841$ ,  $x'_{21} \approx 3.054$  and  $x'_{01} \approx 3.832$ . Furthermore, the frequency range that is excited lies between  $f_l = 1.75 \times 10^9$  Hz (which is the lowest allowable propagation frequency) and  $f_h = 5.2 \times 10^9$  Hz. Given these parameters, we expect from waveguide theory for the three lowest modes to be  $\sim 1.76 \times 10^9$ ,  $2.92 \times 10^9$ ,  $3.66 \times 10^9$ Hz, which is what is found from extractModesViaOperator.py. Repeating these steps for modes 1 to 24 yields the dispersion relation shown in Fig. 2.64. The overlap between the simulation results and theoretical values is evident in Fig. 2.64.



Fig. 2.64: Dispersion relation found by changing *mode* in XSim. The solid line is theoretical dispersion relation  $\omega^2 = k_x^2 c^2 + \omega_{mn}^2$ . The 'X's represent data from the simulation results.

### **Convergence Study**

To demonstrate that the simulation results converge to the theoretical value, we have performed a series of simulations in which DX = DY = DZ are varied. The values chosen are DX = 0.25, 0.33, 0.5 and 0.733 cm. We then computed the lowest propagation frequency using *extractModesViaOperator.py* and plotted this frequency versus  $DX^2$ . Using Richardson extrapolation, we then compute the lowest propagation frequency for  $DX^2 = 0$ , which provides a better comparison with the theoretical frequency than a simulation with finite  $DX^2$ . The field calculation error scales approximately with  $DX^2$  so a plot of frequency versus  $DX^2$  should be a line. The linear correlation between frequency and  $DX^2$  is shown in Fig. 2.65. The extrapolated frequency at  $DX^2 = 0$  is  $\sim 1.75686 \times 10^9$  Hz. The theoretical value is  $\sim 1.75681 \times 10^9$ . Therefore, we see from Fig. 2.65 that the computation of the lowest propagation frequency for the DC mode converges with order  $DX^2$ , which is expected since the field solve is 2nd order accurate. Furthermore, using Richardson extrapolation, we see that the difference between theory and simulation, with  $DX^2 = 0$ , is 0.003 %.



Fig. 2.65: Plot demonstrating convergence of frequency calculation.

### 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.
- Click on *Reload Data*

The first step will be to ensure that we are driving the current density as expected and that the E- and B- fields are "ringing" as a result of driving the current density. Follow these steps to perform this check:

- Click on the Add a Data View pull-down menu at the top of the visualization window
- Click on *History*. This will open a new tab.
- Under Graph 1 plot *jMid\_1*. This is the *y* component of the current density
• Under Graph 2 plot *eMid\_1*.

Your plots should be similar to Fig. 2.66. The current density is driven for a short time period. However, because we are driving resonant modes, the current density excites the transverse electric field and parallel magnetic field. You can also plot  $eMid_0$ ,  $bMid_1$ , and  $bMid_2$  to compare with Fig. 2.66. Since  $eMid_0$  is ~ 0, we are driving a TE waveguide.



Fig. 2.66: History plots showing the modes driven in this simulation

We next wish to examine the spectral characteristics of the current density, which is driven between *FREQ\_MIN* and *FREQ\_MAX*. Therefore, the Fourier Transform of  $J_y$  or  $J_z$  should be greatest in this frequency range. Returning to the History visualization window, under Graphs 2, 3, and 4, change the plotted quantity to *none*, so that only Graph 1 shows a plot. Now check the "Fourier Amplitudes (dB)" box in the upper left corner. Finally, check the "Zoom" box on the right side of the visualization window and highlight from 0 to  $10^{10}$  Hz which will expand the lower frequency part of the plot. After you have zoomed in on the plot, re-check the "Navigate" box. The result of these steps should lead to a plot that looks similar to Fig. 2.67. The driving frequencies lie between *FREQ\_MIN* and *FREQ\_MAX* as expected. The rate at which the Fourier signal dampens below *FREQ\_MIN* and above *FREQ\_MAX* depends on the Gaussian envelope and the parameter called *OMEGA\_SIGMA*.

## **Further Experiments**

The result of varying *mode* from 0 to 24 is shown in Fig. 2.64. One experiment you can perform is to reproduce Fig. 2.64 by running 25 simulations with *mode* varying from 0 to 24. Each simulation takes just a few minutes so the 25 simulations takes about one hour. For each simulation record the three lowest frequencies that are excited using extractModesViaOperator.py.

Another experiment would be to change the dimensions of the waveguide. The constant *InnerRadius* is used to determine the lowest frequency that will propagate in the waveguide. Therefore, changing *InnerRadius* will automatically compute *FREQ\_CUTOFF\_TE*. You will then need to change *BGNY*, *ENDY*, *BGNZ*, and *ENDZ* so that the primitive fits within the simulation domain. The primitive geometry called "pipe0" scales with the constant *InnerRadius*.

A third experiment would be to modify *FREQ\_MIN* and *FREQ\_MAX* and compute the dispersion curve for these new values. No mode will propagate below *FREQ\_CUTOFF\_TE* so do not set *FREQ\_MIN* below *FREQ\_CUTOFF\_TE*.



Fig. 2.67: Fourier Transform of the current density showing that the current density is mainly driven between *FREQ\_MIN* and *FREQ\_MAX* as expected.

Lastly, you can try to propagate a TM mode instead of a TE mode.

## 2.2.3 Pillbox Cavity (pillboxCavity.sdf)

## Keywords:

## Pillbox cavity, Figures of merit, Transit time factor, Geometry factor

## **Problem description**

This XSimEM example demonstrates the usage of XSim 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.

## **Opening the Simulation**

The pillbox cavity example is accessed from within XSimComposer by the following actions:

- Go to  $File \rightarrow New \rightarrow From Example...$
- In the resulting Examples window expand the XSim for Electromagnetics 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*.



Fig. 2.68: Visualizing the periodic cavity geometry in the Setup Window.

## **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*  $\rightarrow$  *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 *Run Mode* 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 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. 2.69.

XSim -	Pillbox Cavity		0 X
File Edit	Tools View Help Window		
	Runtime Options	Logs and Output Files	
	Parameters Run Mode	Run Dump and Stop Force Stop	Clear Log
	Values entered here will override values from the simulation file.	Engine Log File Browser	
weicome	Time Step	kl1 multiFields dumped at 2024-03-21-14:18:37.648.	-
<b>11</b>	Value: 1 15002210076625e-12	No electromagnetic fields to dump.	
**	value: 1.1380822100/8623e-12	No collisions to dump.	
Setup	Use variable:	Domain Dump   completed at 2024-03-21-14:18:37.649.	
:=	Number of Steps		
13	Value: 30000 Default Value (30000)	Main loop ended at clock time 2024-03-21-14:18:37.649	
Prepare	Use Variable:	peleting domain	
502	Dump Periodicity	Deleting domain.	
	Value: 30000 Default Value (30000)	There were 0 Notices encountered in this run.	
Run	Use Variable:	There were 1 Warnings encountered in this run.	
-	Destant at Down Municipal	See above for more information.	
	Restart at Dump Number	VORFAL completed at clock time 2024-03-21-14:18:37.781. NOTF: This similarion can be run if all of the parkages ' GPH YSimRase YSimFM' are enabled. Other parkage combinations may be possible as w	e11
Design	Value:	Please contact support twoor, com to investigate package combinations to meet your needs.	
	Use Variable:	No expand symbols file. Will not limit expansion.	
	V Dump at Time Zero	Importing pillboxCavity from D: Users\qar\Documents\Tech-X\Sim-1.0\simulations\pillboxCavity_EM.	
Anakan	No Particle Sorting	Importing enume from C: Program Files Tech-XSim-1.0. dev/Contents/engine/share/macros.	
Analyze	Disable Per-Rank Output	Importing mathphys from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
	Custom Due Cestines	Importing VSim from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
	Custom Kun Options	Importing deviceselect from C:)Program Files)Tech-XX51m-1.0.0dev)Contents)engine(share)macros.	
Visualize	Reset to Setup Values	Importing logicals from Ci\Program Files/Tech-XiXSim-1.0.0dev/Contents/tengine/share/macros.	
		Importing units from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing utilities from C:\Program Files\Tech=X\X3im-1.0.0dev\Contents\engine\share\macros.	
Help		Importing Values from C. Program Files Tech-XXSin-1.0. Odev(Ontents)endineshare/matters.	
		Importing embcs from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing plasmaDielectric from C:\Program Files/Tech-XXSim-1.0.0dev\Contents\engine\share\macros.	
		Importing metrics from C: Program Files Tech-X:XSIm-1.0.0dev) contents > engine   share   macros.	
		Importing delayeval from C:\Program Files\Tech-X\X3im-1.0.0dev\Contents\engine\share\macros.	
		Importing esmatrix from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing electrostatics from C:\Program Files\Tech-XX35im-1.0.Odev\Contents\engine\share\macros.	
		Importing implicite tector magnetics from files tector and files tector. Source tector is a file internation of the source of th	
		Importing grids from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing timing from C:\Frogram Files/Tech-XXXSim-1.0.0dev/Contents\engine\share\macros.	
		Importing histories from Civrogram Files/ieCn-Akain-10.00ev/contents/engine/share/macros	
		Importing multifields from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing stfuncs from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing emfilters from C:\Program Files\Tech-X:XSim-1.0.0dev\Contents\engine\share\macros.	
		Finished with 'pillbox.dwity.pre' processes.	
		END ENGINE OUTPUT	
		Ingane completed successfully. To see results, click on the "Misualize" icon in the icon panel.	
		Total simulation wall-clock time was 53.912 seconds.	
			-
		u-	
📿 Run:	SUCCESS Simulation engine finished successfully		Show Log

Fig. 2.69: The Run window at the end of a successful execution.

## 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. 2.70. 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.

🔀 XSim - I	Allbox Cavity	- 0	×
File Edit	Tools View Help Window		
	Runtime Options	Logs and Output Files	-
V	Parameters Run Mode	Run Dump and Stop Force Stop Clear	Log
	Values entered here will override values from the simulation file.	Engine Log File Browser	
Welcome	Time Step	All multiFields dumped at 2024-03-21-14:19:10.657.	-
<b>1</b>	Value: 1.158082210076625e-12	No electromagnetic fields to dump.	_
	Use Variable:	No collisions to dump.	
Setup		Domain: Dump 21 completed at 2024-03-21-14:19:10.658.	
	Number of Steps	Main loop ended at clock time 2024-03-21-14:19:10.658	
Decement	Use Unrighter	Deleting domain	
Prepare		Deleting downin	
20	Dump Periodicity	CUTFUT SUMMARY:	
Run	Value: 100 Detaut Value (30000)	There were 0 Notices encountered in this run.	
Ruit	Use Variable:	See above for more information.	
	Restart at Dump Number	VORPAL completed at clock time 2024-03-21-14:19:10.790.	
Design	Value: 1	NOTE: This simulation can be run if all of the packages ' OFU ssimulate ssimult are enabled. Other package combinations may be possible as well. Please contact support@txcorp.com to investigate package combinations to meet your needs.	
	Use Variable:	No expand symbols file. Will not limit expansion.	
	Dump at Time Zero	Importing pillboxCavity from D:\Users\qav\Documents\Tech-X\XSim=1.0\osimulations\pillboxCavity_EN. Importing verbosity from C:\Fromars Files\Tech-X\XSim=1.0\dev\Contents\engine\barcanstelmacros.	
Analyze	No Particle Sorting	Importing enums from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
~	Disable Per-Rank Output	Importing mathphys from Ci\Program Files/Tech-XXSim-1.0.0dev(Contents)engine\share\macros.	
	Custom Run Options	Importing devices elect from Ci\Program File> Tech-XXSIm-1.0.0dev(Concents)engine)smate(mattes).	
Visualize	Reset to Setup Values	Importing listUtilities from C:\Program Files\Tech-X\X3im-1.0.0dev/Contents\engine\shar\macros.	
		Importing inducts from C:/program Files/lecn-X:Asim-1.0.dow/Contents/engine/smare/macros.	
		Importing utilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
Help		Importing Vaputlitles from C:\Programs Files\Tech-XXSIm-1.0.0dev/Contents\engine\Shate\macros. Importing V3imEn from C:\Programs Files\Tech-XXSim-1.0.0dev/Contents\engine\Shate\macros.	
		Importing embcs from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing plasmaDielectric from C:\Program File>\Tech-XiXSim=-1.0.0dev\Content>\engine\share\macros. Importing delayeval from C:\Program File>\Tech-XiXSim=-1.0.0dev\Contents\engine\share\macros.	
		Importing matrices from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing delayeval from C:\Program Files\Tech-X\XSim-1.0.0dev Contents\engine\share\macros.	
		Importing electrostatics from C:\Program Files\Tech-X:XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing implicitelectromagnetics from C:\Program Files\Tech=XXSIm=1.0.0dev\Contents\engine\share\macros.	
		Importing grids for Ciprogram Files/IEChTA(z)III-10-10429(Contents)engine(snate)macros.	
		Importing timing from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing shapes from C:)rFogram FileS:[ech-A:kbim-1.0.0aev[contents]engine]smareimacros.	
		Importing multifields from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing stfuncs from C:)Program Files)Tech-X)XSim-1.0.0dev/Contents/engine/share/macros. Importing emfileers from C:)Program Files)Tech-XXSim-1.0.0dev/Contents/engine/share/macros.	
		Lines from 'pillboxCavity.pre' processed.	
		Finished with 'pillboxCavity.pre'.	
		Engine completed successfully.	
		Total simulation wall-clock time was 20.004 seconds.	
			•
		-	
🕑 Run:	SUCCESS Simulation engine finished successfully	Show	Log

Fig. 2.70: The Run window at the end of a successful execution.

## Step 4: Computing the eigenmodes

- Go to the analyzer window by selecting Analyze in the left column.
- Select *extractModes.py* from the list of available analyzers. Then click "Open" on the top 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"
  - numberUniformPoints: "20"
  - numberRandomPoints: "100"
  - construct: "checked"
  - overwrite existing files: "checked"

🔀 XSim - F	Pillbox Cavity	- 0 >
File Edit	Tools View Help Window	
	Analysis & Controls	Analysis & Results
X	Search: extractModes	extractModes.py 🖸
Welcome	Available Analyzers: Open	Apply To: Primary Run
~	extractModes.py	Run Automatically After Engine. Order: 1
<b>*</b>	extractModesViaOperator.py	simulationName pillborCavity Outputs
setup		field E
E.		
Prepare		beghDump 2 Python variables defined by command line arguments for this analyzer: simulationName: : pillboxCevity [string]
¢.,		Use Variable: v Itield :: E [V9Mdd5.Field] beginbump :: 2 [int]
Run		endbump [21   endbump :: 21 (int] nModes :: 5 (int]
Ξ		Use Variable:
Design		rModes 5 construct, :: Traise (bool) construct, :: Traise (bool)
0		Use Variable:
Analyze		numberUniformPts 20
		Use Variable: freq [Hz] invo SVD Amplitude
Vicuolito		numberRandomPts 100 (0) 1.00018=02 1.617e-04 7 2.913e-02 (0) 1.00018=02 1.617e-04 (0) 1.00018=02 1.617e-04 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=02 (0) 1.00018=00000000000000000000000000000000
visualize		Use Variable: v (13) 1.423/9er03 1.050e-02 4 7.114e-02
		randomSample [7] [3] 3.62326e409 1.169e-02 3 1.100ze-01 (4) 4.31740e+09 8.867e-01 3 6.451e-02
Help		construct 🔽
		overwrmth 🔽
		[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_EigenE_0.vsh5
		[FilterDiagonalizationMethod.py] Retaining positive mode by convention for mode: 1
		[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_EigenE_1.vsh5
		[FilterDiagonalizationMethod.py] Retaining positive mode by convention for mode: 2
		[FilterDiagonalizationWethod.py] Wrote file pillboxCavity_EigenE_2.vsh5
		[FilterDiagonalizationMethod.py] Retaining positive mode by convention for mode: 3
		[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_EigenE_3.vsh5
		The following variables can be used in the above analyzer options: [FilterDiagonalizationMethod.py] Choosing positive mode by convention for mode: 4
		[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_EigenE_4.vsh5
		Industry completed successfully
	Delete Import Custom Analyzer	
📿 Analio	ze: ANALYZER SUCCESS Analyzer finished successfully	y. StowL

Double-check your entries against what is shown in Fig. 2.71.

Fig. 2.71: 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 with the following parameter.

- field: "B"

After the analysis is finished, and scrolling down in the *Outputs* log pane you should see what is shown in Fig. 2.72.

• 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.

#### Step 5: Computing the transit time factor

- Select *computeTransitTimeFactor.py* from the available analyzers and press "Open" on the top 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"

XSim - I	Pillbox Cavity	- 0	~
File Edit	Tools View Help Window		
	Analysis & Controls	Analysis & Results	
X	Search: extractModes	extractModes.py 🔝 extractModes.py 🔼	
	Available Availables	Apply To: Primary Run V	
Welcome	Available Analyzers:Open	Run Automatically After Engine. Order: 1 Analyze Stop Clear Outp	ut
- 🗘	extractModes.py extractModesViaOperator.py	simulationName officer/Zwity Outputs	
Setup		Run Directory: D:\Users\qar\Documents\Tech-X\XSim-1.0\simulations/pillboxCavity_EH	-
IA.		field B	
Prepare		beghDump 2 Python variables defined by command line arguments for this analyzer:	
\$		Use Variable:	
Run		erdDump [21 red]	
=		Uee Variable:	
Design		Nbodes 5 reaction of the second secon	
0		Use Variable:	
Analyze		numberUniformPts 20	
~		Use Variable:	
a dan sa diana		numberRandomPts 100 (0) 1.00011e+09 1.915e-04 8 1.947e-02	
Visualize		Use Variable: (1) 1.44796e-09 3.151e-02 3918 2.900e-02 (2) 2.45659e-09 9.000e-03 2 3.962e-02	
		randomSample [ [3] 3.00743e109 4.123e-01 3 4.123e-02 (4) 3.60075e109 2.123e-02 3 5.159e-02	
Help		construct 🗟	
		overwrhe 🖓	
		Refrech Variable List	
		[FilterDiagonalizationMethod.py] Retaining positive mode by convention for mode: 1	
		[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_TigenB_1.vsh5	
		[FilterDiagonalizationMethod.py] Retaining positive mode by convention for mode: 2	
		[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_EigenB_2.vsh5	
		[FilterDiagonalizationMethod.py] Retaining positive mode by convention for mode: 3	
		[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_EigenB_3.vsh5	
		The following variables can be used in the above analyzer options: [[FilterDisgonalizationHethod.py] Choosing positive mode by convention for mode: 4	
		[FilterDiagonalizationMethod.py] Wrote file pillboxCavity_EigenB_4.vsh5	
		Analysis completed successfully	
	Delete Import Custom Analyzer		•
		• •	
🛛 🕑 Analy:	ze: ANALYZER SUCCESS Analyzer finished successfully.		Log

Fig. 2.72: The Outputs pane after Analyzing to determine the eigenmodes of the magnetic field.

And compare against what is shown in Fig. 2.73

- 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 analytic value of  $2/\pi$ .

## 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.
- Press the *Analyze* button which is located in the upper right corner.
- If you have selected the closed cavity, the geometry factor at the mode frequency of 999.97 MHz should be very close the analytic value of 257.

🔀 XSim - I	Pillbox Cavity		- <b>o</b> ×
File Edit	Tools View Help Window		
	Analysis & Controls	Analysis & Results	
X	Search: computeTransitTimeFactor	extractModes.py 🖸   extractModes.py 🖾 computeTransitTi	meFactor.py 🗵
Welcome	Available Analyzers: Open	Apply To: Primary Run 💌	Analyze Stop Clear Output
ø	computeTransitTimeFactor.py	Kun Automatically Alter Engine. Order:	
Setup		simulationName pillboxCavity	beta=BETA, -B BETA
115		beginDump 0	Fraction of speed of light. axis=AXIS, -a AXIS
Prepare		Use Variable:	Axis along which to compute time transit factor offsetx0=OFFSETX0, -0 OFFSETX0
ø.,		endDump 0	Distance from center to offset axis along the x0 direction.
Run		Use Variable:	offsetx1=OFFSETX1, -1 OFFSETX1 Distance from center to offset axis along the x1
=		beta 1	direction.
Design		Use Variable:	whether a dataset or group should be overwritten if it already exists.
0		axis 0	Accelerating voltage, time-independent accelerating voltage and transit time
Analyze		Use Variable:	printed to the screen.
		offsetx0 0	Product for and lear-
Visualize		Use Variable:	C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\bin\computeTransitTimeFactor.py simulationName="nillhorCavity"beginDumn="0"endlumn="0"beta="0"offsety0="0"
		offsetx1  0	offsetx1="0"overwrite Bun Start Time: Thu Mar 21 14:19:59 2024
Holp		Use Variable:	Run Directory: D:\Users\qar\Documents\Tech-X\XSim-1.0\simulations/pillboxCavity_EH
		overwrae iv	
		Kerresh Variable Lists	Python variables defined by command line arguments for this analyzer: simulationName :: pillboxCavity [string]
			beginDump :: 0 [int]
			endbump :: 0 [int] beta :: 1 [float]
			axis :: 0 [int]
			offsetxU :: 0 [float]
			overwrite :: True (bool)
			Mode file = pillboxCavity_EigenE_0.vsh5
			Node frequency, f = 1.00018e+03 NHz Time Independent &ccelerating Voltage, V0 = 2.79305e-01 V
		The following variables can be used in the above analyzer options:	Accelerating Voltage, Vacc = 1.77807e-01 V Transit time factor, T=Vacc/V0 = 6.36604e-01
			Computing using axis passing through x, y, z = 7.495e-02, 0.000e+00, 0.000e+00
			Analysis completed successfully
	Delete Import Custom Analyzer		J
🕑 Analy:	ze: ANALYZER SUCCESS Analyzer finished successfully.		Show Log

Fig. 2.73: Computing the transit time factor for the eigenmode of interest.

Tree Die Werk Heige Werker       Machanie Kennende         Werkern       Machanie Kennende       Machanie Kennende         Werkern       Machanie Kennende       Machanie Kennende       Machanie Kennende	🔀 XSim - F	Pillbox Cavity		- o ×
Name         Adaptive & controls           Wetch         Sectors         George Advectors         Adaptive & decision (p - )         Adaptive & decision (p - )           Wetch         George Advectors         Adaptive & decision (p - )         Adaptive & decision (p - )         Adaptive & decision (p - )           Wetch         George Advectors         Margine & decision (p - )         Adaptive & decision (p - )         Adaptive & decision (p - )           Wetch         George Advectors         Margine & decision (p - )         Adaptive & decision (p - )         Adaptive & decision (p - )           Wetch         George Advectors	File Edit	Tools View Help Window		
Vector		Analysis & Controls	Analysis & Results	
National Analyzie:       Out         Personal:       Description         Personal:       Personal:         Personal:	X	Search: computeCavityG	extractModes.py 🖸   extractModes.py 🖾   computeTransitTi	meFactor.py 🛛 computeCavityG.py 🔽
Comparison       Instruction (and in the above analyzer open)         Perganne       Control         Perganane       Control	Welcome	Available Analyzers: Open	Apply To: Primary Run 💌	Analyze Stop Clear Output
Betap       Impact Custom Analyse         Betap       Impact Custom Analyse         Decision       Impact Custom Analyse         Decision       Impact Custom Analyse         Decision       Impact Custom Analyse	<b>1</b>	computeCavityG.py	Run Automatically After Engine. Order:	
important       geometry [system]         important       we wratele         important       we wratele <th>Setup</th> <th></th> <th>simulationName pilboxCavity</th> <th>beginDump :: 0 [int]</th>	Setup		simulationName pilboxCavity	beginDump :: 0 [int]
Prepara         Rom         University         Rom	:=		geometry cylCavity	endDump :: 2 [int]
Wind Window       Window       Image: State of the second	Prenare		beginDump 0	
Rn         Cong	ø.		Use Variable:	 Node file = pillboxCavity_EigenB_0.vsh5
Windows       Refresh Variable: Import Castom Analyzer         Mindows       Refresh Variable: Import Castom	Pup		endDump 2	Reading XSim geometry file.
Refront Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab         Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab         Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab         Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab         Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab         Wounder       Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab         Wounder       Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab         Wounder       Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab         Wounder       Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab         Wounder       Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab         Wounder       Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab         Wounder       Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriable Lab         Wounder       Perfond Voriable Lab       Perfond Voriable Lab       Perfond Voriab         Wounde	-		Use Variable:	Reorganizing grid mask field to compNinorC.
Address         Q         Address         Value	Dorign		Refresh Variable Lists	Node frequency, f = 1000.1MHz
Cutow       Import Custom Analyzer	Design			B^2 integrated over cavity surface = 0.14674
Nakyon         Visuality         Visuality         Nakyon         Visuality         Visuality         Nakyon         Visuality         Nakyon         Visuality         Nakyon         Visuality         Nakyon         Visuality         Nakyon         Visuality         Nakyon         Nakyon <th></th> <th></th> <th></th> <th>B^2 integrated over domain = 0.0050494</th>				B^2 integrated over domain = 0.0050494
Column       Import Custom Analyzer	Analyze			granted, pillbovenity Baurf O vabs
Window       Import Costom Analyzer         Colution       Import Costom Analyzer				created. piriboxcarrey_sourr_o.vans
Volume Period Heav Heav Heav Heav Lower Lordon Analyzer Lower Lordon L				
Pering XSin geometry file.         Procreating or is may file that the down analyzer option.         Mode frequency, f = 1498Min         P2 integrated over county surface = 0.095394         P2 integrated over county surface = 0.095394         P2 integrated over county surface = 0.095394         P2 integrated over county surface = 20.005511         County Tartace = 20.00         The following variables can be used in the above analyzer option:         The following variables can be used in the above analyzer option:         County       Import Custom Analyzer	Visualize			Node file = pillboxCavity_EigenB_1.vsh5
Perior       Import Custom Analyzer         Colutor       Import Custom Analyzer				Reading XSim geometry file. Reorganizing grid mask field to compNinorC.
Coluto     Import Custom Analyzer	Help			Node frequency, f = 1448MHz
Columb       Import Custom Analyzer				B^2 integrated over cavity surface = 0.095294
Celete     Import Custom Analyzer				B^2 integrated over domain = 0.002351
Celete     Import Custom Analyzer				Geometry Factor = 282.06
Oblinit         Import Custom Analyzer				center. presonary_soury_soury_
Celote     Import Custom Analyzer				
Celote     Import Custom Analyzer				Mode file = pillboxCavity_EigenB_2.vsh5 Reading XSim geometry file.
Colute         Import Custom Analyzer				Reorganizing grid mask field to compMinorC.
Debrin     Import Custom Analyzer				Mode frequency, f = 2456.6MHz
Color         Import Custom Analyzer				B^2 integrated over cavity surface = 0.035132
Outrus         Import Custom Analyzer				B^2 integrated over domain = 0.00074877
Cebro Import Custom Analyzer			The tollowing variables can be used in the above analyzer options:	Geometry Factor = 413.4
Color         Import Custom Analyzar				created: pillboxCavity_Bsurf_2.vsh5
Doloto Import Custom Analyzer				
Delate         Import Custom Analyzer         I         I         I		1		Analysis completed successfully
		Delete Import Custom Analyzer	<u></u>	)
C Analyze: ANALYZER SUCCESS Analyzer fnished successfully Show Log	Analyz	ze: ANALYZER SUCCESS Analyzer finished successfully		Show Loc



## 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:

- Expand Scalar Data
- Expand Bsurf
- Check *Bsurf\_magnitude*
- Click the *Plane Controls* button at the bottom of the *Visualization Controls* pane on the left of the Composer window.
- 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. 2.75



Fig. 2.75: The magnitude of the magnetic field on the wall of the cavity

To see a more quantitative visualization of the eigenmode fields, as shown in Fig. 2.76, do the following:

- Add a Field Analysis Data View
- Select the *E\_x* (*EigenE*) as a field
- 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.



Fig. 2.76: 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).

## 2.2.4 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.

Three waveguides are demonstrated in this example .

## **Opening the Simulation**

The Rectangular Waveguide example is accessed from within XSimComposer by the following actions:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the XSim for Electromagnetics 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. 2.77.



Fig. 2.77: 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 has been 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 has been placed on the simulation boundary. It is important that this plate extends 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 has been 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**

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 that you are using these run parameters:
  - Time Step: 1.829541541469147e-12
  - Number of Steps: 400
  - Dump Periodicity: 20
  - Dump at Time Zero: Checked

• Click on the Run button in the upper left 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."

🔀 XSim -	Rectangular Waveguide		×
File Edit	Tools View Help Window		
	Runtime Options	Logs and Output Files	
X	Parameters Run Mode	Run Dump and Stop Force Stop	Log
	Values entered here will override values from the simulation file.	Engine Log   File Browser	
weicome	Default Value (1.829541541469147a-12)	All multiFields dumped at 2024-03-22-09:01:00.672.	-
<b>\$</b>	Value: 1.829541541469147e-12	No celetromagnetic fields to dump.	
Setup	Use Variable:	No ionization processes to dump.	
:=	Number of Steps	pomain: pump 20 completed at 2024-03-22-09:01:00.673.	
13	Value: 400 Default Value (400)	Rain loop ended at clock time 2024-03-22-09:01:00.673	
Prepare	Use Variable:	Pereving comman	
8.	Dump Periodicity	Deleting domain.	
.8	Value: 20 Default Value (20)	There were 0 Notices encountered in this run.	
Run	Use Variable:	There were 1 Warnings encountered in this run.	
	-Restart at Dumo Number	See above for more information. WORDEL complete at clock two _2024_03_22_00:01:01 762	
	Value:	NOTE: This simulation can be run if all of the packages ' GPU XSimBase XSimEH' are enabled. Other package combinations may be possible as well.	
Design	Line Variable	Please contact support@txcorp.com to investigate package combinations to meet your needs.	
		No expand symbols file. Will not limit expansion.	
U C	Dump at Time Zero	Importing verbosity from C: Program Files/Tech-XXSIm-1.0.0dev/Contents/engine/share/macros.	
Analyze	No Particle Sorting	Importing enums from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
	Disable Per-Rank Output	Importing mathphys from C:\Program Files\Tech-X\X3im-1.0.0dev\Contents\engine\share\macros.	
	Custom Run Options	Importing vsim ifom C:) Frogram FileS/isch-X:x51m-1.0.0dev/contents/engine/snare/macros.	
Vicualita		Importing listUtilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
visualize	Reset to Setup Values	Importing logicals from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing units from C:\Program Files\Tech-X\XSim-1.0.0dev/Contents\engine\share\macros	
		Importing voltilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\shrebmacros.	
Help		Importing VSimEm from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing embcs from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing plasmauleiecric from Citrogram Files lech-Alain-J. Odevicontentstengineisnaretmacros.	
		Importing matrices from C:\Program Files\Tech-X\ZSim-1.0.0dev\Contents\engine\share\macros.	
		Importing delayeval from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing emaatrix from C:\Program Files\Tech-XXX3im-1.0.0dev/Contents\engine\share\macros.	
		Importing implicited crossations from C: Program Files/ied=Xisim=1.0.0dev/Contents/tengine/smatte/macros.	
		Importing tfsfboxxd from C:\Program Files\Tech-XXXSim-1.0.0dev\Contents\engine\share\macros.	
		Importing grids from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing timing from C:)Program Files)Tech-XXSim-1.0.0dev(Contents)engine)share)macros.	
		Importing shares from C:\Program Files\Tech-XXSim-1.0.0dev(Contents)engine\Share\macros.	
		Importing multifields from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing stfuncs from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Lipporting emfliters from C:\Program Files\Tech-X\XSIm-1.0.Udev\Contents\engine\Share\macros.	
		Finished with 'rectangularWaveguide.pre'.	
		END ENGINE OUTPUT	
		Engine completed successfully.	
		To see results, click on the "Visualize" icon in the icon panel.	1
		Total simulation wall-clock time was 49.56 seconds.	÷
		<u></u>	<u> </u>
Ruma	SUCCESS Simulation anning finished successfully		100
WRUN:	ouccess amulatori engine mished successfully	Show	LUG

Fig. 2.78: The Run Window at the end of execution.

#### **Visualizing the Results**

After a successful 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, check the *Clip Plot* box. Now step through time using the Dump slider on the bottom of the right pane. This is shown below.

## **Further Experiments**

Waveguides can be added or subtracted to this simulation.

## 2.2.5 Rectangular Metal Waveguide Dispersion (rectMetalWaveguideDisp.sdf)

Keywords:

Waveguide, Dispersion Relation, Fourier Transform, Phase Shifting Boundary Conditions



Fig. 2.79: The  $E_y$  field propagating out of the two waveguides centered on the z axis. The effects of the third waveguide can be viewed by adjusting the "Origin of Normal Vector" parameter under the Plane Controls button.

## **Problem description**

This XSimEM example demonstrates several unique capabilities of XSim that can be used to efficiently model waveguides. One unique capability is the use of phase shifting periodic boundary conditions. These work by adding a phase to a wave at a boundary to mimic a much longer physical dimension. We also demonstrate how to use an analyzer called extractModesViaOperator.py to accurately compute the excited modes in the waveguide, even if some of the modes are much weaker than the dominant mode. In this example, we use extractModesViaOperator.py to compute the dispersion relation ( $\omega$  vs k) and compare the numerically determined dispersion relation with the theoretical dispersion relation using standard waveguide theory.

This example involves five steps. In the first step, the waveguide is "pinged" with a short pulse in the current density that excites a range of modes. The Fourier transform then shows the range of frequencies of the modes. In the second step, the waveguide is excited using a sinc pulse function multiplied by a Gaussian envelope to excite a flat band of frequencies with sharp cutoffs at either end. The excitation current density is in the transverse directions (z and y) which excites an electric field primarily in the transverse direction. In the third step, the data is restarted from the end of the second run and saved at shorter time intervals in order to resolve the frequencies of interest. The output from the third step is used by the extractModesViaOperator.py analyzer to compute the eigenmodes in step 4. Finally, in step 5, the dispersion relation is computed by varying the wavelength which is resolved in the simulation.

## **Opening the Simulation**

The rectMetalWaveguideDisp example is accessed from within XSimComposer by the following actions:

- Go to  $File \rightarrow New \rightarrow From Example...$
- In the resulting Examples window expand the XSim for Electromagnetics option.
- Expand the Cavities and Waveguides option.
- Select "Rectangular Metal Waveguide 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.

The properties and values that create the simulation are accessible in the left pane when the Setup Window is selected as shown in Fig. 2.80. The right pane shows a 3D view of the selected geometry components, grids and current distributions.

The geometry can be visualized by expanding "Geometries" in the left pane. The hollow rectangular waveguide can be seen more clearly by de-selecting the "Grid" option. The length in the direction of propagation (x) is a small fraction of either transverse direction. This is possible because we are using "phase shifting periodic" boundary conditions in the *x*-direction. The phase shifting BCs are selected under "Basic Settings".

The simulation is excited with a sinc hat function, which has a formula of

$$(t, f_l, f_h, \delta_f, t_{off}) = H(t_{off} - t) \exp(-0.5\delta_f^2(t - 0.5t_{off})^2) \times \frac{\sin(2\pi f_h(t - 0.5t_{off})) - \sin(2\pi f_l(t - 0.5t_{off}))}{(2\pi f_h - 2\pi f_l)(t - 0.5t_{off})}$$

 $\delta_f$  is calculated according to. frequencyGap = (frequency high - frequency low)\*frequency gap factor

numSigma = sqrt(-2.0\*log(suppression factor))

sigmaT = (TWOPI\*frequencyGap)/numSigma

As mentioned in the introduction, this function has a Fourier spectrum that is fairly flat over the desired range,  $f_l < f_h$ , of frequencies and falls off rapidly over a frequency width of  $\delta_f$ , so that it is nearly zero for  $f < f_l - \delta_f$  or  $f > f_h + \delta_f$ .

## Phase Shifting Boundary Conditions and Phase Shift

Before discussing phase-shifting periodic boundary conditions, we first review ordinary periodic BCs. In this discussion, periodic BC's are applied in the x-direction. With normal periodic BC's, there are two criteria in resolving a wave (or mode) of interest. (1) There must be enough grid points along the wavelength to resolve the spatial profile. Typically we sample at least 20 cells along a wavelength ( $\lambda_x$ ), or  $DX = \lambda_x/20$ , and (2) The length of the simulation domain, represented by  $L_x$ , must be one wavelength long,  $L_x = \lambda_x$ . Periodic BC's in the x-direction means that  $F(0) = F(L_x)$ , where F is any field quantity. Now introduce a grid that extends from 0,1,...,nx and let x = 0 at grid point 0 and  $x = L_x$  at grid point nx. Periodicity on the grid implies F(0) = F(nx). Finally, let's assume that  $F(x) \sim \sin(k_x x)$ . Then, applying the condition for periodicity,  $\sin(0) = \sin(k_x L_x)$ , which is exactly met if  $L_x = \lambda_x$ .

With phase-shifting periodic BC's, we are no longer required to meet the second criteria discussed in the preceding paragraph. To see this, assume  $L_x < \lambda_x$ . Then the periodicity condition can still be met by setting  $k_x L_x - \phi_0 = 0$ , where  $\phi_0$  is the phase shift equal to  $k_x L_x$ , which is the exact phase shift we have chosen to apply in XSim for this example. The numerical implementation is more challenging than the conceptual picture we just discussed. To implement phase-shifting periodic BC's, we need to treat the fields as complex numbers and set  $F(L_x) = \exp(i\phi_0)F(0)$ . For the grid, we pick 2 cells such that  $L_x = 2DX$ .

By using phase-shifting periodic BC's, we can simulate different physical lengths without changing the simulation length  $L_x$  through the phase shift  $\phi_0 = k_x L_x$ . Because  $k_x = 2\pi/L_x$ , we can solve for  $\omega(k_x)$  without requiring a simulation of length  $L_x$ .



Fig. 2.80: Initial Setup Window for the Waveguide Dispersion example. The block in the middle is the region in which the current density is driven.

## **Running the Simulation and Analyzing Results**

We now walk through the five steps discussed in the Introduction. In the first step, we test to determine the minimum frequency that will propagate through the waveguide. For the rectangular waveguide in this example, we know the analytical result which is the lowest cutoff frequency. However, we still do this step to demonstrate how to determine the lowest frequency that will propagate for cases that are not analytic. In the second step, the current density "rings up" to a maximum value, then rings down to 0. Because we are exciting modes at resonant frequencies of the cavity, the *E*-and *B*-fields continue to oscillate after the excitation is turned off. We wish to determine the resonant modes using Eigen mode analysis using data saved in Step 3 when the externally driven source is turned off and the cavity is "ringing" at its natural frequencies. In step 4, extractModesViaOperator.py is used to compute the excitation frequencies in the cavity at a given mode. Finally, in step 5, you will change the value of "mode", which is defined under "Constants" to compute the dominant frequencies which are excited in the cavity at a given wavelength. We assume the maximum wavelength which can be resolved in a traditional periodic simulation is 0.2 m and denote this as *lambdaMax*. We then define the parameter  $k_x = \frac{mode}{12} \frac{2\pi}{lambdaMax}$  and run 25 simulations with *mode* =0,1,2,...,24. Finally, we set the phase shift to  $k_x \times L_x$  to ensure the correct phase shift is applied for the phase shifting periodic BC's. In this way, we are able to compare the simulation with waveguide theory without explicitly changing the length of the simulation domain in the *x*-direction.

## Step 1: Determining the lowest propagation frequency - The "Ping" Run

The Ping run is used to determine the lowest propagation frequency in the waveguide. In this simulation, we can analytically compute the lowest propagation frequency at a given k based on the allowable modes in a square wave guide. We can then compare the computed lowest propagation frequency with theory. However, for arbitrarily-shaped waveguides, this step is necessary since no theory is available to compute the lowest propagation frequency. We have defined a PINGON constant (with values of 0 or 1) in order to easily transition from the "Ping" run to the "Excitation" run in Step 2. The default is PINGON=1 to do the "Ping" run first. For Step 1, perform the following steps

- Click on the Run Window.
- This simulation may be accelerated by changing the Run Mode to Parallel.
- Click the Run button on the upper left corner of the Logs and Output Files pane

The run has completed when you see the output, "Engine completed successfully." A snapshot of the simulation run completion is shown in Fig. 2.81.

🔀 XSim -	Rectangular Metal Waveguide Dispersion	- 0 X
File Edit	Tools View Help Window	
	Runtime Options	Logs and Output Files
X	Parameters Run Mode	Run Dump and Stop Force Stop Clear Log
	values entered here will override values from the simulation hie.	Engine Log File Browser
weicome	Time Step Onfault Volum (2.590(720)(267007046-(2)	All multiFields dumped at 2024-03-21-12:47:09.206.
<b>1</b>	Value: 2 5901720126790794e-12	No electromagnetic fields to dump.
	I ke Variable	No collisions to dump. No ionization processes to dump.
Setup		Domain: Dump 5 completed at 2024-03-21-12:47:09.207.
15	Number of Steps	
:20	Value:  5000 Default Value (5000)	Tain loop ended at clock time 2024-03-21-12:4/109.207
Prepare	Use Variable:	
102	Dump Periodicity	Deleting domain.
<b>1</b>	Value: 1000 Default Value()	OUTPUT SUMMARY:
Run	Use Variable:	There were 1 Warnings encountered in this run.
-		See above for more information.
	Restart at bump number	VORPAL completed at clock time 2024-03-21-12:47:09.232.
Design	Value:	Please contact support@xxcorp.com to investigate package combination to meet your meeds.
	Use Variable:	No expand symbols file. Will not limit expansion.
	🔽 Dump at Time Zero	Importing rectMetalWaveguideDisp from D:\Users)qar\Documents\Tech-XXSim-1.0\simulations\rectMetalWaveguideDisp_EM. Importing uptholity from C:\Usergram Files\Tech-ViSim-1.0.Qety(Contents)tech-WaveguideDisp_EM.
Analyze	No Particle Sorting	Importing enums for City forma Files Theorie and a devolution of the state of the s
-	Disable Per-Rank Output	Importing mathphys from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.
	Custom Run Ontions	Importing Usim from C:\Program Files\Tech=X\XSim=1.0.Odev\Contents\engine\share\macros.
1 farrallan		Importing listUtilities from C: Program Files Tech-X/XSim-1.0.0dev/Contents/engine/share/macros.
visualize	Reset to Setup Values	Importing logicals from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.
		Importing units from C:\Program Files\Tech-XXSim-1.0.0dev\Contents\engine\share\macros.
•••		Importing volities from civrogram Files/Tech-XISIm-10.0dev/Contents/engine(share)mactos.
Help		Importing VSimEm from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.
		Importing makes from C:\Program Files\Tech-XXSim-1.0.0dev\Contents\engine\share\marcos.
		Importing plasmaticity of the state of the s
		Importing matrices from C:\Program Files\Tech-X\X5im-1.0.0dev\Contents\engine\share\macros.
		Importing delayeval from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.
		Importing electrostatics from C:\Program Files/Tech-XXSIm-1.0.0dev(Content)stangingingingingingingingingingingingingin
		Importing implicitelectromagnetics from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.
		Importing ffsfboxxd from Ci\Program Files/Tech-XXSIm-1.0.0dev/Contents/engine/share/macros.
		Importing times from C: Program Files/lech-AlkSim-1.0.000v/Contents/engine/share/macros.
		Importing histories from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.
		Importing shapes from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.
		Importing multifields from C:\Program Files\tech-XiXim-1.0.Udev\Contents\engine\smartextmaclos.
		Importing emfilters from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.
		Lines from 'rectRetalWaveguideDisp.pre' processed.
		Pinismed with 'recretaiwaveguideyisp.pre'.
		Ingine completed successfully.
		Total simulation wall-clock time was 18.964 seconds.
		u <sup>*</sup>
📿 Run:	SUCCESS Simulation engine finished successfully	Show Loa

Fig. 2.81: The Run window at the end of a Step 1.

After the run has completed, click on the *Visualize* tab on the left side of the visualization window. After the data has loaded, click on *Add a Data View* and then select *History*. Perform the following steps to analyze the history:

- Graph 1 select *jMid\_1*
- Graph 2 select *eMid\_1*

In Graph 1, to see the data, click on the "Zoom" option above the plot. Left click with your mouse on the upper left corner of the graph and drag the "zoom" box to about  $0.2 \times 10^{-9}$  s (hold the left mouse button when you drag). You should see a step function in the current density beginning at t=0 and ending abruptly after about 20 time steps. This is the "ping" that we impose on the simulation. To determine the lowest frequency mode that we excite, click the "Fourier Amplitudes (dB)" box in Graph 2. Again, zoom in on the left most portion of the plot. You will see that the first peak occurs at a frequency of about  $1.5 \times 10^9$  Hz, which is the lowest cutoff frequency in the simulation and is what we find analytically using waveguide theory. Therefore, we have confirmed that XSim reproduces linear theory. You can

confidently use XSim to numerically determine the lowest cutoff frequency for non-analytically solvable shapes. The visualization window after the above steps have been performed is shown in Fig. 2.82



Fig. 2.82: History plots showing the lowest cutoff frequency at  $1.5 \times 10^9$  Hz as the result of pinging the waveguide with an abruptly applied current density.

## **Step 2: Excitation**

From Step 1, we know that the lowest propagation frequency is  $\sim 1.5 \times 10^9$  Hz. We can now check that the lowest cutoff frequency (*frequency low* in *SincHat*) is correct. The parameter *FREQ\_MIN* is passed in to *frequency low* and is indeed  $\sim 1.5 \times 10^9$  Hz. We can now proceed to excite the waveguide with confidence knowing that lowest excitation frequency is correct. To excite the waveguide, perform the following steps:

- Go to the Setup Window and expand the Constants option
- Set PINGON to 0
- Click on Save and Setup in the upper right corner of the visualization window
- Go to the Run Window by pressing the Run button in the left column of buttons.
- Set Number of Steps to 21000. This ensures that the simulation runs long enough for the current density to "ring up" then "ring down" to 0.
- Leave the "Dump Periodity" at 1000.
- This simulation may be accelerated by running on multiple MPI ranks. The parallel options are in the *Run Mode* 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." A snapshot of the simulation run completion is shown in Fig. 2.83.



Fig. 2.83: The Run window at the end of a Step 2.

## Step 3: Evolving the excited cavity

The purpose of Step 3 is to continue the simulation with only the *E*- and *B*- fields excited due to the externally imposed current density from Step 2. We will also save more data which we can analyze using Eigenmode analysis for Step 4.

- Go to the Setup Window and expand the *Basic Settings* option.
- Change "number of steps" to "10000".
- Change "steps between dumps" to "100".
- Change "dump in groups of" to "3".
- Click on Save and Setup in the upper right corner of the visualization window.
- Go to the Run Window and click the "Reset to Setup Values" button in the lower right corner of the "Run Options" section.
- In the Run Options section, uncheck the Dump at Time Zero box and set the Restart at Dump Number to 21.
- NOTE: the "Dump Periodicity" must be blank for the "dump in groups of 3" setting to work. In this case, the simulation will revert to the values defined in "Basic Settings" which are to dump every 100 time steps in groups of 3 (21000,21001,21002, ..., 21100,21101,21102,..., etc). If you fill in an integer into Dump Periodicity, the default gets overwritten and the data are not dumped in groups of 3, which is required for the extractModesVia-Operator.py analyzer. So, there should be 300 new dumps making a total of 321 dumps.
- 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. 2.84. When this run is finished, the last step should be step 31000.

XSim -	Rectangular Metal Waveguide Dispersion	-	• •
File Edit	loois View Help Window		_
	Runtime Options	Logs and Output Files	
	Parameters Run Mode	Run Dump and Stop Force Stop	Clear Log
	Values entered here will override values from the simulation file.	Engine Log File Browser	
Treaconne	Time Step	All multiFields dumped at 2024-03-28-15:21:24.692.	-
<b>1</b>	Default Value (2.5901720126790794e-12)	No electromagnetic fields to dump.	
	Value: 2.5901720126790794e-12	No ionizione to dump.	
Setup	Use Variable:	Domain: Dump 321 completed at 2024-03-28-15:21:24.692.	
13	Number of Steps	Main loop ended at clock time 2024-03-28-15:21:24.692	
Dronaro	Value: 10000 Default Value (10000)	Deleting domain	
Trepare	Use Variable:	Talating downin	
23.	Durin Bardi di b	OUTPUT SUMMARY:	
- 6	Dump Periodicity	There were 0 Notices encountered in this run.	
Run	Value: Default Value()	There were 1 Warnings encountered in this run.	
-	Use Variable:	See above for more information.	
	Restart at Dump Number	NOTE: This simulation can be run if all of the packages ' GPU XSInKM' are enabled. Other package combinations may be possible as well. Please	
Design	Value: 21	contact support@txcorp.com to investigate package combinations to meet your needs.	
-	Lise Variable	No expand symbols file. Will not limit expansion.	
$\odot$		Importing rectMetalWaveguideDisp from D: Users\qar\DocumentsTech-X\XSim-1.0\simulationsYectMetalWaveguideDisp_EM.	
Analumo	F Dump at Time Zero	Importing enums from C:\readman Files\Tech=Xissim=1.0.0dev\Contents\engine\smalletmatcs.	
Analyze	No Particle Sorting	Importing mathphys from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
	Disable Per-Rank Output	Importing VSim from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
	Ourteen Burn Orations	Importing deviceselect from C:)Program Files/Tech-XXXSim-1.0.0dev/Contents/engine/share/macros.	
Visualize	custom an options	Importing locals from C: Program Files/Tech-XX31m-1.0.0dev/contents/engine/share/martos.	
	Reset to Setup Values	Importing units from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing utilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
Help		Importing vputilities from C:)Program Files/Tech-X)XSim-1.0.0dev/Contents/engine/share/macros.	
		Importing which from C:/Program FilesTech-XXSim-1.0.0dev/Contents/engine/share/macros.	
		Importing plasmaDielectric from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing delayeval from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing matrices from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing emetry from C. (Program Files) Tech-XISIn-1.0. Odev/Contents/engine(share)macros.	
		Importing electrostatics from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing implicitelectromagnetics from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing this down of the control of the theory of theory of the theory	
		Importing time from C:Program Files/Tech-X/Sim-1.0.0dev/Contents/engine/smatematros.	
		Importing histories from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing shapes from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing multifields from C:) Program Files) Tech-XX X3im-1.0.00ev/Contents/engine) share/macros. Tumorting stimuc; from C:) Program Files) Tech-XX X3im-1.0.00ev/Contents/engine) bare/macros.	
		Importing emilters from C:\Program Files\tech-X\Zimin-10.0dev(contents)engine\tenzinetcimeters.	
		Lines from 'rectMetalWaveguideDisp.pre' processed.	
		Finished with 'rectMetalWaveguideDisp.pre'.	
		END ENGINE OUTPUT	
		Engine completed successfully.	
		To see results, click on the "Visualize" icon in the icon panel.	
		Total simulation wall-clock time was 59.286 seconds.	_
			-
🕑 Run:	SUCCESS Simulation engine finished successfully		Show Log

Fig. 2.84: The Run window at the end of Step 3.

#### Step 4: Computing the eigenmodes

- Go to the analyzer window by selecting Analyze in the left column.
- Select *extractModesViaOperator.py* from the list of available analyzers. Then click "Open" on the top right of the *Analysis Controls* pane.
- Compute the electric field eigenfunctions. After the analyzer loads, ensure the following parameters are entered:
  - simulationName: "rectMetalWaveguideDisp"
  - outputsimName: leave blank
  - realFields: "E"
  - imagFields: leave blank
  - secondFieldFactor: leave blank
  - operator: "d2dt2"
  - dumpRange: "21:321"
  - cellSamples: ":,5:25:5,5:45:5"
  - cutoff: "1e-12"
  - maxNumModes: "-1"
  - initialModeNumber: "0"
  - normalizeModes: checked
  - testing: leave blank
  - overwrite: checked

Also, check the "Overwrite Existing Files" box. The dump range runs only over the data saved in step 3 and we are only sampling every 5 cells in the *z*- and *y*- directions which is adequate to compute the eigen modes. Double-check your entries against what is shown in Fig. 2.85. After you run the analyzer, you will need to scroll up to find the computed frequencies. The screenshot shown in Fig. 2.85 is from a visualization window that is scrolled up so that the computed frequencies can be compared with what you ran. Figure Fig. 2.85 shows that there are 6 modes, but two are degenerate resulting in 5 unique modes. One indication that you are using extractModesViaOperator.py correctly is that the imaginary part of the frequency (which represents attenuation of the wave) is nearly 0 or at least much smaller than the real part. The lowest excited mode is  $\sim 1.5 \times 10^9$  Hz, which is discussed further below in the context of standard waveguide theory.



Fig. 2.85: Computing the electric field eigenfunctions and frequencies using the extractModesViaOperator.py analyzer.

## **Step 5: Computing the Dispersion Relation**

Once the Eigenmode analysis is complete in Step 4, it is necessary to record the three lowest-order modes. The preloaded value of *mode* is 0, which means we are simulating  $k_x = 0$ . The waveguide dispersion relation is given by  $\omega^2 = k_x^2 c^2 + \omega_{mn}^2$ , where  $\omega_{mn}^2 = c^2 \pi^2 ((m/a)^2 + (n/b)^2)$ , and where *a* and *b* are the *y* and *z* dimensions, respectively and *m* and *n* are integers. In this simulation, we analyzed the (m, n) = (1,0), (0,1) and (1,1) modes. Furthermore, the frequency range that is excited lies between  $f_l = 1.5 \times 10^9$  Hz (which is the lowest allowable propagation frequency) and  $f_h = 4.5 \times 10^9$  Hz. Given these parameters, we expect from waveguide theory for the three lowest cutoffs to be  $\sim 1.5 \times 10^9, 3 \times 10^9, 3.35 \times 10^9$  Hz, which is what is found from extractModesViaOperator.py. Repeating these steps for modes 1 to 24 yields the dispersion relation shown in Fig. 2.86. The overlap between the simulation results and theoretical values is evident in Fig. 2.86.



Fig. 2.86: Dispersion relation found by changing *mode* in XSim. The solid line is the theoretical dispersion relation  $\omega^2 = k_x^2 c^2 + \omega_{mn}^2$ . The 'X's represent data from the simulation results.

#### **Convergence Study**

To demonstrate that the simulation results converge to the theoretical value, we have performed a series of simulations in which DX = DY = DZ are varied. The values chosen are DX = 0.205, 0.41, 0.50 and 0.615 cm. We then computed the lowest propagation frequency using *extractModesViaOperator.py* and plotted this frequency versus  $DX^2$ . Using Richardson extrapolation, we then compute the lowest propagation frequency for  $DX^2 = 0$ , which provides a better comparison with the theoretical frequency than a simulation with finite  $DX^2$ . The field calculation error scales approximately with  $DX^2$  so a plot of frequency versus  $DX^2$  should be a line. The linear correlation between frequency and  $DX^2$  is shown in Fig. 2.87. The extrapolated frequency at  $DX^2 = 0$  is  $\sim 1.4992 \times 10^9$  Hz. The theoretical value is  $\sim 1.4990 \times 10^9$ . We see from Fig. 2.87 that the computation of the lowest propagation frequency for the DC mode converges with order  $DX^2$ , which is expected since the field solve is 2nd order accurate. Furthermore, using Richardson extrapolation, we see that the difference between theory and simulation, with  $DX^2 = 0$ , is 0.02 %.

#### 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 first step will be to ensure that we are driving the current density as expected and that the E- and B- fields are "ringing" as a result of driving the current density. Follow these steps to perform this check:

- Click on the Add a Data View pull-down menu at the top of the visualization window
- Click on *History*. This will open a new tab.
- Click on Reload Data.
- Under Graph 1 plot *jMid\_1*. This is the *y* component of the current density
- Under Graph 2 plot *eMid\_1*.



Fig. 2.87: Plot demonstrating convergence of frequency calculation.

• Under Graph 2 click the reset button to put the axis scaling back from the first plots make in Step 1 above.

Your plots should be similar to Fig. 2.88. The current density is driven for a short time period. However, because we are driving resonant modes, the current density excites the transverse electric field and parallel magnetic field. You can also plot *eMid\_0*, *bMid\_1*, and *bMid\_2* to compare with Fig. 2.88. We have driven a TE mode since *eMid\_0* is nearly 0.

We next wish to examine the spectral characteristics of the current density, which is driven between *FREQ\_MIN* and *FREQ\_MAX*. Therefore, the Fourier Transform of  $J_y$  or  $J_z$  should be greatest in this frequency range. Returning to the History visualization window, under Graphs 2, 3, and 4, change the plotted quantity to *none*, so that only Graph 1 shows a plot. Now check the "Fourier Amplitudes (dB)" box in the upper left corner. Finally, check the "Zoom" box on the right side of the visualization window and highlight from 0 to  $10^{10}$  which will expand the lower frequency part of the plot. After you have zoomed in on the plot, re-check the "Navigate" box. The result of these steps should lead to a plot that looks similar to Fig. 2.89. The driving frequencies lie between *FREQ\_MIN* and *FREQ\_MAX* as expected. The rate at which the Fourier signal dampens below *FREQ\_MIN* and above *FREQ\_MAX* depends on the Gaussian envelope and the parameter called *OMEGA\_SIGMA*.

## **Further Experiments**

The result of varying *mode* from 0 to 24 is shown in Fig. 2.86. One experiment you can perform is to reproduce Fig. 2.86 by running 25 simulations with *mode* varying from 0 to 24. Each simulation takes just a few minutes so the 25 simulations take about one hour. For each simulation record the three lowest frequencies that are excited using extractModesViaOperator.py.

Another experiment would be to change the dimensions of the waveguide. The Constants *a* and *b* are used to determine the lowest frequency that will propagate in the waveguide. Therefore, changing *a* and *b* will automatically compute *FREQ\_CUTOFF\_TE*. You will then need to change *BGNY*, *ENDY*, *BGNZ*, and *ENDZ* accordingly. Finally, you will need to modify the dimensions and position of the primitives constructed under *Geometries* which are used to construct the waveguide.

A third experiment would be to modify *FREQ\_MIN* and *FREQ\_MAX* and compute the dispersion curve for these new values. No mode will propagate below *FREQ\_CUTOFF\_TE* so do not set *FREQ\_MIN* below *FREQ\_CUTOFF\_TE*.



Fig. 2.88: History plots showing the modes driven in this simulation



Fig. 2.89: Fourier Transform of the current density showing that the current density is mainly driven between *FREQ\_MIN* and *FREQ\_MAX* as expected.

Lastly, you can try to excite a TM mode instead of a TE mode.

## 2.2.6 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. Xsim 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. 2.90.

## **Opening the Simulation**

The Scattering Matrix example is accessed from within XsimComposer by the following actions:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the XSim for Electromagnetics 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 the waveguide in the 3D View. Fig. 2.91.

## **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 antenna is constructed of two planar current sources with each current source one cell thick in the x-direction and directly adjacent to each other along x. The amplitudes and phases of the current sources are tuned so that the electric and magnetic fields which launch in the -x direction cancel whereas in the +x direction, the fields add to a non-trivial value. 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. 2.90: Some relevant parameters for the S-Matrix Box Cavity example.



Fig. 2.91: Setup Window for the Scattering Matrix example.

This example is parameterized in the waveguide and DUT geometry specification, allowing for easy modification to either. 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 "Fourier Amplitudes" capabilities in XsimComposer, or if single frequency, then the S11 and S21 values are just the amplitudes of the signals.

The x axis is the axial direction of the waveguide, with the parameters WAVEGUIDE\_LENGTH, APER-TURE\_LENGTH, and CAVITY\_LENGTH controlling the lengths of each component. These parameters are also used to control the position of each component, allowing a change to one of them to properly adjust the component positions.

The height (Y axis) of each component is standardized to the parameter WR90\_HEIGHT, and centered around the Y axis.

The waveguide and aperture widths are centered around the Z axis, while the cavity is not.

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 spatial profile of the current source is consistent with the TE01 mode. Since we are launching a TE01 mode, the perpendicular component of the electric field is 0 at the boundaries. Because the larger waveguide dimension is along z the lowest order mode coincides with the y-component of the electric field being excited. Therefore the function "waveguideEyProfile" is the spatial function used in the current source in order to launch the desired waveguide 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.
- Check that you are using these run parameters:
  - *Time Step*: 1.2250750302237241e-12
  - Number of Steps: 17899
  - Dump Periodicity: 182
  - Dump at Time Zero: Checked
- 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 result is shown in Fig. 2.92.

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. 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 constant, FREQCENTER, specifies either the single frequency or the center frequency of the band. The constant, FREQBANDWIDTH, provides the bandwidth or is set to 0 if a single-frequency simulation is desired.

With a single frequency simulation, the constant, 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.



Fig. 2.92: The Run Window at the end of execution.

With a finite frequency band, the same constant, NUMBEROFCYCLESTODRIVE, can be adjusted upward to increase the detail and sharpness of the S-Matrix variation with frequency. In particular, increasing NUMBEROFCYCLESTO-DRIVE causes the FFT of the S11 and S21 histories to attain a sharper cutoff at FREQHI and FREQLO. The constant, 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 XsimComposer. 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 *myCalibrationWaveguide* 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 (MAL's) are not working well enough, or that the waveguide's "mode-Profile" description is not accurate enough, and/or that there is not enough grid resolution. To decrease reflections at the MAL boundaries, you can decrease the "damping factor" in the MAL boundary condition. If you do this, you may need to increase the length of the MAL. Another potential issue is the value of TRISE, which is the length of time over which the current source is turned in the single frequency runs. Increasing TRISE could lead to a better measure of S11 and S21.

To adjust the DRIVENORMALIZATION constant, 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 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 shown by adding a *History* data view. Once the history graphs are displayed check *Fourier Amplitudes* to get an FFT of the data. To look at the results from 8 to 12 GHz, click *Limits* in the upper right corner of each graph and set the X-axis lower limit to 6e9, and upper limit to 14e9.



Fig. 2.93: Fourier transforms of the histories S11\_Voltage and S21\_Voltage as a function of frequency (in GHz).

## **Further Experiments**

Experiment with finding optimal placement of the pole in the transmitted signal.

# 2.3 Photonics

## 2.3.1 Dielectric Waveguide Mode Calculation (dielectricWaveguideModeCalc.sdf)

Keywords:

Mode Extraction, Photonic Waveguide, Guided Mode, Semiconductor

## **Problem Description**

This example demonstrates the process for extracting the effective index and fields of a guided mode by directly solving an eigenvalue equation. The use of permittivity averaging enables second order accuracy in our solution. The waveguide axis runs parallel to the x-axis, and is surrounded by a background cladding with a greater permittivity. We will run the simulation for 1 step and then use the dielectricWaveguideModeCalc\_invEps\_0.h5 file to solve for the guided modes using the computeDielectricModes.py analyzer. This analyzer will find the fundamental mode of this waveguide and output this mode into a separate .vsh5 file. This mode file is then used to launch the extracted mode into the simulation using the Wave Launcher.

Eigenmodes in such a simulation have the form:

$$\mathbf{E}(\mathbf{x},t) = \mathbf{E}(y,z)e^{i(kx-\omega t)}$$

The 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 XSimEM license.

## **Opening the Simulation**

To open this example open an instance of XSimComposer and follow the steps below:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting Examples window expand the XSim 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 Variables**

This example contains a number of variables defined to make the simulation easily modifiable.

- EPS\_SILICON and EPS\_SILICA: Relative permittivities of silicon and silica. These constants are used in multiple parameters and in the accompanying Python file for solving the waveguide mode.
- WAVELENGTH\_VAC: Wavelength of the input signal. This wavelength is also used for the calculation of the fundamental guided mode of the device.

The *Materials* section contains just silicon and silica. The *Geometries* section 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. This simulation makes use of XSim's Wave Launcher to launch a unidirectional wave down the waveguide with a mode defined by the file generated by the computeWaveguideModes.py analyzer.

## Setting up the Pre-run Simulation

As delivered, the system is set up to generate the data needed to run the computeDielectricModes.py analyzer. To ensure that your simulation has second order accuracy, under *Basic Settings*, verify that the *dielectric solver* field is set to *permittivity averaging*. This algorithm is a powerful XSim feature. This setting is shown in Fig. 2.94. It is also necessary to make sure the wave launcher is deactivated before running for the first simulation since the mode file has not been generated until after running the computeWaveguideModes.py analyzer. To do it, expand *Field Dynamics*, expand *CurrentDistributions* and verify that *waveLauncher1* is *{inactive}*.



Fig. 2.94: Choosing the second order accurate, *permittivity averaging* for the *dielectric solver* field under *Basic Settings* and wave launcher *{inactive}*.

#### **Running the Pre-run 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. If you changed something in the setup, you will be asked to Save. Click *Save* upon the request to save.
- In the left pane :
  - Set Number of Steps to 1
  - Set Dump Periodicity to 1.
  - Check Dump at Time Zero.
- 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 result is shown in Fig. 2.95.

🔀 XSim -	Dielectric Waveguide Mode Calculation	-	o ×
File Edit	Tools View Help Window		
	Runtime Options	Logs and Output Files	
X	Parameters Run Mode	Run Dump and Stop Force Stop	Clear Log
Wolcomo	Values entered here will override values from the simulation hie.	Engine Log   File Browser	
weicome	Default Value (5.170243475571357e-17)	Deleting domain	<u> </u>
<b>.</b>	Value: 5.170243475571357e-17	Deleting domain.	
Setup	Use Variable:	OUTPUT SUMMARY: There were O Notices encountered in this run.	
:==	Number of Steps	There were 1 Warnings encountered in this run.	
:3	Value: 1 Default Value (12500)	See above for more information.	
Prepare	Use Variable:	NOTE: This simulation can be run if all of the packages ' GPU XSimBase' are enabled. Other package combinations may be	
502	Dump Periodicity	possible as well. Please contact support@txcorp.com to investigate package combinations to meet your needs.	
	Value: 1 Default Value (1250)	No expand symmols rile. Will not inmit expansion. Inmorting dielectric@aveguideBodeCalc from D:Users\gar\Documents\Tech-	
Run	Use Variable:	X\XSim-1.0\simulations\dielectricWaveguideModeCalc_EN.	
-	Restart at Dump Number	Importing verbosity from C:\Program Files\Tech-X:XSim-1.0.Odev\Contents\engine\Share\macros. Importing enume from C:\Program Files\Tech-X:XSim-1.0.Odev\Contents\engine\Share\macros.	
	Value:	Importing mathphys from C:\Program Files\Tech-X\X3Im-1.0.0dev\Contents\enginejnetshare\macros.	
Design	Use Variable:	Importing VSim from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing deviceselect from C:\Program Files\tech-X\xsim-1.0.0dev\contents\engine\shate\macros. Importing distUtilities from C:\Program Files\tech-X\xsim-1.0.0dev\contents\engine\shate\macros.	
ex,	Dump at Time Zero	Importing logicals from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
Analyze	No Particle Sorting	Importing units from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
~	Disable Per-Rank Output	Importing vputities from C:\Program Files\Tech-XiXIII.0.00ev(concents\tengine(smateshare)matros.	
	Custom Run Options	Importing VSimEm from C:\Program Files\Tech-X\XSim-1.0.Odev\Contents\engine\share\macros.	
Visualize	Reset to Setup Values	Importing embods from C:\Program Files\Tech-XXXSim-1.0.0dev\Concents\engine\share\macros. Importing blasmabielectric from C:\Program Files\Tech-XXXSim-1.0.0dev\Concents\engine\share\macros.	
		Importing delayeval from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing matrices from C:\Program Files\Tech-X\XSim-1.0.0dev(Contents\engine)share\macros. Importing delayes[from C:\Program Files\Tech-X\XSim-1.0.0dev(Contents\engine)share\macros.	
Help		Importing estatist from C:\Program Files\Tech-X\X3im-1.0.0dev\Contents\engine\shate\macros.	
		Importing electrostatics from C: Program Files Tech-X/XSim-10.0dev/Contents engine share macros.	
		Importing implicitelectromagnetics from C:\Program Files\Tech-X\X3im-1.0.0dev(contents)engine(share\macros. Invorting tisfboxed from C:\Program Files\Tech-X\X3im-1.0.0dev(contents)engine(share\macros.	
		Importing grids from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing thing from C:\Program Files\Tech-X\XSim-1.0.0dev(Contents)engine\share\macros.	
		Importing shapes from C:\Program FilesTech-X;XSim=1.0.0dev(Contents/engine)share(matches.	
		Importing multifields from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing stiuncs from C:\Program Files\Tech-X\XSim-1.0.0dev/Contents\engine\share\macros. Importing emiliers from C:\Program Files\Tech-X\XSim-1.0.0dev/Contents\engine\share\macros.	
		Lines from 'dielectricWaveguideNodeCalc.pre' processed.	1
		Finished with 'dielectricWaveguideModeCalc.pre'.	
		ERD ENGINE OUTPUT	
		Engine completed successfully.	
		Total simulation well-clock time we de 423 encode	
		I TOTAL SHIMARCAME WART-CAUCH CARE WAS TITTE SECOND.	•
	-	u	
📿 Run:	SUCCESS Simulation engine finished successfully		Show Log

Fig. 2.95: Running the simulation for one step to get the permittivity data for the analyzer.

#### Solving for the Eigenmodes

After performing the above actions, continue as follows:

- Proceed to the Analyze Window by clicking the Analyze button on the left.
- Select computeWaveguideModes.py and click Open under the list.
- Update the analyzer fields accordingly by choosing the corresponding variables under Use Variable :
  - simulationName: dielectricWaveguideModeCalc
  - datasetName: invEps

- *transverseSliceX*: PORT\_X\_INPUT
- transverseSliceLY: PORT\_YBGN
- transverseSliceUY: PORT\_YEND
- transverseSliceLZ: PORT\_ZBGN
- transverseSliceUZ: PORT\_ZEND
- vacWavelength: 1.55e-6
- nModes: 1
- writeFieldProfile: D
- modeFileName: port1
- Normalize: checked
- compMajorC: unchecked
- Overwrite: checked
- Click *Analyze* in the top right corner.

The analyzer will only find guided modes. The results should resemble Fig. 2.96. We see that the analyzer found the fundamental mode and gives the effective index of the mode under *Neff*. This parameter is necessary for using the *Wave Launcher* to launch a unidirectional wave. This value is saved as a parameter as *NEFF\_FROM\_ANALYZER*.

XSim - Dielectric Waveguide Mode Calculation	
File Edit Tools View Help Window	
Analysis & Controls	Analysis & Results
Search: computeWaveguideModes	computeSParamsFromHists.py 😨 computeWaveguideModes.py 🖸
Welcome Analyzers. Open	Run Automatically After Engine. Order: 1      Analyze Stop Clear Output
Weicowe     Available Analyzers:     Open       Setup     Compute Waveguide Modes say compute Waveguide Modes say       Prepare     Ren       Design     Ren       Ustandze     Ren       Weiser     Ren	Apply To: Primary Run
	Analysis completed successfully
Delate Juneary Contrast Analysis	
Analyze: ANALYZER SUCCESS Analyzer finished successfully	Show Lon

Fig. 2.96: The analyzer window after a successful run of computeWaveguideModes.py.

## Setting up the Main Simulation

After performing the above actions, continue as follows:

- Proceed to the Setup Window by pressing the Setup button in the left column of buttons.
- Activate the wave launcher
  - Expand Field Dynamics
  - Expand CurrentDistributions
  - Right Click on waveLauncher1
  - Left click on Activate
- Click on Save and setup in the top right corner.

This setting is shown in Fig. 2.97.

Tools View Help Window							
Editor							
						🙆 Circulation C	atun is Rearly Cave ar
						Sinuador 3	etup is Reauly Save al
dielectricWaveguideModeCalc.sdf dielec	tricWaveguideModeCalc.pre dielectricWavegui	deModeCalc.in					
Simulation		- 1 ×	the partice tool	Clip Chevy Capita Uties: Calida	-	Deest Lieuu	Descarting
Description		§	livo active con	Cip Silow Scale View Solids		Reset View	Properces
Constants		2					
Parameters		8					
Basic Settings		_					
- Functions		68	м				
SpaceTimeFunctions		- P	p p				
Materials		gat					
Geometries		2					
Grids		ie ie					
Field Dynamics		ate	<b>4 *</b>				
+ hields		ž					
<ul> <li>FieldboundaryConditions</li> </ul>							
- currences unputions							
Histories							
C TRANKS							
				3e-06			
				0.20000 00.05	1	2 n OF	
				302002.30-03		00-00	
	the former for the	11 I					
	Undo Add Multiple Remove A	dd					
Property	Value	-					
kind	Wave Launcher						
description							
color							
effective index of refraction	NEFF_FROM_ANALYZER						
directionality	unidirectional						
launching direction (if unidirectional)	positive						
surface	yz plane						
- xCoordinate	PORT_X_INPUT						
yMin	PORT_YBGN						
yMax.	PORT_YEND						
zMin	PURIZEGN						
zMax	PORI_ZEND						
🖭 spatial profile	The defined						
The name	porti_EigenU_uvsh5	_					
El temporal prome	1 0	_					
ampinume	ERECUENCY LO	- 1					
- frequency low	a concentration of the second s	The second se					
Frequency low							

Fig. 2.97: Choosing the second order accurate, permittivity averaging for the dielectric solver field under Basic Settings.

## **Running the Main 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. You will be asked to Save. Click *Save* upon the request to save.
- In the left pane, click on *Reset to Setup Values* to run the full simulation which launches this wave down the waveguide.
- Check Dump at Time Zero.
- 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 result is shown in Fig. 2.98.

XSim -	Dielectric Waveguide Mode Calculation	-	• •
File Edit	Tools View Help Window		
	Runtime Options	Logs and Output Files	
X	Parameters Run Mode	Run Dump and Stop Force Stop	Clear Log
Welcome	Values entered here will override values from the simulation file.	Engine Log File Browser	
	Default Value (5 170243475571357e-17)	Deleting domain	<u> </u>
<b>\$</b>	Value: 5.170243475571357e-17	Deleting domain.	
Setup	Use Variable:	OUTPUT SUMMARY:	
·	Number of Steps	There were U Notices encountered in this run.	
1B	Value: 12500 Default Value (12500)	See above for more information.	
Dropara	Liea Variabla	VORPAL completed at clock time 2024-03-20-12:07:21.203.	
prepare		NOTE: This simulation can be run if all of the packages ' GPU XSimBase' are enabled. Other package combinations may b	e
- 23 a	Dump Periodicity	No exand symbols file. Will not limit example.	
- 8	Value: 1250 Default Value (1250)	Importing dielectricWaveguideHodeCalc from D:\Users\qar\Documents\Tech-	
Run	Use Variable:	X\XSim-1.0\simulations\dielectricWaveguideModeCalc_EN.	
-	Restart at Dump Number	Importing verbosity from C:\Program Files\Tech-X\XSim-1.0.ddev\Contents\engine\share\macros.	
	Value	Importing mathematics from c.\program Files Tech-XiSim-1.0.0dev(Contents)engine(share)macros.	
Design	Line Vaviables	Importing VSim from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
	Use variable:	Importing deviceselect from C:\Program Files\Tech-XXSim-1.0.0dev\Contents\engine\share\macros.	
	🔽 Dump at Time Zero	Importing instutinities from C:\Program Files\Tech=X\Sim=1.0.0dev\Contents\engine\share\macros.	
Analyze	No Particle Sorting	Importing units from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
7 undry LC	Disable Per-Rank Output	Importing utilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
	Custom Run Options	Importing vputilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros. Importing VSimEm from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
visualize	Reset to Setup Values	Importing embcs from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros. Importing plasmaDielectric from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing delayeval from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing delayeral from C:\Program Files\tech-XXSIm-1.0.0dev(Contents\tengine\share\matrix)are=	
Help		Importing esmetrix from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing electrostatics from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing implicitelectromagnetics from C:\Program Files\Tech-X\X3im-1.0.0dev(Contents\engine\Share\macros. Importing is feithered from C:\Program Files\Tech-X\X3im-1.0.0dev(Contents\engine\Share\macros.	
		Importing grids from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing timing from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing histories from C:\Program Files\Tech-XXXSim-1.0.0dev\Contents\engine\share\macros.	
		Importing multifields from C:)Program filesilech-AikSim-L.U.UdeviContentsiengineishareimecros	
		Importing structs from C:\Program Files\Tech-X\XSim-1.0.0dev(Contents)engine\share\macros.	
		Importing emfilters from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Lines from 'dielectricWaveguideModeCalc.pre' processed.	
		Engine completed successfully.	
		Total simulation wall-clock time was 1218.94 accords.	
		· · · · · · · · · · · · · · · · · ·	<u> </u>
🕑 Run:	SUCCESS Simulation engine finished successfully		Show Log

Fig. 2.98: Running the full simulation which launches this wave down the waveguide.

## **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.

You may need to *Reload Data* (bottom left). Visualize an eigenmode by following these steps:

- From the Add a Data View dropdown, select Data Overview.
- Expand Scalar Data, expand EigenD, and select EigenD\_magnitude.

The resulting visualization pane should look like Fig. 2.99.

## **Analyzing the Results**

When designing waveguides, one might be interested in calculating the S parameters and spectral power of the wave traveling down the waveguide. XSim features an analyzer that will compute both of these quantities.

- After visualizing the data and making sure the simulation is behaving as expected, proceed to the *Analyze* Window.
- Click on the computeSParamsFromHists.py analyzer to open it and fill in the necessary parameters as follows:
  - *simulationName* dielectricWaveguideModeCalc
  - firstStep: 0
  - lastStep: 12500

🔀 XSim - I	Dielectric Waveguide Mode Calculation		- 0	
File Edit	Tools View Help Window			
	Add a Data View 💌 Choose a Run: PrimaryRun	•	Unload Data Relo	ad Data
X	Data Overview 🗵 🔋 1-D Fields 🖾 🛛			
Welcome	Visualization Controls	Visualization Results		
Welcome Setup IIII Prepare Q Run Design Q Analyze Visualize	Vessalization Controls	Vessbardion Results           Save         Labels         Limits         Rendering         Coord Swap         Reset View         Preserve Aspect Ratio         Auto Reset           Var: EigerD, mognitud         0         0         0         0         0         0           Var: EigerD, mognitud         0         0         0         0         0         0           Max: 1006         0.500         0         0         0         0         0           Max: 1006         -400         -600         -600         -600         -600         0.5           -1.0         -0.5         0.0         0.5         x (10^n-6 units)         0.5	1.0	
	Annotation Level: Axes & Legends			10
🕑 Visual	ize: READY Visualization available	-	ę	show Log

Fig. 2.99: The visualization pane showing the magnitude of the D field of the fundamental mode.

- stepOffset: 0
- maxWavelength: WAVELENGTH\_MAX
- minWavelength: WAVELENGTH\_MIN
- inDirection: 0
- inSlabE: slab0\_E
- inSlabB: slab0\_B
- inSign: 1
- outDirection: 0
- outSlabE: slab1\_E
- outSlabB: slab1\_B
- outSign: 1
- *outputFileName* : blank
- outputSuffix: blank
- overwrite: checked
- Click *Analyze* in the top right corner.

The results should resemble Fig. 2.100

Upon successful completion of the analyzer, proceed to the Visualize tab.

- From here, click on the drop down menu called Add a Data View and select 1-D Fields.
- To visualize the S-parameters and the input power:

City City			_ 0 ^					
File Edit	loois view Help Window			-				
	Analysis & Controls	Analysis & Results						
X	Search: computeSParamsFromHists	computeSParamsFromHists.py 🖾   computeWaveguide	Modes.py 🖸 computeSParamsFromHists.py 🔽					
Welcome	Available Analyzers: Open	Apply To: Primary Run  Analyze Stop						
1	computeSParamsFromHists.py	I Run Automatically After Engine. Order: 1						
Setup		simulationName dielectricWaveguideModeCalc	Cutputs	r				
:=		firstStep 0	maxwavelength :: 1.35000000000000-06 [float] minWavelength :: 1.25000000000000e-06 [float]	L				
Prenare		Use Variable:	inDirection :: 0 [int] inSlabE :: slabO_E [string]	L				
d'		lastStep 12500	inSign :: 1 [int]	L				
~¢		Use Variable:	outDirection :: 0 [int] outSlabE :: slab1_E [string]	L				
Run		stepOffset 0	outSlabB :: slab1_B [string] outSign :: 1 [int]	L				
-		Use Variable:	outputFileName :: None [string] outputSuffix :: None [string]	L				
Design		maxWavelength 1.85000000000000e-06	compMajorC :: False [bool] overwrite :: True [bool]	L				
Q		Use Variable:	self.outputFileName = dielectricWaveguideModeCalc_SParameters.vsh5	L				
Analyze		minWavelength 1.25000000000000-06	3 dimensions.	L				
6		Use Variable:	step range = (0, 12500).	L				
Visualize		inDirection 0	Calculating power for histories slab0_E and slab0_E.	L				
2		Use Variable:	[7.01184834e-08 2.50000000e-08 2.50000000e-08] Calculating S-parameters from histories slab1_E and slab1_E.	L				
Неір		inSlabE slab0_E	[7.01184834e-08 2.50000000e-08 2.50000000e-08]	L				
			Writing file, dielectricWaveguideNodeCalc_SParameters.vsh5. dielectricWaveguideNodeCalc_SParameters.vsh5	d.				
		insiade  siadu_e	File dielectricWaveguideModeCalc SParameters.vsh5 successfully written.	I.				
		inSign 1	Channel :	I.				
		· · · · · · · · · · · · · · · · · · ·	[1.46780303e-06 1.45676692e-06 1.44589552e-06] [0.99998671 0.99998726 0.9999796 ]	I.				
		The following variables can be used in the above analyzer	Maximum transmission is 1 or -0.00 db	I.				
		options:	at wavelength = 1.4616e-06 m	I.				
			Done					
			Analysis committed successfully					
	Delete Import Custom Analyzer		· · · · · · · · · · · · · · · · · · ·					
				Ē				
🕑 Anah	Analyze: ANALYZER SUCCESS Analyzer finished successfully Show Log							

Fig. 2.100: The analyzer window after a successful run of computeSParamsFromHists.py.

- Click Add Curve
- Select S\_slab1\_EAndslab1\_B
- Click Apply, followed by Ok.
- Click on the *Add Window* button and repeat the same process as before but this time select *slab0\_EAndslab0\_BPower* from the drop down menu.

The resulting plots will have the S-parameters per wavelength on top and the spectal input power on the bottom, as in Fig. 2.101

## **Further Experiments**

Change the geometry on the Setup window and rerun the simulation and analyzer to see the effects on the modes.

Once you have your desired mode, launch it down the waveguide using the procedure laid out above.

One can run a full convergence study of eigenmode effective indices by changing the resolution in the Setup window and re-running the simulation and mode extraction script. A plot of the effective index as a function of transverse cell area is shown in Fig. 2.102. The linear relationship shows the second order accuracy of our dielectric algorithms.


Fig. 2.101: The visualization window after adding in the plots of the S-parameters and the spectral input power.



Fig. 2.102: The effective index as a function of transverse cell area for an eigenmode.

# 2.3.2 Microring Resonator with Gaussian Launcher (ringResonatorGaussian-Launch.sdf)

Keywords:

Ring Resonator, Mode Launcher, MAL, Guided Mode, Photonic Device, Semiconductor

#### **Problem Description**

The Ring Resonator consists 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.

The approximate fundamental guided mode profile is launched as a wide band pulse in the input waveguide. This mode is a simple 2D Gaussian distribution centered at the center of the waveguide. This approximate mode is accurate enough for this simulation. The exact mode can be calculated using the computeWaveguideModes analyzer. To go through the mode solve process check out the Multimode Fiber Mode Calculation example. We will use the computeSParamsViaOverlapIntegral.py analyzer to determine the transmission coefficients at the thru-port and drop port.

This simulation can be performed with a XSimEM license.

# **Opening the Simulation**

The Ring Resonator example is accessed from within XSimComposer by the following actions:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the XSim for Electromagnetics option.
- Expand the *Photonics* option.
- Select *Ring Resonator* 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 available in the Setup Window as shown in Fig. 2.103. 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 Constants:

- WAVEL\_MIN = the smallest wavelength in the frequency band
- WAVEL\_MAX = the largest wavelength in the frequency band
- RADIUS\_RING = radius of the ring
- WIDTH\_WAVEGUIDE = Width of waveguides
- HEIGHT\_WAVEGUIDE = Height of waveguides
- WIDTH\_GAP = Width of the gap between ring and waveguides



Fig. 2.103: The Setup window for the ring resonator example showing the external launching field.

This simulation applies a wide frequency band signal to the approximate fundamental spatial mode profile. The signal is defined under *SpaceTimeFunctions* and then assigned under *Field Dynamics*, *CurrentDistributions*, *separableDistributionCurrent* as shown in Fig. 2.103.

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.

Under *Basic Settings* you can see that the *dielectric solver* is set to *permittivity averaging*. This feature enables second order accuracy for simulations using dielectrics.

# **Running the Simulation**

When you have saved the setup, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- Check that you are using these run parameters:
  - Time Step: 7.343490630124558e-17
  - Number of Steps: 60000
  - Dump Periodicity: 6000
  - Dump at Time Zero: Checked
- 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". As seen in Fig. 2.104.



Fig. 2.104: Run window at completion.

### **Analyzing the Results**

Using post analysis scripts, one can extract the transmission coefficients. This simulation uses 4 "EM Field on Plane" histories (slab0-slab4) and this data is used to calculate S parameters between these 4 locations using the analysis script computeSParamsViaOverlapIntegral.py.

Follow these steps:

- Proceed to the Analyze Window by clicking the Analyze button on the left.
- Select computeSParamsViaOverlapIntegral.py and click Open under the list.

Now update the analyzer fields accordingly.

- *maxWavelength*: 1.7e-06 (or use the corresponding parameter from the setup)
- *minWavelength*: 1.4e-06 (or use the corresponding parameter from the setup)
- inPlane: slab0
- *outPlane*: slab1 (or slab3)
- *outputFileSuffix*: S01 (or S03)

The remaining parameter default values will work. Hit the *Analyze* button in the top right corner. After a successful run the window should resemble Fig. 2.105. Continue by analyzing the drop-port. To do this change the *outSlabE* and *outSlabB* fields to slab3\_E and slab3\_B, respectively.

🔀 XSim - I	Xim - Microning Resonator with Gaussian Launcher — 🔿 X					
File Edit	Tools View Help Window					
	Analysis & Controls	Analysis & Results				
X	Search: computeSParamsViaOverlapIntegral	computeSParamsVlaOverlapIntegraLpy 🔽	1			
Welcome	Available Analyzers: Open	Apply To: Primary Run 🔟 Analyze Stop Clear Out	out			
	comnuteSParamsViaQuerlanIntegral mu	Run Automatically After Engine. Order: 1 🛨				
Ç.	compares and instruction approximation approximation and approximat	simulationName ringResonatorGaussianLaunchOutputs				
iii.		minWavelength 14e-6 A .vsh5 file with with the overlap integral plotted as a function of wavelength.	-			
:=0		Use Variable:				
Prepare		Executing analyzer:				
¢.,		maxwavelengm 1./e-6 CVI/Contents\engine\bin\computeSParamsViaOverlapIntegral.py				
Run		Use Variable:				
		inPlane slab0 overwrite				
		Run Start Time: Wed Mar 20 11:05:46 2024 Run Directory: D; Ugers/car/Documents/Tech-X\XSim-1.0\simulations/				
Design		outPlane slab1 ringResonatorGaussianLaunch_EM				
Q		firstStepinPlane				
Analyze		Use Variable: Python variables defined by command line arguments for this analyser:				
~		freStarOutBlace minWeylength: 1.4e-6 [flot]				
		maxWavelength :: 1.7e-6 [float]				
visualize		Use Variable: InFlame :: slab0 [VsHdt5.history] outPlame :: slab1 [VsHdt5.history]				
		numSteps firstStepInPlane :: None [int]				
		Use Variable:				
Help		outputFileSuffix :: Sol [string]				
		outputFileSuffx [S01 compare]				
		compMajorC C				
		overwrite 🔽 Frequency domain is [1.763485e+14,2.141375e+14]				
		Refrech Variable Lists				
		Analyzing output planes for time steps [0,60002]				
		The following variables can be used in the above analyzer Computing overlap, integral				
		Writing results to ringResonatorGaussianLaunch_OverlapIntegral_S01.vsh5				
	,	Analysis completed successfully	÷			
	Delete Import Custom Analyzer	Li Di				
🕜 Analy	ze: ANALYZER SUCCESS Analyzer finished successfully	Sho	w Log			

Fig. 2.105: Analyze window with output from computeSParamsViaOverlapIntegral.py.

#### 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 magnetic field by performing the following:

- Near the top left corner of the window, select Data Overview from the Add a Data View drop-down.
- Expand *Scalar Data*, then *B*, then select *B\_y*
- In the bottom left, select *Clip Plot*, and set z=0 in *Plane Controls*
- Finally, move the dump slider on the bottom of the window to watch the light propagate.

The results are shown in Fig. 2.106.

One can visualize the transmission coefficients by performing the following:

- Near the top left corner of the window, select 1-D Fields from the Add a Data View drop-down.
- In the Plot Control panel select the Slab1 and Slab3 SParams data for Graphs 1 and 2, respectively.
- Set the other Graphs to None

The results are shown in Fig. 2.107. As expected, we see coupling of resonant wavelengths with the ring.



Fig. 2.106: Visualization of magnetic field.



Fig. 2.107: Visualization of the s-parameters.

# 2.3.3 Dielectric Waveguide With Roughness (dielectricWaveguideModeCalcRough.sdf)

Keywords:

Photonic Waveguide, Guided Mode

#### **Problem Description**

This example demonstrates how to add roughness to a waveguide and how to implement a variable grid to resolve roughness that would otherwise be too computationally expensive to resolve. Since roughness features of a waveguide are typically on the order of nanometers, a very high resolution (and therefore small cell size) is required. By using a variable grid, a user is able to increase the resolution where needed and decrease the resolution elsewhere to maintain the computational efficiency of a uniform grid.

#### **Opening the Simulation**

To open this example open an instance of XSimComposer and follow the steps below:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the *XSim for Electromagnetics* option.
- Expand the *Photonics* option.
- Select Dielectric Waveguide With Roughness 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 Variables**

This example contains a number of variables defined to make the simulation easily modifiable.

- EPS\_SILICON and EPS\_SILICA: Relative permittivities of silicon and silica. These constants are used in multiple parameters and in the accompanying Python file for solving the waveguide mode.
- WAVELENGTH\_CENTER: Center wavelength of the input signal. This wavelength is also used for the calculation of the fundamental guided mode of the device.
- DY\_REFINED: The cell size desired within the refinement region.
- USE\_DY\_REFINED: A switch that when set to 1 will replace DY with DY\_REFINED in calculations such as the CFL time step. By default this is set to 0.

The *Materials* section contains just silicon and silica. The *Geometries* section includes the GDS waveguide. 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. This simulation makes use of XSim's Wave Launcher to launch a unidirectional wave down the waveguide with a mode defined by the file generated by the computeWaveguideModes.py analyzer.

## Setting up the Roughness

As delivered, this example comes equipped with a geometry imported from a GDS file. For more on importing a geometry from a GDS file, see the Y Splitter example documentation. At the moment roughness is only supported on geometries made from importing a GDS. To add roughness to the waveguide, proceed as follows:

- Expand the geometry object in the Geometries section by clicking on the + symbol next to waveguideGeom.
- Select the Boundary0 object under waveguideGeom.
- Switch the *roughness type* option from *roughnessDisabled* to *stochasticODE*. Fill in the following parameters under *stochasticODE* as follows:
  - RMS amplitude: 50e-9
  - normal tolerance: 45
  - Ramp-up distance: 0.0
  - Ramp-down distance: 0.0
  - ODE Correlation: 1000e-9
  - ODE step size: 140e-9
  - Random seed: Leave as is
  - export to CSV:

The geometry in the 3D visualization view should look like:

dielectricWaveguideModeCalcRough.sc	df dielectricWaveguideModeCalcRough.pre dielect	ricWaveguide	ModeCalcRough.in				
Simulation		3	No active tool	Clip Show Scale View Solids	▼ +2 ▼	Reset View	Propertie
Description		<li>Second second secon</li>					
Constants		8					
Parameters Basic Settings							
Functions		8	м				
SpaceTimeFunctions		- Q	P				
Materials     Geometries		Dat					
CSG		20					
CAD CAD		- Part	<del>z x</del>				
<ul> <li>waveguideGeom</li> </ul>		ž					
Boundary0		-					
Grids							
<ul> <li>Field Dynamics</li> </ul>		_					
(±) Histories		_					
				3e-06			
				2020082 90.05		30-05	
				0000000000			
	Undo Add Multiple Remove Add	4 I					
( <u></u>							
Property	Value						
layer	1	_					
material	Slicon	_					
<ul> <li>roughness type</li> </ul>	stochasticODE						
- RMS amplitude	5e-08						
normal tolerance	45						
ramp down distance	0.0	_					
ODE correlation	1e-06						
ODE step size	1.4e-07						
export to CSV	1234308.0	_					
and a set of		_					

Fig. 2.108: The waveguide with roughness added to the upper and lower walls in the y direction.

## **Running the Uniform Grid 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. If you changed something in the setup, you will be asked to Save. Click *Save* upon the request to save.
- In the left pane : \* Set Number of Steps to 12500. \* Set Dump Periodicity to 1250. \* Check Dump at Time Zero.
- 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 result is shown in Fig. 2.109.

🔀 XSim - I	Dielectric Waveguide With Roughness	-	o ×
File Edit	Tools View Help Window		
	Runtime Options	Logs and Output Files	
X	Parameters Run Mode	Run Dump and Stop Force Stop	Clear Log
	Values entered here will override values from the simulation file.	Engine Log File Browser	
Welcome	Time Step	All multiFields dumped at 2024-03-22-13:16:41.411.	-
<b>1</b>	Value (5.170243475571357e-17)	No electromagnetic fields to dump.	
~	Like Variable:	No collisions to dump.	
Setup		Domain: Dump 10 completed at 2024-03-22-13:16:41.411.	
5	Number of Steps	Wain loop anded at clock time 2024-02-22-12:16:41 411	
100	Value: 12500 Detault Value (12500)	Deleting domain	
Prepare	Use Variable:		
8.	Dump Periodicity	Deteting domain.	
-0	Value: 1250 Default Value (1250)	There were 0 Notices encountered in this run.	
Run	Use Variable:	There were 1 Warnings encouncered in this run.	
	Restart at Dump Number	VORPAL completed at clock time 2024-03-22-13:16:42.461.	
	Value:	NOTE: This simulation can be run if all of the packages ' GPU XSimBase' are enabled. Other package combinations may be possible as well. Pla	ease
Design	Use Variable:	No expand symbols file. Will not limit expansion.	
	Dump at Time Zero	Importing dielectricWaveguideModeCalcRough from D:\Users\qar\Documents\Tech-X\XSim-1.0\simulations\dielectricWaveguideModeCalcRough_EN.	
A	No Particle Sorting	Importing verbosity from C:)Frogram Files)Tech-XXSim-1.0.0dev/Contents)engine share macros.	
Analyze	C Disable Per-Rank Output	Importing mathphys from C:\Program Files\Tech-X\X3im-1.0.0dev\Contents\engine\share\macros.	
	Custom Run Ontione	Importing VSim from C:\Program Files\Tech-X\XSim-1.0.Odev\Contents\engine\share\macros.	
Vicualita		Importing listUtilities from C:\Program FileS\Tech-X\ZSim-To-1.0.0dev\Contents\engine\share\macros.	
visualize	Reset to Setup Values	Importing logicals from C:\Program Files\Tech-X\XSim=1.0.0dev\Contents\engine\share\macros.	
		Importing units from C:\Program Files\Tech-A\x51m-1.0.00eV\Contents\engine\Share\macros.	
Holo		Importing vputilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
нер		Importing VSimEm from C:\Programs Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing plasmaDielectric from C:\Program Files\Tech-XXXSim-1.0.0dev\Contents\engine\share\macros.	
		Importing delayeval from C:\Program Files\Tech-XXSim-1.0.0dev/Contents\engine\share\mmacros.	
		Importing delayers from Civrogram Files Tech-XXSIm-10.0dev/Contents/engine/smare/macros.	
		Importing esmatrix from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing electrostatics from C:)Program Files)Tech-XXSim-1.0.0dev/Contents)engine)share)macros. Importing implicite/ectromagnetics from C:)Program Files)Tech-XXSim-1.0.0dev/Contents)engine)share)macros.	
		Importing tfsfboxxd from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing grids from C:\Program Files\Tech-XXSim-1.0.0dev\Contents\engine\share\macros.	
		Importing timing item (iteragiam Files) tech-sitsim i.o.odev(contents) engine(shate) macros.	
		Importing shapes from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing multifields from C:\Programs Files\Tech-X\XSim-1.0.0dev\Contents\engine\Share\macros.	
		Importing emfilters from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Lines from 'dielectricWaveguideModeCalcRough.pre' processed.	
		Finished with 'dielectric@wveguideHodeCalcRough.pre'.	
		Engine completed successfully.	
		Total simulation wall-clock time was 1552.03 seconds.	
		(1)	-
		*	
C Dunu	CUCCECC Circulation onging finished automofills		Chowles

Fig. 2.109: Running the full simulation.

#### **Visualizing the Uniform Grid 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.

You may need to *Reload Data* (bottom left). Visualize the electric field and a cross section of the geometry by following these steps:

- From the Add a Data View dropdown, select Data Overview.
- Expand *Scalar Data*, expand *E*, and select *E\_y*.
- Expand Geometries and select waveguideDualTriangles.

The resulting visualization pane should look like Fig. 2.110.

Slide the dump selector at the bottom of the window to the right to watch the wave travel down the waveguide. It is clear that with this level of roughness that the transmission of the wave will not be great.



Fig. 2.110: The visualization pane showing the y component of the E field with the geometry on top.

### Setting Up the Variable Grid

In XSim, a variable grid is implemented into the simulation by adding in a *Refinement Region*. This is a 3D box in which the cell sizes inside can be changed to increase the resolution in one or multiple directions. The refinement region will then take the total number of cells in each direction and will automatically adjust the size of the cells outside of the refinement region to smoothly fit in the rest of the cells into the domain. In this simulation, the resolution in y is increased. In this example, a refinement region has already been created and is inactive. To activate this refinement region, proceed as follows:

- Expand the Grids and Grid sections.
- Right click on refinementRegion0 and click Activate.

This refinement region was created by performing the follwing actions:

- Expand the Grids section and right click on Grid.
- Select Create Refinement Region.

A new option will appear under *Grid* called *refinementRegion0*. To make the box surround the waveguide and increase the resolution in y, fill in the following options in the refinement region as follows:

- xMin: GRID\_X\_BGN
- xMax: GRID\_X\_END
- cellSizeX: DX
- yMin: -3.5e-7
- yMax: 3.5e-7
- cellSizeY: DY
- zMin: -2e-7

- zMax: 2e-7
- cellSizeZ: DZ

Now to change the resolution in the y direction, expand the *Parameters* section and select *USE\_DY\_REFINED*. Set this parameter equal to 1. This parameter acts as a switch that will replace DY with a smaller value specified by DY\_REFINED in all calculations. Since the grid has changed, a new mode file needs to be calculated. Expand the *Field Dynamics* section and under *CurrentDistributions*, deactivate *waveLauncher0* by right clicking on it and select *Deactivate*. Next, expand the *Geometries* tab and expand the *waveguideGeom* geometry. Select *Boundary0* and set *roughness type* to "roughnessDisabled". Save the simulation and continue to the run tab. Now to generate the necessary data to recalculate the fundamental mode of the waveguide, proceed as follows:

- Click on the Reset to Setup Values button.
- Set the *Number of Steps* and *Dump Periodicity* to 1.
- Make sure that *Dump at Time Zero* is selected.
- Click Run.

Upon successful completion of the engine, proceed to the next section.

# Solving for the Eigenmode

After performing the above actions, continue as follows:

- Proceed to the Analyze Window by clicking the Analyze button on the left.
- Select *computeWaveguideModes.py* and click *Open* under the list.
- Update the analyzer fields accordingly by choosing the corresponding variables under Use Variable :
  - simulationName: dielectricWaveguideModeCalcRough
  - datasetName: invEps
  - *transverseSliceX*: PORT\_X\_INPUT
  - transverseSliceLY: PORT\_YBGN
  - *transverseSliceUY*: PORT\_YEND
  - transverseSliceLZ: PORT\_ZBGN
  - transverseSliceUZ: PORT\_ZEND
  - vacWavelength: 1.55e-6
  - nModes: 1
  - writeFieldProfile: D
  - modeFileName: port1
  - Normalize: checked
  - compMajorC: unchecked
  - Overwrite: checked
- Click *Analyze* in the top right corner.

The analyzer will only find guided modes. The results should resemble Fig. 2.111. We see that the analyzer found the fundamental mode and gives the effective index of the mode under *Neff*. This parameter is necessary for using the *Wave Launcher* to launch a unidirectional wave. This value is saved as a constant as *NEFF\_FROM\_ANALYZER*.

File Edit	le Edit Tools View Help Window						
	Analysis & Controls	Analysis & Results					
X	Search:	compute/WareguideMicke, pr 🖬 compute/FaramiPromHills pr 🔲					
Welcome	Available Analyzers: Ope	Republication and the second					
\$	compute2DanternaGainAndPhase.py compute2avity6.py compute2avity6.py	simulation/lame_delectricWaveguideModeCaleSouth					
Setup	computeSParameters.py computeSParamsFromHists.py	datasetiame [mtgs	-				
Prepare	computes/aramsviaUverapintegrat.py computeTransitTimeFactor.py computeWaveguideModes.py	ParnyerseSikes 7.750000000000-05					
φ.,	convertSlabToPointHistories.py extractModes.py extractModesViaOnecator.ny	Use Variable:					
Run	performLowPassFilter.py performTwoHistoryArithmetic.py	YannerneSkelY         1.050000000000-05           CiVercomer Viles/Tech-XIXBin-1.0.0dev/Contents/engine/bin/computeBaces.py					
H	truncateHistory.py	Ue variable:					
Design		Use Variable:					
Analyze		YanwerseSiteX2         1.02500000000000000000000000000000000000					
1		Use Variable Python variables defined by command line arguments for this analyser:					
Visualize		Tansenesikutz [L0200000000000-05					
<b>(2)</b>		vacuatory [1.55+6]					
Help		Use Variable:					
		rModes 1 mBodes :: 1 [un] writeFieldForlie :: D [string]					
		Un variable:					
		modeling boot					
		normalize 🔽 Pinding modes with effective indices in the range (1.425, 3.46)					
		compMajorC Minime of cell dimensional nia dy = 1.2044180304130824a-08					
		overwrite 🖓					
		The following variables can be used in the above analyzer options:					
		11703368318900.296975 0.000000 2.668745 0					
		Analysis completed successfully					
	Delete Import Custom Anal		1				
🕑 Anatyz	e: ANALYZER SUCCESS Analyzer finished succ						

Fig. 2.111: The analyzer window after a successful run of computeWaveguideModes.py.

### Setting up the Variable Grid Simulation

After performing the above actions, continue as follows:

- Proceed to the Setup Window by pressing the Setup button in the left column of buttons.
- Activate the wave launcher.
  - Expand Field Dynamics.
  - Expand CurrentDistributions.
  - Right Click on waveLauncher0.
  - Left click on Activate.
- Click on Save and setup in the top right corner.

From the setup tab we will now add in roughness that can be resolved with the variable grid. Add in roughness the same way as before but change the RMS amplitude to 10 nm.

#### **Running the Variable Grid Simulation**

To run the variable grid simulation, proceed 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.
- In the left pane, click on *Reset to Setup Values* to run the full simulation which launches this wave down the waveguide.
- Check Dump at Time Zero.
- Make sure that Number of Steps is set to 24000.
- Make sure that *Dump Periodicity* is set to 2400.

• 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."

## Analyzing the Results

When designing a waveguide and simulating the roughness, one might be interested in calculating the S parameters and spectral power of the wave traveling down the waveguide. XSim features an analyzer that will compute both of these quantities.

- After visualizing the data and making sure the simulation is behaving as expected, proceed to the *Analyze* Window.
- Click on the *computeSParamsFromHists.py* analyzer to open it and fill in the necessary parameters as follows:
  - *simulationName* dielectricWaveguideModeCalcRough
  - firstStep: 0
  - *lastStep*: 12500 (or 24000 for variable grid simulation)
  - stepOffset: 0
  - maxWavelength: WAVELENGTH\_MAX
  - minWavelength: WAVELENGTH\_MIN
  - inDirection: 0
  - inSlabE: slab0\_E
  - inSlabB: slab0\_B
  - inSign: 1
  - *outDirection*: 0
  - outSlabE: slab1\_E
  - outSlabB: slab1\_B
  - outSign: 1
  - *outputFileName* : blank
  - *outputSuffix*: blank
  - overwrite: checked
- Click Analyze in the top right corner.

The results should resemble Fig. 2.112

Upon successful completion of the analyzer, proceed to the Visualize tab.

- From here, click on the drop down menu called Add a Data View and select 1-D Fields.
- To visualize the S-parameters and the input power:
  - Click Add Curve
  - Select S\_slab1\_EAndslab1\_B
  - Click *Apply*, followed by *Ok*.
  - Click on the *Add Window* button and repeat the same process as before but this time select *slab0\_EAndslab0\_BPower* from the drop down menu.

File Edit	Edit Tools View Help Window						
	Analysis & Controls	Analysis & Results					
X	Search:	compute/WaveguideModes.py 🔄 computeSParamsFromHists.py 🔼					
Welcome	Available Analyzers: Open	Apply To: Primary Run 💌	Analyze Stop Clear Output				
1	compute2DantennaGainAndPhase.py	L Run Automatically After Engine. Order:					
Setun	compute/article/romKirchhoffBax.py	maxWavelength 1.85000000000000000000000000000000000000	outpus				
:=	computesParamsFromHists.py	Use Variable:	Python variables defined by command line arguments for this analyzer: simulationName :: dielectricWaveguideWodeCalcRough [string]				
130	computes and state of the second state of the	minWavelength 1.2500000000000e-06	firstStep :: 0 [int] lastStep :: 24000 [int]				
Prepare	convertSlabToPointHistories.py	Use Variable:	stepOffset :: 0 (int)				
- 40 <sub>0</sub>	extractModesViaOperator.py	inDirection 0	minWavelength :: 1.2500000000000000 of [float]				
Run	perform TwoHistoryArithmetic.py	Use Variable:	inflabE::slab_E[string]				
=	truncateHistory.py	inSlabE slab0_E	insign :: 1 [int]				
Design		inSlabB slab0 B	outStabE :: slabi E [string]				
Q			outsign :: 1 [int]				
Analyze		n aga 1	output/iewame :: None [string] outputSuffix :: None [string]				
		Use Variable:	compfajorc :: Faise [hool] overwrite :: True [bool]				
Visualize		outDirection  0	self.outputFileName = dielectricWaveguideModeCalcRough_SParameters.vsh5				
		Use Variable:	3 dimensions.				
Help		outSlabE slab1_E	step range = (0, 24000). Calculating power for histories slab0_E and slab0_B.				
		outSlabB slab1_B	[6.99527187e-08 2.48275862e-08 2.48275862e-08]				
		outSign 1	Calculating S-parameters from histories slabi_E and slabi_B.				
		Use Variable:	[6.99527187e-08 2.48275862e-08 2.48275862e-08]				
		outputFileName	Writing file, dielectricWaveguideBodeCalcRough_SParameters.vsh5.				
		outputSuffix	dielectricWaveguideModeCalcRough_SParameters.vsh5				
		constituter.	File dielectricWaveguideModeCalcRough_SParameters.vsh5 successfully written.				
		Definesh Veschille Liste	[1.50519382e-06 1.49163352e-06 1.47831536e-06]				
		The following variables can be used in the above analyzer options:	at wavelength = 1.4957e-06 m				
			Done				
	Delete Import Custom Analyzer						
		3					
🕝 Analy	ze: ANALYZER SUCCESS Analyzer finished successfully.		Show Log				

Fig. 2.112: The analyzer window after a successful run of computeSParamsFromHists.py.

The resulting plots will have the S-parameters per wavelength on top and the spectal input power on the bottom, as in Fig. 2.113

# **Further Experiments**

Change the *DY\_REFINED* parameter to analyze different roughness amplitudes. Change the variable grid to increase the resolution in the propagation direction and then increase the roughness in this dimension as well.

# 2.3.4 Microring Resonator with Mode Launcher (ringResonatorModeLaunch.sdf)

Keywords:

Ring Resonator, Mode Launcher, MAL, Guided Mode, Photonic Device, Semiconductor

#### **Problem Description**

The Ring 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.

The fundamental guided mode profile is launched as a wide band pulse in the input waveguide. This mode is imported from the file save\_EigenD\_0.vsh5, produced by the computeWaveguideModes.py analyzer. To go through the mode solve process check out the Multimode Fiber Mode Calculation example. We will use the computeSParamsViaOver-lapIntegral.py analyzer to determine the transmission coefficients at the thru-port and drop port.



Fig. 2.113: The visualization window after adding in the plots of the S-parameters and the spectral input power.

### **Opening the Simulation**

The Ring Resonator example is accessed from within XSimComposer by the following actions:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the XSim for Electromagnetics option.
- Expand the *Photonics* option.
- Select Ring Resonator 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 available in the Setup Window as shown in Fig. 2.114. 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 Constants:

- RESOLUTION\_XY = the inverse of the number of cells per wavelength
- RESOLUTION\_Z = the inverse of the number of cells per waveguide height in the z dimension
- WAVELENGTH\_CENTER = the central wavelength used in the excitation
- WIDTH\_EXCITATION = the width in wavelength space of the wide band signal
- RADIUS\_RING = radius of the ring
- WIDTH\_WAVEGUIDE = Width of waveguides

X XSim - Microring Resonator with Mode Launcher		
File Edit Tools View Help Window		
Editor		
	Simulation Setup is Ready Save and Setur	'n
ringResonatorModeLaunch.sdf ringResonatorModeLaunch.pre ringResonatorM	Modelaunch.in	
Simulation	A k No active tool View Solids View Solids View Properties	1
Description		-
PI		
PIO2		
- MU0		
Prepare ELEMCHARGE ELECTRASS		
PROTMASS		
- MUONMASS	82	
Run – EPSILON0		
- C2 FLECCHARGE		
- ELECMASSEV	5	
Design – WAVEL_MAX		
RESOLUTION		
- WAVEGUIDE_PERMITTIVITY		
Analyze SUPPRESSION_FACTOR		
RADIUS_RING		
HEIGHT_WAVEGUIDE		
Visualize - XCELLS_MODE		
- WIDTH_GAP		
Help	0188 5	
Undo Add Multiple Remove Add	.d (10~∞m)	
Describe		
Value		
	nanometer micrometer millimeter meter kilometer	
	<u> </u>	
P		۲
Setup: COMPLETED Click run to continue	Show L	g

Fig. 2.114: The Setup window for the ring resonator example showing the external mode launching field.

- HEIGHT\_WAVEGUIDE = Height of waveguides
- WIDTH\_GAP = Width of the gap between ring and waveguides

This simulation applies a wide frequency band signal to the fundamental spatial mode profile. The signal is defined under *SpaceTimeFunctions* and then assigned under *Field Dynamics*, *Fields*, *externalModeLaunchingField1* as shown in Fig. 2.114.

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.

Under *Basic Settings* you can see that the *dielectric solver* is set to *permittivity averaging*. This feature enables second order accuracy for simulations using dielectrics.

#### **Running the Simulation**

When you have saved the setup, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- Check that you are using these run parameters:
  - Time Step: 7.343490630124558e-17
  - Number of Steps: 60000
  - Dump Periodicity: 6000
  - Dump at Time Zero: Checked
- 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". As seen in Fig. 2.115.

🔀 XSim - I	Microring Resonator with Mode Launcher		0 X
File Edit	Tools View Help Window		
	Runtime Options	Logs and Output Files	
X	Parameters Run Mode	Run Dump and Stop Force Stop	Clear Log
	Values entered here will override values from the simulation file.	Engine Log File Browser	
Welcome	Time Step	Nain loop ended at clock time 2024-03-20-09:36:20.533	-
<b>1</b>	Value (7.343490630124558e-17)	Deleting domain	
Setup	Use Variable:	Deleting domain.	
-	-Number of Steps	OUTPUT SUMMARY:	
13	Value: 60000 Default Value (60000)	There were I Warnings encountered in this run.	
Prenare	Use Variable:	See above for more information.	
ricpore		VORPAL completed at clock time 2024-03-20-09:36:20.820 (PH XSimBase) are enabled. Other package combinations may be	
- 10 <sub>00</sub>	Dump Periodicity	possible as well. Please contact support@txcorp.com to investigate package combinations to meet your needs.	
*	Value: 6000 Default Value (6000)	No expand symbols file. Will not limit expansion.	
Run	Use Variable:	Importing ringResonatorModeLaunch from D: USers (qar) bocuments/Tech-X/XSim-1.0/simulations/ringResonatorModeLaunch EM.	
	Restart at Dump Number	Importing events from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\Share\macros.	
	Value:	Importing mathphys from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
Design	Lite Variable:	Importing VSim from C:\Program Files\Tech-X\XSim-1.0.Odev\Contents\engine\share\macros.	
		Importing deviceselect from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\Share\macros	
<b>O</b>	Dump at Time Zero	Importing logicals from C:\Program Files\Tech-X\SSim-1.0.0dev\Contents\engine\share\macros.	
Analyze	No Particle Sorting	Importing units from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
~	Disable Per-Rank Output	Importing utilities from C:\Program Files\Tech-XXSim-1.0.0dev\Contents\engine\share\macros.	
	Custom Run Options	Importing Vputilties from C:\Program Files\Tech-XXSim-1.0.0dev(contents)engine\shartmacrimetros. Importing VSime from C:\Program Files\Tech-XXSim-1.0.0dev(contents)engine\shartmacrimetros.	
Vicualiza		Importing embcs from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
VISUAIIZE	Reset to Setup Values	Importing plasmaDielectric from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
2		Importing delayeval from C:)Frogram FileS; Tech-X; XSim-1.0.00ev)(contentS)engine)share; macros.	
		Importing delayeval from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
Help		Importing esmatrix from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing electrostatics from C:\Program Files\Tech=X\XSim=1.0.0dev\Contents\engine\share\macros	
		Importing the boxd from C: Vrogram Files Tech XX sim-1.0.0dev(Contents)engine (share)macros.	
		Importing grids from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing timing from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing histories from C:)Program Files/Tech-X;XSim-1.0.0dev/Contents/engine/share/macros.	
		Importing multifields from C:\Program Files\Tech-XXSim-1.0.0dev\Contents\engine\sharts.	
		Importing stfuncs from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing emfilters from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Lines from 'ringResonatorRodeLaunch.pre' processed.	
		Engine completed successfully.	
		Total simulation valledick the was 1179-22 seconds.	_
			-
💙 Run:	SUCCESS Simulation engine finished successfully		Show Log

Fig. 2.115: Run window at completion.

# **Analyzing the Results**

Using post analysis scripts, one can extract the transmission coefficients. This simulation uses 4 "EM Field on Plane" histories (slab0-slab4) and this data is used to calculate S parameters between these 4 locations using the analysis script computeSParamsViaOverlapIntegral.py.

Follow these steps:

- Proceed to the Analyze Window by clicking the Analyze button on the left.
- Select computeSParamsViaOverlapIntegral.py and click Open under the list.

Now update the analyzer fields accordingly.

- *maxWavelength*: 1.7e-6 (or use the corresponding parameter from the setup)
- *minWavelength*: 1.4e-6 (or use the corresponding parameter from the setup)
- inPlane: slab0
- *outPlane*: slab1 (or slab3)
- *outputFileSuffix*: S01 (or S03)

The remaining parameter default values will work. Hit the *Analyze* button in the top tight corner. After a successful run the window should resemble Fig. 2.116.

Xim - Microring Resonator with Mode Launcher - O X						
File Edit Tools View Help Window						
Analysis & Controls	Analysis & Results					
Search: computeSParamsViaOverlapIntegral	computeSParamsViaOverlapIntegral.py 🗵	1				
Welcome Available Analyzers: Open	Apply To: Primary Run 💌 Analyze Stop Ck	ear Output				
computeSParamsViaOverlapIntegral.py	simulationName (rinoResonatorModeLaunch					
Setup		-				
	minWavelength 1.4e-6					
Prepare	Use Variable: Executing analyzer:					
\$ <sub>6</sub>	maxWavelength 1.7e-6 C:\Program Files\Tech- X\XSim-1.0.0dev\Contents\engine\bin\computeSParamsViaOverlapIntegral.py					
Run	Use Variable: maxWavelength="1.7e-6"	1"				
=	inPlane slabO overwrite Run Start Time: Wed Mar 20 09:36:42 2024					
Design	outPlane slab1 Run Directory: D:\Users\qar\Documents\Tech-X\XSim-1.0\simulations/ ringResonatorNodeLaunch_EN					
Q.	firstStepInPlane					
Analyze	Use Variable:  Python variables defined by command line arguments for this analyzer: simulationName :: ringResonatorNodeLaunch [string]					
6	firstStepOutPlane minWavelength :: 1.4e-6 [float] maxWavelength :: 1.7e-6 [float]					
Visualize	Use Variable: inPlane :: slab0 [VsHdf5.History] outPlane :: slab1 [VsHdf5.History]					
	numSteps firstStepInPlane :: None [int]					
нер	Use Variable:					
	cutputFileSuffix S01 compMajorC :: False [bool]					
	compMajorC					
	overwrite 🔽 Frequency domain is [1.763485e+14,2.141375e+14]					
	Refrech Variable Liste					
	Analyzing output planes for time steps (0,60002)					
	options: Computing overlap integral					
	Writing results to ringResonatorNodeLaunch_OverlapIntegral_S01.vsh5					
Delate Treased Outers testing	Analysis completed successfully	-				
Import Custom Analyzer						
Analyze: ANALYZER SUCCESS Analyzer finished successfu	n and a second sec	Show Log				

Fig. 2.116: Analyze window with output from computeSParamsViaOverlapIntegral.py.

#### 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 magnetic field by performing the following:

- Near the top left corner of the window, select Data Overview from the Add a Data View drop-down.
- Expand Scalar Data, then B, then select B\_magnitude
- In the bottom left, select *Clip All Plots*, and set z=0 in *Plane Controls*
- Finally, move the dump slider on the bottom of the window to watch the light propagate

The results are shown in Fig. 2.117.

One can visualize the transmission coefficients by performing the following:

- Near the top left corner of the window, select 1-D Fields from the Add a Data View drop-down.
- In the Plot Control panel select the Slab1 and Slab3 SParams data for Graphs 1 and 2, respectively.
- Set the other Graphs to None

The results are shown in Fig. 2.118. As expected, we see coupling of resonant wavelengths with the ring.



Fig. 2.117: Visualization of magnetic field.



Fig. 2.118: Visualization of the s-parameters.

# 2.3.5 Y Splitter (ySplitter.sdf)

Keywords:

## S Parameter, Mode Launcher, MAL, Guided Mode, Photonic Device, Semiconductor

# **Problem Description**

The Y Splitter makes use of a gds file of a passive photonic device, intended to split the power of a single incoming wave into two separate branches.

The device itself is silicon, surrounded by silicon dioxide.

This examples makes use of a "Photonics simulation template" intended to allow for rapid setup of any passive photonic device, provided the ports are aligned with the X-Axis. Matched Absorbing Layers (MALs) are used to dampen the E, B and D fields near the boundary of the simulation.

The fundamental guided mode profile is launched as a wide band pulse in the input waveguide. This mode is imported from the file port1\_EigenD\_0.vsh5, produced by the computeWaveguideModes.py analyzer. We will use the computeSParamsViaOverlapIntegral.py analyzer to determine the transmission coefficients at the two exit ports of the device.

This simulation out of the box is set to a minimal resolution, allowing for a faster simulation time, with 8 cells per wavelength (in Silicon) used. For typical use cases a value of 12-18 cells per wavelength is recommended.

This simulation can be performed with a XSimEM license.

# **Opening the Simulation**

The Y Splitter example is accessed from within XSimComposer by the following actions:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the *XSim for Electromagnetics* option.
- Expand the *Photonics* option.
- Select *Y Splitter* 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 available in the Setup Window as shown in Fig. 2.119. 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. These are also intended to be reset for a different gds file and will redefine the simulation size and parameters

General Simulation Constants:

- WAVELENGTH\_SINGLE = Used for a single wavelength simulation and calculation of simulation information based on wavelength.
- WAVELENGTH\_MIN = Minimum wavelength for a multi-wavelength simulation.
- WAVELENGTH\_MAX = Maximum wavelength for a multi-wavelength simulation.



Fig. 2.119: The Setup window for the y splitter example showing the imported gds file.

- WAVELENGTH\_XMAL = Number of wavelengths of MAL in the X-direction.
- WAVELENGTH\_YMAL = Number of wavelengths of MAL in the Y-direction.
- WAVELENGTH\_ZMAL = Number of wavelengths of MAL in the Z-direction.
- PORT1\_X = Location of port 1 (excitation port) on the X-axis.
- PORT1\_PHYS\_YBGN = Beginning of Port 1 on the Y-axis.
- PORT1\_PHYS\_YEND = End of Port 1 on the Y-axis.
- PORT\_PHYS\_ZBGN = Beginning of all ports on the Z-axis
- PORT\_PHYS\_ZEND = End of all ports on the Z-axis
- PORT2\_X = Location of port 2 on the X-axis.
- PORT2\_PHYS\_YBGN = Beginning of Port 2 on the Y-axis.
- PORT2\_PHYS\_YEND = End of Port 2 on the Y-axis.
- PORT3\_X = Location of port 3 on the X-axis.
- PORT3\_PHYS\_YBGN = Beginning of Port 3 on the Y-axis.
- PORT3\_PHYS\_YEND = End of Port 3 on the Y-axis.
- PORT4\_X = Location of port 4 on the X-axis.
- PORT4\_PHYS\_YBGN = Beginning of Port 4 on the Y-axis.
- PORT4\_PHYS\_YEND = End of Port 4 on the Y-axis.
- PORT5\_X = Location of port 5 on the X-axis.
- PORT5\_PHYS\_YBGN = Beginning of Port 5 on the Y-axis.
- PORT5\_PHYS\_YEND = End of Port 5 on the Y-axis.

- PORT1\_LEFT\_SIDE = Set to 1 if on the left side of the simulation, -1 if on the right.
- PORT2\_LEFT\_SIDE = Set to 1 if on the left side of the simulation, -1 if on the right.
- PORT3\_LEFT\_SIDE = Set to 1 if on the left side of the simulation, -1 if on the right.
- PORT4\_LEFT\_SIDE = Set to 1 if on the left side of the simulation, -1 if on the right.
- PORT5\_LEFT\_SIDE = Set to 1 if on the left side of the simulation, -1 if on the right.
- MIN\_Y\_COORDINATE = Minimum Y coordinate of the component, used if less than any of the ports.
- MAX\_Y\_COORDINATE = Maximum Y coordinate of the component, used if greater than any of the ports.
- CELLS\_PER\_WAVELENGTH\_AXIAL = Number of cells per wavelength (in Silicon) along the X axis.
- CELLS\_PER\_WAVELENGTH\_CROSSSECTION = Number of cells per wavelength (in Silicon) along the Y and Z axis.

This simulation applies a wide frequency band signal to the fundamental spatial mode profile. The signal is defined under *SpaceTimeFunctions* and then assigned under *Field Dynamics*, *Fields*, *port1* 

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.

Under *Basic Settings* you can see that the *dielectric solver* is set to *permittivity averaging*. This feature enables second order accuracy for simulations using dielectrics.

#### **GDSII File Import**

The GDS File has already been imported, however this example serves as a good demonstration of how to do so

It may be imported like any other CAD file, however a second dialog box will appear, shown below.

	GDS2 Layer Information							
Import?	Index	Name	Start (1e-09m)	Thickness (1e-09m)	End (1e-09m)	Sidewall (deg)	Material	
Q	1	Layer001	0	200	200	90	Silicon	•
							,	
Reset to	Reset to current values Cancel Import							
L								

#### Fig. 2.120: GDS layer selection.

This allows for selection if the layer should be imported, its starting position on the Z axis, and thickness. Sidewall angles can also be set however this is a feature that is still under active development at this time and is recommended to be kept at 90 degrees.

## **Running the Simulation**

When you have saved the setup, continue as follows:

- Proceed to the run window by pressing the Run button in the left column of buttons.
- Check that you are using these run parameters:
  - *Time Step*: 1.5124169044445238e-16
  - Number of Steps: 3000
  - Dump Periodicity: 500
  - Dump at Time Zero: Checked
- 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". As seen in Fig. 2.121.

🔀 XSim -	ySplitter		0 X
File Edit	Tools View Help Window		
	Runtime Options	Logs and Output Files	
V	Parameters Run Mode	Run Dump and Step Force Step	Clear Log
Χ	Values entered here will override values from the simulation file.	Engine Log File Browsor	
Welcome	Time Step	Dumping invEps.invEps at 2024-03-19-12:56:29.384.	-
<b>d</b>	Detaul Value (1.5124169044440238e-16)	Dumped invEps.invEps at 2024-03-19-12:56:29.544.	_
	Value, 1.51241050444452500-10	Dumped invEps at 2024-03-19-12:55:29.544.	
Setup	Use variable:	No electromagnetic fields to dump.	
	Number of Steps	No collisions to dump.	
	Value: 3000 Default Value (3000)	No ionization processes to dump. Domain Dump 6 committed at 2024-03-19-12:55:29.596.	
Prepare	Use Variable:		
8.	Dump Periodicity	Main loop ended at clock time 2024-03-19-12:56:29.546 Deleting dowain	
.8	Value: 500 Default Value (500)	estering downant	
Run	Use Variable:	Deleting domain.	
	Restart at Dump Number	There were 0 Notices encountered in this run.	
	Value:	There were 0 Warnings encountered in this run.	
Design	Use Variable:	VOMPAL completed at clock time 2024-03-19-12:36129.707. NOTE: This simulation can be unif all of the packages (%)inflage' are enabled. Other package combinations may be possible as well. Please contact supportfixcorp.com to	
	E Dumo at Timo Toro	investigate package combinations to meet your needs.	-
	No Particle Cortino	No expand symbols file. Will not limit expansion.	
Analyze	Disable Per-Rank Output	Importing verbaity from C: Pergema FilesTech-X: Sim-1.0.0ev/Contents(engle) hard hercos.	
1	Custom Due Catlons	Importing enums from C:\Program Files\Tech-X\ZSim-1.0.0dev\Contents\engine\share\macros.	
	cusion run opoors	Importing Matinghys Irom (') Frogram FileS (FeCh-ALSIM-1.0. Odev/Contents)engine/Smartematrom. Importing VSim from (') Frogram FileS Tech-ALSIM-1.0. Odev/Contents)engine/Smartematrom.	
Visualize	Reset to Setup Values	Importing deviceselect from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing listUtilities from C:\program Files\Tech=XXSim-1.0.0dev\Contents\engine\share\macros.	
		Importing injurials from CiProgram FileSien-Alaim-1.0.0dev/concents/engine/smartematros.	
Help		Importing utilities from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing Viputilities from C: Program FilesTech-XXSim-1.0.0dev:ContentsTengine)shareTemacros.	
		Importing embcs from C:\Program Files\Tech-X)XSim-1.0.Odev\Contents\engine\share\macros.	
		Importing plasmabilectric from C:)Program Files/Tech-X/XSim-1.0.0dev/Contents/engine/share/imacros.	
		Importing deringeval from Criveogram Files/ieco-XLSIm-1.0.dev/Contents/engine/smate/imactos.	
		Importing delayeval from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing emartix from C:)Program File3/Tech-X1X3im-1.0.0dev/Contents)engine)share\marros.	
		Importing implicited theorem is from CiProgram Filed Tech-XXSim-1.0.0449 (Contents) engine [share]macros.	
		Importing tfsfboxxd from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing grids from C:)Program Files/Tech-X:XSim-1.0.0dev)Contents/engine)share/immoros.	
		Importing that is from C. Programs That's test test to dev) Contents from the instrume instrume.	
		Importing shapes from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing sultifields from C:)Program Files)Tech-X:XSim-1.0.0dev/Contents)engine)sharehmacros.	
		Importing entitlers from C/Frograms Files/Tech-X/XSim-1.0.0dev/ContentS/engine/hhare/macros.	
		Lines from 'ySplitter.pre' processed.	
		Finished with 'ySplitter.pre'.	
		Engine completed successfully.	
		Total simulation valid-clock time vas 422.000 seconds.	
		<u></u>	•
Pup-	SUCCESS Simulation agains finished a proset like		Show Log

Fig. 2.121: Run window at completion.

#### **Analyzing the Results**

Using post analysis scripts, one can extract the transmission coefficients. This simulation uses 3 "EM Field on Plane" histories (Port1, Port2, Port3) and this data is used to calculate S parameters between these 3 locations using the analysis script computeSParamsViaOverlapIntegral.py.

Follow these steps:

- Proceed to the Analyze Window by clicking the Analyze button on the left.
- Select computeSParamsViaOverlapIntegral.py and click Open under the list.

Now update the analyzer fields accordingly.

• *maxWavelength*: 1.7e-6 (or use the corresponding parameter from the setup)

- *minWavelength*: 1.4e-6 (or use the corresponding parameter from the setup)
- *inPlane*: Port1
- outPlane: Port2 (or Port3)

The remaining parameter default values will work. Hit the *Analyze* button in the top right corner. After a successful run the window should resemble Fig. 2.122.

🔀 XSim - :	ySplitter		- • ×
File Edit	Tools View Help Window		
	Analysis & Controls	Analysis & Results	
X	Search: computeSParamsViaOverlapIntegral	computeSParamsViaOverlapIntegral.py 🔀	
	and the sector of the sector o	Apply To: Primary Run 💌	
weicome		Run Automatically After Engine. Order:	1 4 Analyze Stop Clear Output
<b>1</b>	computeSParamsViaOverlapIntegral.py		ame l
Setun		simulationName ySplitter	first step.
		ministruminanth 1 day 06	numSteps=NUMSTEPS, -L NUMSTEPS
			Number of time steps to analyze for noth planes, perault is the maximum number of steps.
Prepare		Use Variable:	outputFileSuffix=OUTPUTFILESUFFIX, -D OUTPUTFILESUFFIX
		maxWavelength 1.7e-06	compHajorC, -Z
		Use Variable:	The indexing order for written datasets. If checked, use "compNajorC". If unchecked, use the default
Kull		inPlane Port1	"compHinorC".
			Whether a dataset or group should be overwritten if it
Design		outPlane Port2	already exists.
0		firetStanInPlane	A .vshS file with with the overlap integral plotted as a function of wavelength.
~			
Analyze		Use Variable:j	Executing analyzer: C:Drown Files Tach-VIXSIn-1 0 Dday/Contents)anginabin/computeSparand/isDuarianIntegral ngSimilarionNames"uSDitter"
1		firstStepOutPlane	-minBavelength*1.4e-06" -maxEavelength*1.7e-06" -inFlame*Fort1outFlame*Fort2overvite
fierraliza		Use Variable:	Run Start Time: Tue Mar 19 12:55:50 2024 Run Directory: D.: Umersioart Documental Tech-X)XSim-1.0)gimulations/vSplitter KN
- CHOMEC		numétron	
2		numbreps j	
Help		Use Variable:	Python variables defined by command line arguments for this analyzer:
		outputFileSuffix	simulationwame :: yoplitter [string] minWavelength :: 1.4e-06 [float]
		weather of E	maxWavelength :: 1.7e-06 [float]
		compwagore i	outPlane :: Pott [Vald5.history]
		overwrite 🔽	firstStepInPlane :: None [int]
		Refresh Variable	numbers :: None [int]
			outputFileSuffix :: [string]
			overwite :: Tutte [bool]
			WARNING: Input and output planes are different sizes. Slicing larger plane to match.
			Frequency domain is [1.763485e+14,2.141375e+14] Analyzing input planes for time steps [0.3002]
		×	
			Analyzing output planes for time steps [0,3002]
		The following variables can be used in the above analyzer options:	Computing overlap integral
			Writing results to ySplitter_OverlapIntegral.vshS
	1		Analysis completed successfully
	Delete Import Custom Analyzer	<u>  </u>	
_			
🕑 Analy	ze: ANALYZER SUCCESS Analyzer finished successfully	L	Show Log

Fig. 2.122: Analyze window with output from computeSParamsViaOverlapIntegral.py.

#### Visualizing the results

After performing the above actions proceed to the Visualize Window by pressing the *Visualize* button in the left column of buttons.

- Near the top left corner of the window, select 1-D Fields from the Add a Data View drop-down.
- In the Plot Control panel select the Port2 and Port3 SParams data for graph 1.
- Set the other Graphs to None

The results are shown in Fig. 2.123. As expected, we see equivalent power split between the two ports across the wavelength range

#### **Further Experiments**

This example in its current form uses a lower resolution than one might typically use when designing a waveguide. Try increasing the resolution, recalculating the mode and the two s-parameter curves should grow closer in agreement.

This example can be easily adapted for different GDS files.



Fig. 2.123: Visualization of the s-parameters.

# 2.3.6 Three Material Waveguide (threeMaterialWaveguide.sdf)

Keywords:

Photonic Waveguide, Guided Mode, Semiconductor

# **Problem Description**

This example demonstrates how to calculate and launch the fundamental mode of a rectangular silicon waveguide using a unidirectional wave launcher and a broadband signal. In this example, the waveguide itself is sitting on top of a silica slab and is surrounded on the other 3 walls by quartz.

This simulation can be performed with a XSimEM license.

# **Opening the Simulation**

To open this example open an instance of XSimComposer and follow the steps below:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the XSim for Electromagnetics option.
- Expand the *Photonics* option.
- Select Three Material 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.

### **Simulation Variables**

This example contains a number of variables defined to make the simulation easily modifiable.

- CPWX and CPWYZ: CPWX controls the number of cells per vacuum wavelength in the x direction and CPWYZ controls the number of cells per vacuum wavelength in both the y and z directions. Increasing one or both of these values will increase the resolution of the simulation.
- WAVELENGTH: Center wavelength of the input signal. This wavelength is also used for the calculation of the fundamental guided mode of the device.
- WAVEGUIDE\_WIDTH and WAVEGUIDE\_HEIGHT: These two parameters control the cross sectional area of the waveguide.

The *Materials* section contains silicon, silica, and quartz. The *Geometries* section includes the CSG waveguide, the silica slab, and their defining parameters. In this example, filling the rest of the empty space with quartz is done by going to the *Basic Settings* section and setting the *background permittivity* variable to 4.5. 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. This simulation makes use of XSim's Wave Launcher to launch a unidirectional wave down the waveguide with a mode defined by the file generated by the computeWaveguideModes.py analyzer.

### Solving for the Eigenmodes

To calculate the fundamental mode of the silicon waveguide, first make sure that in the *Setup* tab that *wavelauncher0* is set to *inactive*. Save the simulation and proceed to the *Run* tab. Set *Number of Steps* and *Dump Periodocity* to 1 and make sure that *Dump at Time Zero* is checked. Click on the *Run* button. Upon successful completion of the engine, proceed as follows:

- Proceed to the Analyze Window by clicking the Analyze button on the left.
- Select computeWaveguideModes.py and click Open under the list.
- Update the analyzer fields accordingly by choosing the corresponding variables under Use Variable :
  - simulationName: threeMaterialWaveguide
  - datasetName: invEps
  - transverseSliceX: 0
  - transverseSliceLY: PORT\_BGNYZ
  - *transverseSliceUY*: PORT\_ENDYZ
  - transverseSliceLZ: PORT\_BGNYZ
  - transverseSliceUZ: PORT\_ENDYZ
  - vacWavelength: 1.55e-6
  - nModes: 1
  - writeFieldProfile: D
  - modeFileName: port1
  - Normalize: checked
  - compMajorC: unchecked
  - Overwrite: checked
- Click *Analyze* in the top right corner.

The analyzer will only find guided modes. The results should resemble Fig. 2.124. We see that the analyzer found the fundamental mode and gives the effective index of the mode under *Neff*. This parameter is necessary for using the *Wave Launcher* to launch a unidirectional wave. This value is saved as a parameter as *NEFF*.

🔀 XSim - '	XSim - Three Material Waveguide — O X					
File Edit	Tools View Help Window					
	Analysis & Controls	Analysis & Results				
X	Search: computeWaveguideModes	compute/WaveguideModes.py 🔽	1			
Welcome	Available Analyzers: Open	Apply To: Primary Run	Analyze Stop Clear Output			
ø	computeWaveguideModes.py computeWaveguideModes/by/Grid.pv					
Setup		simulationName threeMaterialWaveguide parts.				
ii.		datasetName imEps				
Prepare		transverseSiceX 0 C:\Program Files\Ten-X\XSim-1.0.0dev\Contents\engine\bin\cor C:\Program Files\Ten-X\XSim-1.0.0dev\Contents\engine\bin\cor	sputeWaveguideHodes.py			
<b>\$</b> _a		Use Variable:	"			
Run		transverseSiloeLY -1.550000000000e-06 vacVarseSiloeL2== 1.55000000000000e=0= -=transverseSiloeL2== 1.55000000000000e=0= vacVarseSiloeL2== 1.550000000000000e=0= vacVarseSiloeL2== 1.55000000000000e=0= vacVarseSiloeL2== 1.55000000000000e=0= vacVarseSiloeL2== 1.55000000000000e=0= vacVarseSiloeL2== 1.55000000000000e=0= vacVarseSiloeL2== 1.55000000000000e=0= vacVarseSiloeL2== 1.55000000000000e=0= vacVarseSiloeL2== 1.550000000000e=0= vacVarseSiloeL2== 1.5500000000000e=0= vacVarseSiloeL2== 1.5500000000000e=0= vacVarseSiloeL2== 1.55000000000000e=0= vacVarseSiloeL2== vacVarseSiloeL2== 1.5500000000000e=0= vacVarseSiloeL2== vacV	modeFileName="port1"normalize			
=		Use Variable:	tions/threeMaterialWaveguide EM			
Design		transverseSliceUY 1.5500000000000-06				
Q		Use Variable: Python variables defined by command line arguments for this is	analyzer:			
Analyze		transverseSilceLZ -1.5500000000000-06 simulationName :: threeNaterialWaveguide [string] datasetName :: invEps [string]				
Ĩ.		Use Variable:				
Visualize		transverseSiceUZ [1.5500000000000000000000000000000000000				
2		Use Variable:				
Help		vacWaveWingth [1:50=0				
		normalize :: True [bool]				
		overwrite :: True (bool)				
		writeFieldProfile D				
		Finding modes with effective indices in the range (1.425, 3.4 Minima of cell dimensions:	48)			
		moderievame port1 min dy = 6.4583333333264e-08 min dz = 6.4583333333264e-08				
		normalize 🔽 min dy*dz = 4.171006944444355e-15				
		compMajorC Kx*2_r Kx*2_i Neff File #	-			
		183849725508364.812500 0.000000 3.344901 0				
		In the following ten source under it we adone at strate update.	-			
	1	Analysis completed successfully	-			
	Delete Import Custom Analyzer					
📿 Anaka	ze: ANALYZER SUCCESS Analyzer finished successfully	N	Show Log			

Fig. 2.124: The analyzer window after a successful run of computeWaveguideModes.py.

#### Setting up the Simulation

After performing the above actions, continue as follows:

- Proceed to the Setup Window by pressing the Setup button in the left column of buttons.
- Activate the wave launcher
  - Expand Field Dynamics
  - Expand CurrentDistributions
  - Right Click on waveLauncher0
  - Left click on Activate
- Click on Save and setup in the top right corner.

This setting is shown in Fig. 2.125.

						Simulation Setup is Ready Sa
threeMaterialWaveguide.sdf threeMate	rialWavequide.ore threeMaterialWavequ	de.in				
Simulation			≥ No active tool	Clin Show Scale Waw Solide	¥ 42 ¥	Parat View Proper
Description			E his acute doi	Show Scale   Yiew Solida		Troper
Constants			6			
Parameters			m			
Basic Settings						
- Functions			8 1			
SpaceTimeFunctions			- P			
Materials			Jat			
Geometries			2			
Grids						
<ul> <li>Field Dynamics</li> </ul>						
Fields			ž			
PielaboundaryConditions			—			
- Currencostroutions						
Histories						
1000000						
				5e-06		
				1		
				-10 06		
				Fe 06 Here to		2e-05
				56-00		
	. I	1				
	Undo Add Multiple Remove	Add				
Property	Value	<u> </u>				
- kind	Wave Launcher					
description						
color						
effective index of refraction	NEFF					
directionality	unidirectional					
launching direction (if unidirectional)	positive					
- surface	yz plane	_				
	PORT_BGNX					
yMin	PORT_BGNYZ					
yMax	PORT_ENDYZ					
zMin	PORT_BGNYZ					
zMax	PORI_ENDYZ					
Spatial profile	tile defined					
ine name	port_tigenu_uvsno					
temporal profile	La Sagona					
temporal profile     amplitude	1.0					

Fig. 2.125: Location of wavelauncher0 within the setup tree.

### **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. You will be asked to Save. Click *Save* upon the request to save.
- In the left pane, click on *Reset to Setup Values* to run the full simulation which launches this wave down the waveguide.
- Check Dump at Time Zero.
- 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 result is shown in Fig. 2.126.

# **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.

You may need to *Reload Data* (bottom left). Visualize an eigenmode by following these steps:

- From the Add a Data View dropdown, select Data Overview.
- Expand Scalar Data, expand EigenD, and select EigenD\_magnitude.

The resulting visualization pane should look like Fig. 2.127.

To visualize the E field as it travels down the waveguide, proceed as follows:



Fig. 2.126: Running the full simulation which launches the wave down the waveguide.



Fig. 2.127: The visualization pane showing the magnitude of the D field of the fundamental mode.

- From *Data Overview*, expand *Scalar Data*, expand *E*, then select *E\_z*.
- Slide the dump number selector over to the right to watch the wave travel down the waveguide.

The resulting visualization pane should look like Fig. 2.128.

🔀 XSim -	Three Material Waveguide		- 0 ×
File Edit	Tools View Help Window		
	Add a Data View 💌 Choose a Run: PrimaryRun	•	Unload Data Reload Data
X	Data Overview1 🖸 Data Overview2 😫		
Welcome	Visualization Controls	Visualization Results	
		- 31	
- <del>12</del>	Variables	Save Labels Limits Axis Scale Rendering +2  Reset View  Preserve Aspect Ratio  Auto Reset	
Setup	😑 Scalar Data 🔺		
三	⊕-B ⊕-D		
Prenare	B-E		
~		New Section of the se	
**ø	U €.y V €.z		
Run	EigenD EigenDl	2 2 3 3 3 4 5 4 5 4 5 4 5 4 5 4 5 4 5 4 5 4	
Ŧ	terinv€ps	- 3.000+09	
Design	substrateInOut     waveguideInOut		
	waveLauncher0MinusField     waveLauncher0MinusFieldCrossDay		
e		Mar: 5.012-d0 Mar: -6.008-00	
Analyze		3.0	
	Clip Plot     Plane Controls	20	
Visualize	1 Log scale Color		
9	Display Contours 5		
Heln	Opacity , , , , ?		
	Color Table hot_desaturated	-3.0	
	Limits	are 10 -5 States 5 10 15	
	Set Minimum 0		
	Reset Minimum		
	Set Maximum  0		
	Keset Maximum		
	Reset to Default		
			1
		Dump: 5	
	Annotation Level: Axes & Legends 💌	b · · · · · · · · · · · · · · · · · · ·	10
		<u>u</u>	
🕑 Visua	Ize: READY Visualization available		Show Log

Fig. 2.128: The visualization pane showing the z component of the E field.

If instead you wish to view a cross section of the waveguide and E field, proceed as follows:

- From *Data Overview*, expand *Scalar Data*, expand *E*, then select *E\_z*.
- In the visualization controls section, click on the *Plane Controls* button. Deselect Plane3 and select Plane1. Click the *Apply* button. Click *Ok*.
- In the visualization results view, to the left of the *Reset View* button is a drop down menu. Change this to +x.
- To make the field outside of the waveguide, change the minimum and maximum values under *Limits* to -5e-6 and 5e-6.

The resulting visualization pane should look like Fig. 2.129.

#### **Further Experiments**

Change the geometry on the Setup window and rerun the simulation and analyzer to see the effects on the modes. Change the materials used and see how the field inside and outside the waveguide changes.

Asim -	Three Material Waveguide		- 0 ^
File Edit	Tools View Help Window		
	Add a Data View 💌 Choose a Run: PrimaryRun		Unload Data Reload Data
X	Data Overview 1	Description 2	
	Manufaction Controls	Manufard W	
Welcome	Visualization Controls	Visualization Fersula's	
<b>1</b>		34	
Setun	Variables	Save Labels Limits Axis Scale Rendering +x  Reset View   Preserve Aspect Ratio   Auto Reset	
e	E Scalar Data		
国	⊕-D		
Prepare	E magnitude	Development	
~	-86	Vor: E.z 5.000e-06	
~~a	U €.y I €.z		
Run	B EigenD	- 2.500+06	
-	⊕ inv£ps	0.000	
	B−J E: substateleCut		
Design	waveguidelnOut	-2.500+06	
	waveLauncher0MinusField     waveLauncher0MinusFieldCrossDay	-8.000+00	
×.		Mar: 5.02+00 Min: 5.00+00	
Analyze			
	Clip Plot Plane Controls	<b>1</b> 10	
	F Log Scale Color		
Visualize	E Display Contrains		
2			
Help	opacity , , , , , , , , , , , , , , , , , , ,	10	
псер	Color Table hot_desaturated •		
	Limits	20	
	Set Minimum -5e-6		
	Reset Minimum		
	Ed Set Maximum Eo.6	5.0	
	Deset Maximum		
	Reset Maximum	3.0 2.0 1.0 9.10 - 2.0 - 3.0 0	
	Report to Default		
	·	Dump: 5	
	territoria territo función de la constante de la	· · · · · · · · · · · · · · · · · · ·	1
	Annotation Level: JAxes & Legends	0	10
📿 Visual	Ize: READY Visualization available		Show Log

Fig. 2.129: The visualization pane showing the z component of the E field in a cross section of the domain.

# 2.4 Scattering

# 2.4.1 Scattering off a Metal Sphere (metalSphere.sdf)

Keywords:

Mie Scattering Off Metal Sphere

#### **Problem description**

The example describes the scattering of electromagnetic waves off a metal sphere. This phenomenon is often referred to as Mie scattering, where the radius of the sphere is comparable to the wavelength of the incident radiation.

An incident plane wave is launched toward the sphere. XSim computes the resulting fields in the vicinity of the sphere, within a computational domain by applying the proper boundary conditions around the surface of the sphere. The waves that exit the computational domain are absorbed into MAL layers.

The fields for points far away from the sphere, and beyond the computational domain are computed with the help of an analyzer that is part of the XSim distribution. The histories of the electric and magnetic fields are recorded along a closed surface known as a Kirchhoff box that lies within the MAL layers. This field information is then used to compute fields far away from the sphere center by applying the Kirchhoff integral theorem.

In this example, the radius of the metal sphere is set to equal 0.3367 times the wavelength of the incident radiation, which is 1m in length.

The wave is launched from the positive z direction, and the incident wave electric field is polarized along the x direction.

This example is set up as a cube, entirely parameterized off the NUM\_WAVELENGTHS, WAVELENGTH and CELLS\_PER\_WAVELENGTH Constant. It is designed so that it may be easily adapted to take the radar cross section

of any geometry, at any wavelength.

Two wavelengths on all sides are devoted to the MAL absorbing boundary conditions. The resolution of the grid corresponds to 16 cells per wavelength, near the middle of the typically used 10-20 cells. The time step (Parameter DT) is chosen to be very close to the Courant condition limit, calculated using the DX, CFL\_NUM and DMFRAC Parameters.

The recording time for the Kirchhoff box is calculated using the parameters TBGNKBOX and TENDKBOX. TBGNKBOX is set by adding the turn on time of the excitation source, and 2\* the time it would take to cross the diagonal of the RCS Box. TENDKBOX is set by taking TBGNKBOX, and then adding the amount of time to cross the entire RCS box, +2.5 periods. This allows for the collection of 2.5 periods of data.

The number of timesteps (Parameter NUM\_STEPS) for the simulation corresponds to TENDKBOX/DT. Note that the number of steps in the simulation must be set by hand in **Basic Settings**, and verify that the value used in the Run Panel corresponds to this parameter.

### **Opening the Simulation**

The Mie Scattering, Metal Sphere example is accessed from within XSimComposer by the following actions:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the XSim for Electromagnetics option.
- Expand the *Scattering* option.
- Select "Mie Scattering Metal Sphere" 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. 2.130. 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 any 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.

The Setup Window shows the Kirchhoff box, with the metal sphere at the center. To see the grid, click on the drop down menu of "Grids" on the left side panel. Check the box beside "Grid". In order to view or change the wave frequency, click on the drop down menu "Parameters" and then click on "Frequency". To view or change the direction of the incident wave or polarization, you can click on the drop down menu "Field Dynamics", then the drop down menu "RCSBox". Following this, check the box beside "rcsBox0". The lower left panel displays a table with the wave property and its corresponding value.

# **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 that you are using these run parameters:
  - Time Step: 5.71337236814229e-11
  - Number of Steps: 2015
  - Dump Periodicity: 300
  - Dump at Time Zero: Checked
- Click on the *Run* button in the upper left corner of the right pane.

🔀 XSim - Si	attering off a Metal Sphere						- 0 X
File Edit '	Tools View Help Window						
	Editor						
						Simulation	Setun is Ready Save and Setun
						- Sindadon	Seep is ready
Wolcomo	metalSphere.sdf metalSphere.pre metalSphere.in	-					
Weidonie	Simulation	_ ≥	No active tool	Clip Show Scale View Solids	▼ +2 ▼	Reset View	Properties
<u>50</u>	Description	2 e					
	Constants     Dependence	8					
Setup	FREQUENCY		-				
:=	OMEGA	100	N				
13 I	- XEND	18	ľ				
Prepare	MALTHICKNESS	Da					
	- XENDMAL	aki i	<u>н</u> у				
- <del>2</del>	XENDKIRCHHOFFBOX	ate	<b>4 ^</b>				
	XBGNKIRCHHOFFBOX	Σ					
Run	- XENDTFSFBOX						
	- LX - DOMAINTRANSTITIME						
	TURNONTIME						
Design	- XBGNTFSFBOX						
	- XBGN						
	TEGNKBOX						
Analyze	DL			4			
	NUM_STEPS						
A Marca Plan	Basic Settings						
visualize	Functions						
	Materials	-					
	Geometries						
Help	Eiald Dunamics						
	<ul> <li>FieldBoundaryConditions</li> </ul>						
	MalupperX	-1		-3			
		1					
	Undo Add Multiple Remove Add			-3_			
	Burnet Maler	<b>-</b>		<sup>م</sup> لحر.			
	Property Value	-		4 22			
Setup:	COMPLETED Click run to continue						Show Log

Fig. 2.130: Setup Window for the Mie Scattering example.

🔀 XSim - S	Scattering off a Metal Sphere		σ×
File Edit	Tools View Help Window		
	Runtime Options	Logs and Output Files	
X	Parameters Run Mode	Run Dump and Stop Force Stop	Clear Log
	Values entered here will override values from the simulation file.	Engine Log File Browser	
weicome	Time Step Default Value (5.71337236814229e-11)	Dumped invFaceFracs at 2024-03-21-13:01:07.482.	-
• 🗘	Value: 5.71337236814229e-11	All multipleds dumped at 2024-03-21-15:01:07.485. No electromagnetic fields to dump.	
Setun	Use Variable:	No collisions to dump.	
e	Number of Stens	No ionization processes to dump. Domain: hump 2 completed at 2024-03-21-13:01:07 485.	
13	Value: 2015 Default Value (2015)		
Prepare	Use Variable:	Deleting domain	
40	Dump Periodicity	Deleting domain.	
~~@	Value: 300 Default Value (300)	OUTPUT SUMMARY:	
Run	Use Variable:	There were 1 Warnings encountered in this run.	
-	Restart at Dump Number	See above for more information.	
	Value:	NOTE: It simulation can be run if all of the packages ' GPU XSimBase XSimEN' are enabled. Other package combinations may be possible as we	e11.
Design	Liee Variable	Please contact support@txcorp.com to investigate package combinations to meet your needs.	
		No expand symbols file. Will not limit expansion. Importing metalSphere from D:\Users\carciverlocuments\Tech-X\XSim-1.0\simulations\metalSphere EN.	
	V Dump at Time Zero	Importing verbosity from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\maccos.	
Analyze	No Particle Sorting     Disclob Day, Dayly Output	Importing enumes from C:)Programs Files/Tech-XXSim-1.0.0dev/Contents/engine/share/macros. Importing mathbuys from C:)Programs Files/Tech-XXSim-1.0.0dev/Contents/engine/share/macros.	
	i bisable Pel-Kalik Oulput	Importing VSim from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
	Custom Run Options	Importing deviceselect from C:)Program Files)Tech-X:XXIm-1.0.0dev/Contexts)engine(share)macros.	
Visualize	Reset to Setup Values	Importing logicals from C:\Program File\Tech-Tid Tido Idev\ContentSignationality in the technology is the technology in the technology in the technology is	
		Importing units from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing vputilities from C:\Program Files\Tech-XIXSim-10.0.dev\Contents\engine that eacros.	
нер		Importing VSimE from C:\Program Files\Tech-XXSim-1.0.0dev\Contents\engine\share\macros.	
		Importing plasmabilectric from C: Program Files/lech=Atsim=1c0.000v(Contents)engine(share\macros.	
		Importing delayeval from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing matrices from Civrogram Files/tech-XXSim-1.0.0dev/Contents/engine/shate/macros.	
		Importing esmatrix from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing electrostatics from C:\Programs Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing disflocted comparison from cirrogram files Tech-Alian in the interior content of the files interior and the second sec	
		Importing grids from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing timing from c::/rogram FileS/rech-X/x3im-1.0.0dev/contents/englne/share/macros. Importing histories from c:/Program FileS/rech-X/xSim-1.0.0dev/contents/englne/share/macros.	
		Importing shapes from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Importing multifields from C:\Program Files\Tech-XIXSim-1.0.0dev\Contents engine share\marcs.	
		Importing emfilters from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.	
		Lines from 'metalSphere.pre' processed.	
		Engine completed successfully.	
		To see results, click on the "Visualize" icon in the icon panel.	
		Total simulation wall-clock time was 24.655 seconds.	- I
		U <sup>1</sup>	<u> </u>
📿 Run:	SUCCESS Simulation engine finished successfully		Show Log

Fig. 2.131: The Run Window at the end of simulation.

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 result is shown in Fig. 2.131.

Running in 3D, this simulation uses 2000 time steps. The run takes about 5 minutes on a 4-core 2.3 GHz processor.

#### **Running the Analyzer**

- 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 metalSphere
  - fieldLabel E
  - farFieldRadius 1024.0
  - numPeriods 0.25
  - numFarFieldTimes 2
  - frequency 0.3e9
  - 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)
  - incidentWaveAmplitude 1.0
  - incidentWaveDirection (0,0,1)
  - varyingMeshMaxRadius 1.0
  - principalPlanesOnly checked
- Click the Analyze button near the top right of the window.

#### Visualizing the results

After performing the above actions, click on *Visualize* in the column of buttons at the left. The program will load the data and provide you with certain options.

One of the quantities that is of interest in such scattering phenomena is radar cross section (RCS) measured in dBsm. The RCS, sometimes designated as  $\sigma$ , having units of  $m^2$ , is given as

$$RCS = 4\pi R_s^2 \frac{P_r}{P_i}$$

where  $R_s$  is the radial distance from scatterer,  $P_r$  is the power flux received at the point of interest, and  $P_i$  is the incident power flux. In MKS units the power flux is measured in  $W/m^2$ . RCS in dBsm is given as

$$10\log_{10}(RCS)$$

To obtain plots of this quantity, click on the drop down menu, *Add a Data View*. In this menu, choose 1-D Fields. There is a large list of options to choose from. Figure Fig. 2.133.



Fig. 2.132: The Analyze Window at the end of analyzer execution.





# **Further Experiments**

You may try the example that includes a dielectric coating over the same metal sphere to compare the effects of such a coating over the RCS.

# 2.4.2 Scattering off a Metal Sphere with a Dielectric Coating (dielecCoatedMetal-Sphere.sdf)

Keywords:

#### Mie Scattering Dielectric Coated Metal Sphere

#### **Problem description**

The example describes the scattering of electromagnetic waves off a metal sphere with a dielectric coating, which is a modification of the previous example that described scattering off a pure metal sphere.

An incident plane wave is launched toward the sphere. XSim computes the resulting fields in the vicinity of the sphere, within a computational domain by applying the proper boundary conditions around the surface of the sphere. The waves that exit the computational domain are absorbed into MAL layers.

The fields for points far away from the sphere, beyond the computational domain, are computed with the help of an analyzer that is part of the XSim distribution. The histories of the electric and magnetic fields are recorded along a closed surface known as a Kirchhoff box that lies within the MAL layers. This field information is then used to compute fields far away from the sphere center by applying the Kirchhoff integral theorem.

In this example, the radius of the metal sphere is set to equal 0.3367 times the wavelength of the incident radiation, which is 1m in length. The thickness of the coating is 0.1 times the wavelength. The computational domain extent is set to three times the wavelength in all directions along the three coordinate axes. The thickness of the MAL layer is twice the wavelength. The resolution of the grid corresponds to 24 cells per wavelength. This parameter is chosen such that the thickness of the dielectric layer is resolved. The time step is chosen to be very close to the Courant condition limit. The wave is launched from the positive z direction, and the incident wave electric field is polarized along the x direction. Care needs to be taken so that the simulation is performed for a sufficient number of time steps, so that the wave reaches the surface of the Kirchhoff box where the histories are recorded. Thus, changing the frequency or the resolution of the grid would alter the time step, and the number of steps required to complete the run will need to be changed accordingly.

#### **Opening the Simulation**

The Mie Scattering, Dielectric Coated Metal Sphere example is accessed from within XSimComposer by the following actions:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting Examples window expand the XSim for Electromagnetics option.
- Expand the *Scattering* option.
- Select "Scattering off a Metal Sphere with a Dielectric Coating" 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. 2.134. 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. While the setup is similar to the previous
example of a pure metal sphere, the additional components in this example may be found as follows: Expand the Menu under Material to see the element COATING. As showed in Fig. 2.134, clicking on COATING shown the properties of the dielectric material. In addition, expand the menu under Geometries, followed by the menu under CSG to see the additional geometry element, coating.

🔀 XSim - Sc	attering off a Metal Sphere with a Dielectric Coating					- (	o ×
File Edit T	ools View Help Window						
1	Editor						
X	dielecCoatedMetalSphere.sdf dielecCoatedMetalSphere.pre dielecCoa	tedMetalSphere.in				Simulation Setup is Ready Save a	and Setup
Welcome	Simulation	<b>^</b>	The active tool	Clip Chow Cools Litiou Colide	w la w Bounts	Unu Descertion	1
100	Description		ino active tool	Cip Snow Scale   View Solids	•  +z • Reset v	new Properties	
- <b>V</b>	Constants		á				
Setun	Parameters						
	FREQUENCY						
	- YRGN		8 V				
:	NKCELLS		ata				
Prepare	MALTHICKNESS		<u> </u>				
~	- XENDMAL		2 - x				
- 40 <sub>0</sub>	- YRGNMAI						
Dum	XBGNKIRCHHOFFBOX		2				
Run	XENDTFSFBOX						
	TURNONTIME						
	- TREAKEOY						
Design	TENDKBOX						
	XBGNTFSFBOX						
$\odot$	Basic Settings						
	= SpaceTimeEunctions			4			
Analyze	incidentPlaneWave						
~	Materials			-			
	E Geometries						
Visualize	V metalSphere	_					
	☑ coating			-			
(?)	CAD CAD						
	ends			-			
нер	Field Dynamics			-			
	- Fields						
	Electric Field						
	Current Density	*		-3			
	,			3	4		
	Undo Add Multiple R	emove Add		-3 vet			
	la lui			<sup>م</sup> ليكر.			
	Property Value			4 . <sup>ye</sup>			
				- <b>N</b>			
	1						
Cohen	COMPLETED Click out to continue						Chaulon

Fig. 2.134: Setup Window for the Mie Scattering example.

The Setup Window shows the Kirchhoff box, with the metal sphere at the center. To see the grid, click on the drop down menu of "Grids" on the left side panel. Check the box beside "Grid". In order to view or change the wave frequency, click on the drop down menu "Parameters" and then click on "Frequency". To view or change the direction of the incident wave or polarization, you can click on the drop down menu "Field Dynamics", then the drop down menu "RCSBox". Following this, check the box beside "rcsBox0". The lower left panel displays a table with the wave property and its corresponding value.

#### **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 that you are using these run parameters:
  - Time Step: 3.8115448780607225e-11
  - Number of Steps: 4000
  - Dump Periodicity: 400
  - Dump at Time Zero: Checked
- 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 result is shown in Fig. 2.135.



Fig. 2.135: The Run Window at the end of simulation.

Running in 3D, this simulation uses around 4000 time steps. The run takes about 15 minutes on a 4-core 2.3 GHz processor.

#### **Running the Analyzer**

After completing the run, one can run the analyzer to compute field values at far away points. To bring up the analyser script, Click on the Analyze icon. The panel that appears will be named computeFarFieldFromKirchhoffBox.py, along with a set of text boxes in which the necessary parameters need to be filled. For the default settings of the file you may use the following:

- *simulationName* : dielectricCoatedMetalSphere
- *fieldLabel* : E
- farFieldRadius : 50.0
- numPeriods : 0.25
- numFarFieldTimes : 2
- *frequency* : 0.3e9
- numTheta : 31
- numPhi : 64
- *zeroThetaDirection* : (0,0,1)
- *zeroPhiDirection* : (1,0,0)
- incidentWaveAmplitude : 1.0
- *incidentWaveDirection* : (0,0,1)

- varyingMeshMaxRadius : 1
- *principalPlanesOnly* : checked

After entering the above parameters, also shown in Fig. 2.136, press the "Analyze" button that appears on the upper right side.

XSim - Scattering	off a Metal Sphere with a Dielectric Coating	<u> </u>	- 0 X
e Edit Tools V	iew Help Window		
Analys	is & Controls	Analysis & Results	
Search	computeFarFieldFromKirchhoffBox	computeFarFieldFromKirchhoffBox.py 🛛	
Availat	le Analyzers: Onen	Apply To: Primary Run 💌	tables Size Charles and
	open	Run Automatically After Engine.     Order:	Analyze Stop Clear Output
comp	uteFarFieldFromKirchhoffBox.py	• <b>I</b> •	x that is
otun		simulationName dielecCoatedMetalSphere	General and Phase on Principal Dianes
ecop		este stal la	Elering Power and Finder on Franciscus
		neioLabel je	Part Theory Theory Theory Constraints and the second s
enare		farEleldRadus 50	acaset: Pabsminetacompinetasofiane written.
-parc		De	ataset: PdBsmThetaCompPhiOPlane written.
φ <sub>α</sub>		Use Variable:	ataset: PdBswThetaComoPhi90Plane written.
Run		backgroundEpsRel 1	one writing file: dielecCoatedMetalSphere_PwrAndPhsePrincipalPlanes.vsh5.
		Una Una della Companya della Compa	
		Use variable:	ataset: PdBsmPhiCompTheta90Plane written.
sign		precision double	Parameter Ballow Bb ( 201 and an end and
			acaset: Pubsmphilopiane written.
2		numPeriods 0.25	ataset: PdBsmPhiCompPhi90Plane written.
alyze		Use Variable:	one writing file: dielecCoatedMetalSphere_PwrAndPhsePrincipalPlanes.vsh5.
		numEntipleTimes 2	
		De De	ataset: PdBsmTotalTheta90Plane written.
ualize		Use Variable:	ataset: PdBsmTotalPhiOPlane written.
		frequency 0.3e9	atagat, DdBawTatalDb(00Blane witten
3		D	one writing file: dielecCoatedMetalSphere_PwrAndPhsePrincipalPlanes.vsh5.
lelp		Use variable:	
		numTheta 31	ataset: PhasePhiCompTheta90Plane written.
		Lice Variable:	
			ataset: PhasePhiCompPhiOPlane written.
		numPhi 64 - D	ataset: PhasePhiCompPhi90Plane written.
		Use Variable:	one writing file: dielecCoatedMetalSphere_PwrAndPhsePrincipalPlanes.vshS.
		The bolice time (0.0.1)	
		zerometabirection (0,0,1)	ataset: PhaseThetaCompTheta9OPlane written.
		zeroPhiDirection (1.0.0)	ataset: PhaseThetaCompPhiOPlane written.
			These These Committees and these
		incidentWaveAmplitude 1.0	acase: massimetadomprisofrane wilcon. one writing file: dielecostedHetalSphere PwrAndPhsePrincipalPlanes.vsh5.
		The following variables can be used in the above analyzer options:	computeFarFieldFromKirchhoffBox) Not writing general 1D datasets.
		[1	computerarrieidrromkirchnoffBox] Monostatic power calculation not yet implented.
	1	دة (()	nalysis completed successfully
Dek	ete Import Custom Analyzer		<u> </u>
Analyze: AN	ALYZER SUCCESS Analyzer finished successfully		Show Lor

Fig. 2.136: The Analyze Window at the end of analyzer execution.

#### Visualizing the results

After performing the above actions, click on *Visualize* in the column of buttons at the left. The program will load the data and provide you with certain options.

One of the quantities that is of interest in such scattering phenomena is radar cross section (RCS) measured in dBsm. The RCS, sometimes designated as  $\sigma$ , having units of  $m^2$ , is given as

$$RCS = 4\pi R_s^2 \frac{P_r}{P_i}$$

where  $R_s$  is the radial distance from scatterer,  $P_r$  is the power flux received at the point of interest, and  $P_i$  is the incident power flux. In MKS units the power flux is measure in  $W/m^2$ . RCS in dBsm is given as

 $10log_{10}(RCS)$ 

To obtain plots of this quantity, click on the drop down menu, *Add a Data View*. In this menu, choose 1-D Fields. There is a large list of options to choose from. Figure Fig. 2.137.



Fig. 2.137: The Visualize Window at the end of execution.

#### **Further Experiments**

Alter the thickness or the dielectric constant of the coating to see its effect on the computed RCS.

### 2.4.3 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.

#### **Opening the Simulation**

The Scattering off Multiple Objects example is accessed from within XSimComposer by the following actions:

- Select the  $New \rightarrow From Example...$  menu item in the *File* menu.
- In the resulting *Examples* window expand the XSim 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. 2.138. 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. 2.138: Setup Window for the Scattering off Multiple Objects example.

#### **Simulation Properties**

This simulation includes just one user defined constant, WAVELENGTH, and just two user defined parameters, FRE-QUENCY 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 how the electromagnetic wave can scatter off of different materials.

#### **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 that you are using these run parameters:
  - Time Step: 2.473991708760349e-11
  - Number of Steps: 200
  - Dump Periodicity: 20
  - Dump at Time Zero: Checked
- 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 result is shown in Fig. 2.139.

🔀 XSim - 🛛	Scattering off Multiple Objects			
File Edit	Tools View Help Window			
	Runtime Options	Logs and Output Files		
X	Parameters Run Mode	Run Dump and Stop Force Stop	Clear Log	1
Wolcomo	Values entered here will override values from the simulation file.	Engine Log   File Browser		
weicome	Default Value (1.8295415414691468e-11)	All multiFields dumped at 2024-03-22-10:28:07.112.	<u>^</u>	1
<b>\$</b>	Value: 1.8295415414691468e-11	No electromagnetic rielas to duap.		
Setup	Use Variable:	No ionization processes to dump.		
:=	Number of Steps	Polarin, Publy 10 completed ac 2027-07-22-10-25101.113.		
13	Value: 200 Default Value (1000)	Balarin loop ended at clock time 2024-03-22-10:28:07.113		
Prepare	Use Variable:			
8.	Dump Periodicity	Deleting domain.		
-8	Value: 20 Default Value (100)	There were 0 Notices encountered in this run.		
Run	Use Variable:	See above for more information.		
-	Restart at Dump Number	VORPAL completed at clock time 2024-03-22-10:28:07.182.		
Docion	Value:	NOTE: This simulation can be run if all of the packages ' GPU XSImBase XSImEH' are enabled. Other package combinations may be possible as	vell.	
Design	Use Variable:	No expand symbols file. Will not limit expansion.		
$\Theta$	I Dump at Time Zero	Importing dislecPlusMetalObjs from D:Usersigar)Documents/Tech-X/XSim-1.0/simulations/dislecPlusMetalObjs_EM. Immorting verbosity from C:UPrograms Files/Tech-X/XSim-1.0/dev/Contentslengine/hare/macros.		
Analyze	No Particle Sorting	Importing enums from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
~	Disable Per-Rank Output	Importing mathphys from C:)Program Files/Tech-X)XSim-1.0.0dev/Contents/engine/share/macros. Immorting VSim from C:)Program Files/Tech-X)XSim-1.0.0dev/Contents/engine/share/macros.		
	Custom Run Options	Importing deviceselect from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
Visualize	Reset to Setup Values	Importing issutilities from C:\Program Files\Tech-X\Zim-1.0.0dev\Contents\engine\share\macros. Immorting logical from C:\Program Files\Tech-X\Zim-1.0.0dev\Contents\engine\share\macros.		
		Importing units from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
		Importing utilities from Ci\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros. Importing vutilities from Ci\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
Help		Importing V5imEm from C:\Programs Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
		Importing makes from C:\Program Files\Tech-X)XSim-1.0.0dev\Contents\engine\Share\macros.		
		Importing delayers from C:\Program Files\Tech-XXSim-1.0.0dev(Contents)engine(Share)marcos.		
		Importing matrices from C:\Program Files\Tech=X\X5im=1.0.0dev[Contents]engine\share\macros.		
		Importing ematrix from C:\Program Files\Tech-X X3im-1.0.0dev\Contents engine share\matrix		
		Importing electrostatics from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
		Importing isplicatelectomagnetics from cirrogram risesteen-atometrosum-inconcentorengine smatchmetros. Importing tisfboxid from cirrogram FilesTech-XXSin-i.o.sum-(seque) seque) share macros.		
		Importing grids from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\Share\macros.		
		Importing histories from C: [Frogram Files] Tech-Alson-1.0.0dev(Contents) engine(share) macros.		
		Importing shapes from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
		Importing multifields from C:\Program Files\tech-XiXim-1.0.000V(Contents)engine\snare\textmercos. Importing stfuncs from C:\Program Files\tech-XiXim-1.0.000V(Contents)engine\snare\textmercos.		
		Importing emfilters from C:\Program Files\Tech-X\XSim-1.0.0dev\Contents\engine\share\macros.		
		Lines from 'dielec?iusshetalObja.pre' processed. Finished with 'dielecPiusShetalObja.pre'.		
		END ENGINE OUTPUT		
		Ingine completed successfully.		
		To see results, click on the "Visualize" icon in the icon panel.		1
		IVel Simulation war-clock time was 17.350 Sevenas.	-	1
		<u>U</u> <sup>2</sup>		i
Run	SUCCESS Simulation anning Bnisharl successfully		Chow Lor	Ĵ

Fig. 2.139: 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, as well as the geometries as shown in Fig. 2.140, do the following:

- Expand Scalar Data
- Expand E
- Select *E\_z*
- Check Clip Plot
- Set the color limits to -2 and 2
- Expand Geometries
- Select and check Clip Plot for: AluminaObjectTriangles, PECObjectTriangles, sapphireObjectTriangles

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.



Fig. 2.140: Visualization of the wave as it hits the objects.

#### **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.

## INDEX

# Α

antenna on hand, far field, radiation, dielectric, 17 antennas antennaArray2D, 8 Applied Magnetic Field, 54 2.4 GHz Yagi Uda, 3 Antenna on Predator Drone, 22 Dish Antenna, 35 S-Matrix of Box Cavity, 92 Scattering off Multiple Objects, 144

### В

Boundary Launcher Mie Scattering Dielectric Coated Metal Sphere, 140 Mie Scattering Metal Sphere, 135

# С

Cells Per Wavelength Mie Scattering Dielectric Coated Metal Sphere, 140 Mie Scattering Metal Sphere, 135 Charge Density, 54 2.4 GHz Yagi Uda, 3 Antenna on Predator Drone, 22 Dipole Above Conducting Plane, 28 Dish Antenna, 35 Half-Wave Dipole in Free Space, 38 S-Matrix of Box Cavity, 92 Scattering off Multiple Objects, 144 coaxial cable, 47 coaxial waveguide, 47 Complex Electric Field Dielectric Waveguide Mode Calculation, 97 Dielectric Waveguide Mode Calculation, Roughness, 111 Three Material Waveguide, 129 Complex Magnetic Field Dielectric Waveguide Mode Calculation, 97 Dielectric Waveguide Mode Calculation, Roughness, 111

Three Material Waveguide, 129 CSG Geometry Pillbox Cavity, 70 S-Matrix of Box Cavity, 92 Scattering off Multiple Objects, 144 Waveguide Dispersion, 58, 80 Current Density, 54

# D

Dielectric Coating Mie Scattering Dielectric Coated Metal Sphere, 140 Dipole Current Dipole Above Conducting Plane, 28 Distributed Current, 54 2.4 GHz Yagi Uda, 3 Antenna on Predator Drone, 22 Dipole Antenna, 31 Dish Antenna, 35 Half-Wave Dipole in Free Space, 38 Pillbox Cavity, 70 S-Matrix of Box Cavity, 92 Waveguide Dispersion, 58, 80

### Е

Electric Field, 54 2.4 GHz Yagi Uda, 3 Antenna on Predator Drone, 22 Dipole Above Conducting Plane, 28 Dipole Antenna, 31 Dish Antenna, 35 Half-Wave Dipole in Free Space, 38 Pillbox Cavity, 70 Ring Resonator, 106, 118 S-Matrix of Box Cavity, 92 Scattering off Multiple Objects, 144 Waveguide Dispersion, 58, 80 Electric Field;, 124 Electro Magnetic Solver Mie Scattering Dielectric Coated Metal Sphere, 140 Mie Scattering Metal Sphere, 135

External Magnetic Field Half-Wave Dipole in Free Space, 38

#### F

far field, 43, 50
Far-Field Box
 2.4 GHz Yagi Uda, 3
 Antenna on Predator Drone, 22
Field at Position, 54
 Antenna on Predator Drone, 22
 Pillbox Cavity, 70
 Ring Resonator, 106, 118
 Waveguide Dispersion, 58, 80
Field Boundary Condition, 78
Frequency
 Mie Scattering Dielectric Coated Metal
 Sphere, 140
 Mie Scattering Metal Sphere, 135
Function, 54

### Η

hornAntenna, 43

### I

Imported Geometry Antenna on Predator Drone, 22 Dish Antenna, 35

### Μ

Magnetic Field, 54 2.4 GHz Yaqi Uda, 3 Antenna on Predator Drone, 22 Dipole Above Conducting Plane, 28 Dipole Antenna, 31 Dish Antenna. 35 Half-Wave Dipole in Free Space, 38 Pillbox Cavity, 70 Ring Resonator, 106, 118 S-Matrix of Box Cavity, 92 Scattering off Multiple Objects, 144 Waveguide Dispersion, 58, 80 Magnetic Field Intensity, 54 Dipole Antenna, 31 Magnetic Field: 124 MAL Damping Factor Mie Scattering Dielectric Coated Metal Sphere, 140 Mie Scattering Metal Sphere, 135 Matched Absorbing Layer S-Matrix of Box Cavity, 92 Mode Extraction, 54

### Ρ

patchAntenna, 50

Perfect Electric Conductor Dish Antenna, 35 Mie Scattering Dielectric Coated Metal Sphere, 140 Mie Scattering Metal Sphere, 135 Scattering off Multiple Objects, 144 Phase Shifting Boundary Conditions Waveguide Dispersion, 58, 80 Phase Shifting Periodic, 54 Phi, 54 2.4 GHz Yagi Uda, 3 Antenna on Predator Drone, 22 Dish Antenna, 35 Half-Wave Dipole in Free Space, 38 S-Matrix of Box Cavity, 92 Scattering off Multiple Objects, 144 Poisson Solver, 54 2.4 GHz Yaqi Uda, 3 Antenna on Predator Drone, 22 Dish Antenna, 35 Half-Wave Dipole in Free Space, 38 S-Matrix of Box Cavity, 92 Scattering off Multiple Objects, 144 Port Launcher Ring Resonator, 106, 118 Scattering off Multiple Objects, 144 Port Launcher;, 124 Pseudo-potential S-Matrix of Box Cavity, 92

### R

radiation, 43, 50
Radius of Sphere
 Mie Scattering Dielectric Coated Metal
 Sphere, 140
 Mie Scattering Metal Sphere, 135
Rectangular Waveguide, 78
rectangularWaveguide, 78

### S

sectoralHornAntenna, 43 SpaceTimeFunction, 54 2.4 GHz Yagi Uda, 3 Antenna on Predator Drone, 22 Dipole Above Conducting Plane, 28 Dipole Antenna, 31 Dish Antenna, 35 Half-Wave Dipole in Free Space, 38 Mie Scattering Dielectric Coated Metal Sphere, 140 Mie Scattering Metal Sphere, 135 Pillbox Cavity, 70 Ring Resonator, 106, 118 S-Matrix of Box Cavity, 92

```
Scattering off Multiple Objects, 144
Waveguide Dispersion, 58, 80
SpaceTimeFunction;, 124
```

### Т

Time Dependence Dipole Antenna, 31

### W

Waveguide, 54