All right. Welcome back, everybody. For our third talk of TWSS 2021, our annual user conference here at Tech-X, I'd like to introduce Dr. Svetlana Shasharina. She is one of the co-founders of Tech-X, and one of the originals since 1994. And she will be speaking on using VSim for photonics and the workflow involved with that. So on that note, Sveta welcome.

Thank you. Can you hear me?

We can hear you.

Okay. So I'll be switching between the PowerPoint presentations and live demos of VSim. So first of all, if you have any questions after this presentation, please address them either to me or just using the email for sales@txcorp.com.

So the overview of my presentation, I'll briefly introduce what photonics is. Then I'll talk about how to set up a simulation of photonic devices using VSim, how to calculate modes of such devices, how to run simulations, how to visualize results, and do analysis, particularly how to calculate S-parameters.

So, um, I don't know how people how many people in the audience are familiar with photonics. But generally speaking, the photonics is about creating, manipulating and detecting light. And silicon photonics plays a particularly important role in photonics because silicon is already used in traditional electronic design automation, in traditional manufacturing of integrated circuits. And the same, fabs can be used to fabricate this new type of devices made of silicon. Quite often when people talk about silicon photonics, they also imply other materials, dielectric materials, for example, silicon nitride, so that's about photonics that are outside of VSim.

When you look at VSim, the, in practicality photonics is pretty much electromagnetic simulation using the dielectric materials, and the telecommunication wavelength approximately around 1.5 micron. The applications of photonics are enormous. First of all, the data rates which are requested by modern telecommunications and data centers cannot be supported by metal anymore, so you have to switch to new kind of switches and transmitting media. So basically, the planning system as for 5G applications, for optical computing and for all kinds of telecommunications, for data centers, for self-driving cars, using LIDAR sensors. So that said, that’s why I'm so excited about photonics.

Let's switch gears and talk about how photonics can be modeled in VSim. First of all, the structure of the setup in this sim consists of several elements. The first one is basic settings, then geometry, materials, light source and diagnostics.

So basic settings, let me show you first of all, how it looks in VSim. So basic settings, if you open the same, you will see two, it's the Setup tab, and you will see the preview of the setup and you also see the visualization of the setup. So basic settings go under here, and you'll see multiple things there. But of particular interest, especially for the novice user of VSim, would be dimensionality, which is typically three for this simulations. Then, background permittivity by background, broad, background permittivity we mean the material which surrounds all the visible shape. For example, if you have a silicon wire surrounded by air, you don't want to create the cube of air you just specify the background permittivity being one, or if you have silicon embedded in silica CO2, you specify background permittivity to be corresponding to silica.

Another interesting setting here is the electric solver. And you can see that we have two types of the electric solver here. It's permittivity averaging and point permittivity. To understand what it is, I created these two pictures here. This is a human head which is represented by the point permittivity, which is pretty much stair stepping representation of the boundary. And the average permittivity which creates the smooth representation of the boundary. The smooth representation for the boundary is much more reliable, but it takes more memory. And so you can try uh, playing the point permittivity if you want just quick and dirty simulation and see how everything is going. But for second order accurate results, you will need the permittivity averaging. So, this is the base main basic settings to pay attention to. What's the next step in setting up the simulation. A geometry in VSim can be created either by hand in the setup or can be important as a CAD file. By manual creation I mean just using the primitives which are provided by VSim GUI and I'll show you them in a second. And, and you can also manipulate on them by creating arrays of them like you can create three dimensional, two dimensional, one dimensional array of these primitives. You can also operate on them by applying Boolean operations such as intersections, unions, and subtractions. But for photonics simulations of the most important form of geometry is presented by GDS files. GDS files is the kind of inheritance from electronic design automation and they are, it is a binary format which describes the etching on the wafer. So they are inherently two dimensional. But the the structures which are modeled by VSim are three dimensional so we basically extrude this GDS files to create three dimensional structure. Let me show you an example of GDS files here. So let me get rid of this one. And let me show you a pretty complicated example. This is, this is the example of GDS file which consists of pretty much 29 layers, it was just important in VSim, and you can further edit it by right clicking on the shape, and you can see, edit GDS layers, and you after that this window will appear. And you see all the layers by name, and you can edit z location, which is a start, you know basically the lower coordinates of the layer, the thickness, and you even can edit this sidewall angle here, which is a new feature and I will not dare to do it in the demo mode. So you can also assign materials to each layer if you wanted to. So that's how you edit the 3 dimensional structure here. You can also, it's a pretty heavy it takes a little bit of time. So you can also play with GDS Boolean operations, and when you click on GDS Boolean operation, you can see that you can operate on any layer, you can click on any number of layers of all them evaluate available shapes here and you can do intersection of layers, union, subtraction, and XOR operation. So, why it is useful, some customers whose name I cannot really pronounce, challenged us with the GDS files, which in three dimensions when you try to interpret them in three dimensions, you need to create something which is called Deep edge by intersecting two layers and the shallow edge by subtracting these two layers. So specifically for this customers we created this tool and which allows us to create this changing basically they changed the depth of the etching if needed.

So returning to my presentation here. So, another way to create geometry is to use the primitives which are provided by VSim. For example here I show the ring resonator and the ring resonator is a structure which consists of the input waveguide, the ring, the output waveguide. So basically, you put all you shoot the light at the bottom, it goes through this waveguide that gets captured by the ring and it comes outside of the output ports. So to create the shape For this circle, for example, we just use two cylinders. And we did subtraction on these two cylinders, and how that's how the ring was created. So if you are working with primitives, you can create a bunch of them. You can. just show you a little bit how to do that here geometry, you can just add any number of primitives here and to play with them. But once again, GDS probably is more important for the workflow, relevant to workflows. proceeding. So I already talked about GDS here. Examples of GDS geometries are shown here. Here, on the left, you see the so called y splitter, which is you know, you can see the input here and the light gets split in two directions. A more advanced more interesting device is shown on the right, it's called arrayed waveguide grating it's multiplexing demultiplexing device. And it's it's of use for switches and for telecommunications, you can start with a signal which comes from the top here and consists of multiple wavelengths. And due to the very interesting intricate structure of this waveguides the signal gets split into outcoming waveguides and each waveguide will have a single frequency. So, this is this is why photonics is very interesting, because it allows you to pack a lot of information on one signal and demultiplex at the end. Boolean operations, I showed them a little bit quickly and I just documented everything on the presentation for your perusal later. So, once you create the geometry, you need to assign materials to geometry switching to VSim, we can see the material database and we will switch from the regular view to material database by clicking here. And you can see that VSim comes with several predefined materials. If your material is not here, you can always use custom material and just add it to the simulation in custom material. Then have you can just for the photonics simulation, you can define conductivity, relative permittivity and and you have to go you can even create silicon nitride for example here and assign it to whatever you need. So assigning of materials happens here, let's by clicking on the shape, and after the double clicking on the material here, and you can assign material, you know it was pretty pre-assigned before silicon, but you can be assigned to something else here. So that's how you assign materials in VSim. So, now we have a geometry set up and the geometry, all geometries have materials. So what is next for the photonic simulation is specification of the light source. And light sources in VSim can be created by using a field basically electromagnetic fields or currents. And each source will have a two components to that, it will have the time dependency component and the space distribution component. For the time dependency, you can use a bunch of pre-built functions, and I'll show them in a second. For the space distribution you also can use pre built functions. For example, for many examples, we use just simple Gaussian distribution. Or you can provide an external file with a profile of a mode for example, let me show you on this example that we have here. So this example, has external mode launching field. And you can add this by just right clicking on the field. And you can add field and you can do one more external mode watching field that's how this thing is ended up here in the simulation. And if you look at the parameters of this external mode launching field, you will see that first of all it says description says it's 2D field and this 2D field is located right here it's visualized the temporal variation is pulse. And before that I created this pulse which was a specified as a

One of the available functions here and I used the so called sinks, sinc hat function and sinc hat function is the function of the pulse, which, if you transform it using Fourier transform, you'll end up with the wide frequency band which is limited by the min and max frequency. So, so you can see that the pulse is specified here, by the low frequency, high frequency and other things, you can specify amplitude which is not relevant because everything will be just multiplied by this amplitude. There's other things suppression factor and frequency gap, which I don't recommend to change at this point, I will just use them as they are. So, this is the pulse in the external mode launching field. But this space, this space profile of this source, which is specified here, is specified by the HDf5 file. And the name of the file here is ringResonatorMode\_EigenD\_0.vsh5. So how did I how do I find this mold how I do create create this file, you can do it by using the new feature of VSim, which is called prepare, it's the prepare tab. So let's say set up. And the prepare tab here is it's saving it. So in the future, we could actually imagine multiple types of prepare steps. But right now what we have only one type of prepare setup in and it's just calculates the mode. So you can specify to do this prepare, prepare step, anytime you run the simulation. And here, I just do it once. And you can specify number of modes you're interested in, and the number of and the field that you're interested in, you can specify more, I know that the space I know that the mode of the single waveguide is very simple. And we just need only one mode to calculate the, to find what we need. So once I won't go to do it in real time, because it will take just a little bit too much time, not too much but you know a couple of minutes. But once you specify what you want to find you click on prepare button, the under the hood, VSim will run simulation for one step, it will create the permittivity profile, then it will run the mode calculation for this profile create the file for you. And now you are ready to go because this file for you will be created at this point. So returning to the PowerPoint. So I went through this already, it's just looking at it for your inspection after the presentation you basically using that sinc hat function to create wide band frequency signal, then you add external launching field then you run the Prepare step Create the profile and you're ready to go. But it will not do anything good to you if you don't specify any diagnostics and diagnostics of, the, of interest for photonics are listed here.

So first of all the terminology what what we call history in VSim is, is a diagnostic which combines uh, combines data from each time step. But some simulations of VSim as really, really huge. And we don't want to collect data from the all grades available for the simulation. And we create the smaller sub sections of the simulation for example, little planes or points and we collect data of subset of simulation in such diagnostic which collects data from each time step in a subset of simulation its called a history in VSim. So the history is of interest for photonics or electromagnetic field on the plane, field in a position and point, pointing flux on the plane. So to add them to the simulation, you just scroll down here, go under history, right click on history and you can see all the histories which are of interest to you here. So when I was creating the simulation, I did exactly that. And I added several EM field on a plane. And you can see them here. Let me enlarge them here. So we have uh, 1, 2, 3, 4, 5 uh, histories on the plane. And we're collecting the electric and magnetic field on this plane. And using this electric and magnetic field on the plane we'll calculate the Poynting flux in each of these planes, and then integrate this poynting flux over the plane and translate it to the frequency domain to calculate the so called S parameters. So once the simulation is set up, you go back to what we have here. So we have everything set up, we have geometry, materials, we have the launching field, the source, we have the diagnostics of interest to us in the four slabs here. And so we ready to run, we can go to run panel. And that's how the run panel in VSim looks like. So we have, we can switch between the parameters of the run panel and the Run Mode. And parameters here are the time step. And VSim calculates the timestamp for you. So I would recommend not to play with it, just use the default. Or if you actually want to, to risk a little bit and calculate say, with bigger time step just to see what happens, you can create your own custom time step here. And you can just assign time step from the previously calculated things in the setup here. Then, for the number of steps for the photonics simulation, first, you can just do, you know, basically propagation of the wave till the end of the device. But if you need to do the frequency domain analysis, you need to wait till the signal pretty much dies out. And a good rule of thumb here would be to calculate number of steps that you need to reach the end of the device, you multiply approximately by 10. And here's your number of steps that will give you reasonable S parameters at the end. So now you specify the parameters of the run time, then I would recommend if you have a parallel machine to run in parallel, for example, right now I have a number of processes available for these six. So I will run with six processes here. And of course, I will not run 60,000 steps in front of you, I already pre ran the simulation. And once the simulation is run, you will see this magic, magic thing at the end, you ready to visualize it. And so I typically go to visualize panel here. So when you switch to visualize panel, you can add as many window, Windows as you want. And I added several here, and I would recommend for each type of data that you have, for example, you might have three dimensional data for the simulation results, you might have two dimensional data for the mode, you might have one dimensional data for the S parameters, I would recommend to use separate windows because it's just clear for you what you are visualizing here. So on this one though, I edit data overview and it's three dimensional or with you and you can see that I clipped the plot. And I use the you just one plane here. And it's a plot.

It cuts in pretty much. Basically, basically simulation you can see just in one plane cut. So if you switch to the time different time steps, you'll see how the electromagnetic field propagates and this device. This is Dz. Dy is typically more interesting here, because I need to clip it. So this is a really useful debugging tool. And it's also very useful for creating nice PowerPoint presentation, a little eye candy for fund, funding agency, or your clients. So I like to play with this. Uh another way to look at the simulation data would be to say to look at the materials, which we created by the simulation, and you can do it by plotting inverse epsilon. And doing a couple of cuts here, which I created previously here. So I created the cut on two planes. And once again, it's very useful for debugging. And for other purposes as well. Another thing I created here is two dimensional, basically, I, if you remember, I pre calculated the mode. And here I just visualize the mode here. It's two dimensional data. And it's useful to have a special view for that. So you can play with this you know, let me here, see here, what it did here. So here I just plotted the external mode launching field, you can slightly see it here, it's just right there. And I wanted to make sure that this launching field is going to the input waveguide. So in addition to that, I plotted the input waveguide geometries. So visualization is very useful for debugging. And you can also play with individuals controls here. So each port will have individual control for opacity. You can also specify different color schemes, whichever is pleasing to you. You know, some people which are more artistic, they like the bright ones, some people more scientific, they like magma visualization tool, and so forth. So this is visualization. So we have the setup, we run the simulation, we visualize the results, but what's the most relevant for photonics is the calculating of S-parameters, and I'll switch to S parameters here. So, first of all people who are not familiar with S, S parameters. S parameter is the ratio of in, of two integrated fluxes basically, you have input and output port and you specify the dimension of this ports as just two rectangles, and you calculate, integrate the power flux going through this ports, you translate it into frequency domain using Fourier transform. And after that, you just calculate and you can define as many S parameters as you need, depending on how many how many rectangles or monitors you put on your simulation. So, to calculate a S parameters in VSim, you go to analyze panel, and and you see that we have tons of analyzers, which can, which come with VSim. And the one which is of interest to you is called computeSParamsOverlapIntegral.py. So you find this thing in the list over here, and you click on open here and I already opened before. And once you click on Open this, this analysis script will show its input variables here on the left. And so, basically, to, to perform this analysis, you have to specify the minimum, the minimal wavelength and the maximum wavelength in the vaccum, which is of interest to you, then you specify the input and output slab here. And the rest is for more intricate analysis, and I will not just describe it. At this point, by default, you just use all number of steps available to you. So that's why you don't have to specify anything, then you do click Analyze. And it's just, that's analyzing for you, it just running,

running for you right now, but I just see that I made an error which I'm not going to show to you. So I pre-ran it before and I loaded this data into VSim, and this data appears as 1D fields. And here you see the plots which are pretty typical plots for S parameters calculations for the ring resonator. So on top, you see the, let me show it in the setup. What's going on here basically, as what you see on top is the S matrix between this input and this output. And what you see on, on, on the bottom here is using this output and this output here, at the top here, so this is a good result. And that shows the resonance character of this ring resonator device. So um, so basically that's all I wanted to show you at this point, and you're welcome to ask questions.

Looks like we have any questions? Yeah, looks like we have one question. Asking about the prepare tab. The prepare tab looks really interesting, what do you expect that will be used for.

So, you know, basically, the big vision here for prepare tab, right now, we use just to calculate mode, but I can imagine that we have automated workflow. So prepare tab will be something which will prepare a simulation, which is always needs to be done to, to for example, into optimization loop, if you change any parameter in the setup, you move one ring versus, you know, versus the some other geometry. So it changed anything for setup, you have to recalculate the mode. So the prepare tab, if I click here, always run before the engine run will automatically recreate this mode or do any other pre calculation we need to do in the optimization loop. Similarly, in the Analyze, what I envision here would be another little button saying, always run this post processing step. When you run a simulation, it means that I can then do optimization loop, extract the, uh, I finished the simulation, automatically extract the figure of merit, feed it to the optimization engine, and it will then automatically conclude the optimization loop. So basically this prepare tab and the future automated post processing run, which enable us will enable us optimization.

It looks like we have another question that says about GDS exports. Is there anything under consideration on surface roughness simulations? On what excuse me? On GDS exports? Yes,

so right now, what we can do with GDS experts, we import GDS, then we can do Boolean operations, and we can export it. So that's what we can do. In the future, I believe that we should be able to manipulate the polygons for optimization, because optimization is basically the next step for us and exported into GDS, because GDS is pretty simple format to write.

Great. Other questions? We have. We have Sveta for just a little bit longer. If anyone has anything else they want to type in the q&a. I think that might be it for what we have for today. Okay, thank

you so much. No, thank

you so much for such a good presentation. We will be back tonight at 530 Mountain Time for Dr. Salvador Sosa and Trudy Bolin from the University of New Mexico. And again, if anyone is interested in evaluating our software, please visit the tech x website at txcorp.com and we can get you the latest version for 30 days. Thank you so much and we'll see you all in a few hours.