Fundamentals of Integrated Photonic Components
Fall semester 2025-2026

Instructions for CST simulation
Step-by-step guide

This manual is meant to guide you through the creation of a simulation environment in CST and the extraction of all the relevant results. Read the document carefully before performing a simulation and feel free to give us feedbacks on its organization.
Part 1 of the tutorial focuses on the **creation of the template**: you will select the physics of your problem and specify the wavelength (or frequency) range of the simulation.
Part 2 guides you through the **assembling of the geometry** and the **definition of the parameters of the solver**: you will define shapes, materials and boundary conditions for the geometry, create field sources and field probes, and finally setup the solver by specifying inputs, outputs and which type of calculation CST should perform.
Part 3 explains how to **analyze the results** of the simulation: you will see how to create 1D, 2D and 3D plots and how to export results to be post-processed with other softwares.

Contents

Access CST through VDI 1
1. Creating the template 2
2. Building the geometry, source(s) and boundary condition 6
2.1 CST environment and parameter list 6
2.2 Basic geometry creation and transformation 7
2.4 Creating 3D object by extruding 2D geometry 9
2.5 Material assignment 11
2.7 Defining the background 13
2.8 Waveguide port and other sources 14
2.9 Boundary conditions 16
2.10 Mesh 17
2.11 Post processing, probes and monitors 18
2.12 Setup solvers: time domain and frequency domain 21
2.14 Parametric sweep 24
3. Plotting/extracting the results: 25
Appendix: Tutorial of creating bend waveguide 27

Access CST through VDI

You will perform simulations on a virtual machine. Follow these steps to access it. If you have troubles at some point, don't hesitate to tell us, it might be because you are not added to the list of the course.

1. Install VMware Horizon from https://vdi.epfl.ch/ (might be already installed on the computers in the exercise classrooms.

2. When you connect the first time on the Vmware Horizon you must enter "vdi.epfl.ch" for the **Server**.

3. Login with your Gaspar credentials and you should be able to see the **STI-CST-HP**.

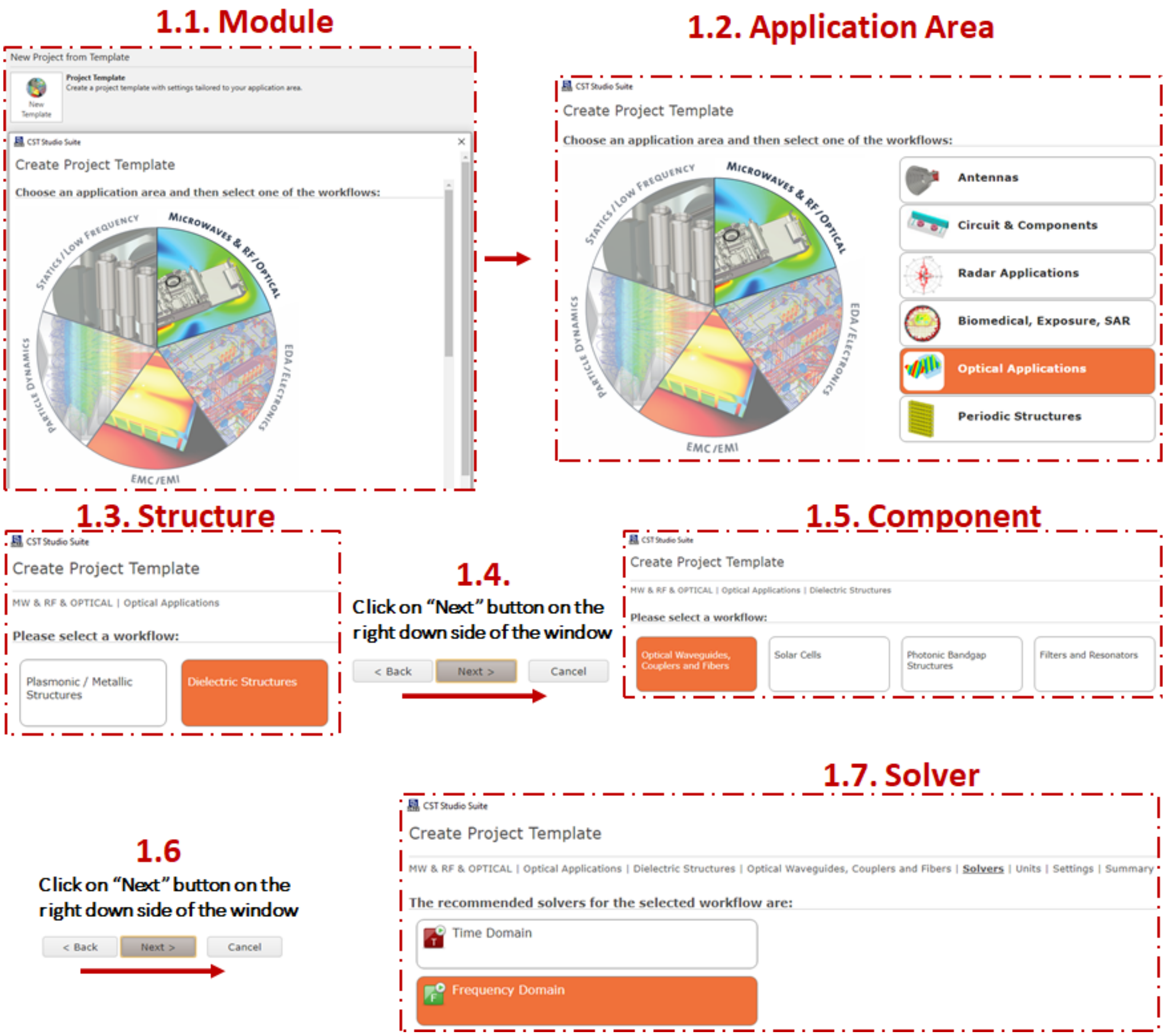
4. Enter STI-CST-HP and search for CST in the program list.

BE CAREFUL: data saved locally on the server **will be deleted**. Remember to save them on a network or through external storage places such as Drive or Dropbox.

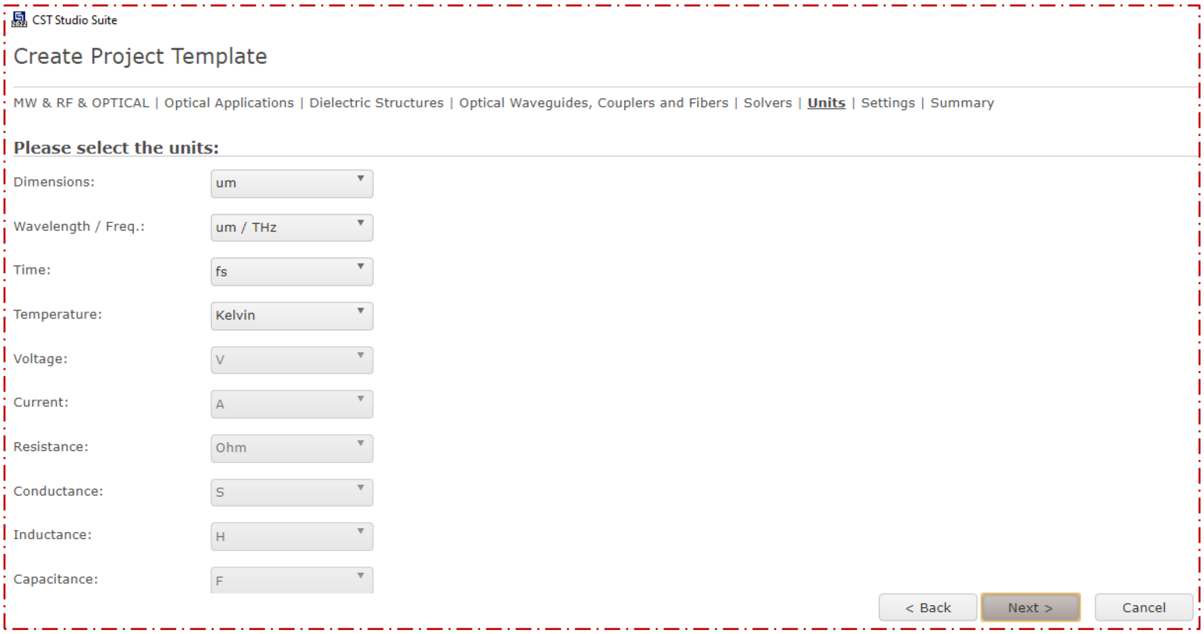
1. Creating the template

To perform simulations in CST, the first step is to define the simulation template. You can follow the steps below, where you will find explanations in the text as well as screenshots of each step in the following pages. Additionally, you can watch the video tutorial provided by the CST Microwave Studio software itself.

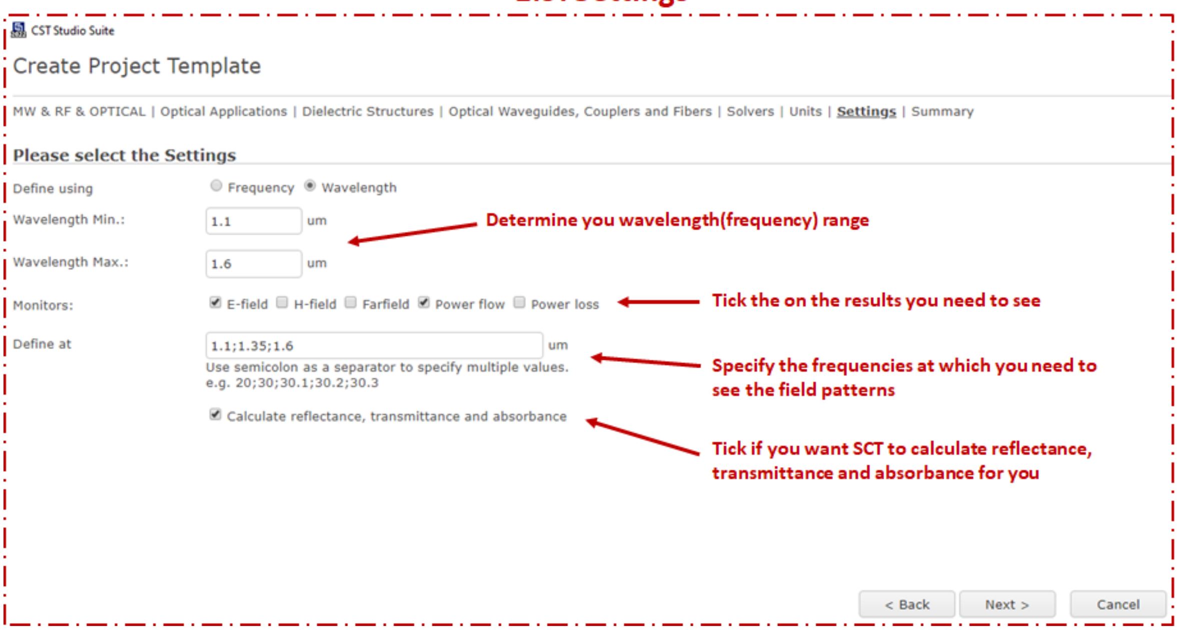
- Steps 1.1 and 1.2: Click on **Project template**. You will see different frameworks to create your model. For simulating all the structures we will analyze during this course, we should select **Microwaves and RF/Optical** then **Optical Applications** framework, then click on **"Next"**.
- Steps 1.3 and 1.4: After choosing this framework, you can now choose **Dielectric structre** to simulate optical waveguides, while in the future exercises we will use **Plasmonic/Metallic structures** for the analysis of bow-tie antennas.
- Steps 1.5 and 1.6: You have now to select the component category: for the dielectric slab waveguide you can choose **"Optical waveguides, Couplers, and Filters"**, while in the future antenna exercise you will choose **Nano antennas** (under the Plasmonic/Metallic structures menu).
- Step 1.7: At this point you can choose the solver: either **Time domain** or **Frequency domain** solvers depending on the type of simulation you want to run (usually specified in the text of the exercises). The latter is usually preferred when you need to perform a simulation for a fixed frequency and not a range as for the other. You will also be able to change the solver later in the software.
- Step 1.8: Choose the desired units for your simulation. We suggest **um, THz and ps** for wavelength (and geometrical dimensions), frequency, and time respectively.
- Step 1.9: Enter the wavelength (or frequency) range in which the simulation will be performed (usually suggested in the text of the exercise). In the **Monitors** section you can choose the results you want CST to calculate, such as electrical or magnetic fields. In the **Define at**, you should specify the calculation wavelength (or frequency) for these monitors. Note that monitors also can be added or modified later in the software.
- Step 1.10: Finally, you just give your template a name! ☺


1.1. Module
New Project from Template
Project Template
Create a project template with settings tailored to your application area.
New Template
CST Studio Suite
Create Project Template
Choose an application area and then select one of the workflows:
Statics / Low Frequency
Microwaves & RF / Optical
EDA / Electronics
EMC / EMI
Particle Dynamics
1.2. Application Area
CST Studio Suite
Create Project Template
Choose an application area and then select one of the workflows:
Antennas
Circuit & Components
Radar Applications
Biomedical, Exposure, SAR
Optical Applications
Periodic Structures
1.3. Structure
CST Studio Suite
Create Project Template
MW & RF & OPTICAL | Optical Applications
Please select a workflow:
Plasmonic / Metallic Structures
Dielectric Structures
1.4.
Click on "Next" button on the right down side of the window
< Back
Next >
Cancel
1.5. Component
CST Studio Suite
Create Project Template
MW & RF & OPTICAL | Optical Applications | Dielectric Structures
Please select a workflow:
Optical Waveguides, Couplers and Fibers
Solar Cells
Photonic Bandgap Structures
Filters and Resonators
1.6
Click on "Next" button on the right down side of the window
< Back
Next >
Cancel
1.7. Solver
CST Studio Suite
Create Project Template
MW & RF & OPTICAL | Optical Applications | Dielectric Structures | Optical Waveguides, Couplers and Fibers | Solvers | Units | Settings | Summary
The recommended solvers for the selected workflow are:
Time Domain
Frequency Domain

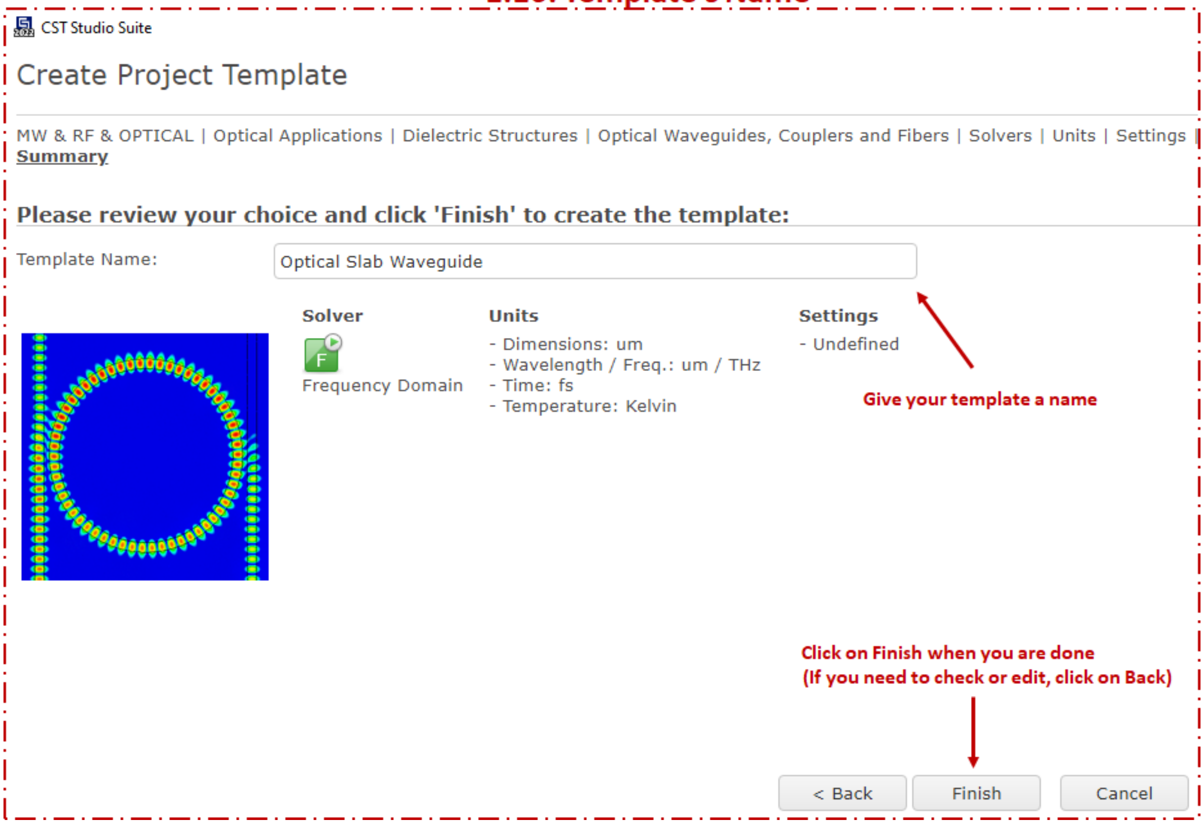
1.8. Units



1.9. Settings



1.10. Template's Name

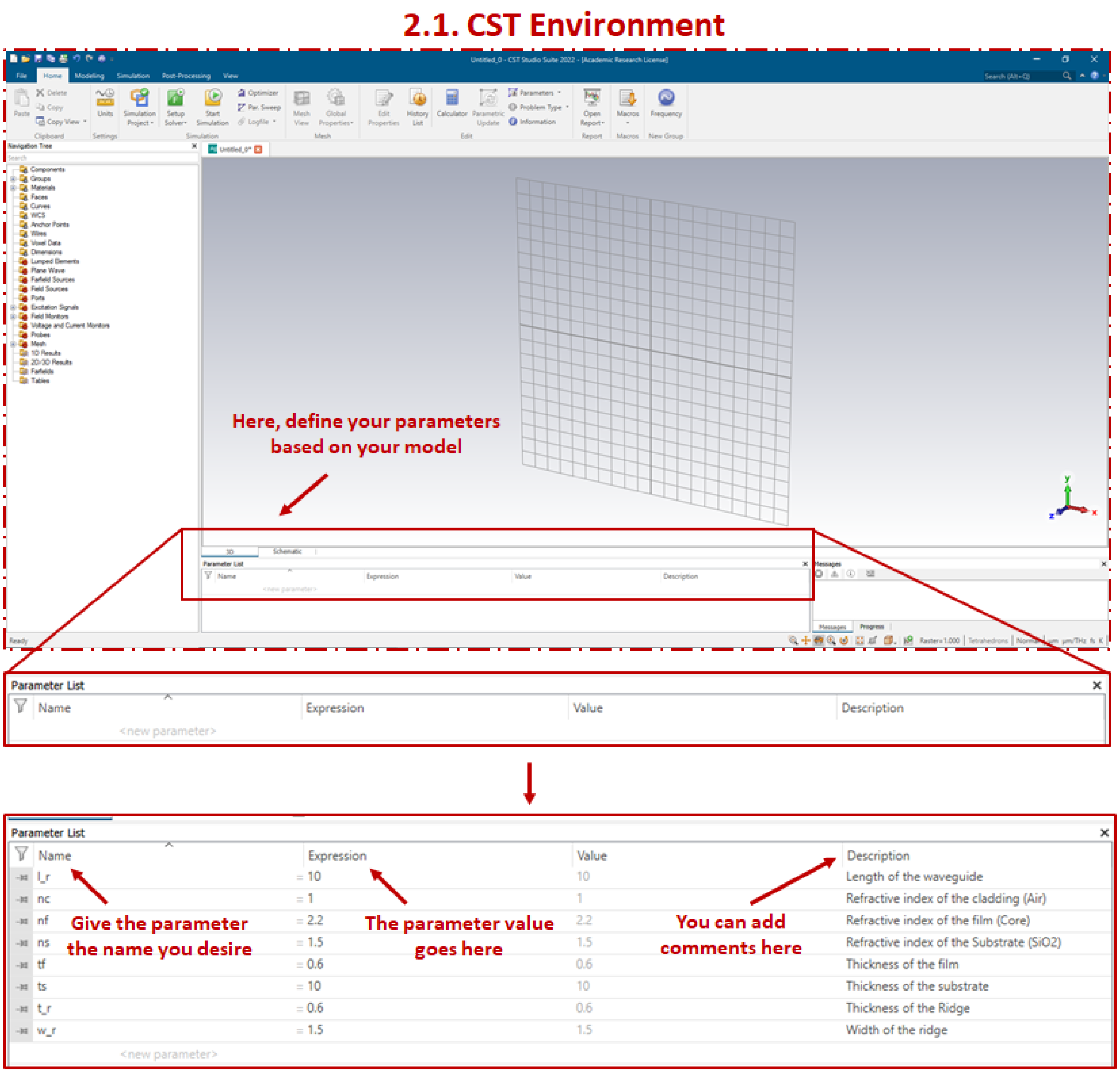


2. Building the geometry, source(s) and boundary condition

Disclaimer: most of the operations here described can be easily performed from the **Navigation Tree** panel found by default on the left of the screen (see the figure in point 2.1). You can right-click on an element of the tree and see the list of available actions you can perform.

2.1 CST environment and parameter list

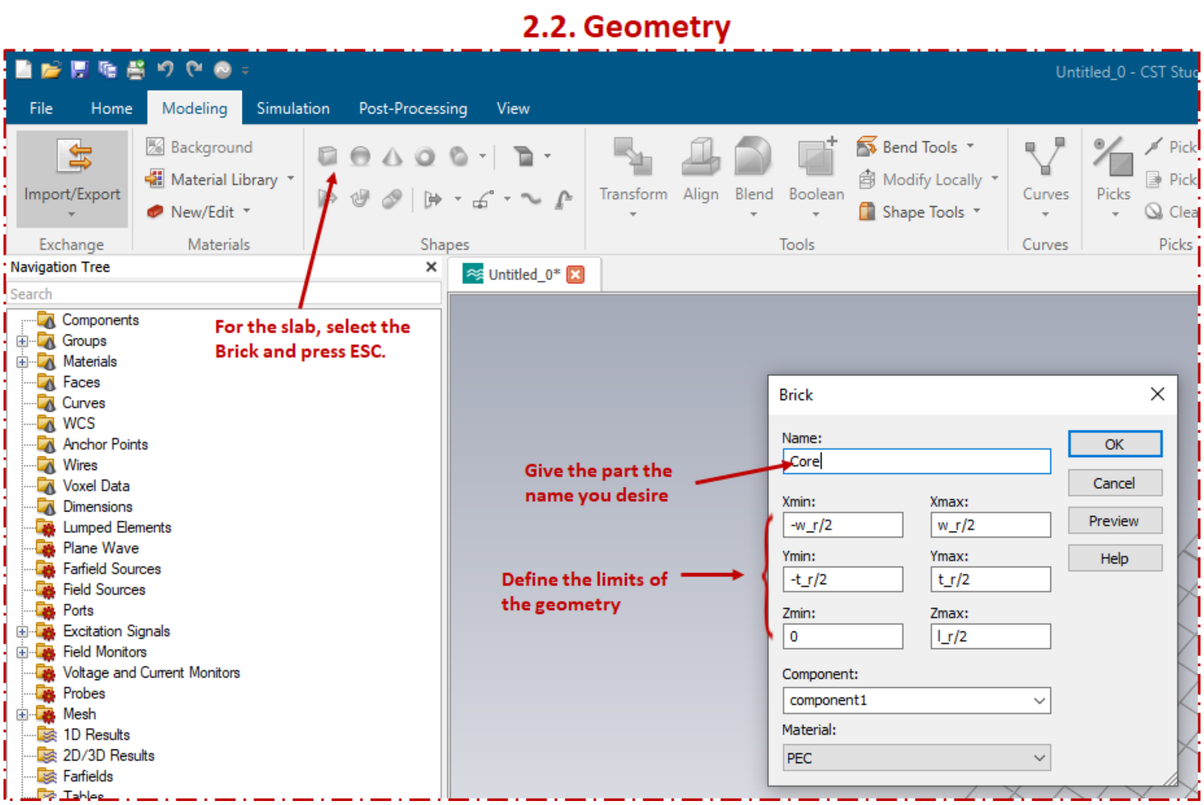
- 2.1) You can now start with the creation of the geometry. To do so, you need to define the parameters of your design in CST. By opening the **Parameters List**, you can specify the **Name** of your parameters, their value under **Expression** (the unit of measure is the one specified before during the creation of the template), and a short comment under **Description**.



2.2 Basic geometry creation and transformation

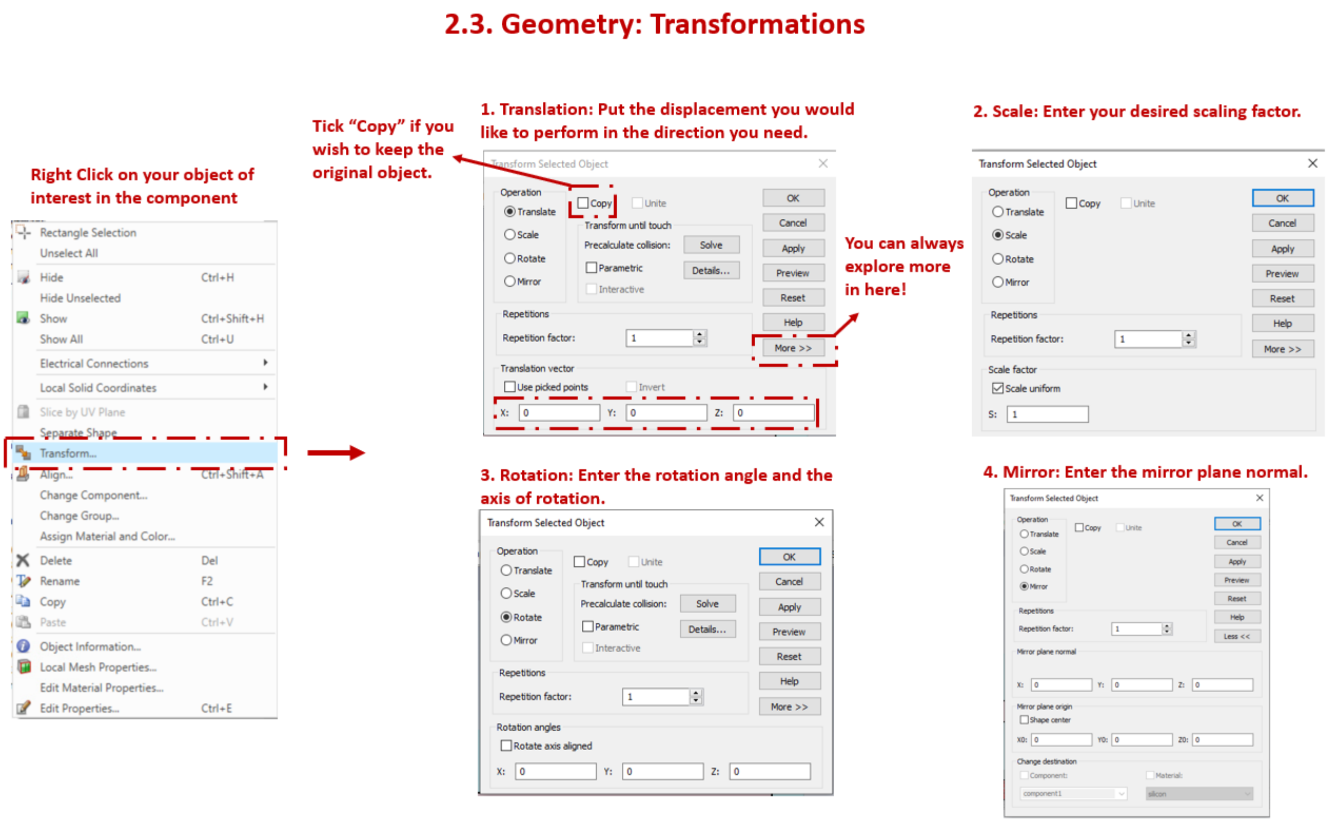
- 2.2) For creating our geometry, we take the slab waveguide exercise as an example, which involves basically a rectangular film. In **Modeling**, click on **Brick** and press **Esc** button on your keyboard to open the **"Brick"** interface. Give your film a name, and define its minimum and maximum coordinates along x, y and z axes. Finally, assign a **Material** to the brick following the guidelines in point 2.5. You will be able to assign a different material also later, from the Navigation Tree.

  (In case a newly generated brick had an intersection with the another one created before, CST would ask you the operation to perform between them: usually, you would need to select the **Trim** option, which will insert the first brick inside the new one but correctly keeping the assigned materials. Many other geometrical options are available for the creation of 3D or 2D structures, as you will also see in steps 2.3 and 2.4. Check the instructions at the **end of the document** for an example on the **creation of a bent waveguide**.)



- 2.3) CST allows to perform geometrical transformation on an existing object. As shown below, you can perform **Translation**, **Scale**, **Rotate**, and **Mirror** on you exiting object. So you don't need to create new objects with the same geometry, only to change their position or scale

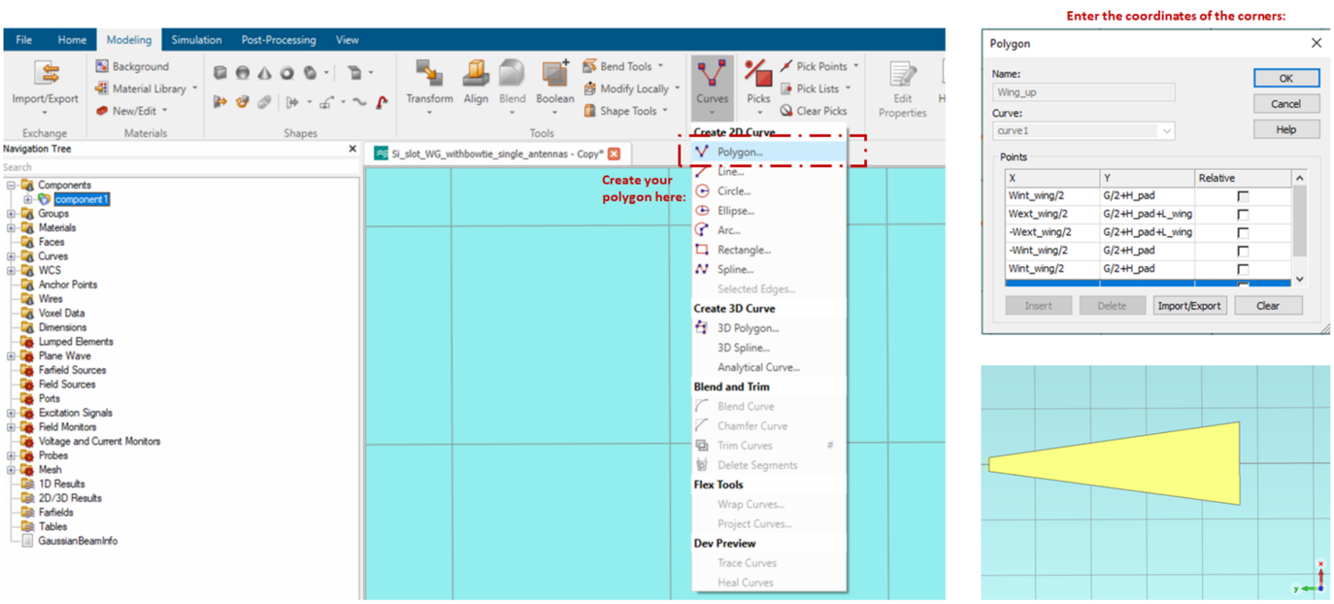
  You can also find the **Transform** menu under **Modeling**, together with other functions to **Align** shapes, create a **Bend** or perform **Boolean** operations between 3D objects. This final option is fundamental when two bricks have an intersection.



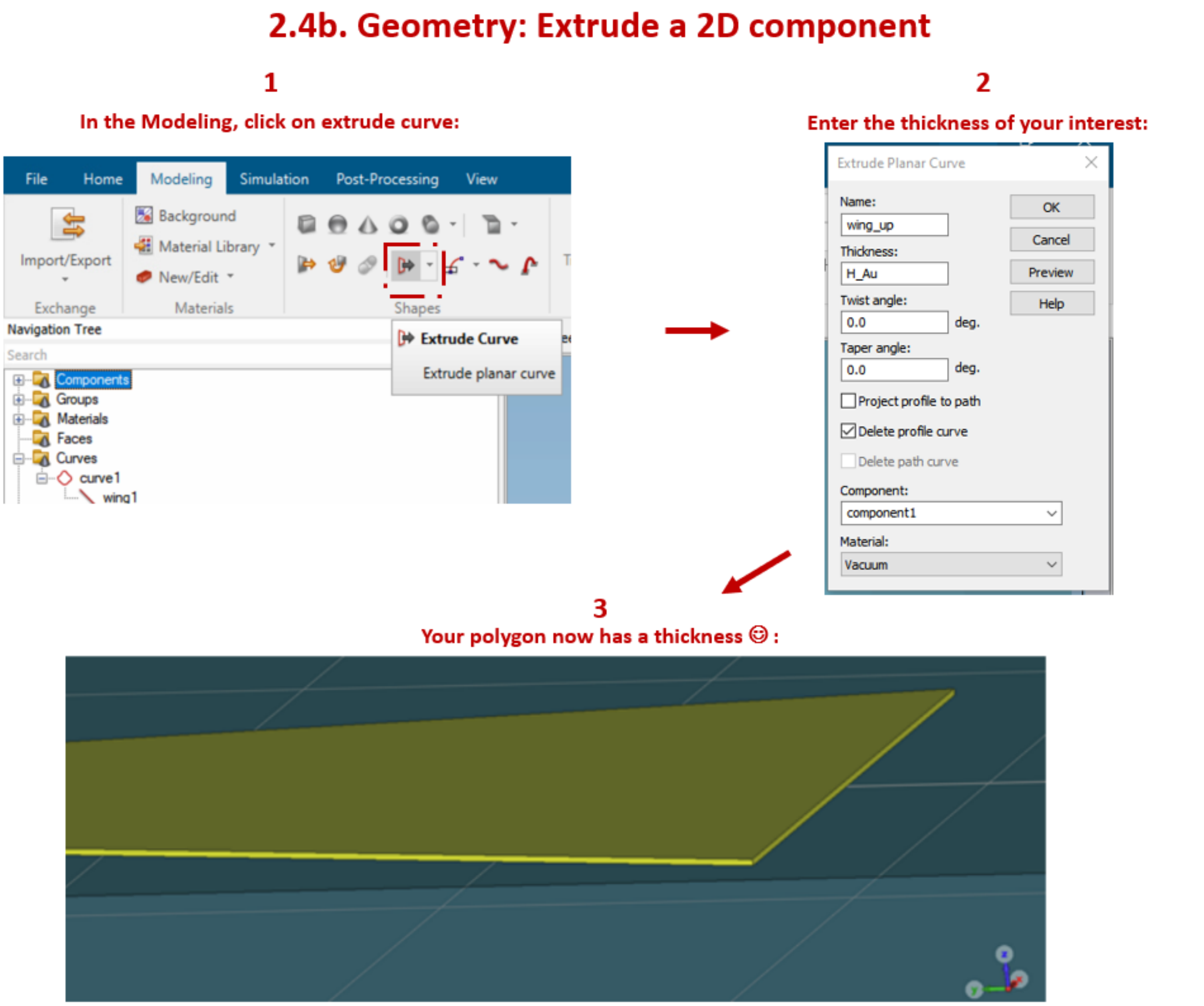
2.4 Creating 3D object by extruding 2D geometry

- 2.4a) You can not only create 3D objects, but also 2D objects with standard curve shapes or a free polygon. You can even define your desired mathematical expression to create an arbitrary curve (but don't worry about this option for this course ☺).
  To create for instance a more complex structure as the wing of a bow-tie antenna, under **Modeling** you can select **Curves** and **Polygon**. A new window will open, where you can enter the coordinates of the vertices of the polygon. **Alert:** to close the polygon, be careful in repeating the first point you specify in the list also as the last one. At this point, you can go on with the extrusion of the curve (see point 2.3b).

2.4a. Geometry: Building a 2D component

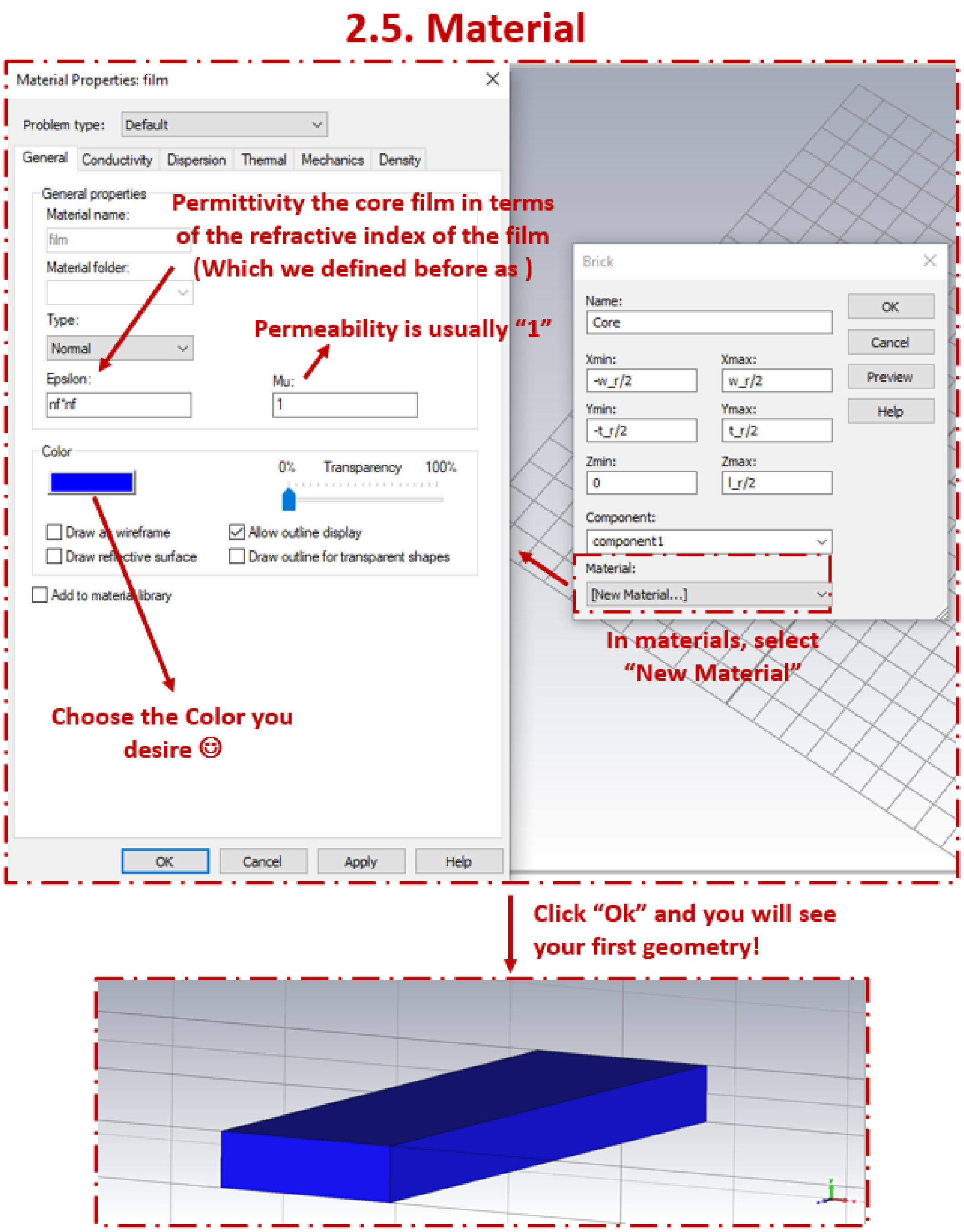


- 2.4b) You can now extrude the curve you have just created: the **Extrude** icon under **Modeling** becomes available only after the creation of the polygon. Click on it and select the curve to extrude. In the new window you can specify the thickness of the final 3D shape (together with a twist angle for more complex geometries) and its material.



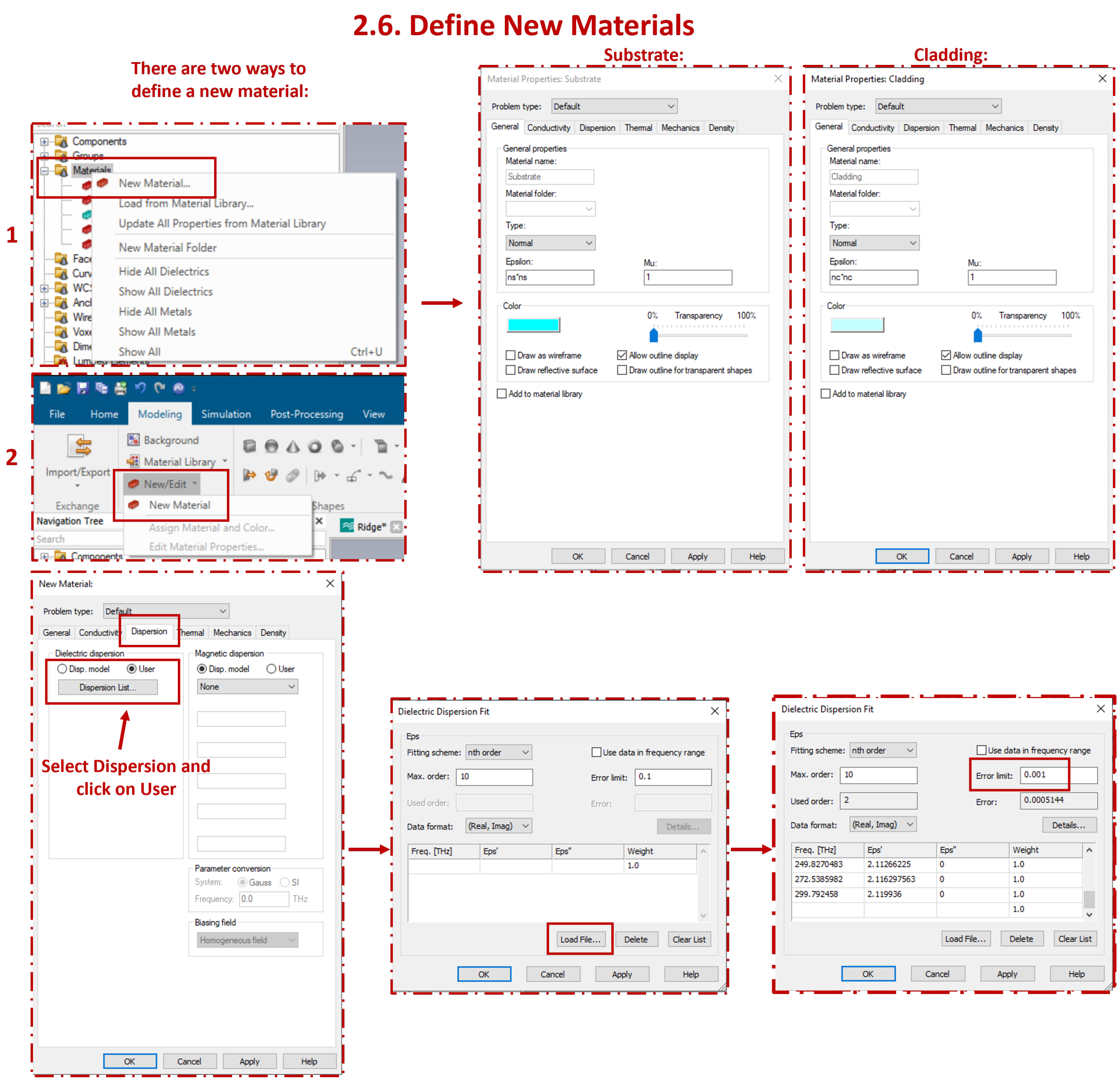
2.5 Material assignment

- 2.5) While creating the brick, you can assign it a new material, or choose a material from your material library. In the first case, specify its permittivity as (for simplicity) the square of the refractive index. For more refined simulations, one can instead specify the dispersion relation of the material in the window about **Dispersion**, as shown in point (2.4b).
  If you want to **assign a different material to a brick after its creation**, right-click on the brick in the Navigation Tree and select **Assign material and color** from the list.



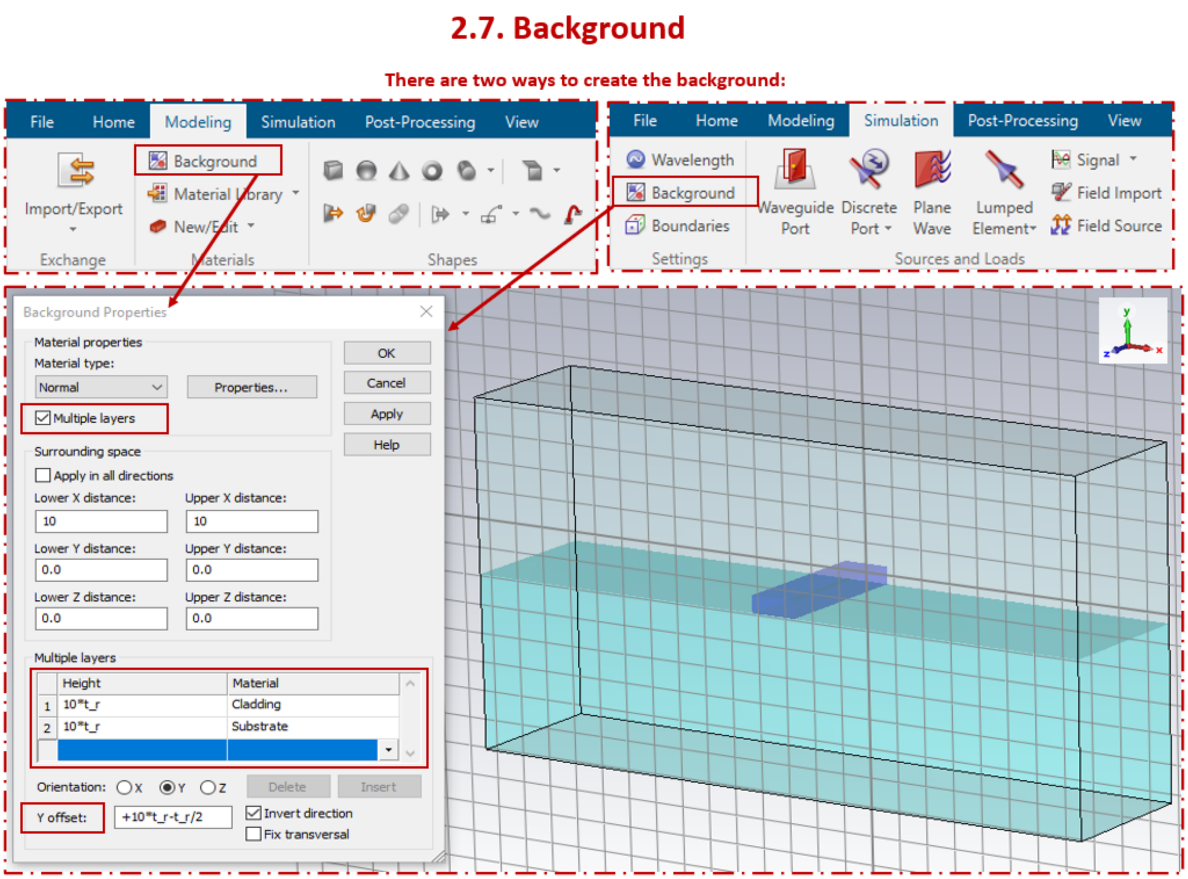
- 2.6) You can select and define materials also from the **Modeling** menu, or even from main tree under on the left of the general interface: right-click on **Materials** and select **New Material**. The same window will open for both cases.

  When defining a new material, you can specify its dispersion relation for a simulation in a large frequency range, or to have a more accurate result even for small ranges. To do so, in the **New material** window, select **Dispersion** in the top list and choose the **User** defined option. Click on **Dispersion list** and then **Load file** to select the dispersion relation (saved usually in a .txt file) and set the **Error limit** to 0.001. This is the error between the dispersion just uploaded and the fit performed by CST: you can check the precision of the fit under **1D Results** (in the **Navigation Tree**) selecting the name of the material you created.



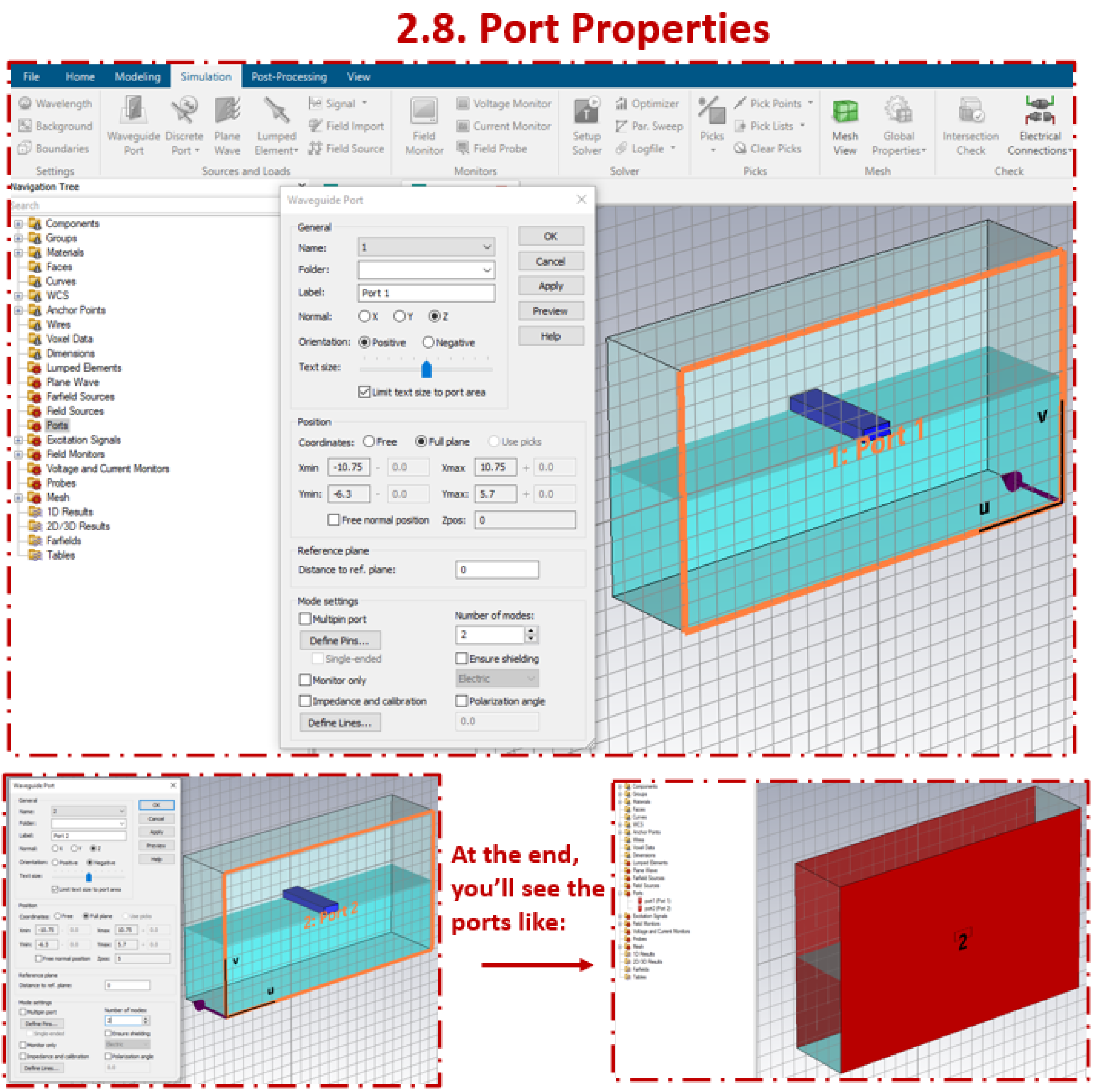
2.7 Defining the background

- 2.7) **After having created the core of your structure**, an efficient procedure for the creation of substrate and cladding, alternative to the creation of 2 dedicated bricks, is to define them at once using the **Background** interface. You can either go through **Modeling** or **Simulation** and click on **Background**. **Background Properties** window will appear: the background will surround your existing geometry and you can here specify its maximum the distance in every direction. Tick **Multiple Layers** and enter the height and material of each layer. At the beginning, the structure's corner will be at the center of the coordinate system, so you need to adjust the **Offset** to bring the background in the same center as the ridge. The orientation of this offset depends on how your structure itself is oriented as it should always be referred to the thickness of the layers: in this example, thicknesses are defined along Y and so does the offset, in other assignment is may vary.

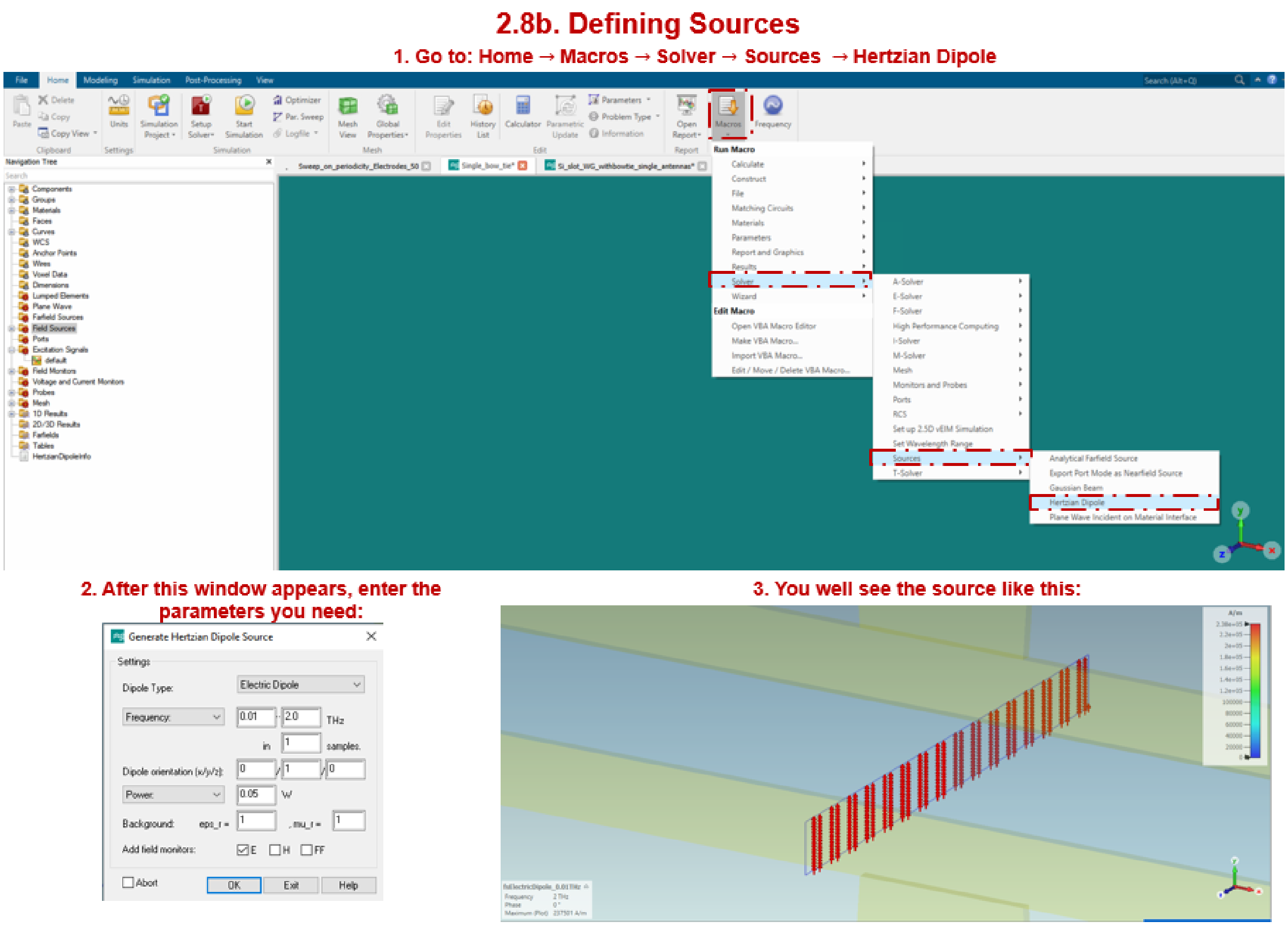


2.8 Waveguide port and other sources

- 2.8a) One important step before running the simulation, is to define a source for the optical wave! To do so, in **Simulation** click on **Waveguide Port**. You can give your "in-port" a name, specify its orientation and **define the number modes that CST will calculate**. In order to retrieve the S-parameters of the structure later on, define another port as an "out-port" by selecting the **Negative** orientation. As the modal analysis depends on the section of the waveguide, if the structure is symmetrical for input and output ports you can analyze the modes on only one of them, since they will be the same for the other.

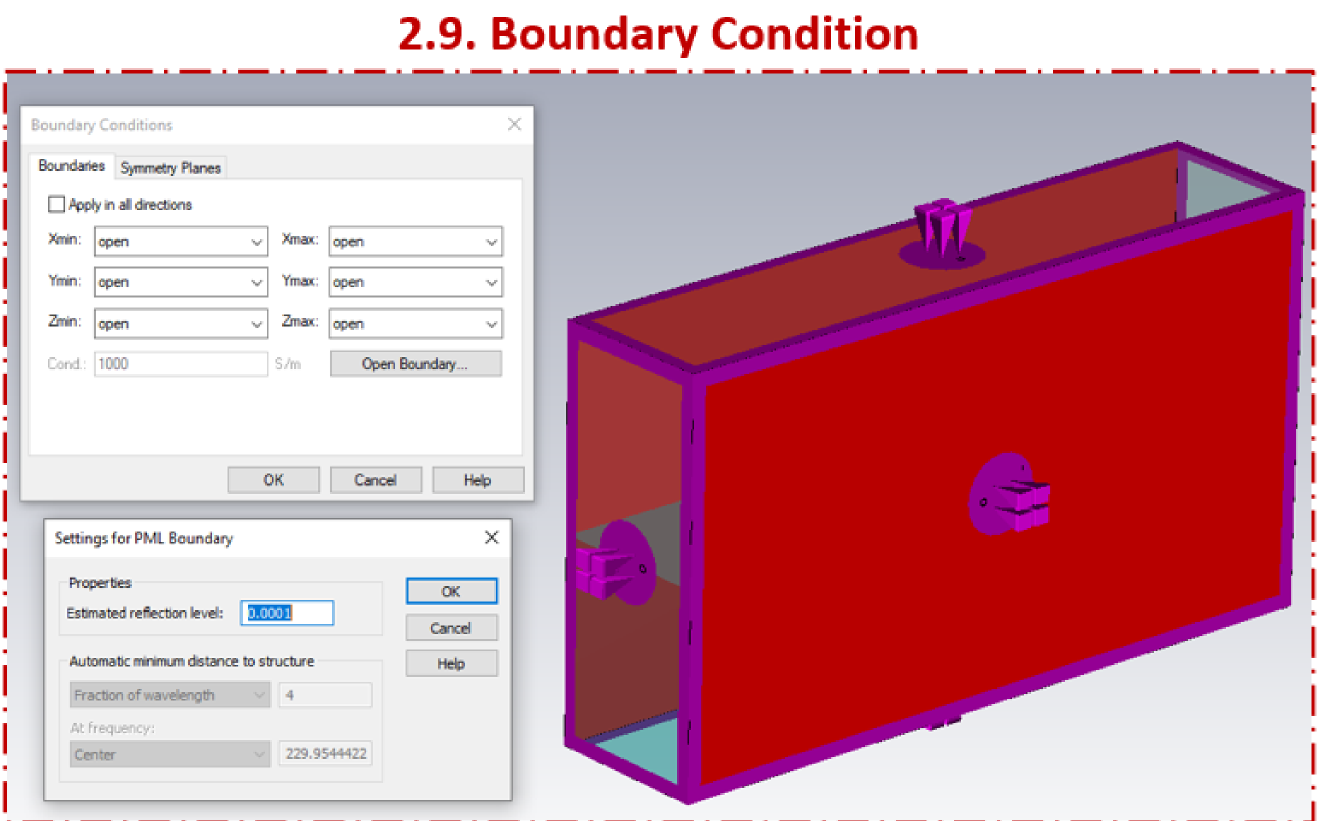


- 2.8b) Besides the waveguide port, in CST you can also define other types of sources for your optical field (i.e., discrete ports, plane waves, lumped elements etc.) that can be found under **Simulation** and **Source and loads**. Moreover, one could define electric dipoles and magnetic dipoles, as for instance the Hertzian dipole (you can learn more about it in BALANIS, Constantine A. Antenna theory: analysis and design, **chapter 4**). To create this particular source, follow **Home** → **Macros** → **Solver** → **Sources** → → **Hertzian Dipole**. You need at this point to enter the parameters of the antenna, including the direction of the field, operating frequency range, background properties and power.
  Using the **Macros** menu, you can find many other interesting functionalities in CST.



2.9 Boundary conditions

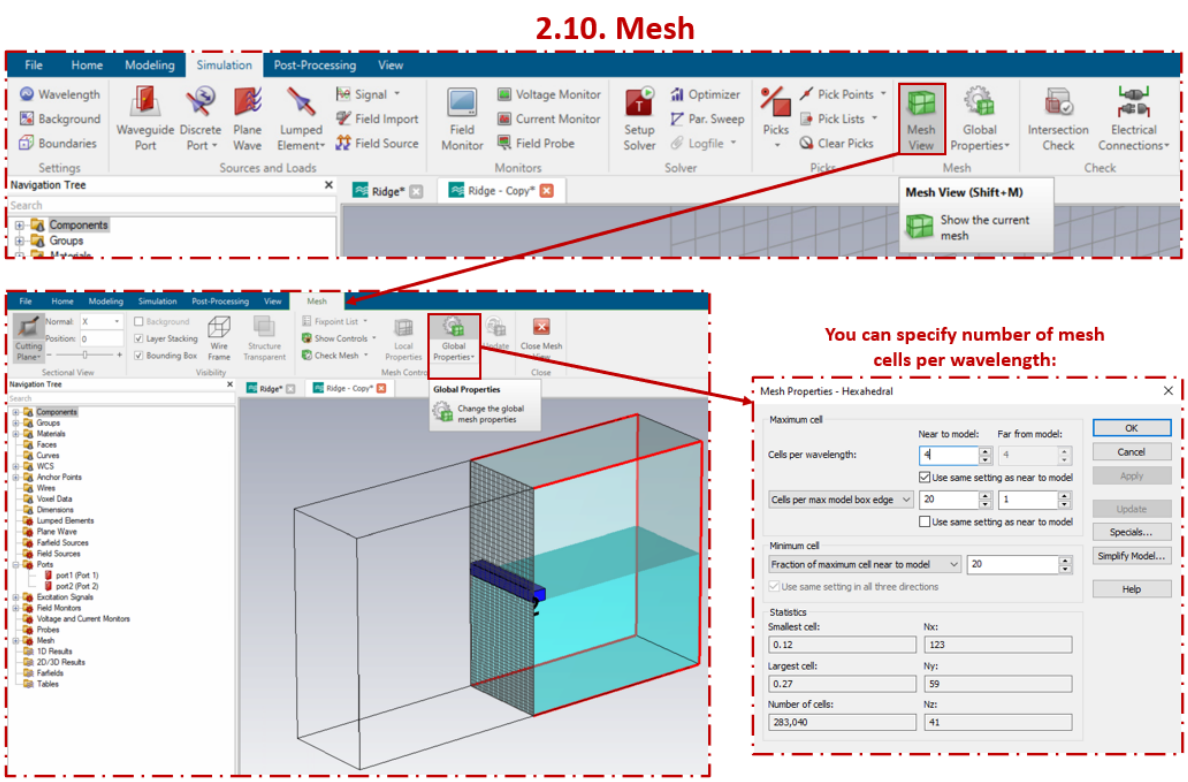
- 2.9) Time to set-up the boundary conditions! The default boundary box in ideally infinite in all directions, therefore you can set an **Open** boundary condition in every direction. Other types of boundary conditions (again not used during this course) might reduce the computational time by exploiting symmetries of the electric or magnetic field or considering a periodic or perfectly conductive wall.



2.10 Mesh

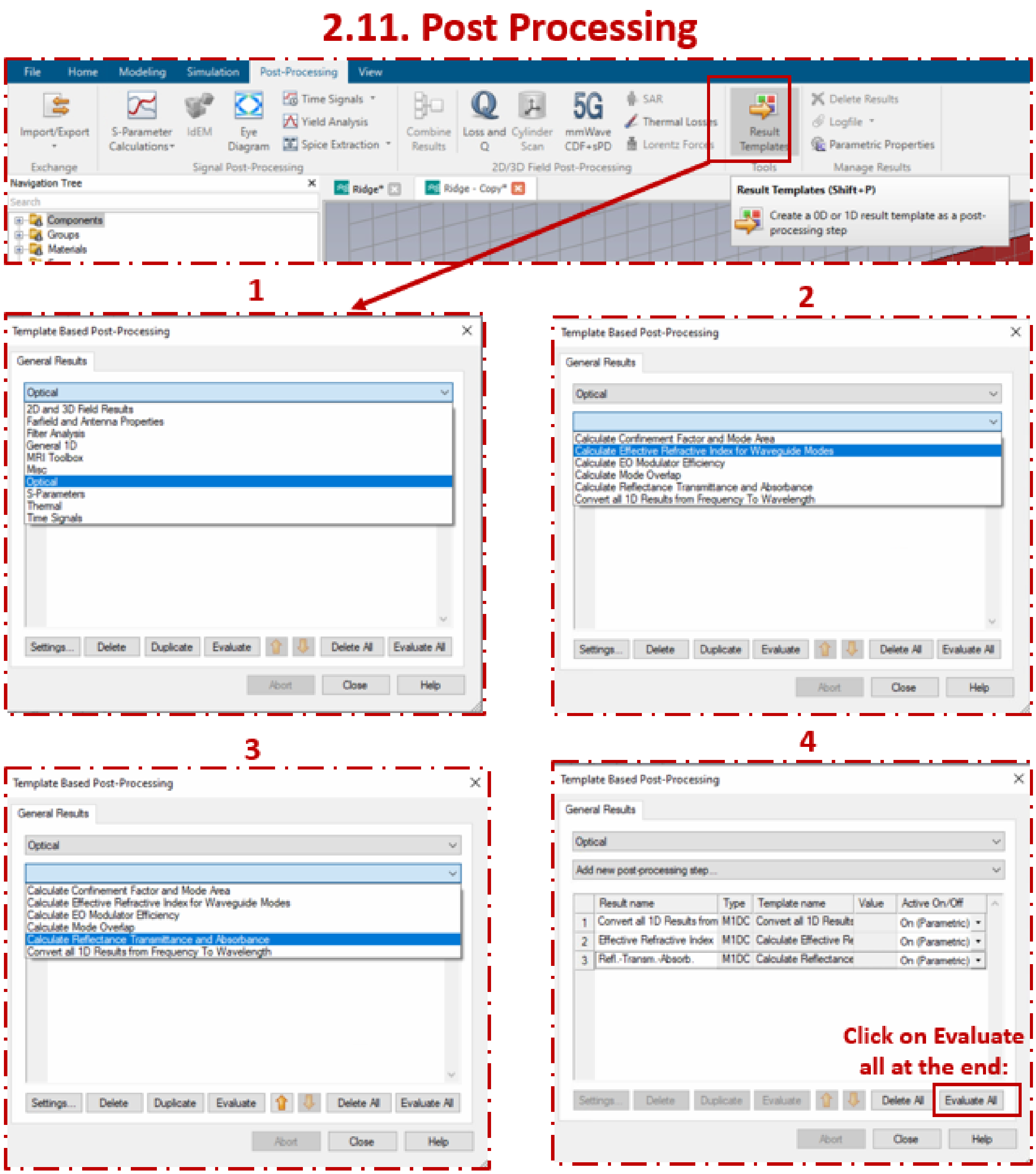
- 2.10) Once the structure of our simulation is complete, we have to define a **Mesh**: mesh setting determines the accuracy of the numerical simulation creating small domains in which the solver can more easily compute the solution. However, a too fine mesh will lead to an extremely long simulation time. As a consequence, there is a trade-off between the error of calculation and the time and memory that the simulation requires. Usually, depending on the structure, some mesh tests are necessary to tune the properties of the mesh: if the default values provided by CST lead to an error, it probably means you need to refine the mesh until the solution can be computed.
  To modify mesh parameters, in **Home** click on **Mesh View** to open the **Mesh** menu, where you can choose the position and orientation of the cut-plane. Then click on **Global Properties**: here you can modify the number of cells in the whole simulation space. In the example shown below, the number of mesh cells per wavelength is chosen to be 4. Change it and see how increasing or decreasing the mesh affects the simulation time and results. Click on **Close Mesh View** to go back to the normal interface.

  For more refined simulations (that we will not be able to perform during this course), there is the possibility to finely tune the local mesh in the region of higher interest (as the core of a waveguide) without affecting the most lateral portions: after selecting one of the solids from the Navigation Tree (found under Component), the **Local Properties** option will become available in the **Mesh menu**. You will be able in this way to refine the mesh for that single solid.

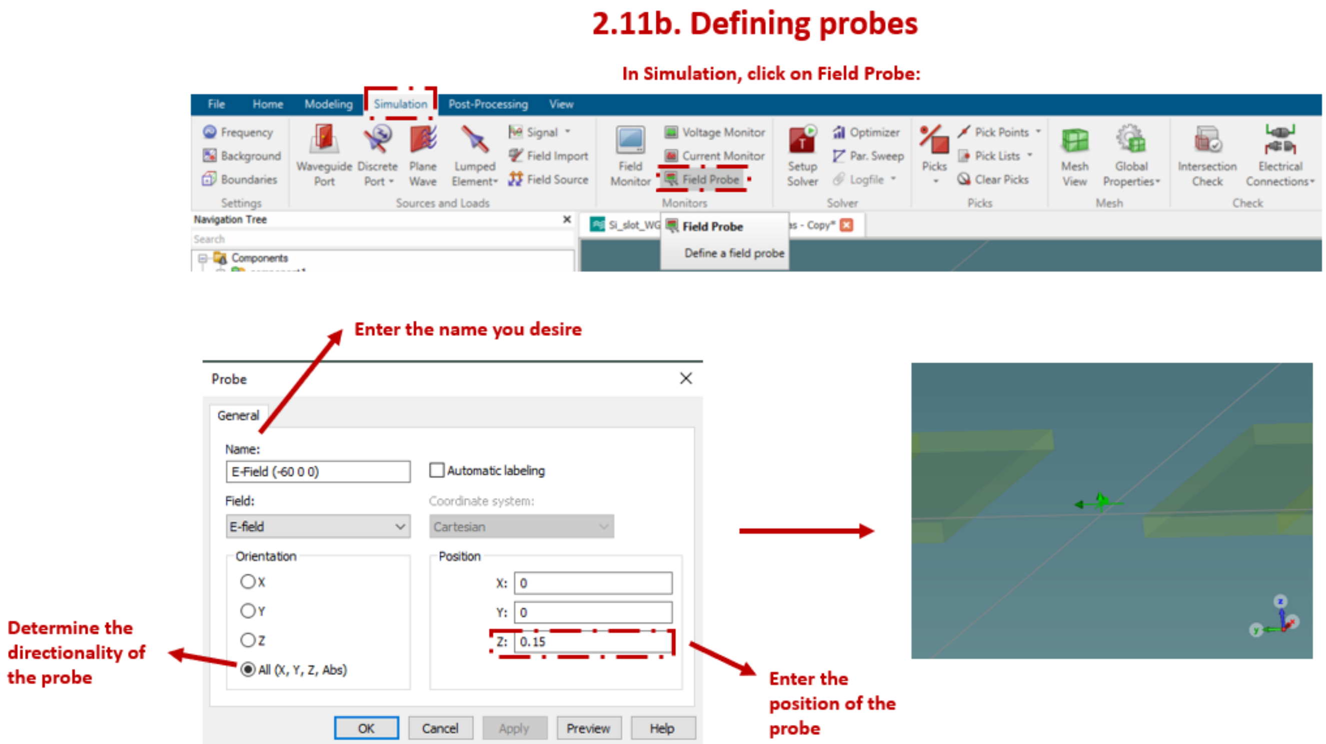


2.11 Post processing, probes and monitors

- 2.11) A powerful tool in CST is the possibility to automatically compute a large variety of parameters during the general simulation: you can use the **Result Templates** window under **Post Processing**. For example, if you need to calculate the effective refractive index of the structure, from **Optical**, choose **Calculate the effective index for waveguide modes**. You can choose other post-processing options as shown in the figure below. However, many operations (as for instance the analysis of the electric field in the structure) are only available after the simulation and can be added to the **Result Templates** list *a posteriori*.

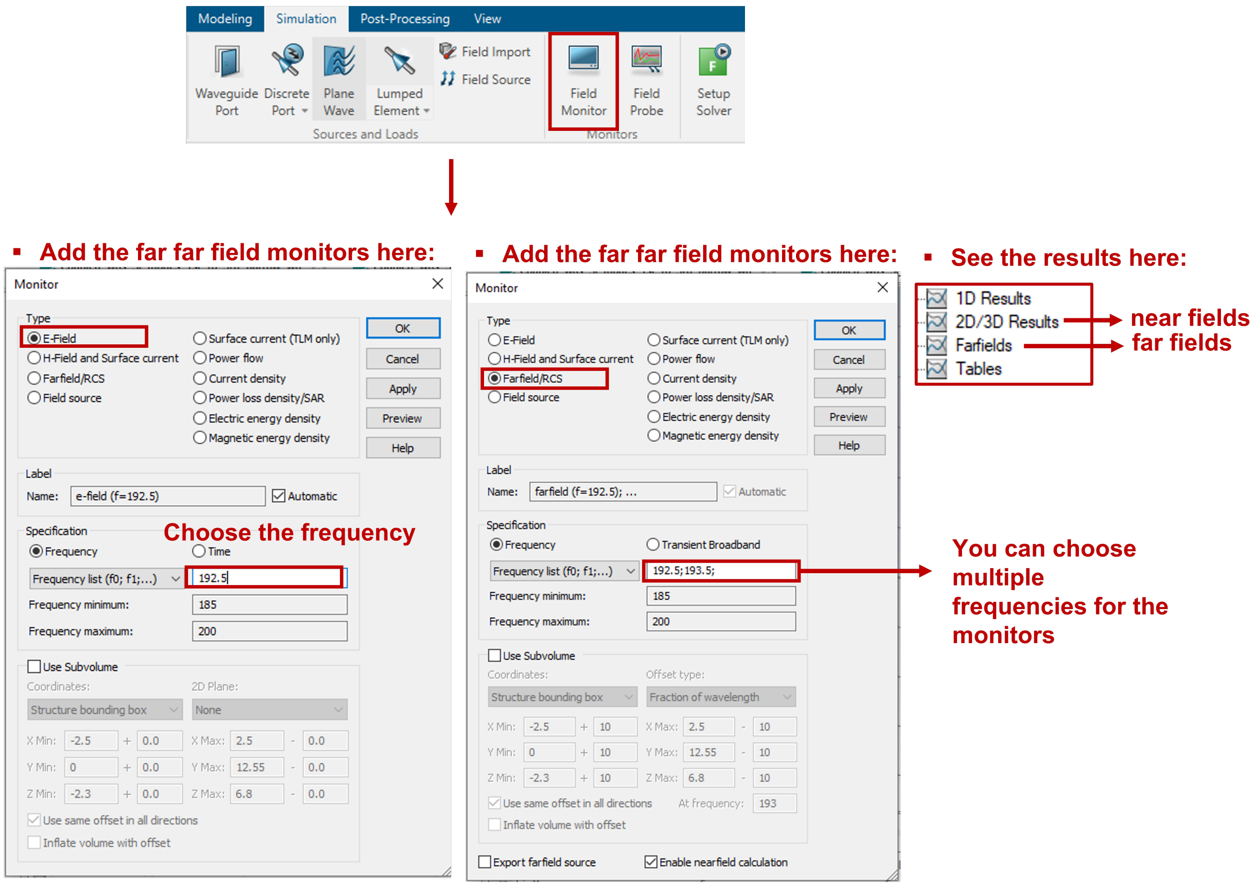


- 2.11b) You can measure the field in specific points by putting probes at the positions you need. Choose **Field Probe** under **Simulation**, then specify its coordinates and which field it should evaluate.



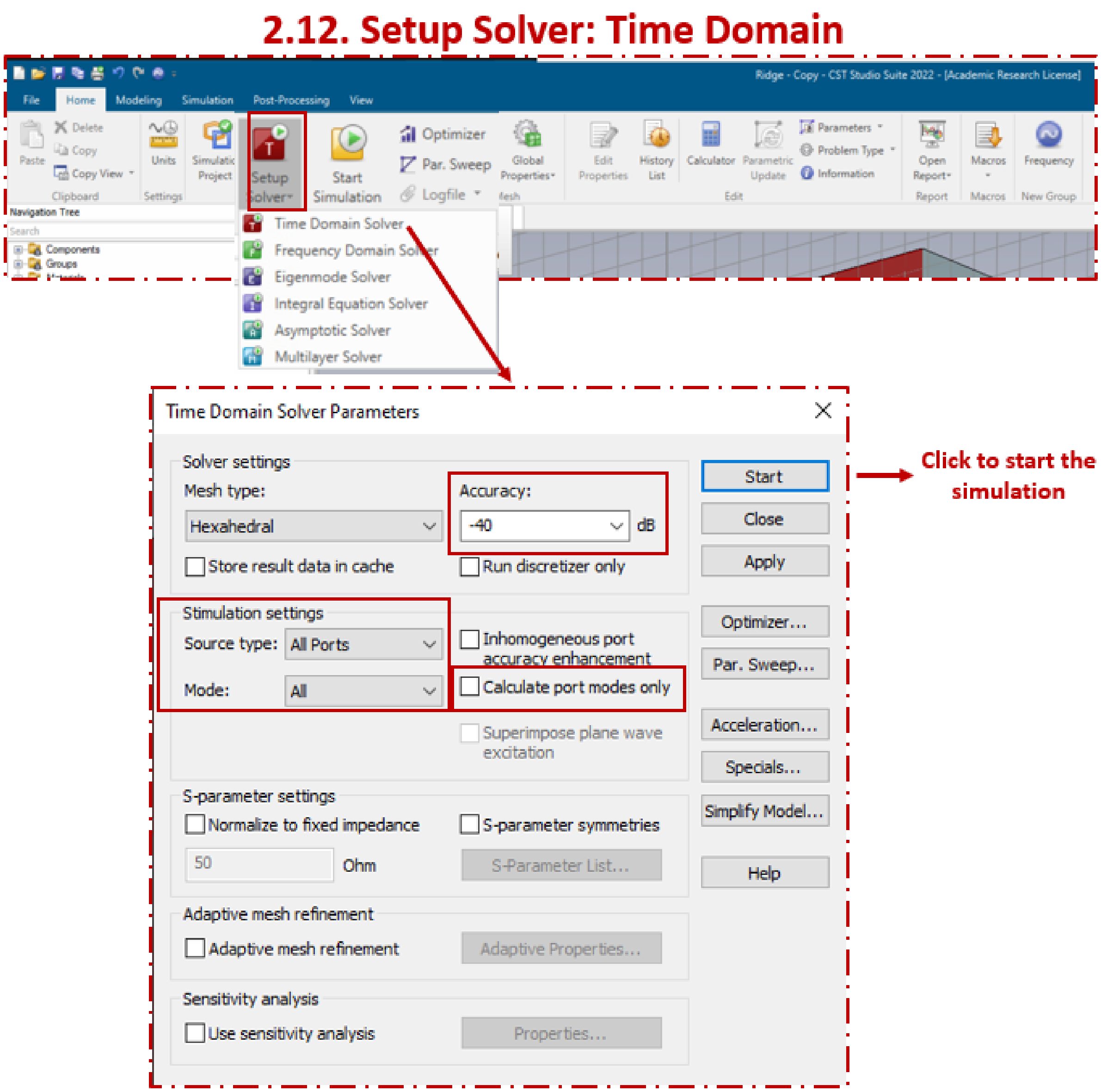
- 2.11c) In order to see, instead, the field distributions in the whole domain, you need to define near field or far field monitors. In **Simulations**, click on **Field Monitors**. In order to see the electric fields at the frequency of your interest, tick **E-Filed** and enter the frequency. You can add multiple frequencies, or even a sweep or a range! In order to see the far-field pattern, click instead on **Far Feild/RCS**. After the simulation is concluded, you will see the results of the near-field monitors under **2D/3D Results** and the results of the far fields monitors under **FarFields**, both in the main **Navigation Tree**.

2.11.c. Field and Far-Field Monitor

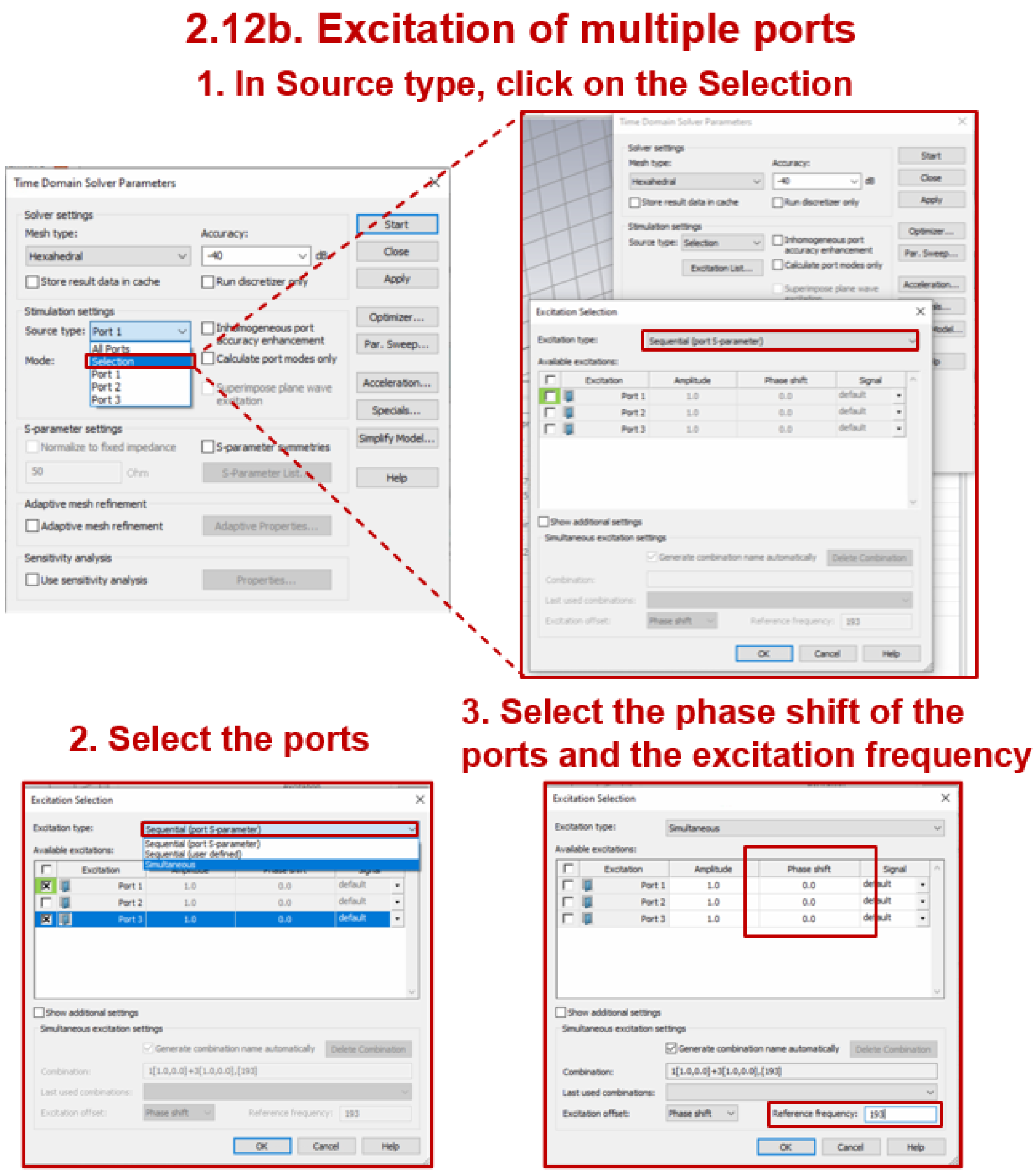


2.12 Setup solvers: time domain and frequency domain

- 2.12) The final step before launching the simulation is to setup the parameters of the solver itself. In **Home**, there are several **Setup Solver** modes, but the two commonly used in this course are the **Time domain setup-solver** and the **Frequency domain setup-solver**. For time domain, you can set the Accuracy of the frequency domain signals calculated by Fourier Transformation of the time signals. For a more accurate simulation, this parameter can be decreased, but it might lead to errors with the mesh set before. With a coarse mesh, simulations with high accuracy (so smaller specified value) may not converge.
  As shown below, the Time Domain window allows you to choose the source of the simulation from the ones you created before by selecting which port and which mode will excite the structure. For a quick modal analysis (which does not require to calculate any propagation along the 3D domain), you can tick the option to **Calculate port modes only**. Modes will be computed at the central wavelength (frequency) of the range specified at the beginning.

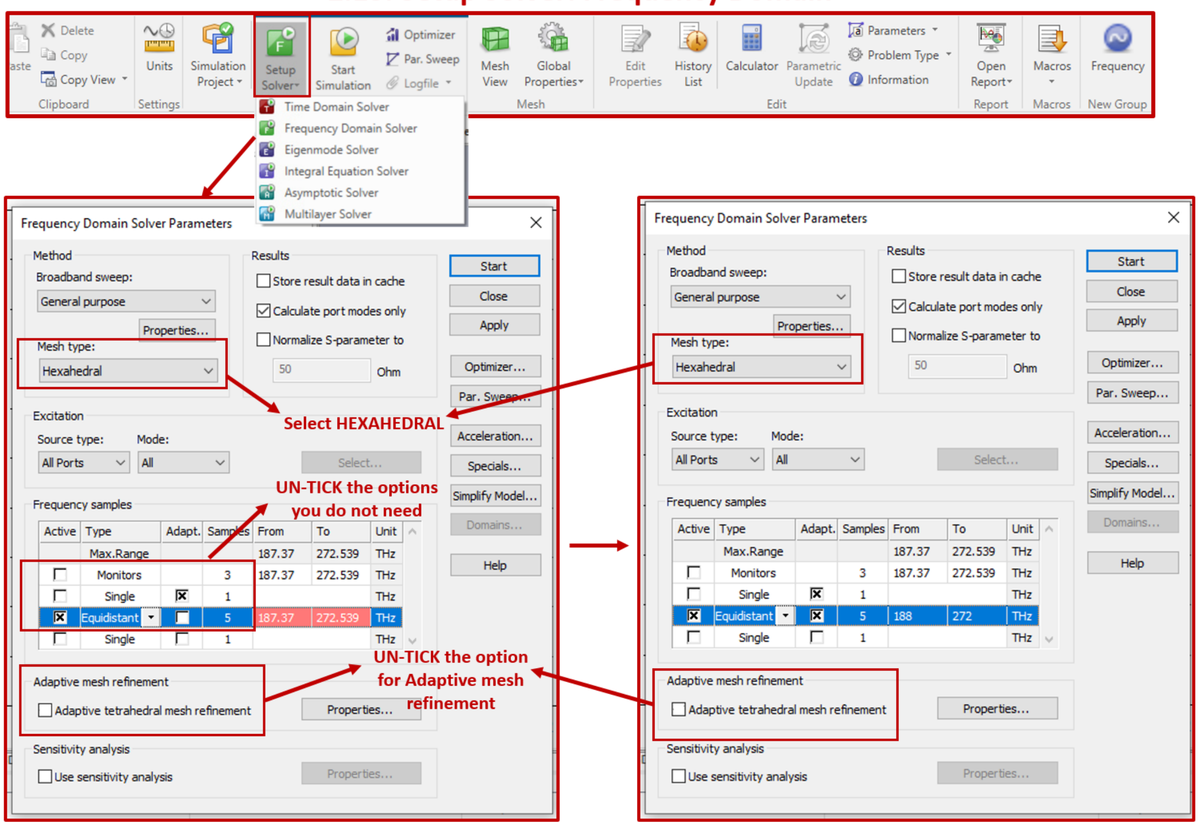


- 2.12b) You can also **excite multiple ports or multiple modes** by following the steps below:



- 2.13) Also in the **Frequency domain setup-solver** you can choose to calculate port modes only. You can also define the number of frequency samples for your simulations (single or equidistant points). By default, if you click on **Calculate port modes only**, CST skips the full frequency sweep and only calculates the effective refractive index of the modes at the central frequency of the one specified under **Simulation**. Instead, if you want to calculate the **group refractive index**, you need to run a complete simulation setting a frequency range (as shown in the pictures) and **UN-TICKING** the option to **Calculate port modes only**. Note that the frequency range in the solver **must be smaller that the maximum range**, otherwise you will get the error shown in red on the bottom left of the picture.
  In order to speed up the simulation **UN-TICK** the option for **Adaptive tetrahedral mesh refinement**, otherwise the software would iterate the simulation trying to automatically adapt the mesh.

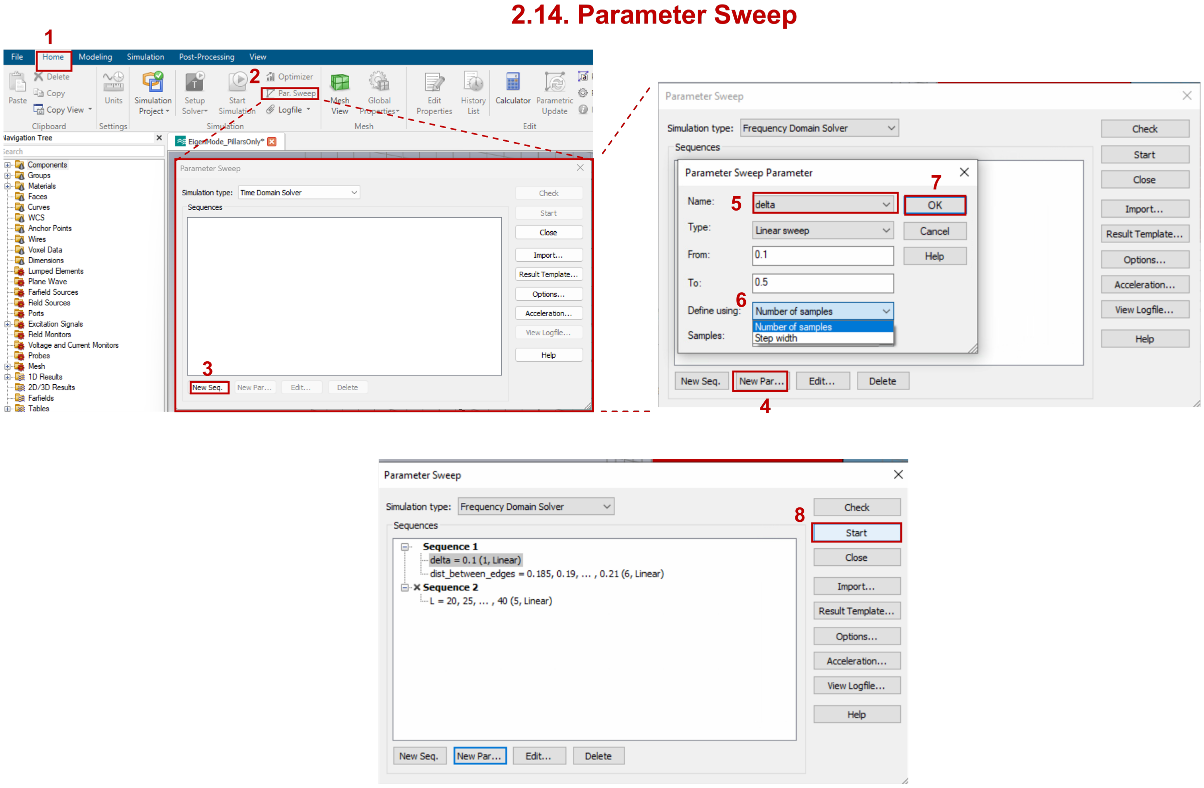
2.13. Setup Solver: Frequency Domain



2.14 Parametric sweep

- 2.14) In order to perform a parametric sweep in CST, in **Home** choose **Par. Sweep** and then, in the new window, click on **New Seq**. You can add different sequences that will be computed separately. Moreover, you have the possibility to perform ”multidimensional” sweeps adding more parameters under the same sequence, all the combinations between them will be computed.
  To choose the parameter you want to sweep, select **New Param** and find its **Name**. You also can determine the **Type** of the sweep, but in this course we will only do **linear** sweeps. Specify starting and ending of the sweep in **From** and **To** boxes, respectively. You can do the sweep either for a predefined **Number of samples**, or in predefined **Step width**, CST will divide the range specified accordingly. At the end, just press start to run the full simulation.

2.14. Parameter Sweep

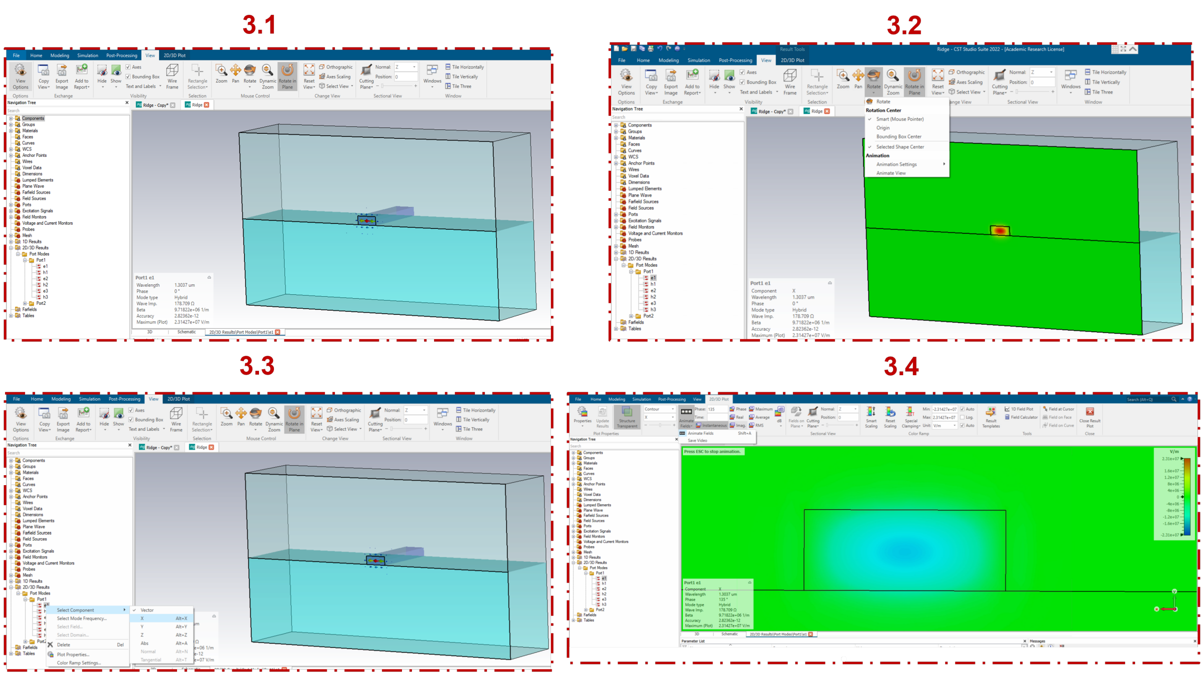


3. Plotting/extracting the results:

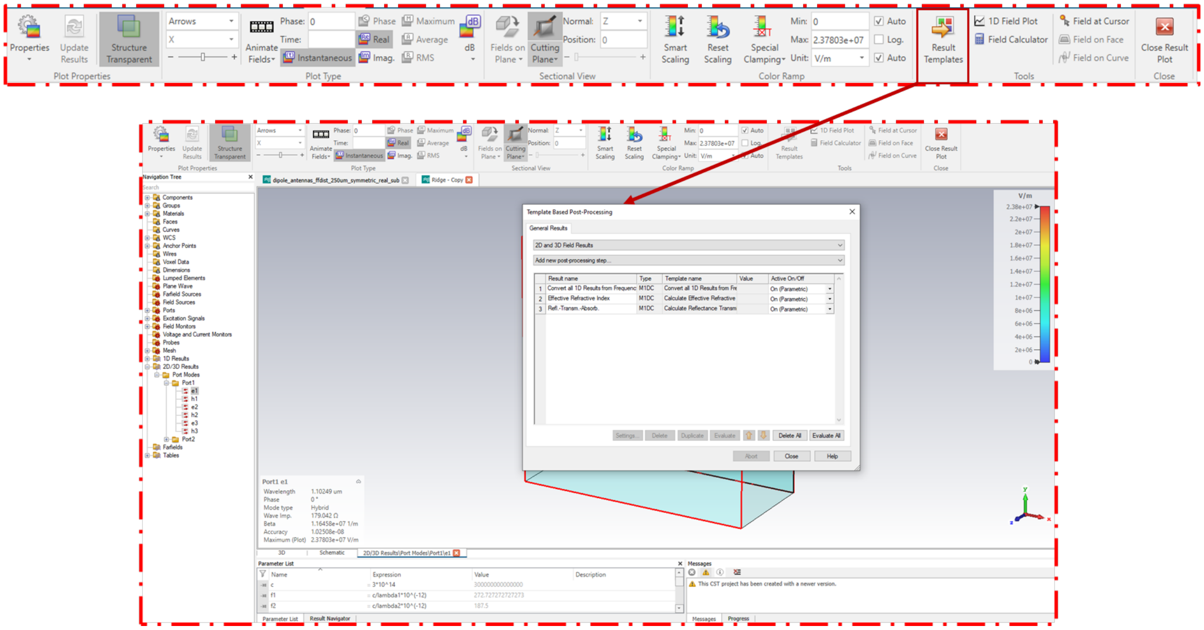
There are several options for plotting and extracting 0D, 1D, 2D, and 3D data from CST simulations. Here, we only mention a few of them:

- 3.1) and 3.2) On the **View** tab, **Mouse Control**, you can choose the desired 3D rotation or in-plane rotation. You can also choose them from the gray taskbar at the right bottom of the screen. In **View** you can also select specific orientations of the geometry (useful for instance to plot field on 2D planes).
- 3.3) You can look at several 1D, 2D/3D plots by clicking on the **1D Results** or **2D/3D Results** on the **Navigation Tree** (left panel). Several options such as field component, mode frequency, color range, and sectional views are available on the **2D/3D plot** panel. Remember that many 2D or 3D results can be obtained only **defining a proper Field monitor** at the desired frequency.
- 3.4) You can also animate the field (by sweeping the phase) and check the propagation along the structure.
- 3.5) As mentioned before, another important tool is the **Result Template** interface under **Post Processing** (also found in **2D/3D Plot** tab when a 2D or 3D result is open), through which you can extract and plot customized 0D, 1D, 2D, and 3D plots from the simulated data.
- 3.6) Data from 1D results plots (and also from 2D or 3D, but we will not need them) can be extracted as a CSV file from **Post Processing** and **Import/Export**, selecting **Plot data ASCII**.
- 3.7) CST can calculate the effective refractive index of the port modes in your structure. You can add this parameter to your list in **Result templates**.

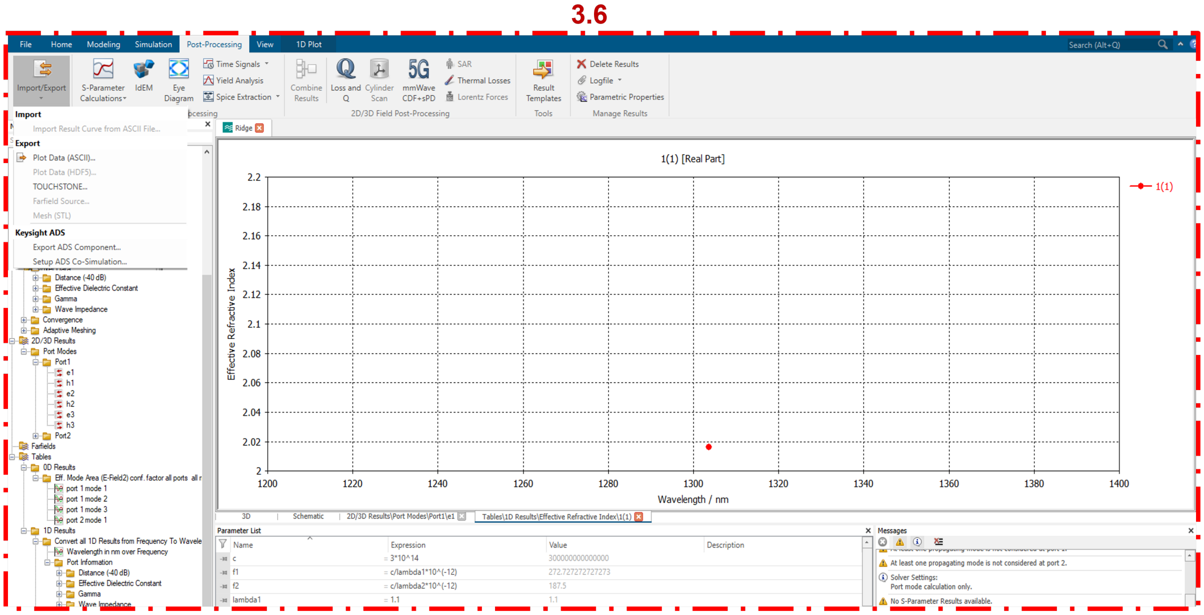
3. Results



3.5


Properties
Update Results
Structure Transparent
Plot Properties
Animate Fields
Phase
Maximum
Real
Average
Instantaneous
Imag.
RMS
Plot Type
Fields on Plane
Cutting Plane
Normal:
Position:
Sectional View
Smart Scaling
Reset Scaling
Special Clamping
Min:
Max:
Unit:
Auto
Log.
Color Ramp
Result Templates
1D Field Plot
Field Calculator
Field at Cursor
Field on Face
Field on Curve
Tools
Close Result Plot
Close
Template Based Post-Processing

3.6

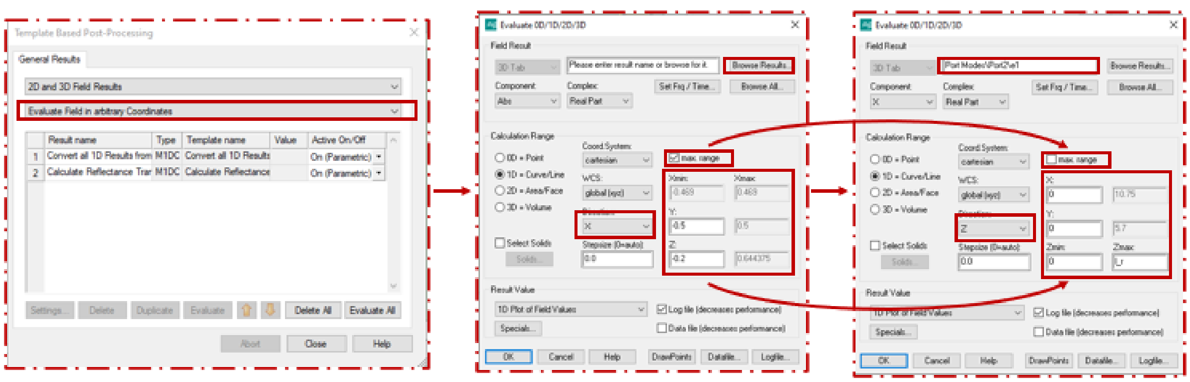

File
Home
Modeling
Simulation
Post-Processing
View
1D Plot
Import/Export
S-Parameter Calculations
IdEM
Eye Diagram
Time Signals
Yield Analysis
Spice Extraction
Combine Results
Loss and Q
Cylinder Scan
mmWave CDF+sPD
SAR
Thermal Losses
Lorentz Forces
2D/3D Field Post-Processing
Result Templates
Tools
Delete Results
Logfile
Parametric Properties
Manage Results
Import
Import Result Curve from ASCII File...
Export
Plot Data (ASCII)...
Plot Data (HDF5)...
TOUCHSTONE...
Farfield Source...
Mesh (STL)
Keysight ADS
Export ADS Component...
Setup ADS Co-Simulation...
1(1) [Real Part]
1(1)
Effective Refractive Index
Wavelength / nm

- 3.8) You can define cut-lines with the following procedure: after the simulation is done, under **Post Processing** select **Results template → 2D and 3D Field Results → Evaluate field in arbitrary coordinates**. A new window will appear, here:

  1. click on **Browse Results** and choose the field result of you interest.
  2. Select the desired component of the field and always choose to calculate its Real part.
  3. Select **1D = Curve Line** and specify the orientation of the line.
  4. Define the coordinates of the line. You can draw the line along the full structure ticking the **Max. range**.
  5. Click Ok and continue.

  Once you have set al the parameters, give a useful name to the cut-line and finally click on **Evaluate** for each cut-line, separately.

3.8. Defining a cut-line



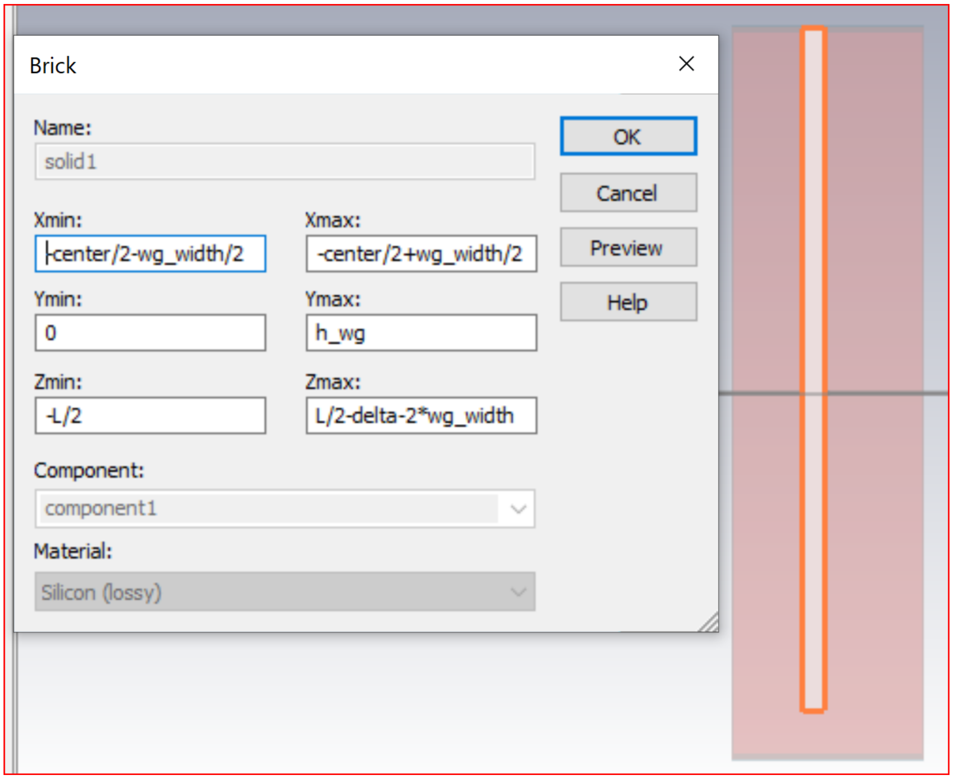
Appendix: Tutorial of creating bend waveguide

See Next pages.

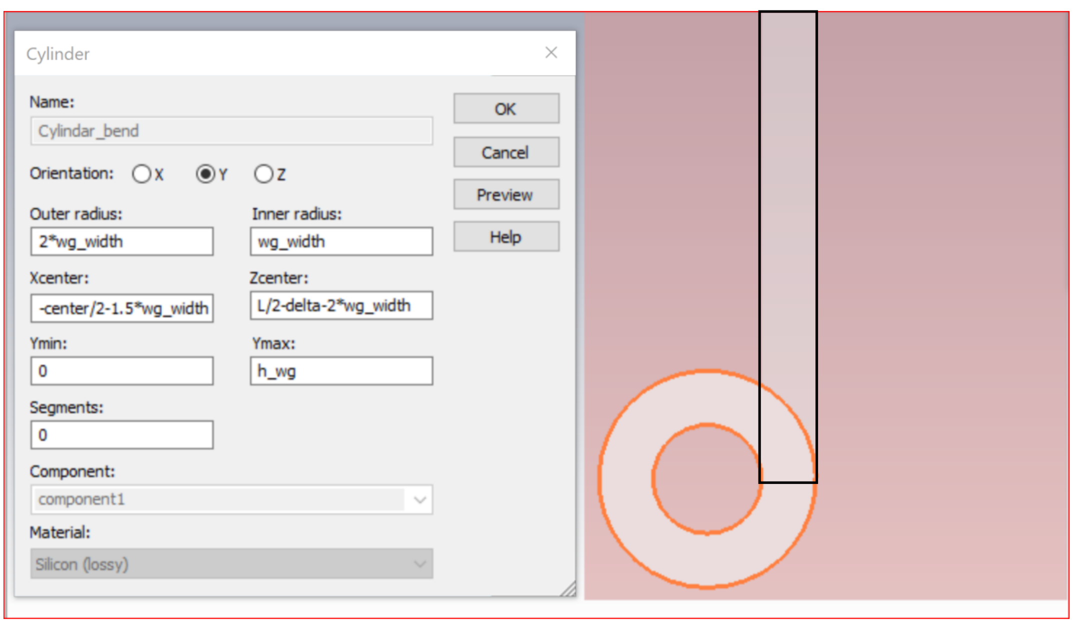
Tutorial to create the bend at the end of a waveguide

1-Make a waveguide. Put the end of the waveguide until $\frac{L}{2} - \delta - 2 \times W_{wg1}$

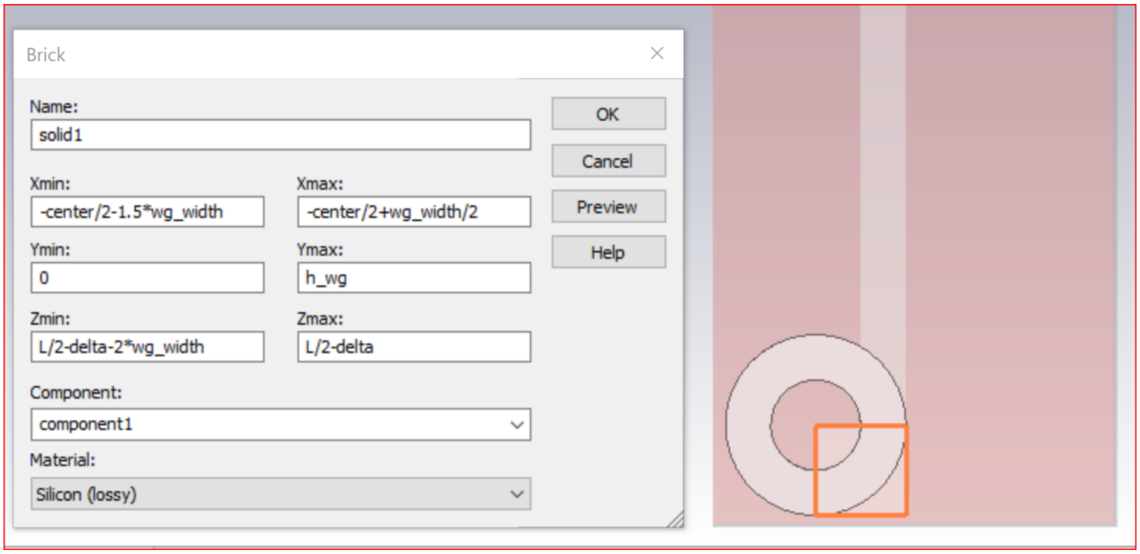
center is dist+W_{wg1}



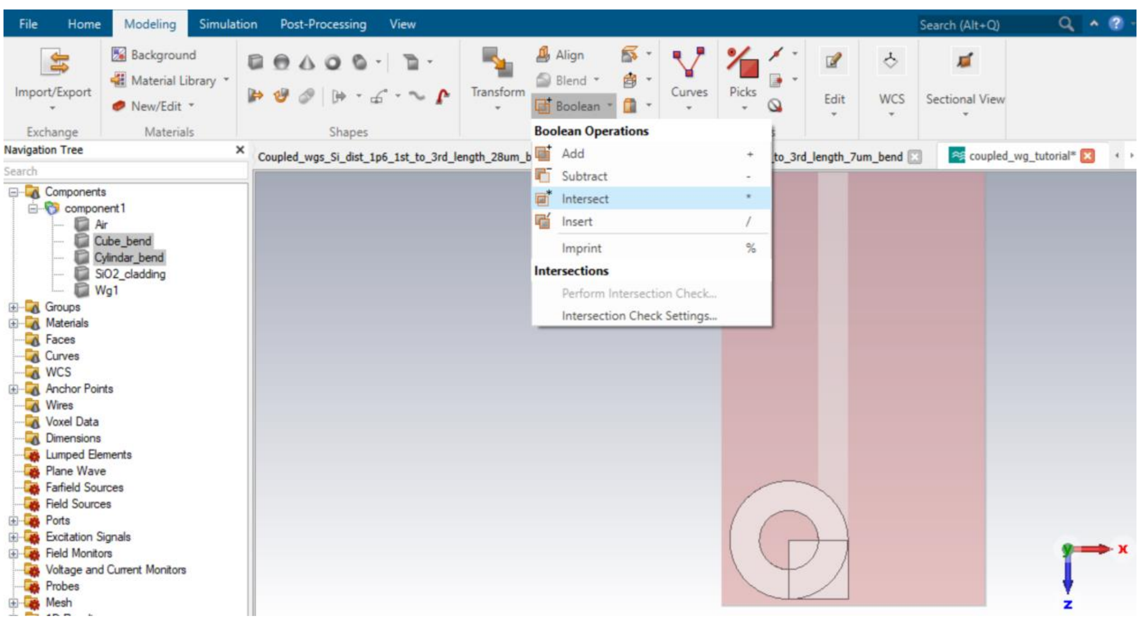
2-Create a Cylinder from Modeling>shapes>Cylinder with the following parameters:



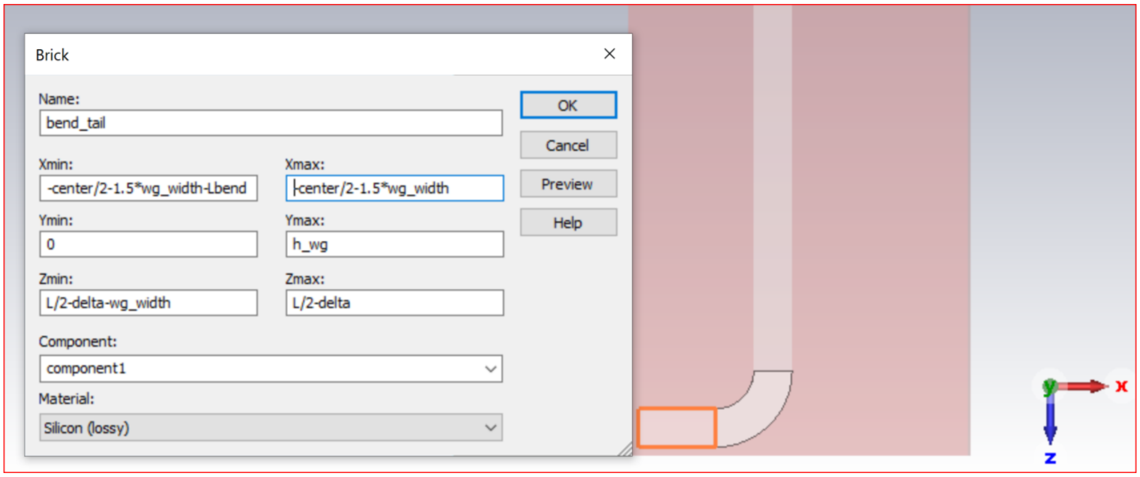
3-Create a Brick with the following parameter:



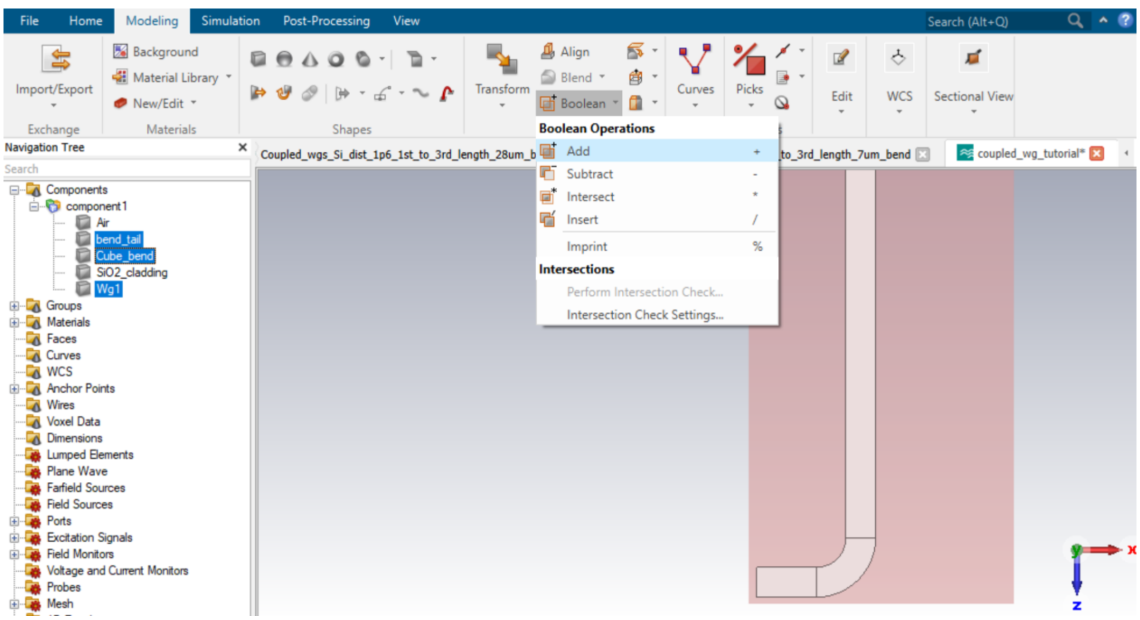
4- Intersect the cube and the cylinder via *Boolean Operations* in *Modeling* tab



5-Create another Brick for the tail of the bend with the following parameters:



6-Merge the waveguide, the curve, and the tail using *Add* in *Boolian Operations:*



7-To have the waveguide inside the cladding, make a copy of waveguide (Wg1_1 in figure). Then, use the *subtract* in *Boolean Operations*. First select the cladding, then choose subtract from the Boolean Operations and finally choose the waveguide (Wg1_1) and then press Enter for subtracting waveguide from the cladding.