

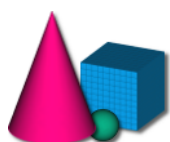
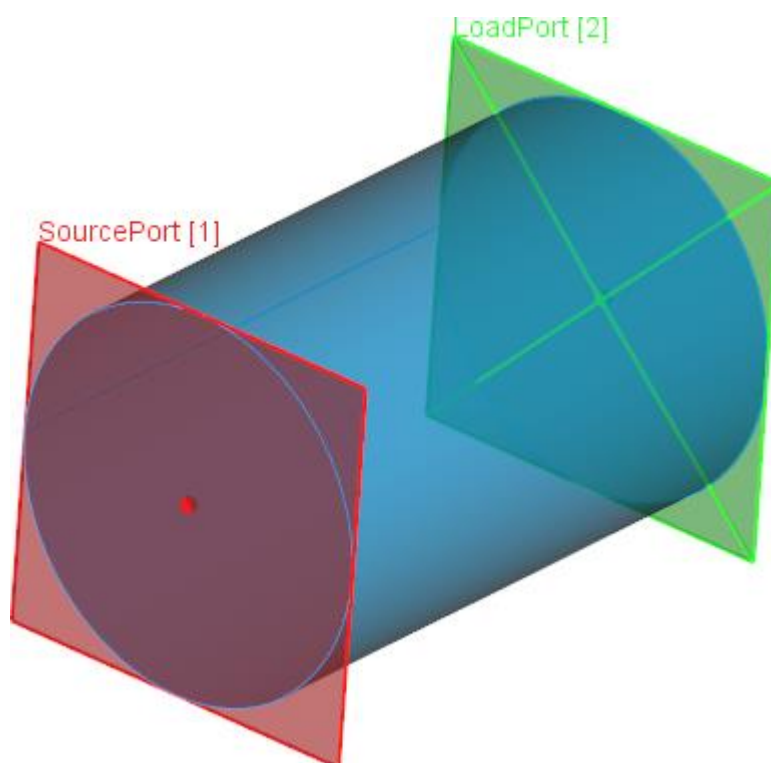
Project Preparation

Table of Contents

MODEL	1
NEW PROJECT	2
DEFINING THE TYPE OF THE STRUCTURE AND BACKGROUND MEDIUM	2
DRAWING THE STRUCTURE	4
DRAWING THE PORTS	8
SETTING BOUNDARY CONDITIONS	13
SETTING UP THE MESH	15
EXPORTING THE PROJECT AND RUNNING THE SIMULATION	16

Model


The model is a circular waveguide excited with a fundamental TE_{11} mode (red transmission line port) and terminated with a load port (green transmission line port) matched to the waveguide. The waveguide is filled with air.

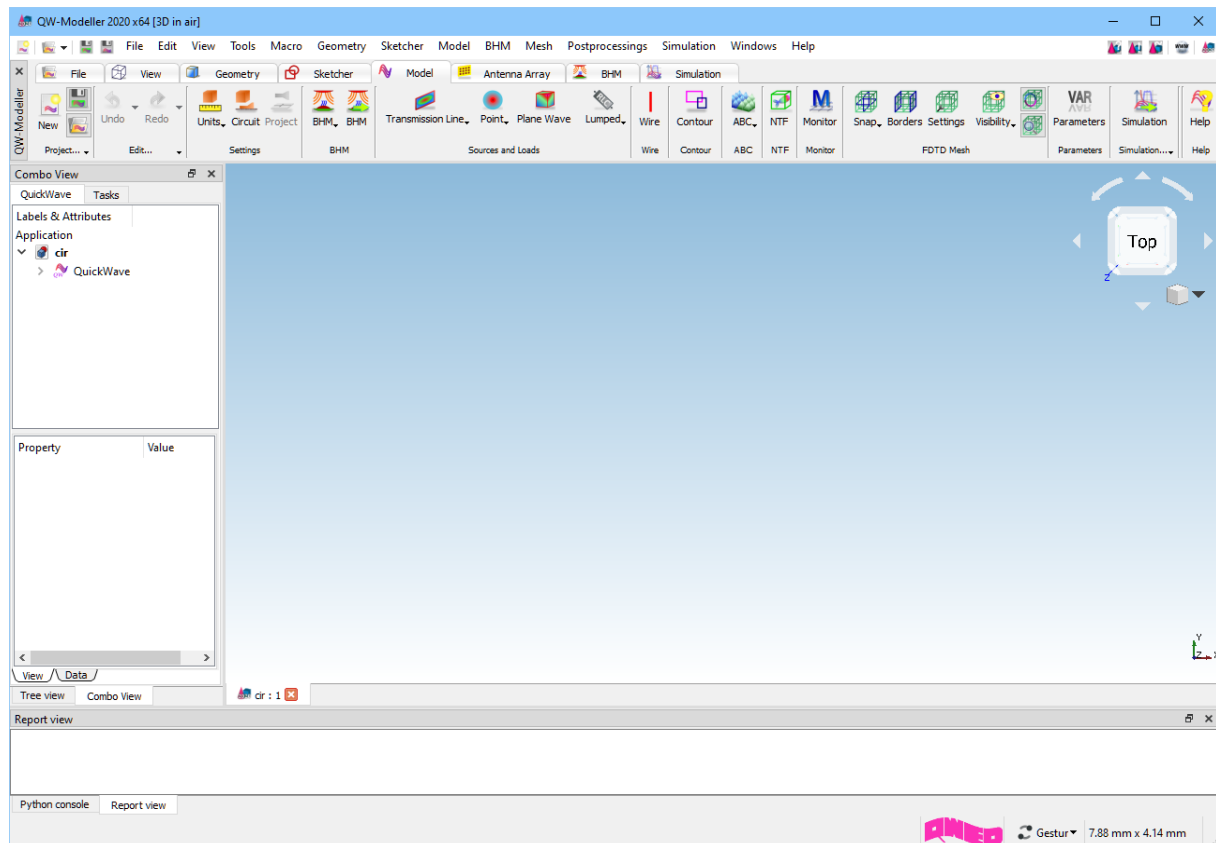


New Project

Having the *QW-Modeller* opened press  **New** button from the Ribbon to create new project. Press




or  button and save it with the *cir.QWpro* name.

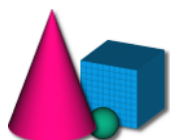


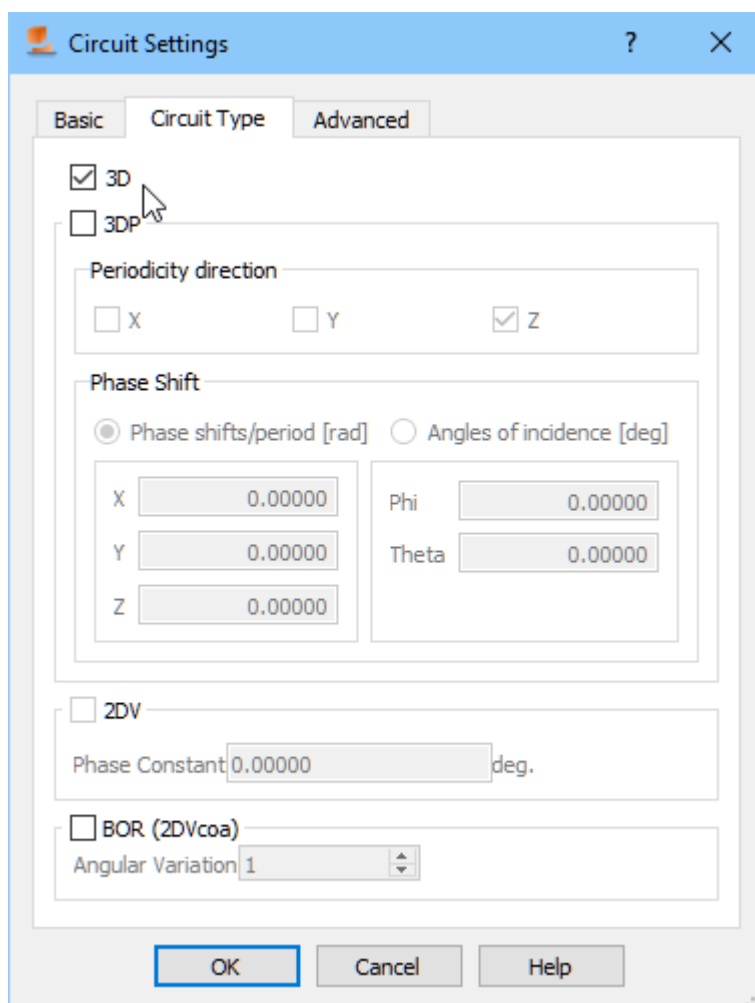
Defining the type of the structure and background medium

Having the *QW-Modeller* opened and a new project document created, we can proceed with defining the type of the structure. First of all, we need to specify if we will be dealing with typical three dimensional (*3D*) or maybe periodic three dimensional (*3DP*) structures. The structure type, called also the *circuit type*, is determined in the *Circuit Settings* dialogue available in the *Model* tab, under



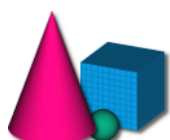
the *Circuit*  push button. In case of our waveguide we will be analysing typical 3D circuit.

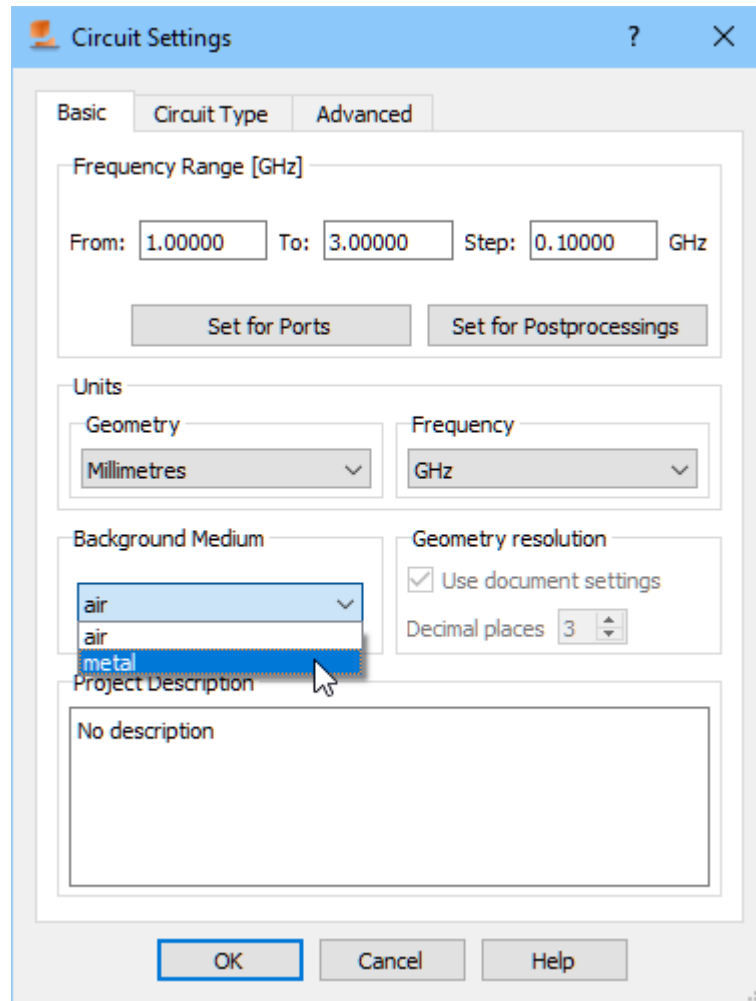




At this stage it is necessary to determine also the background medium for the structure. Since the structure is non-radiating and we are interested only in the fields inside it (it is a closed structure type of problem), we can set the background medium to metal and then draw only the inside of the structure. To set the background medium, we switch to *Basic* tab in the *Circuit Settings* dialogue and choose metal from the *Background Medium* combo box. Please note that there is a choice of only two possible background media, *air* and *metal*.

The frequency range for the analysis should be set to 1-3 GHz.



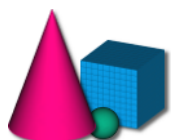


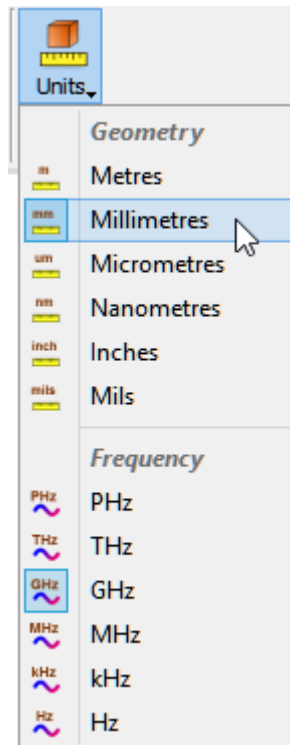
Drawing the structure

At this stage we will be creating the project geometry. First step that needs to be accomplished is choosing the units that will describe the geometry. To do that we go to *Model* tab and from *Units*

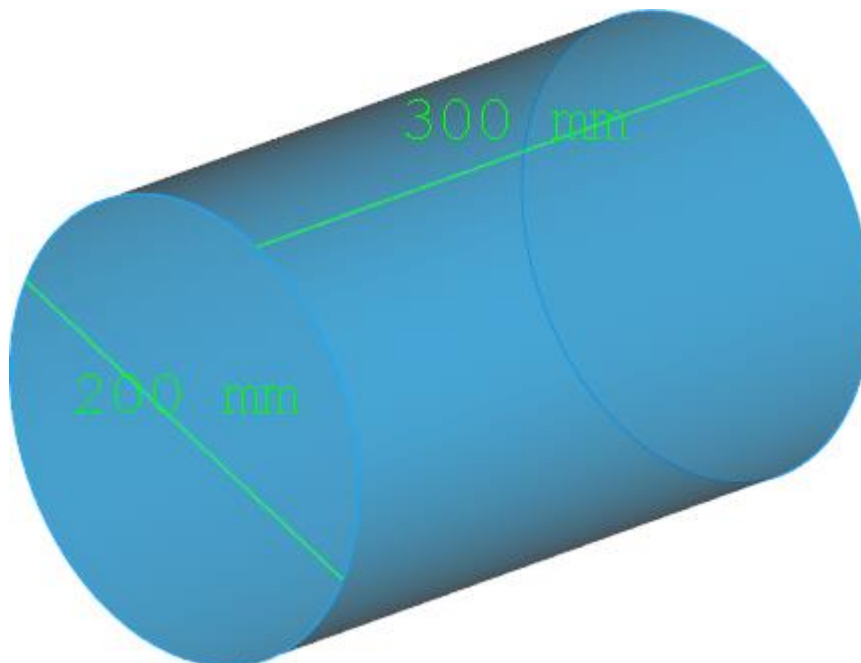



Units button we choose *Millimetres*. The same can be done in the *Basic* tab in the *Circuit Settings* dialogue.

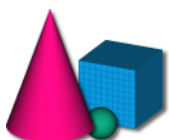


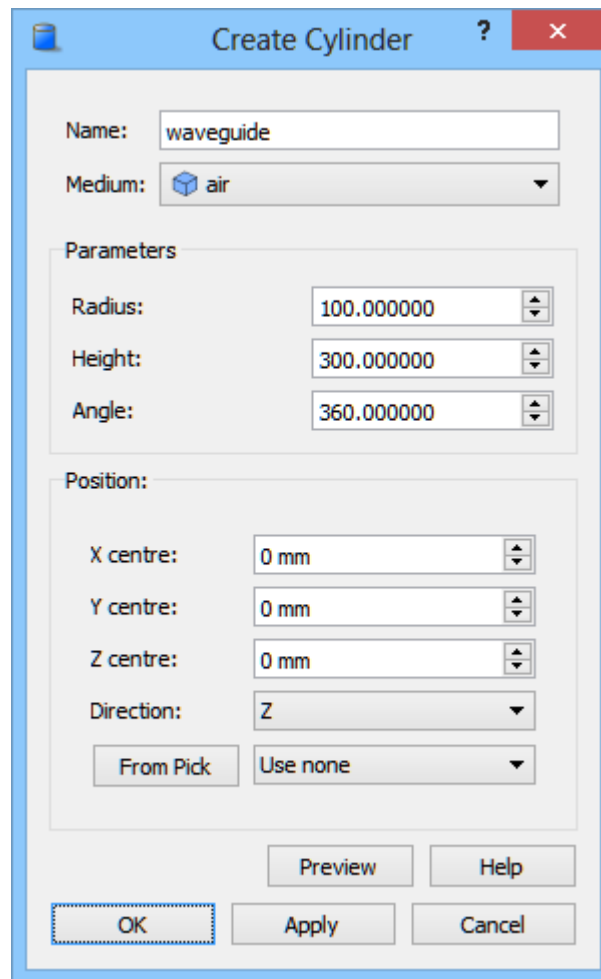


The dimensions of the waveguide are as follow:

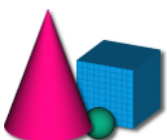


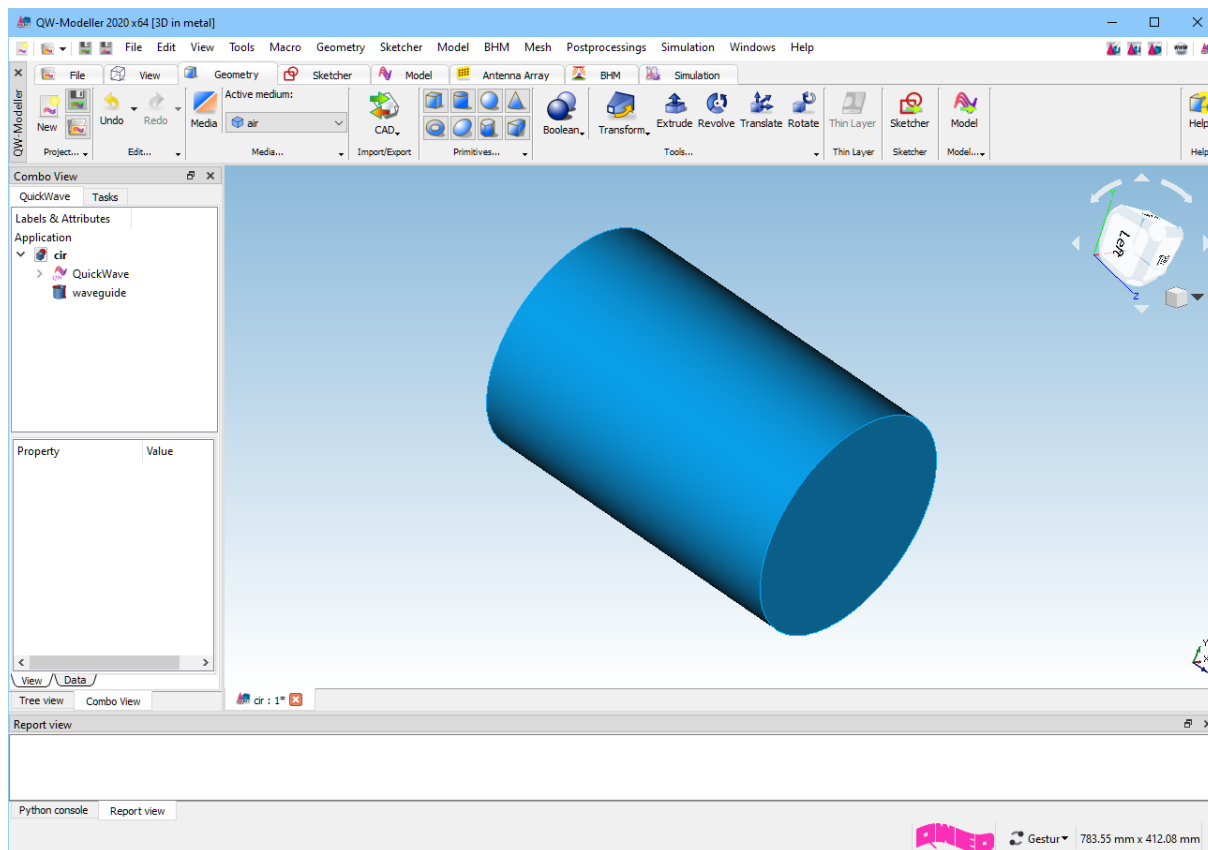
To draw the structure we go to *Geometry* tab. We start from drawing the waveguide's inside. We need to draw only the inside of the structure (the surrounding medium is metal), thus we start from the air cylinder, which stands for the waveguide's inside. To draw the waveguide we press *Create a Cylinder*  button in the *Primitives* section and a dialogue appears.



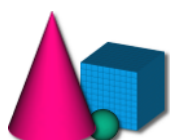


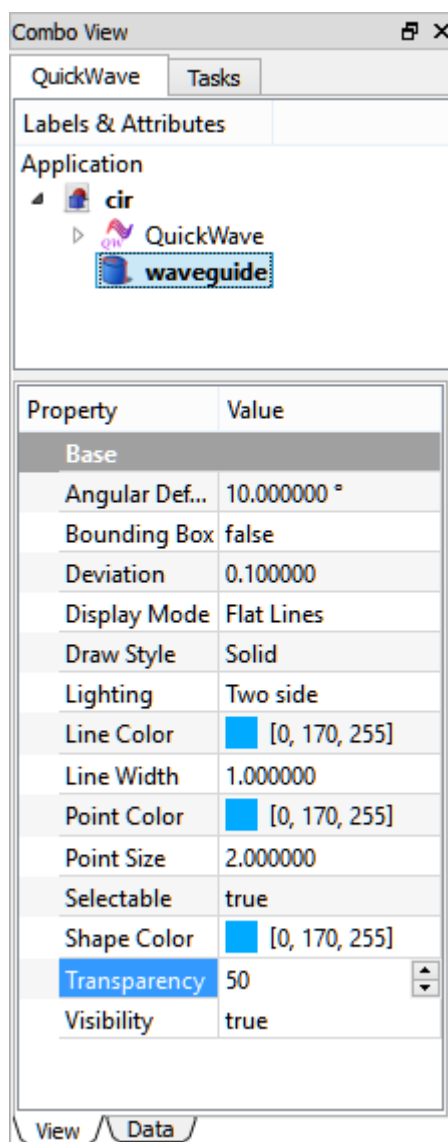
We set the object's name to "waveguide". From the *Medium* combo box we choose air. Now we can proceed to setting the dimensions of the waveguide's inside according to the picture above. With *Position* set to (0,0,0) the waveguide will be drawn so that its centre is placed at these coordinates. After pressing *OK* button the waveguide's inside appears in the project.





For better visualisation of the inside of the waveguide, we change its transparency by selecting *waveguide* object on the *Tree View*, and changing *Transparency* in the *View* tab in *Property Editor* to 50.





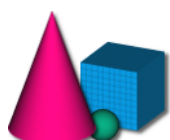


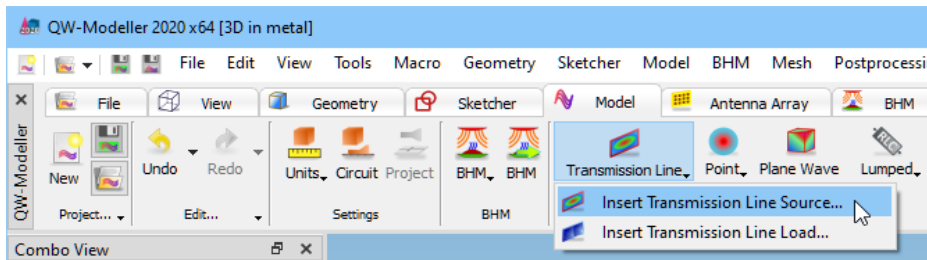
Drawing the ports

In this step we need to draw and define the ports. In this case we need to define two ports, input and output with fundamental TE_{11} mode excitation.

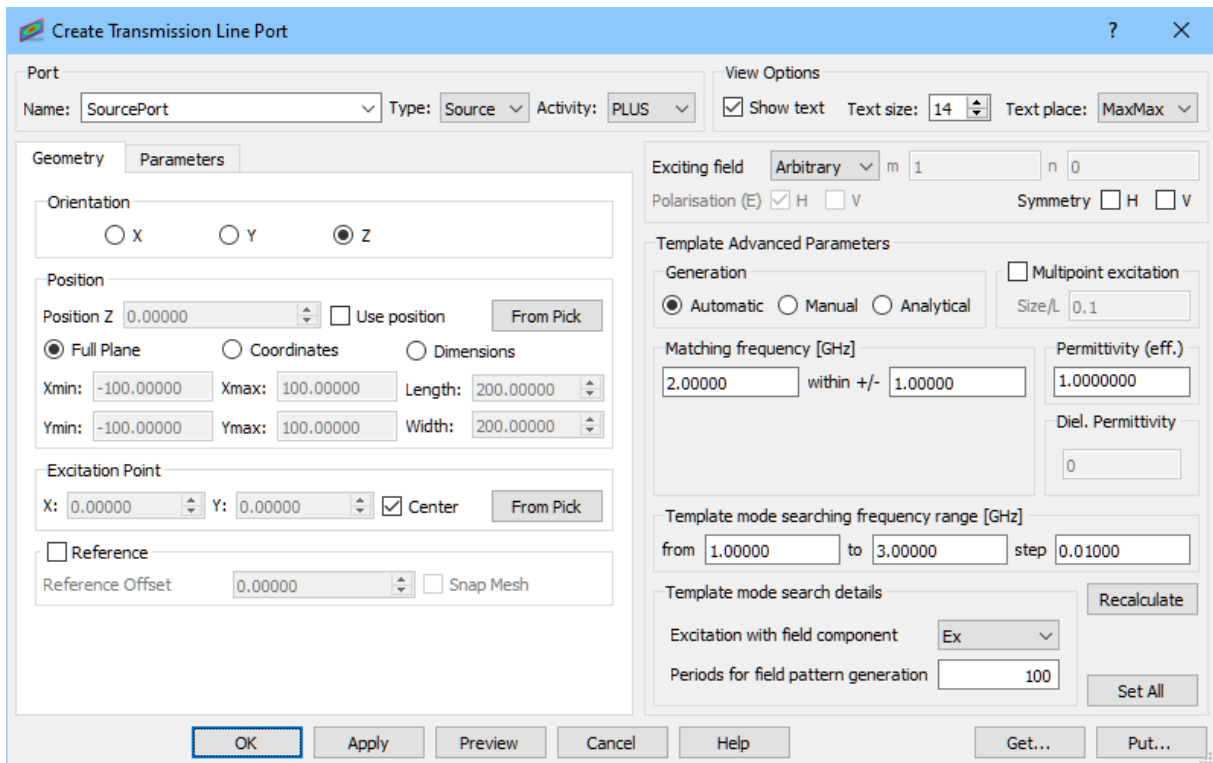
To define the ports we need to go to *Model* tab. As a first one, we will define the waveguide input

port. For that purpose we press *Transmission Line*  *Transmission Line* command button and choose *Insert Transmission Line Source*  *Insert Transmission Line Source...* command.

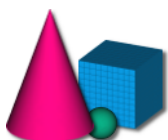


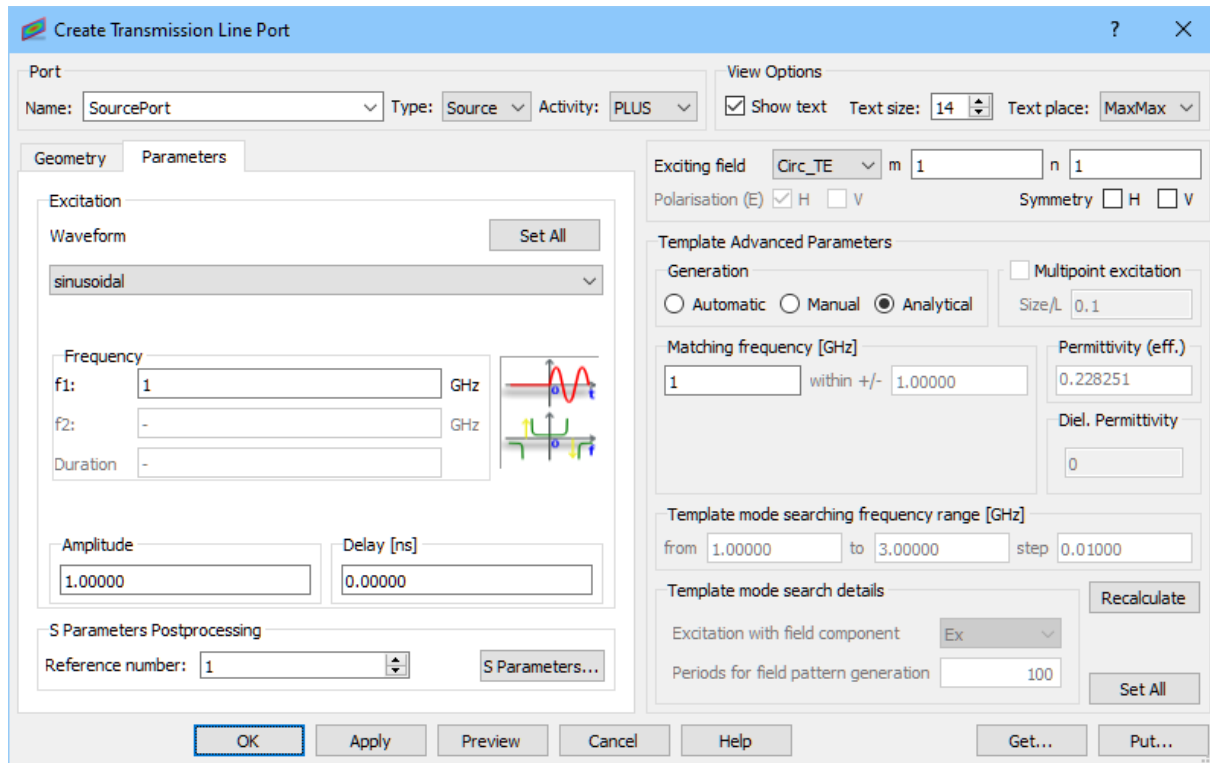


The following dialogue will appear.

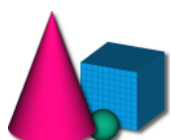
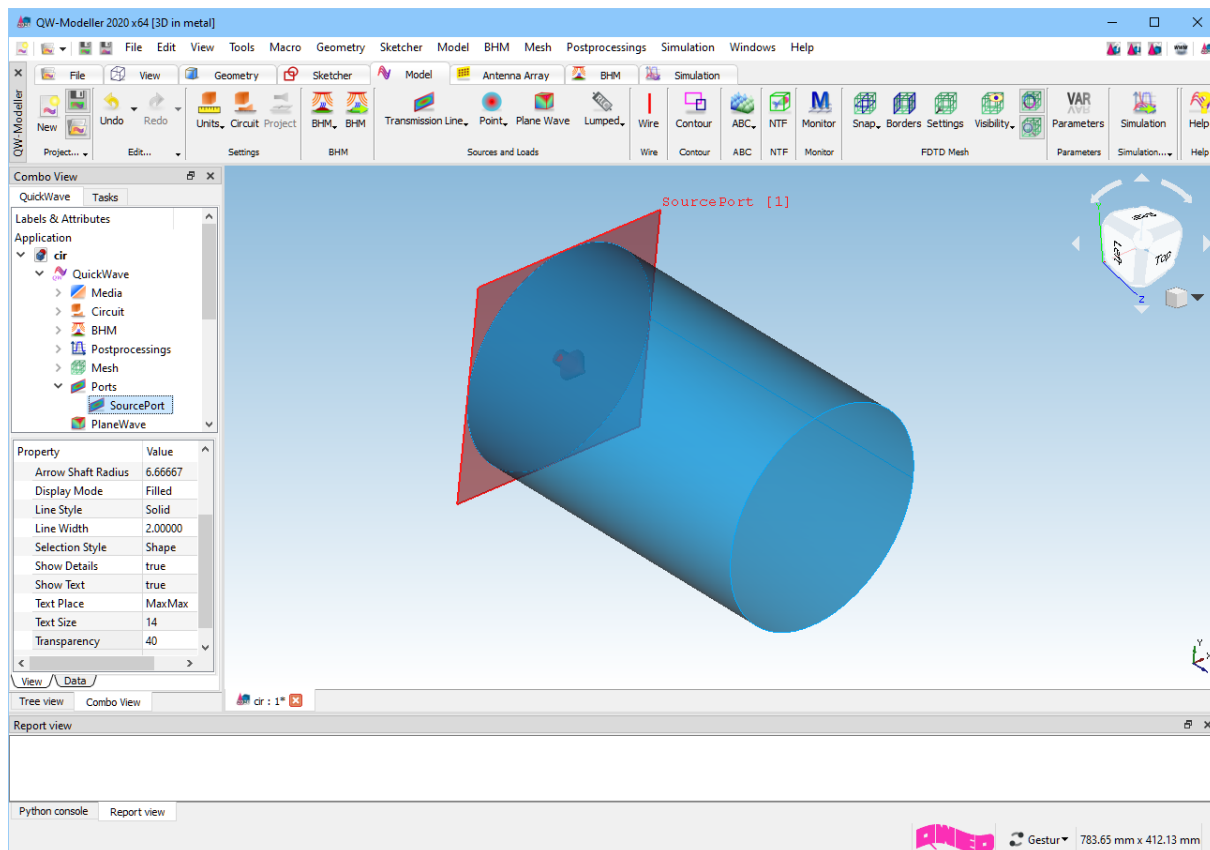


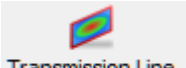

We set the name of the port to *SourcePort*. The default *Type* and *Activity* of the port are properly set for this case. The *Orientation* should be changed to *Z*. *Full Plane* option sets the port dimensions equal to the bounding dimensions of the entire project in the directions perpendicular to the port orientation what is suitable in this case. We uncheck the *Reference* option – it is not necessary since we do not calculate S-Parameters.

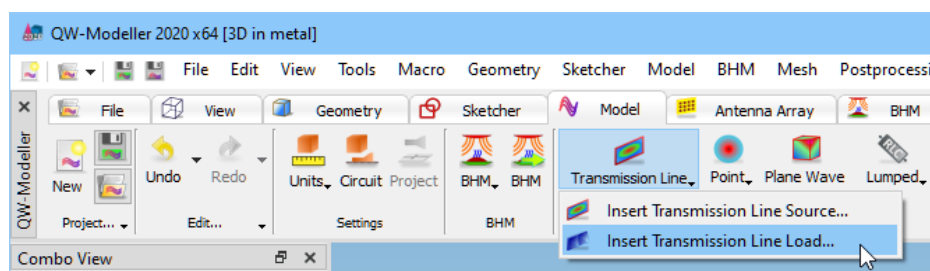




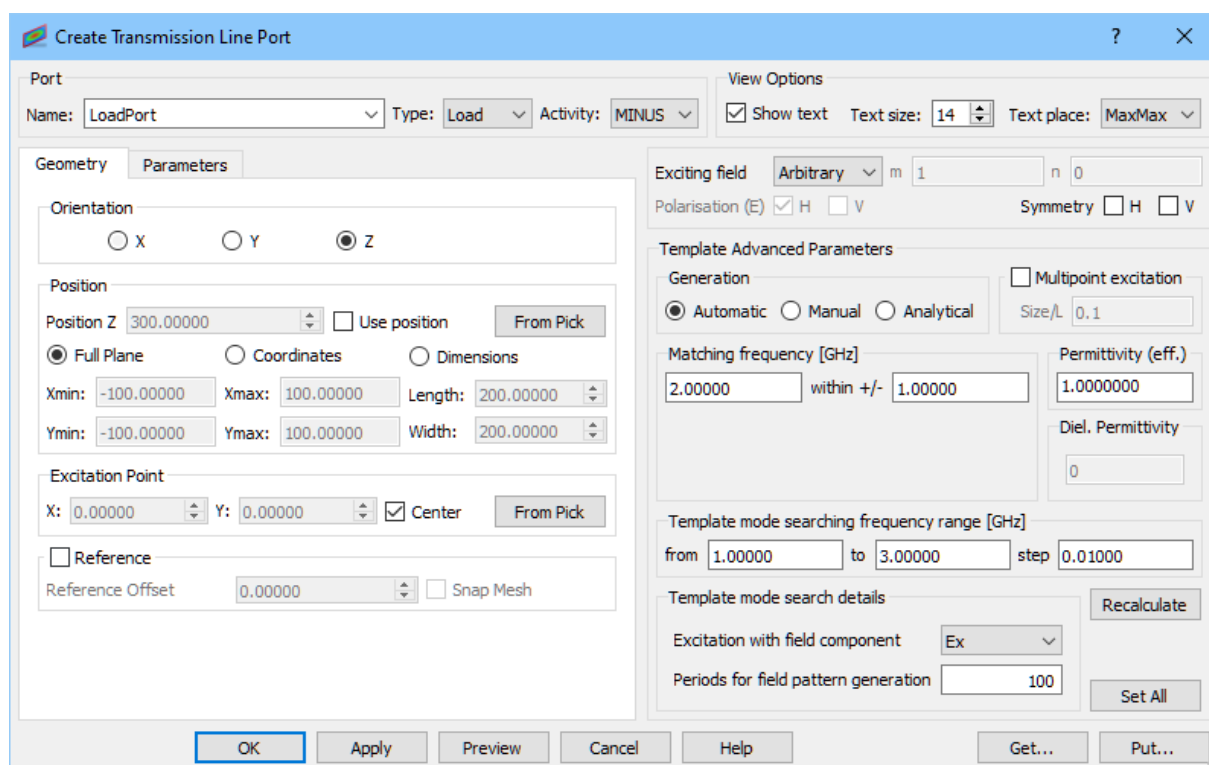
In the *Parameters* tab we set sinusoidal excitation at 1GHz. We excite the structure with a fundamental TE₁₁ mode by setting *Analytical* generation, choosing *Circ_TE* from the *Exciting field* and setting *m* and *n* indexes to 1. Pressing *OK* button accepts the settings.



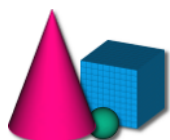
Now press *Transmission Line*  commands button and choose *Insert Transmission Line Load*  *Insert Transmission Line Load...* command.

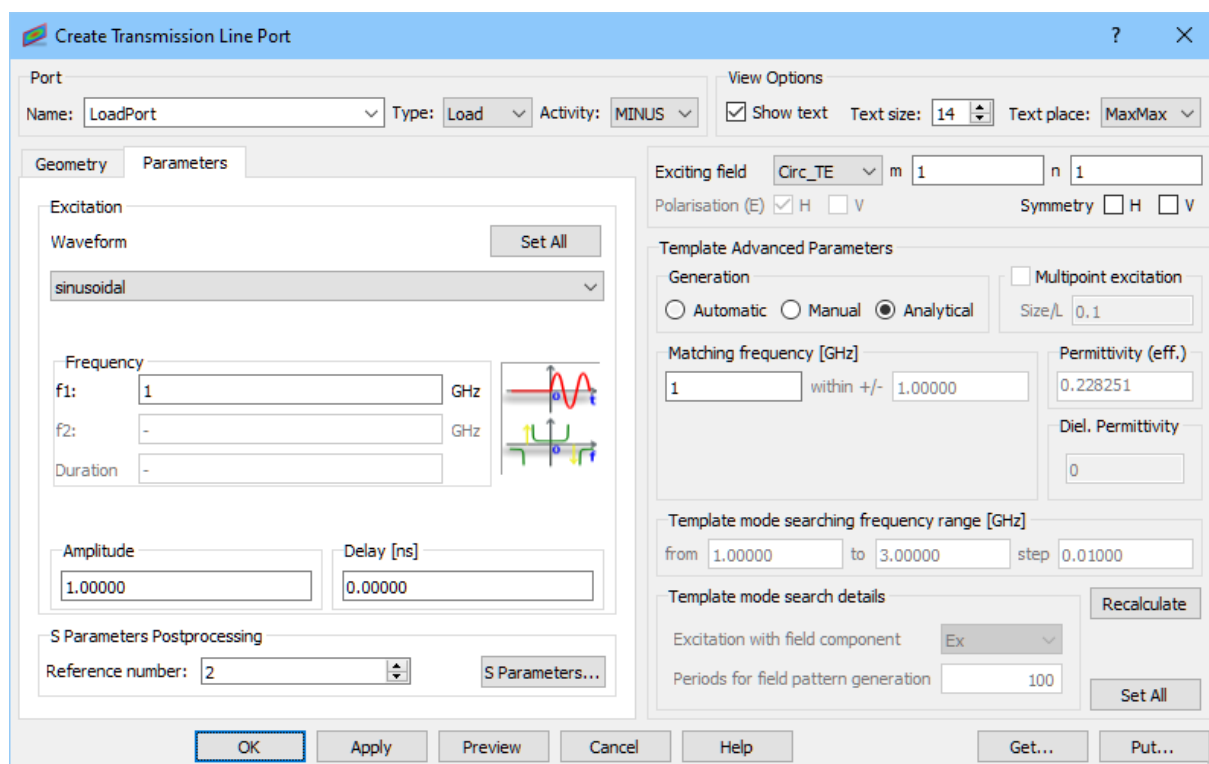


The following dialogue will appear.



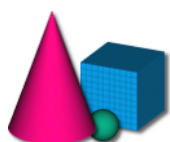
We set the name of the port to *LoadPort*. The default *Type* of the port is properly set for this case. The *Orientation* should be changed to *Z*. We should also change *Activity* to *MINUS* (please note that when *Full Plane* option is checked, port will change its position to the opposite side of the project). We uncheck the *Reference* option – it is not necessary since we do not calculate S-Parameters.



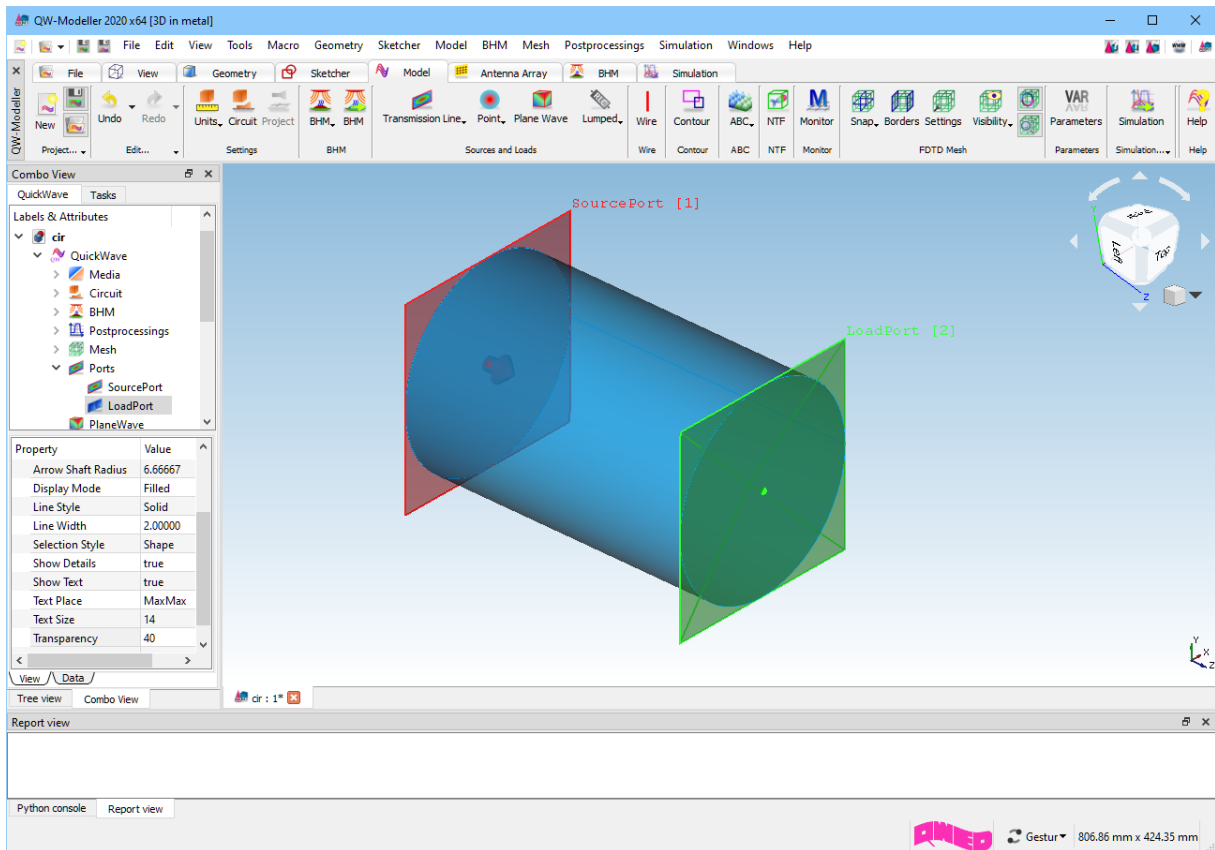


In the *Parameters* tab we set sinusoidal excitation at 1 GHz. We excite the structure with a fundamental TE₁₁ mode by setting *Analytical* generation, choosing *Circ_TE* from the *Exciting field* and setting *m* and *n* indexes to 1. Pressing *OK* button accepts the settings.

The *LoadPort* can be also set by creating a copy of *SourcePort* and changing its parameters.



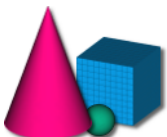
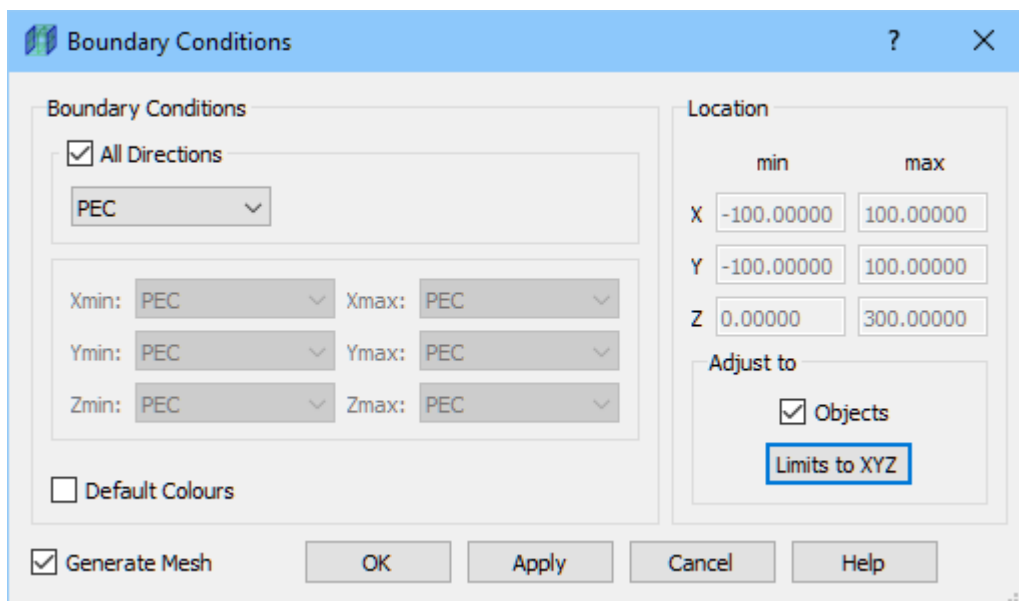
Project Preparation




Setting boundary conditions

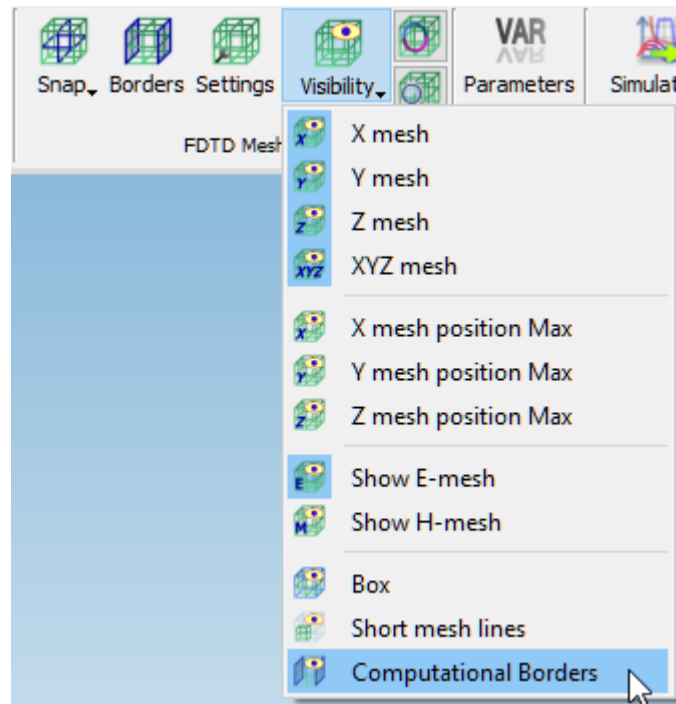


In the *Model* tab we press the *Borders* button. The *Boundary Conditions* dialogue will appear.

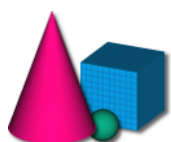
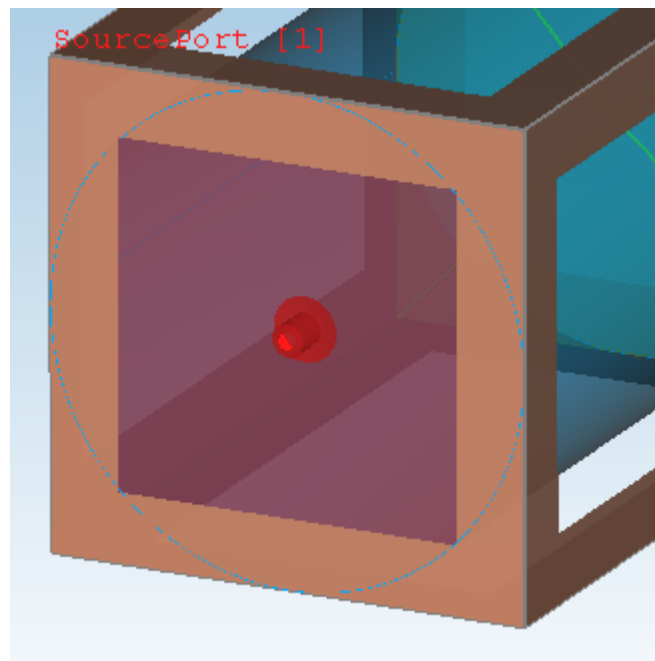


In the *Boundary conditions* frame we leave the default settings (Perfect Electric Conductor boundary conditions in all directions). Please note that boundary conditions' indicators are visible when *Boundary Conditions* dialogue is opened or when visibility for *Computational Borders* is enabled. We

will make *Boundary Conditions* indicators visible using *Visibility*  button.



After zooming the display, we can observe that brown colour indicates PEC boundary conditions.

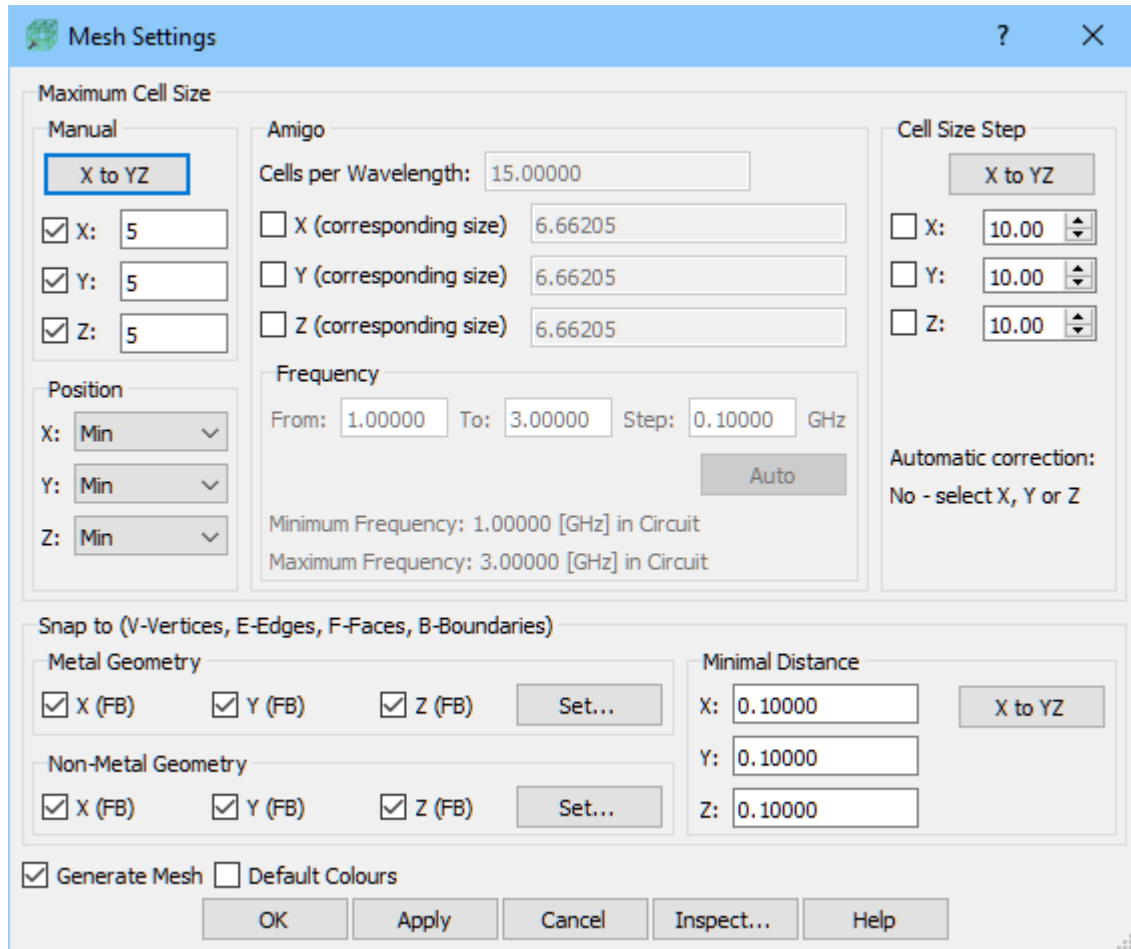


Setting up the mesh

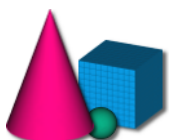
Having the structure's geometry and the ports defined we can now set up the mesh. To do that, in



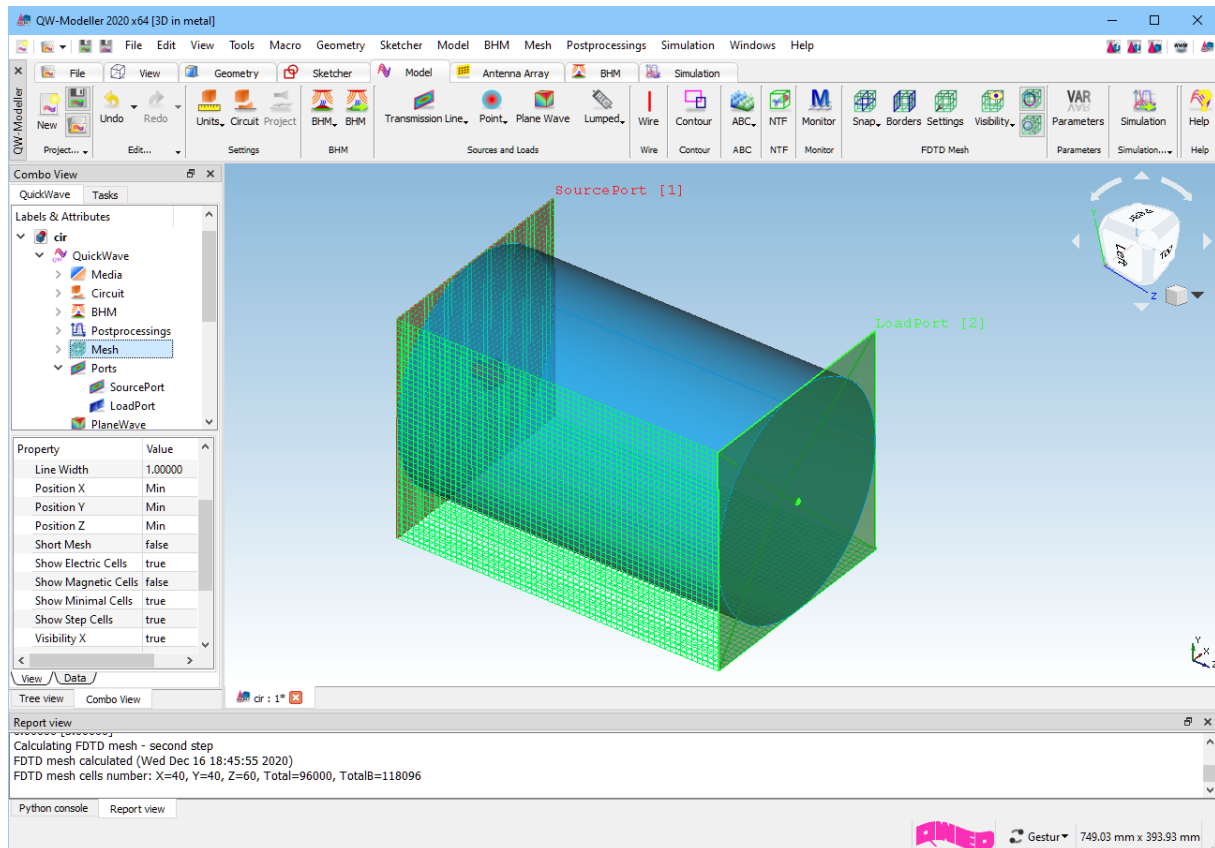
Model tab we press the *Settings* button. The following dialogue will appear.



We set 5mm cell sizes in X, Y and Z direction. The meshing will be enforced in the entire project space.



Project Preparation



Exporting the project and running the simulation

The project is ready and we can run the simulation. For that purpose we switch to *Simulation* tab and



we press **Start** button from the *Export* section. The *QW-Simulator* is run and the simulation starts automatically. For a detailed description of *QW-Simulator* functionalities please refer to *QW-Simulator* manual.

