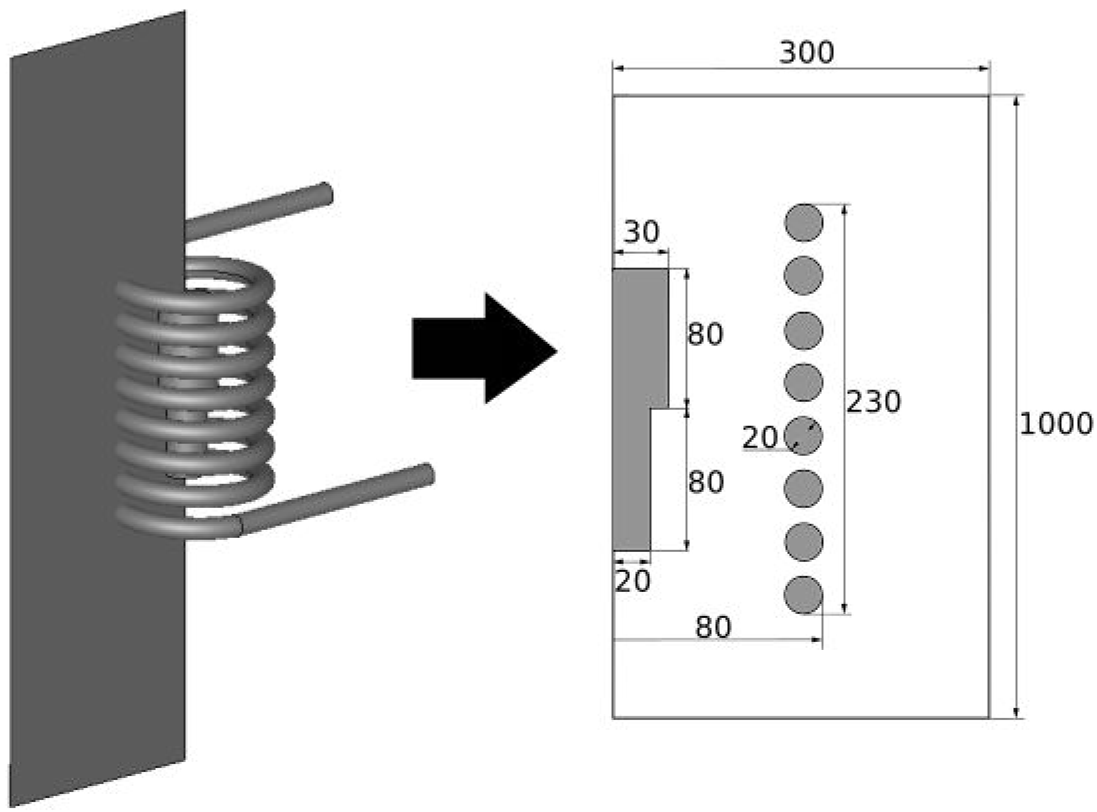
2D Stepped Workpiece Simulation

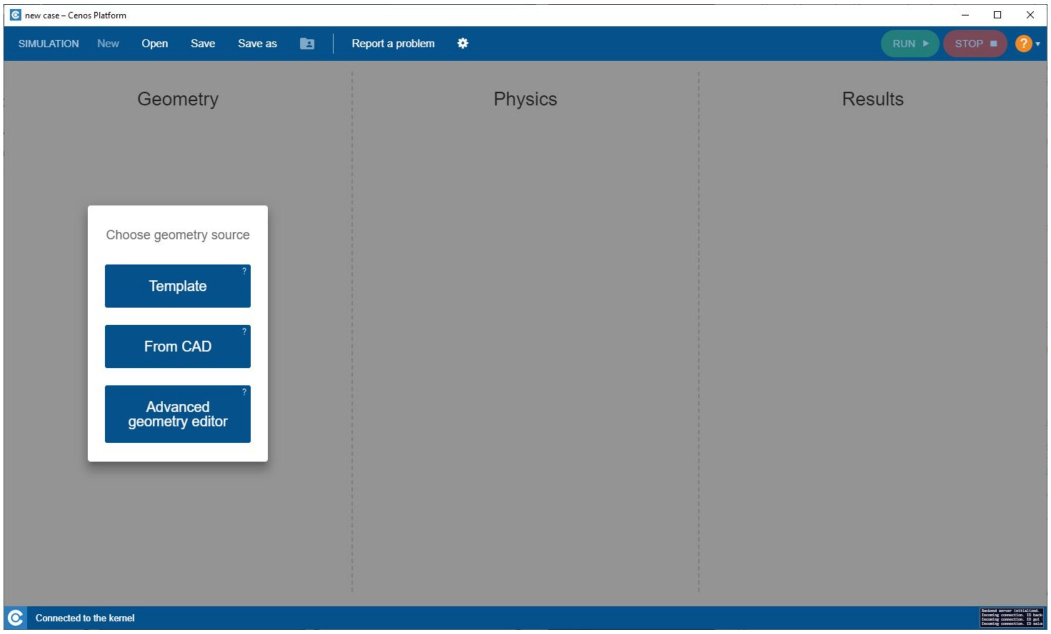
In this tutorial, you will become familiar with the steps needed to perform the preparation, setup and post-processing for **2D axial symmetric induction heating** simulation of **stepped workpiece.** We will create geometry and mesh using CENOS pre-processing tool, then enter specific values and boundary conditions in physics part and in the end evaluate the results using our post-processor.

2D simulation is an easy way to predict thermal and electromagnetic fields within the object of induction heating. In this tutorial an induction heating example of an **AISI 1020** workpiece at **10 kHz** and **4 kA** with radiation and convection boundary conditions is presented.

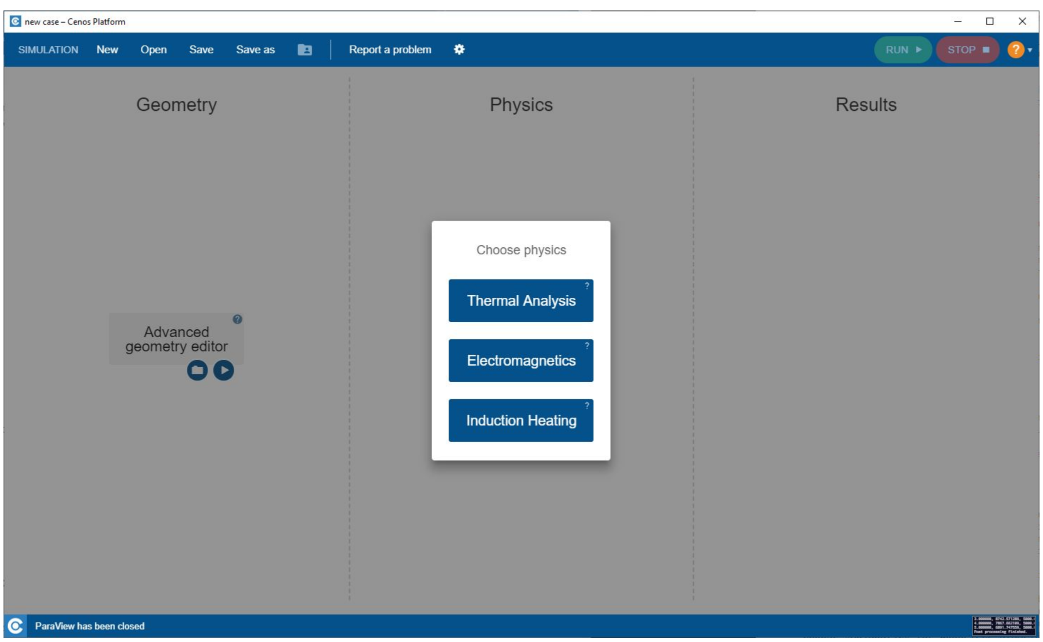


1. Open pre-processor

To manually create geometry and mesh, in CENOS home window click **Advanced geometry editor**.



Click **Induction Heating** to select physics for simulation.

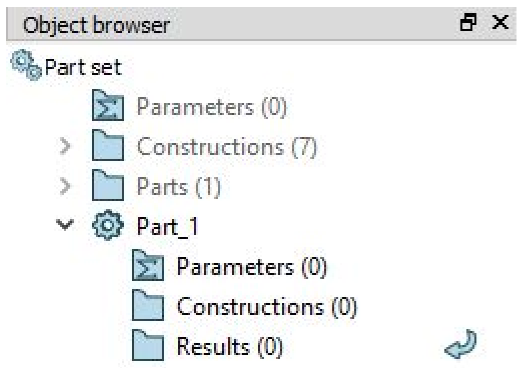


Salome window with already selected **Shaper module** will open.

2. Create geometry and prepare it for meshing

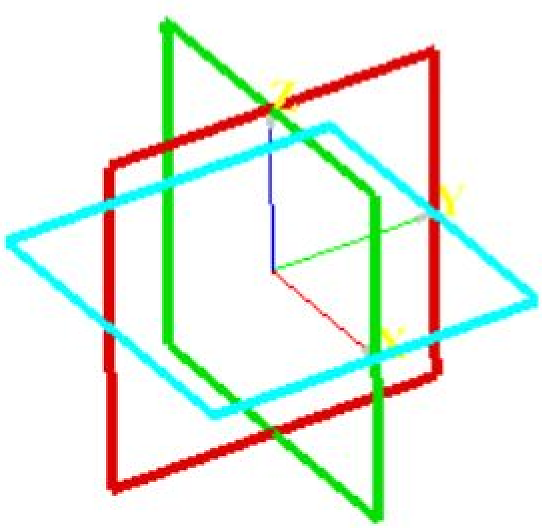
2.1 Create a new sketch

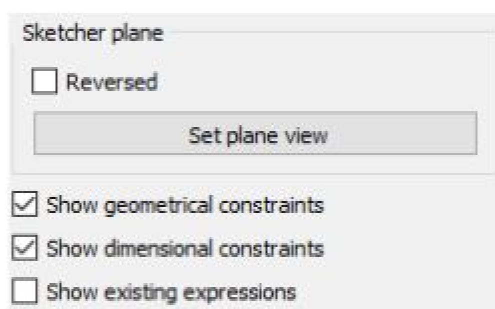
Create a new *Part* by clicking the *New part* () tool. A new part will be added to *Object browser.*



Now create a new *Sketch* by clicking the *Sketch* (

) icon. Select the **XY plane** and click *Set plane view*.





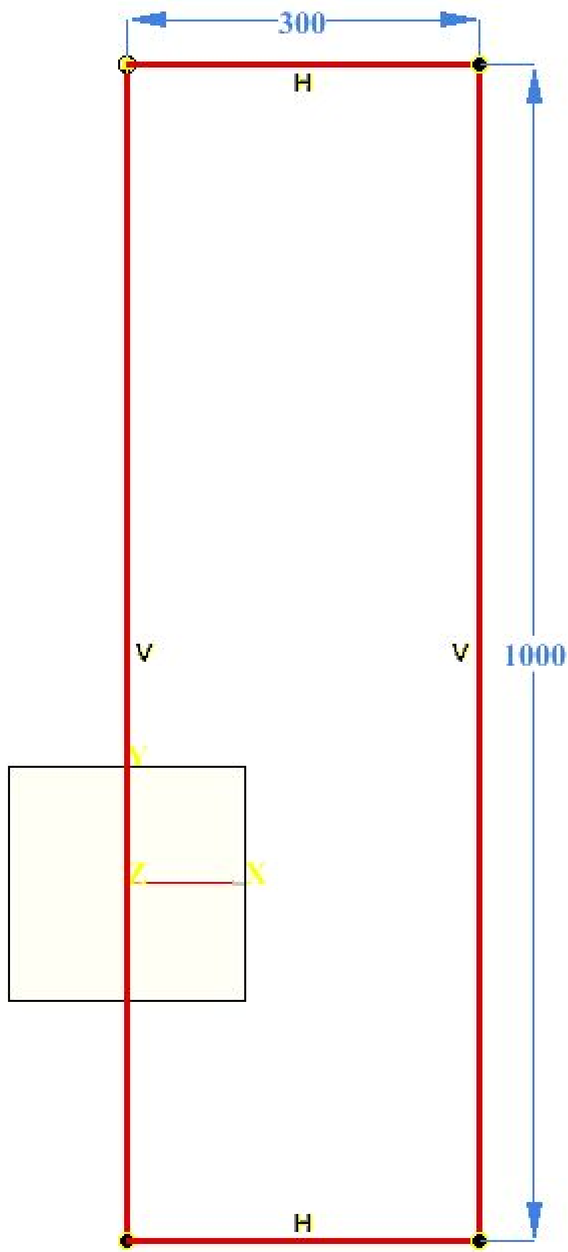
You have now created an **active sketch**, in which you can **start to build your geometry**!

2.2 Create an air box

Select *Rectangle* ( ) tool and with a free hand draw a rectangle which **left edge coincide with OY axis.**

IMPORTANT: To simulate axial-symmetric cases, the **symmetry axis must be Y axis.**

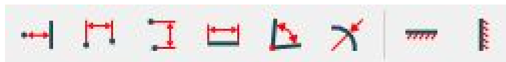
Select *Length* ( ) tool and **define the size of the air box** (300mm x 1000mm).

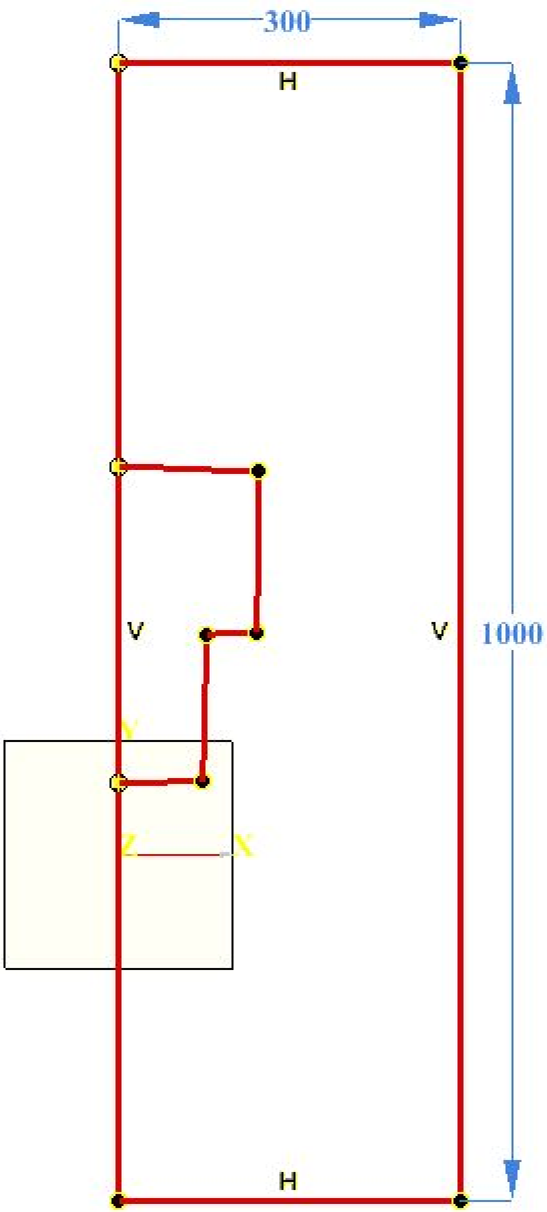


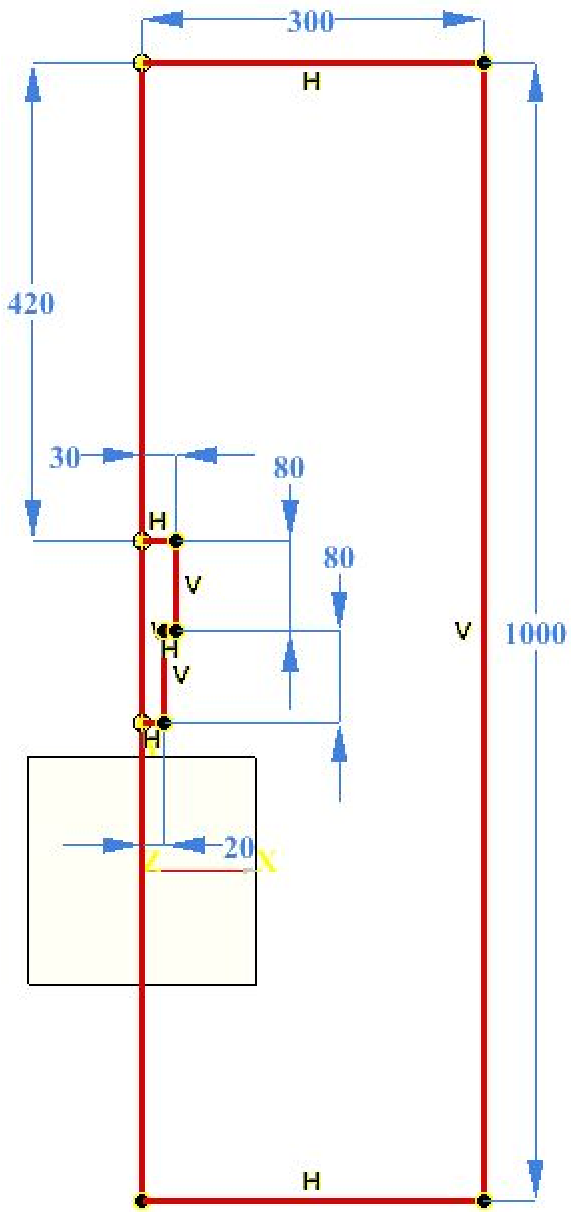
2.3 Create workpiece

Select *Line* () tool and with a free hand draw a *Stepped Shaft* outline.

Then by using *Horizontal* and *Vertical* ties and *Length* and *Distance* tools **define the size and position of the outline** based on the sketch presented in the beginning of this tutorial.





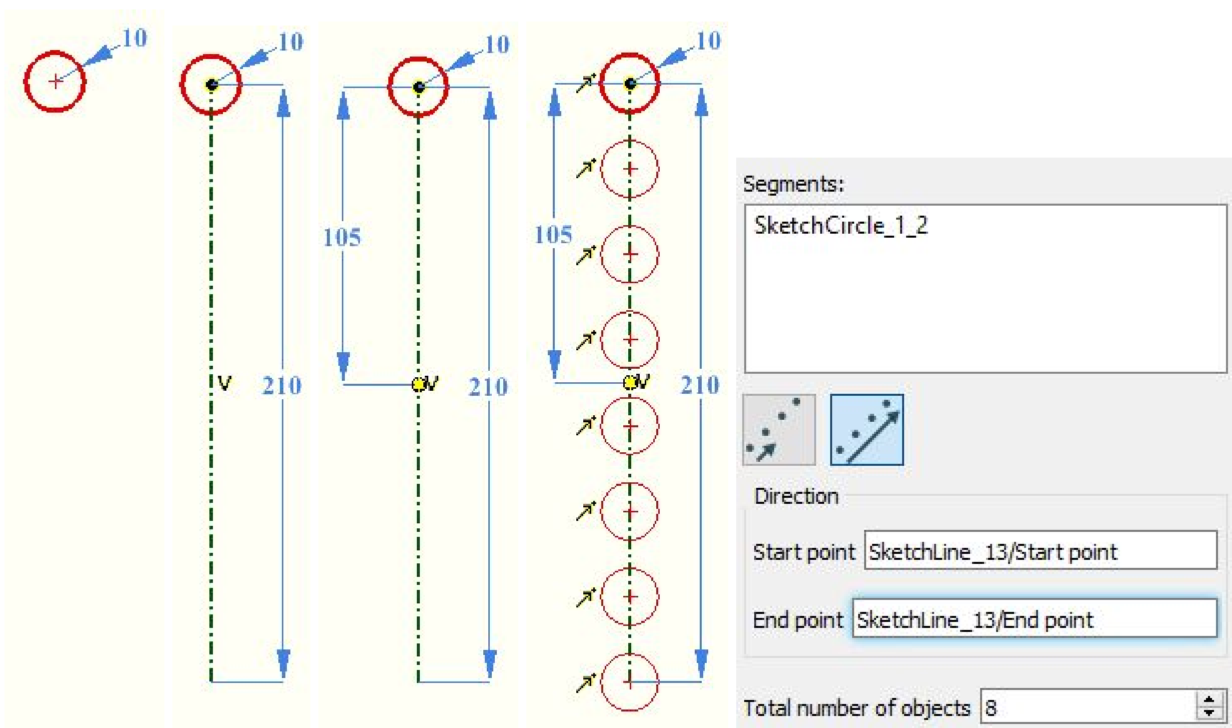


2.4 Create coil windings

First you need to create windings separately and then align them with the workpiece.

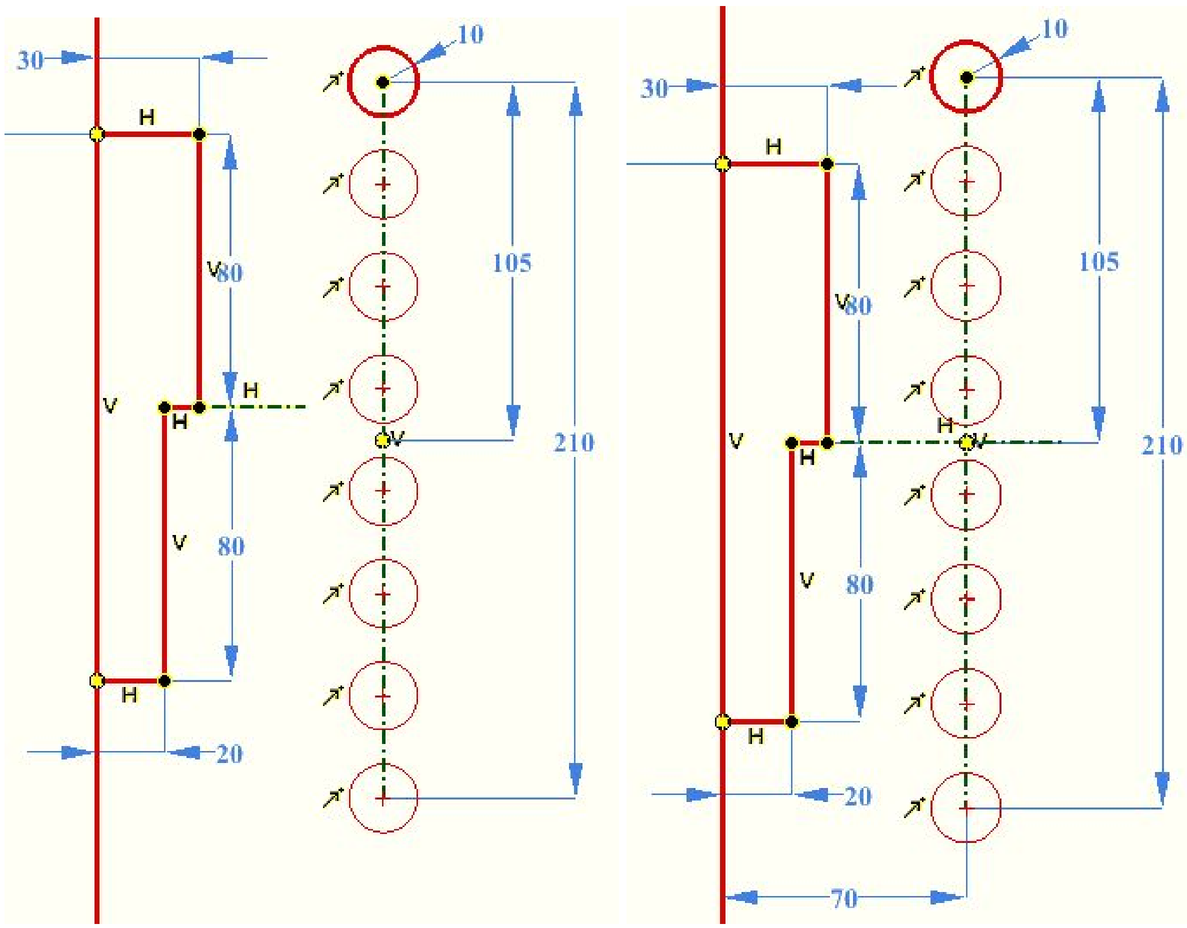
To create coil windings:

1. Select *Circle* ( ) tool and with a free hand **draw a circle.** Using *Radius* ( ) tool **define the size of the circle** (10 mm radius).
2. Select *Line* ( ) and draw an auxiliary line from the center of the circle. Using *Vertical* ( ) and *Length* ( ) tools define the position (**vertical**) and size (**210 mm**) of the line.
3. Select *Point* ( ) tool and **create a point in the middle of the line.** By using *Distance* ( ) tool define the point distance from one end of the line (**105 mm**).
4. Select *Linear copy* ( ) tool. Select circle as *Segments*,the circle centre point as *Start point* and the other end of the line as *End point*, and translate the circle along the line 8 times (*Total number of objects*).



To align windings with the workpiece:

1. Using *Line* () tool draw a horizontal auxiliary line from the shaft step corner. Use the *Horizontal* () constraint to define the position of the line (**horizontal**).
2. Select *Coincident* () tool to align the newly created auxiliary line with the center point of the winding centre line. Set the distance between air box outer edge and winding centre line (**70 mm**)

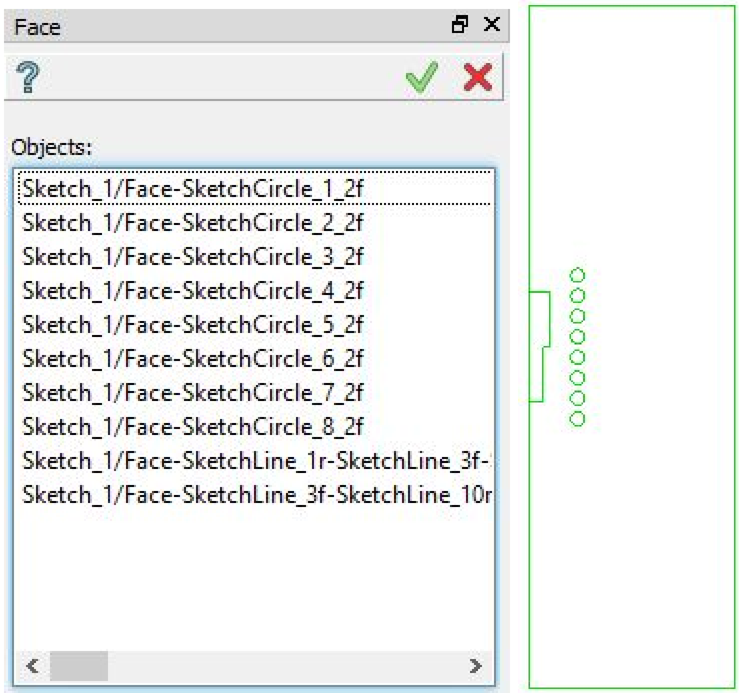


When sketch is finished, click *Apply* () in Sketch window.

2.5 Create faces for your geometry

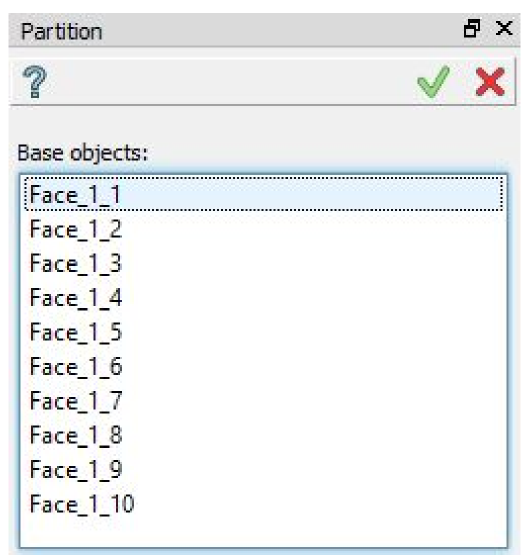
Select *Face* () tool and create faces for each winding, workpiece and air.

IMPORTANT: You can select multiple geometry objects by holding the *Shift* button and clicking on the objects of interest.



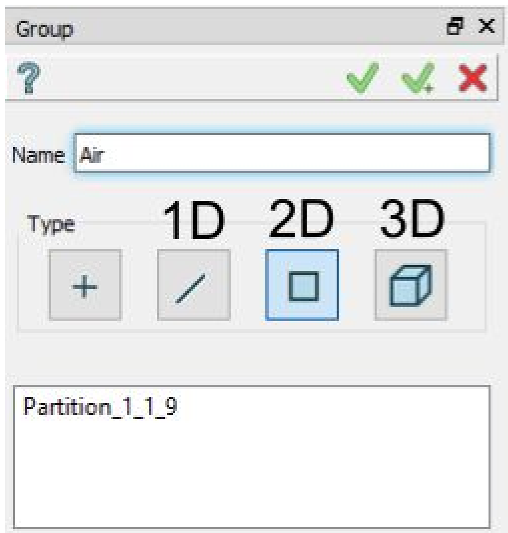
2.6 Create Partition and Groups

Click *Partition* () tool, select previously created faces and join them in one partition.



IMPORTANT: Partition and Groups are vital for simulation setup with CENOS, because mesh creation as well as physics and boundary condition definitions are based on groups created in this part.

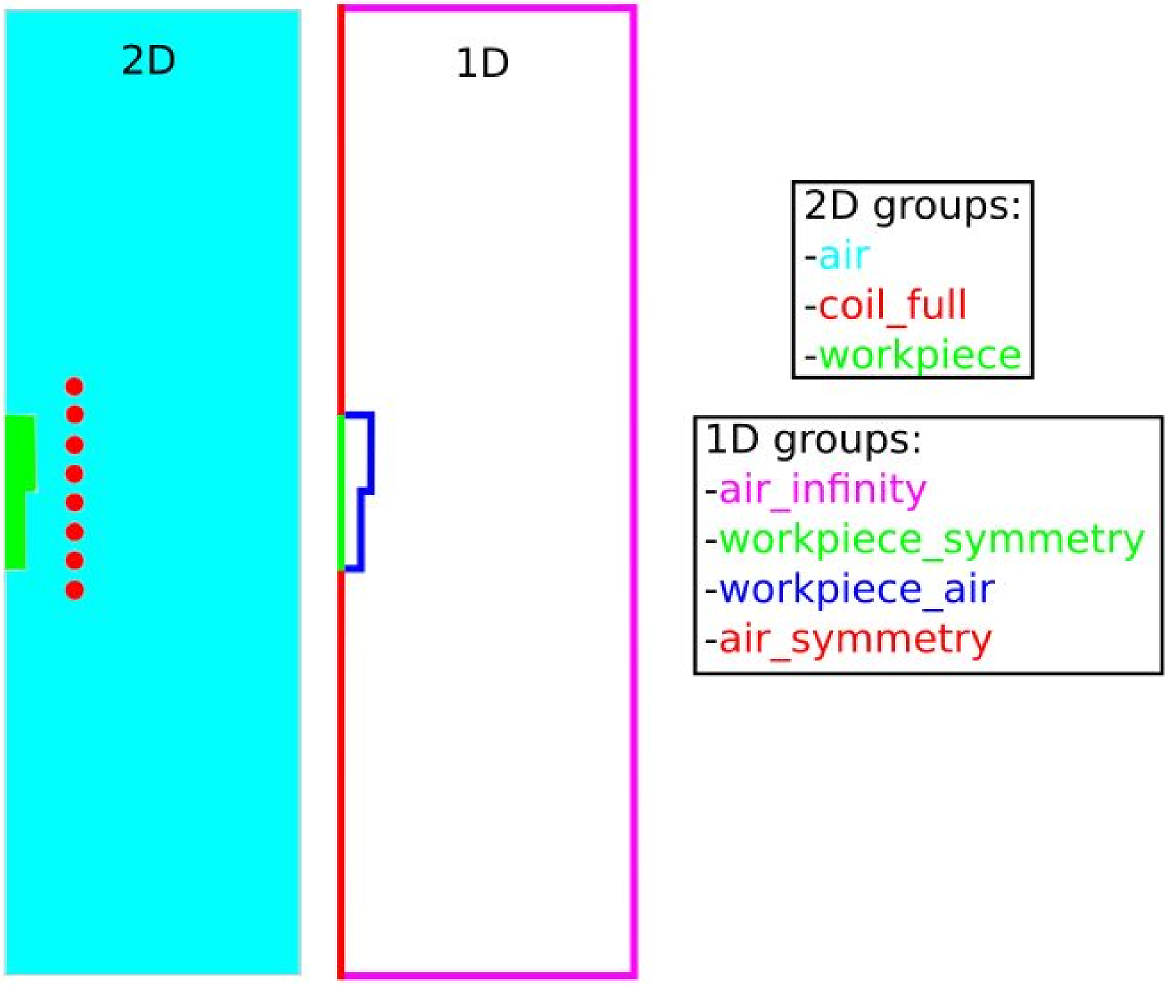
Select *Group* ( ) tool and choose the *Shape Type*. Select one or more shapes from the screen, name the group and click the *Apply and continue* ( ).



For this tutorial we will create eleven 2D groups for domains and four 1D groups for boundary conditions. When creating groups, **select only those objects relevant for the specific group**.

IMPORTANT: For the coil **create one 2D group with all of the windings** in it (*coil_full*), which will be used to ease the meshing of the coil, but also **create separate 2D group for each winding** (*c1*, *c2*...), because these will be used to define current flowing through each winding in the physics setup part.

A detailed breakdown of these groups is as follows:



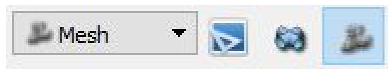
2.7 Export to GEOM

Finally we need to export the geometry created in *Shaper* to GEOM module. Do this by clicking *Export to GEOM* ( ). This will export the *Partition* and *Groups* to GEOM module, which is needed to proceed with mesh creation.

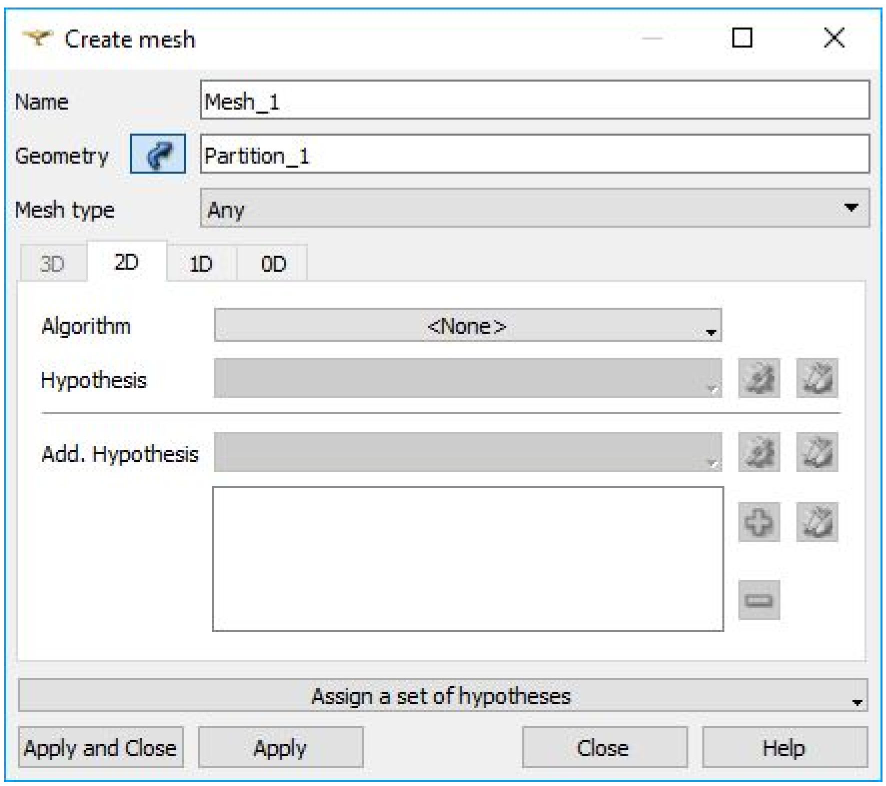
3. Create mesh and export it to CENOS

3.1 Switch to Mesh module and create Mesh

Switch to the Mesh module through *Mesh* icon or select it from the Salome module dropdown menu.



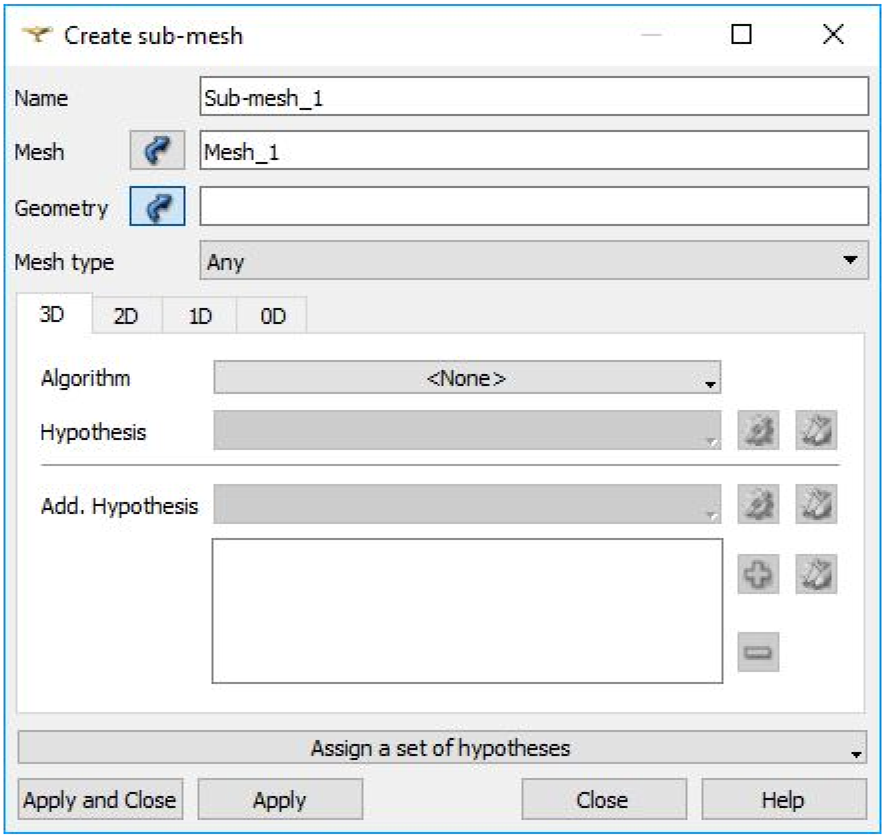
In *Object Browser* from *Geometry* dropdown menu select the previously created *Partition_1_1* and click *Create Mesh* ().



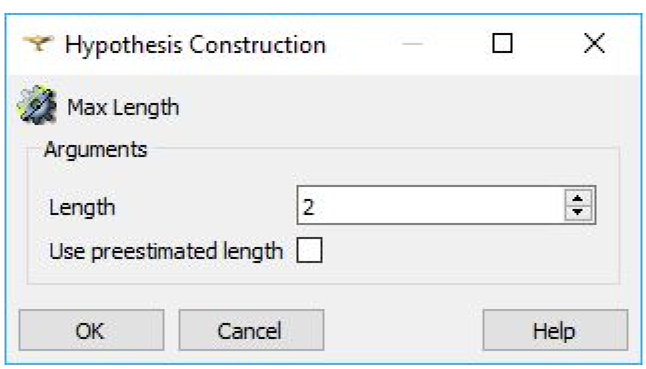
From the *Assign a set of hypothesis* dropdown menu select *2D: Automatic Triangulation* - leave the *Max Length* value default and click *Apply and Close*.

3.2 Create a sub-mesh for the workpiece

Right-click on *Mesh_1* and click *Create Sub-Mesh* or select *Create Sub-mesh* ( ) from the toolbar.

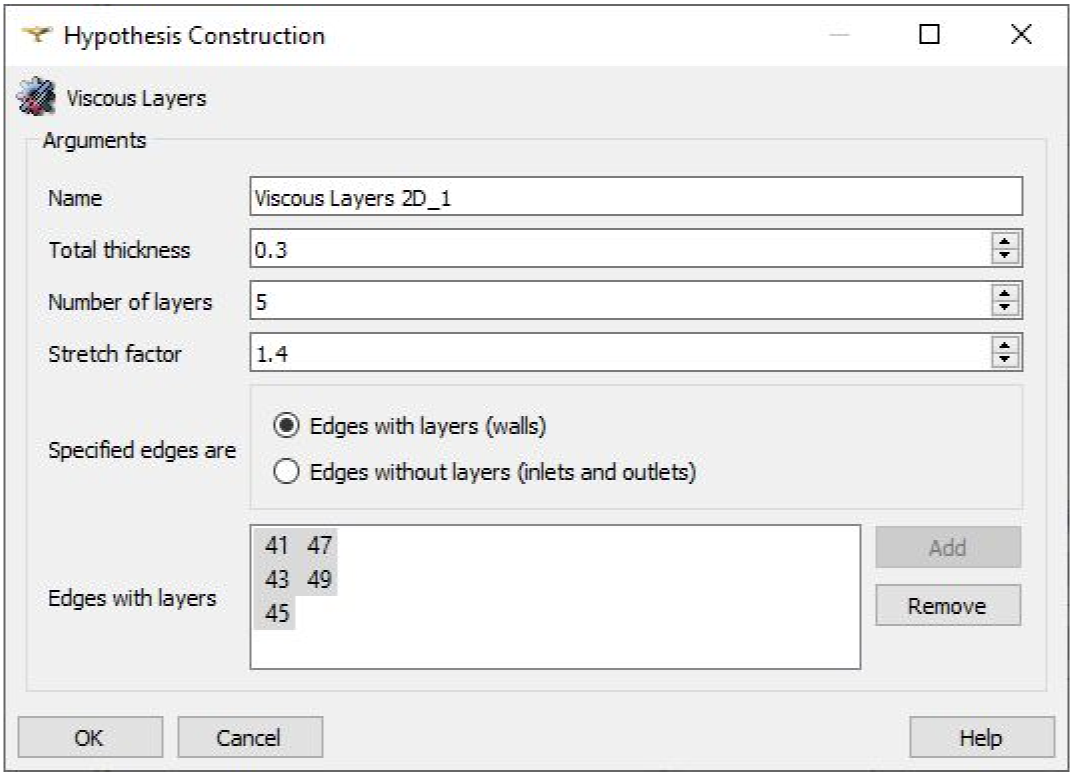


Choose *workpiece* group from the Partition_1 dropdown menu as *Geometry*. From the *Assign a set of hypothesis* dropdown menu choose *2D: Automatic Triangulation*. In the *Hypothesis Construction* window enter **2** for *Max Length*.



Resolve the skin layer on the surface of the workpiece by creating *Viscous Layers*. Click the gear icon ( ) next to *Add. Hypotheses* and select **Viscous Layers 2D**.

Select the group *workpiece_air* from the *Partition_1_1* dropdown menu and click *Add*. Enter **0.3** for *Total thickness*, **5** for *Number of layers*, **1.4** for *Stretch factor* and check the *Edges with layers (walls)* box.



When all is set, click *Apply and Close*.

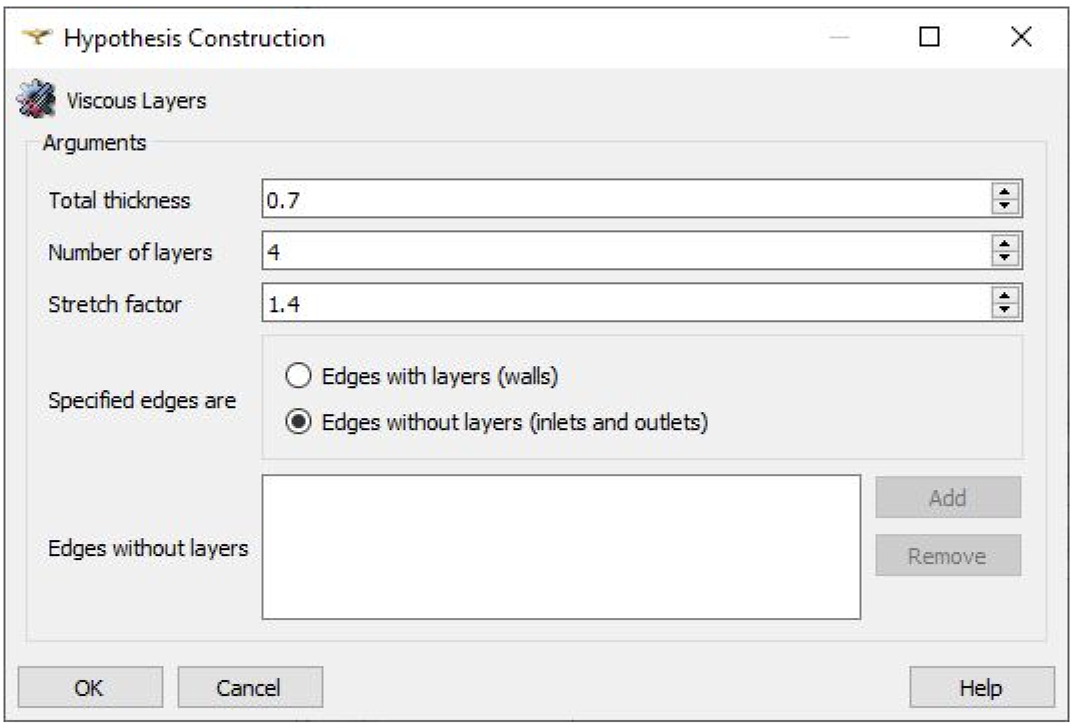
3.3 Create a sub-mesh for the coil

If we had only created groups for each winding separately, we would need to mesh each winding separately, which would be time consuming. For this reason, we created a group with all of the windings together.

Create a sub-mesh and select the *coil_full* group from the *Partition_1* dropdown menu as *Geometry*. From the *Assign a set of hypothesis* dropdown menu choose *2D: Automatic Triangulation* and enter 2 for *Max Length*.

Resolve the skin layer on the surface of the workpiece by creating *Viscous Layers*. Click the gear icon () next to *Add. Hypotheses* and select **Viscous Layers 2D**.

Enter **0.7** for *Total thickness*, **4** for *Number of layers*, **1.4** for *Stretch factor* and check the *Edges without layers (inlets and outlets)* box.

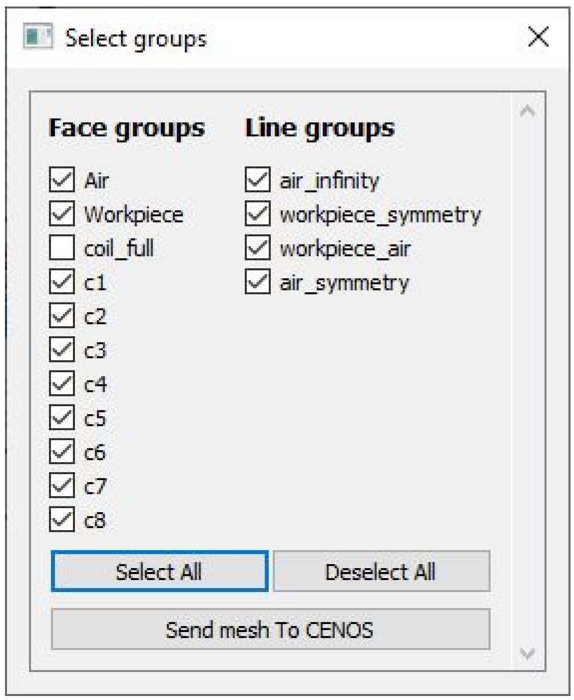


3.4 Calculate and export mesh to CENOS

Right-click on Mesh_1 and click *Compute*. Evaluate the final mesh and export it to CENOS. To do that, select from the dropdown menu under *Tools* → *Plugins* → *Mesh to CENOS* to export your mesh to CENOS.

Before exporting mesh to CENOS, the *Select groups* window will open and you will be asked to select the groups you want to export along with the mesh.

Select all groups relevant for the physics setup, i.e. those who will be defined as domains or boundary conditions. We will select all groups except *coil_full*.

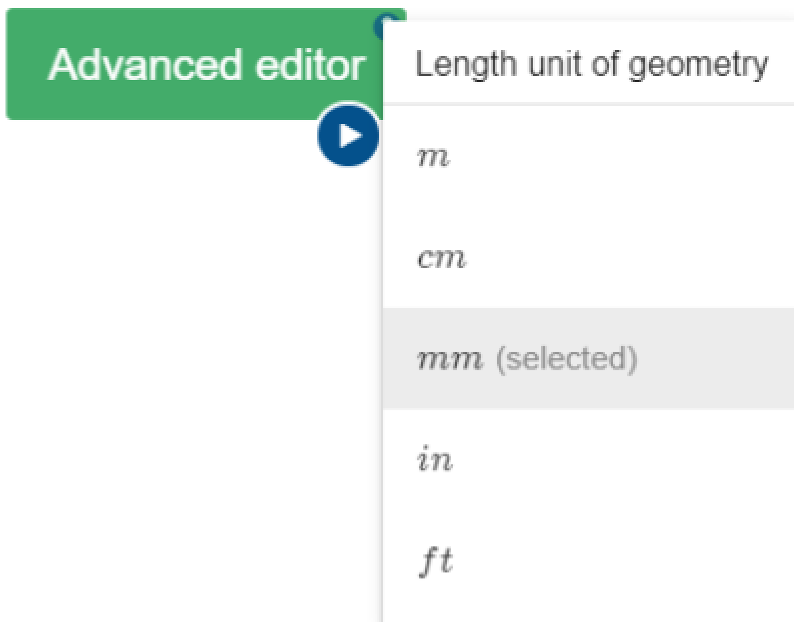


When selected, click *Send mesh to CENOS*.

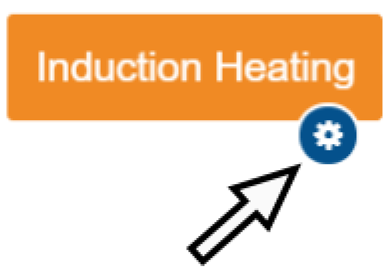
4. Define physics and boundary conditions

4.1 Set units and enter physics setup

Wait until the mesh loads (see the spinner) and **select the units** by clicking on the **gear icon** next to the pre-processing block. In this tutorial we will select **millimeters (mm)**.



Click the **gear icon** under *Induction Heating* block to enter the physics setup.

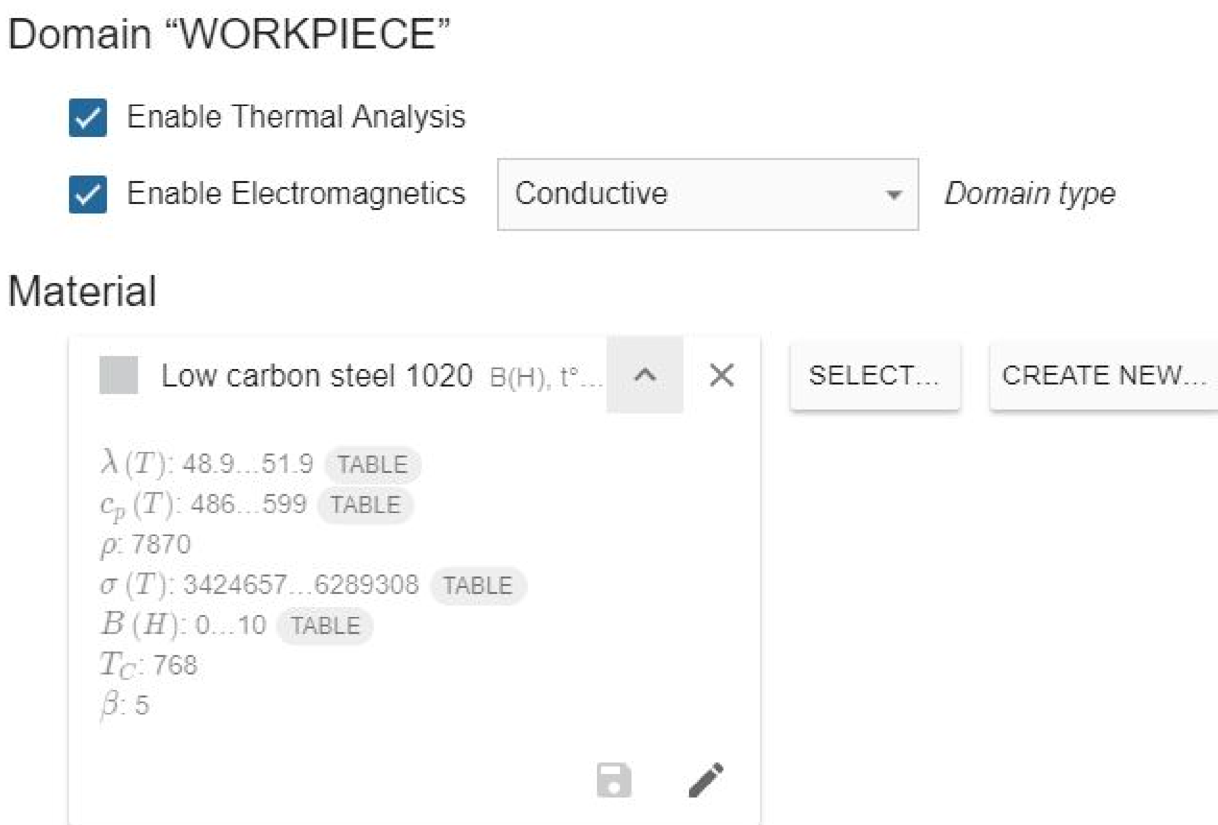


4.2 Simulation control

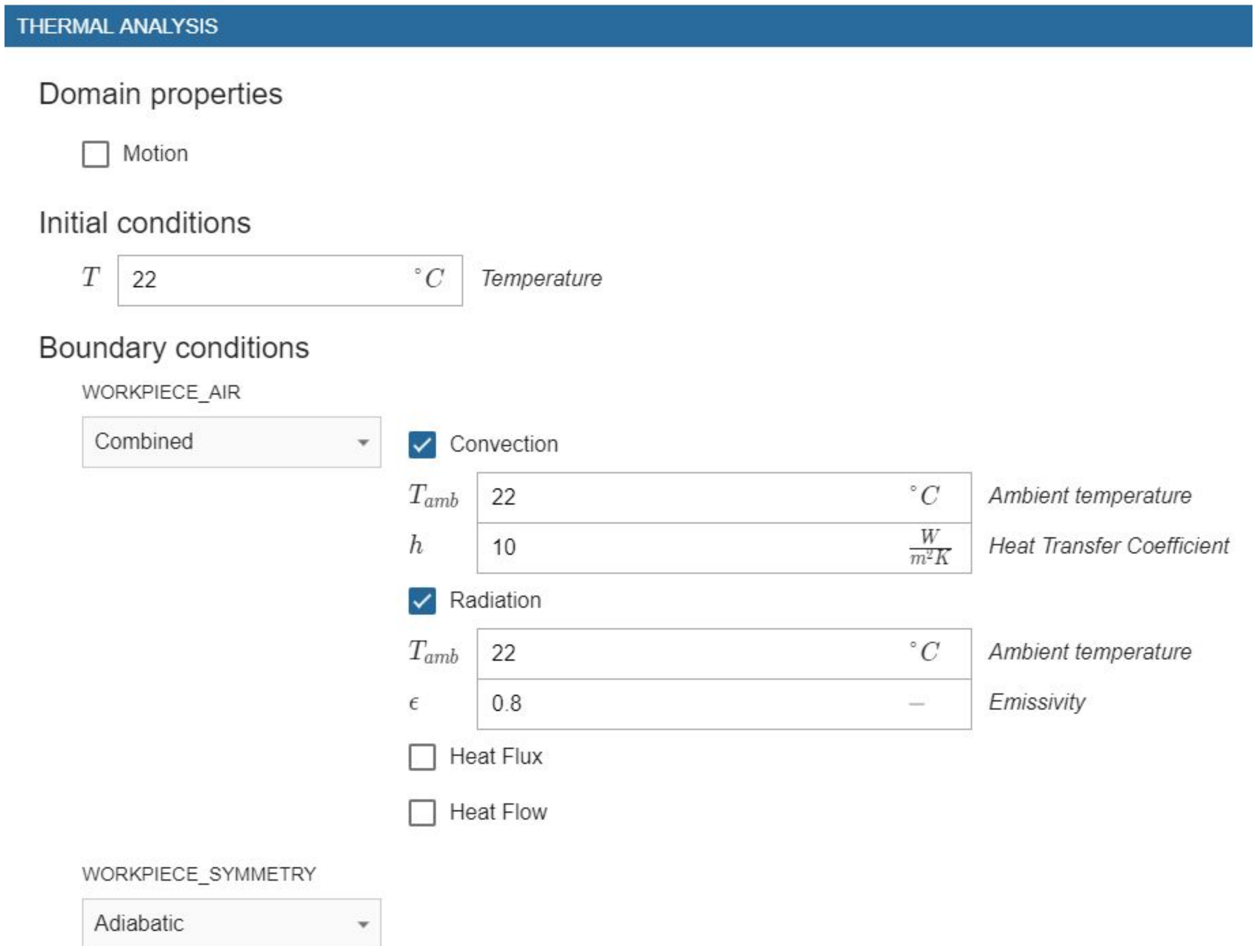
In SIMULATION CONTROL window define the simulation as *axial symmetric* and *transient* with **10 kHz** *frequency*, **10 s** *End time* and **1 s** *time step.* For *Computation algorithm* choose **Accurate**.

4.3 Workpiece definition

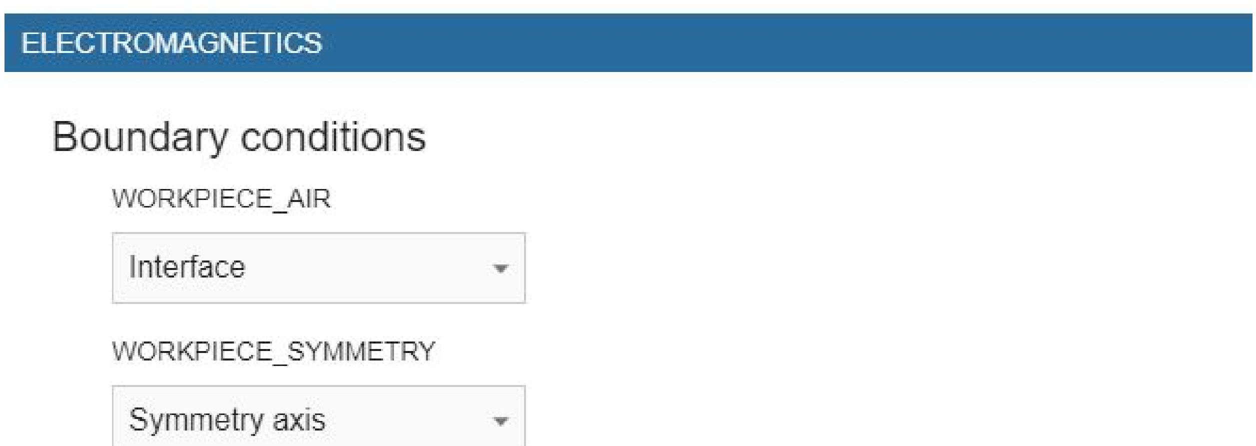
Switch to WORKPIECE in *Domain bar*. Leave *Enable Thermal Analysis* and *Enable Electromagnetics* boxes checked under the Domain “WORKPIECE”. Choose **Conductive** as the domain type. For *Material* click SELECT… and choose **Low carbon steel 1020 B(H), t depend.**



Under THERMAL ANALYSIS for boundary conditions choose *Combined* for WORKPIECE_AIR – check the *Convection* and *Radiation* boxes and enter **10** for *Heat Transfer Coefficient* and **0.8** for *Emissivity*. Choose *Adiabatic* for WORKPIECE_SYMMETRY.



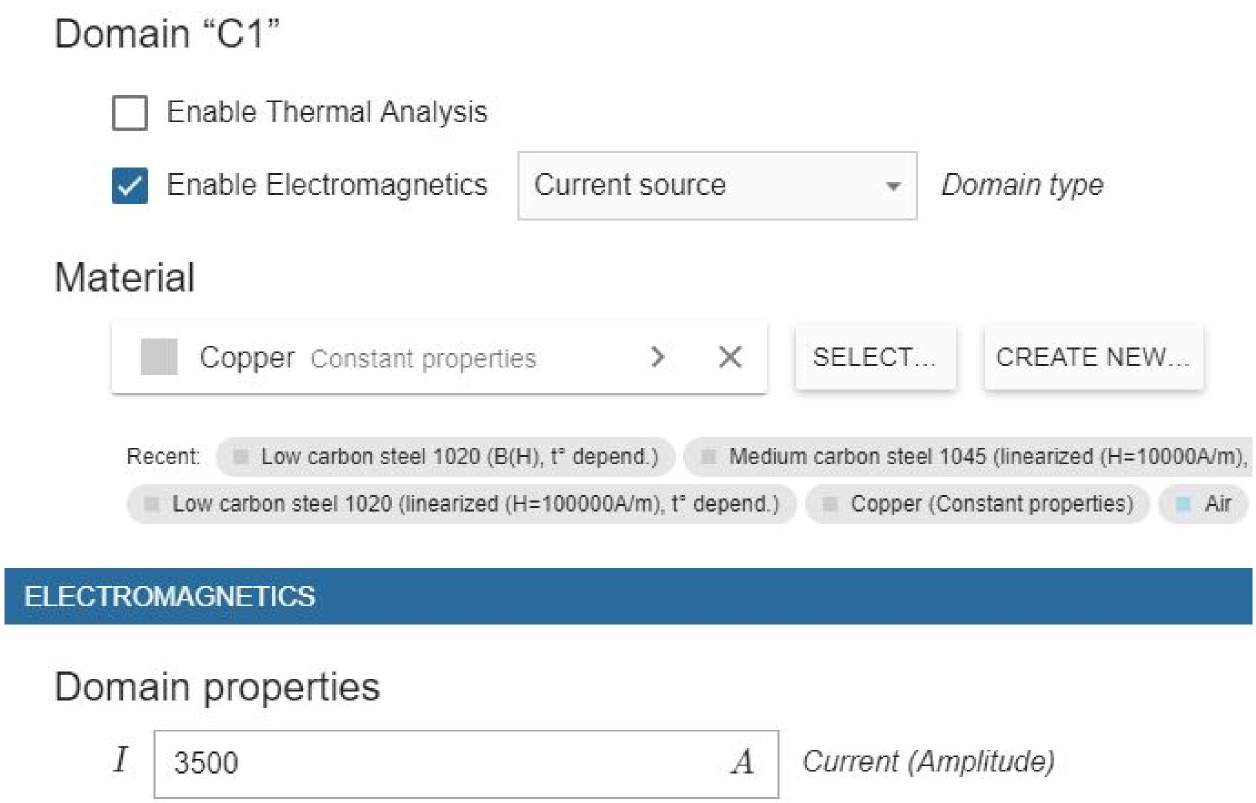
Under ELECTROMAGNETICS choose *Interface* for WORKPIECE_AIR and *Symmetry axis* for WORKPIECE_SYMMETRY.



4.4 Coil definition

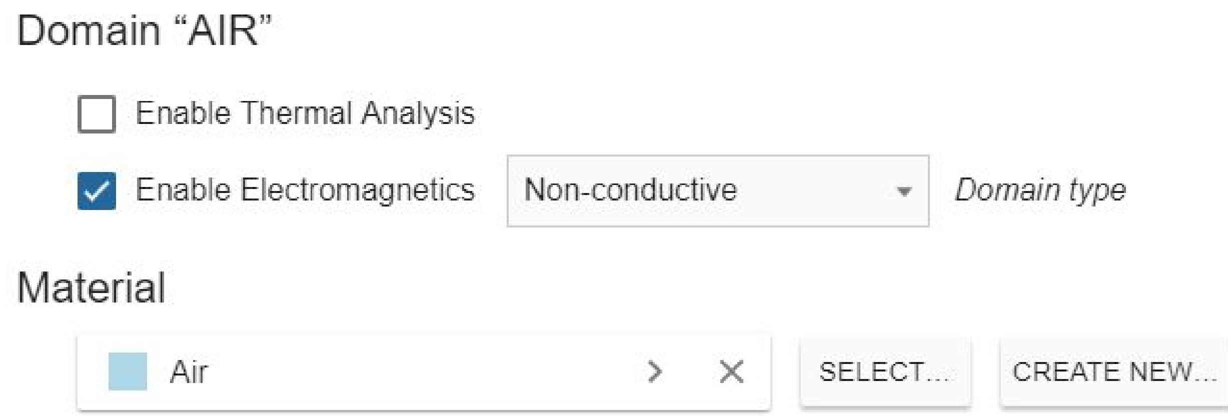
We created 8 different domains for each winding in order to define the current for each of them. To save time, it is possible to group these domains and define them all through one *Setup window*. To do that, select all winding domains and click GROUP.

Disable *Thermal analysis* and select *Current source* for *Domain type*. For *Material* choose *Copper Constant properties* and enter **3500A** for *Current (Amplitude)*.

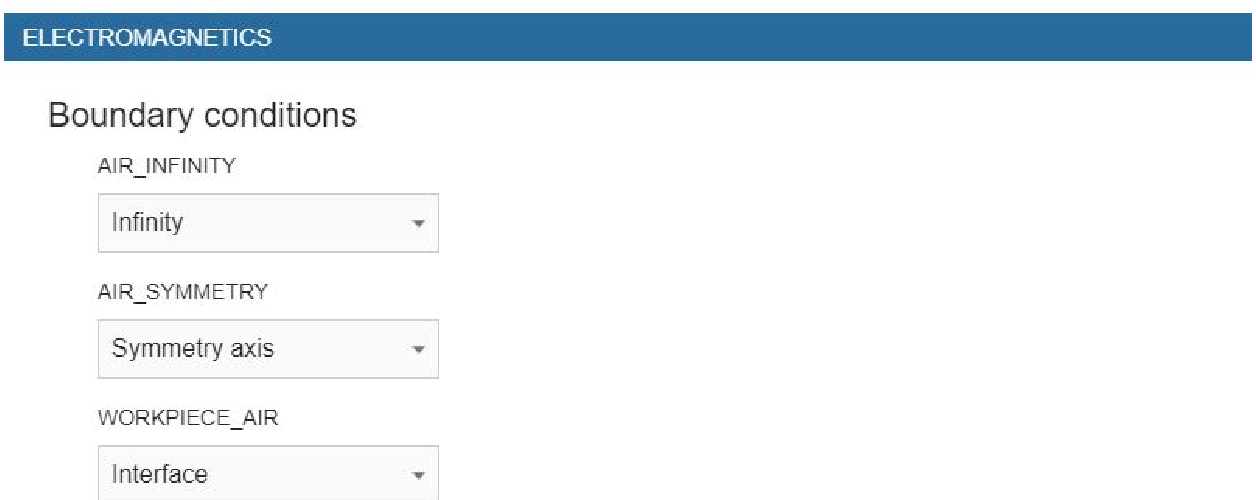


4.5 Air definition

Switch to AIR in *Domain bar*. Disable *Thermal analysis* and select *Non-conductive* as *Domain type*. For *Material* choose *Air*.



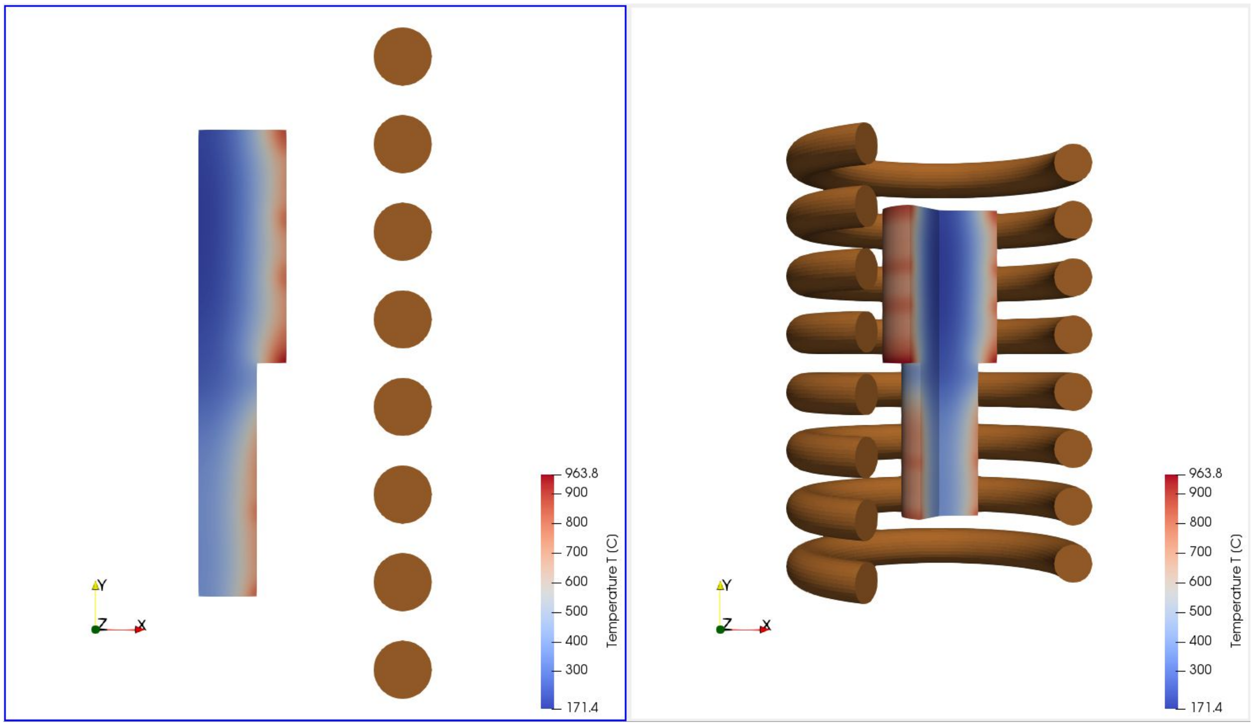
Under ELECTROMAGNETICS choose *Infinity* for AIR_INFINITY, *Symmetry axis* for AIR_SYMMETRY and *Interface* for WORKPIECE_AIR.



When everything is set, **click RUN**.

5. Evaluate results

When CENOS finishes calculation, ParaView window with pre-set temperature result state will open automatically and you will be able to see the temperature field distribution in workpiece in the last time step as well as a 3D revolution of the results to give you better visual interpretation.



Results can be further manipulated by using ParaView filters - find out more in CENOS advanced post-processing article.

This concludes our Induction Heating Template tutorial. For any recommendations or questions contact our support.