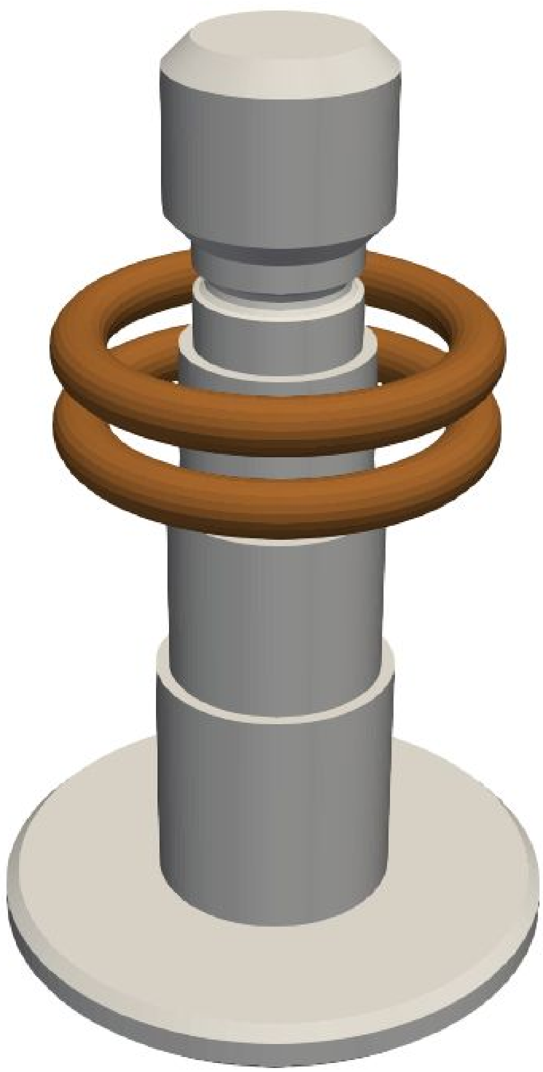
2D Scanning of Transmission Shaft

Induction heating of moving shafts and workpieces of any kind that are not regular in shape and size cannot be simply simulated using motion. CENOS platform incorporates **scanning** as a means of simulating **movement of irregular workpiece through the inductor**.

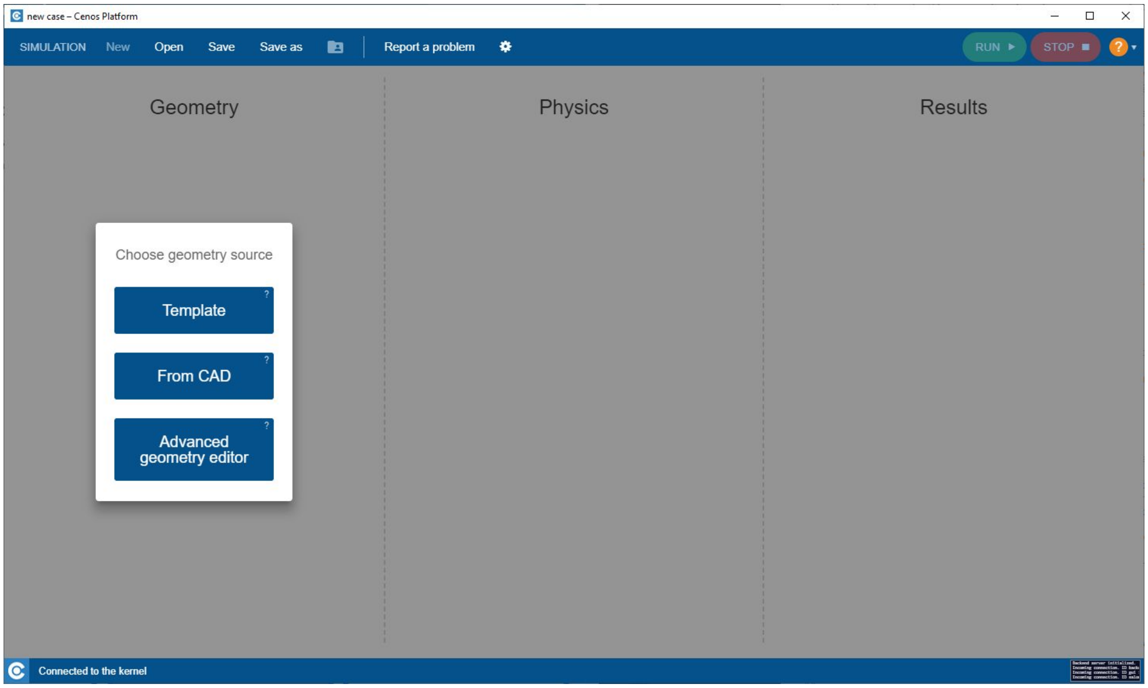
In this tutorial we will learn how to create and set up the simulation of a **30s** long **transmission shaft scanning** at **10 kH**z and **10 kA** with two winding inductor.



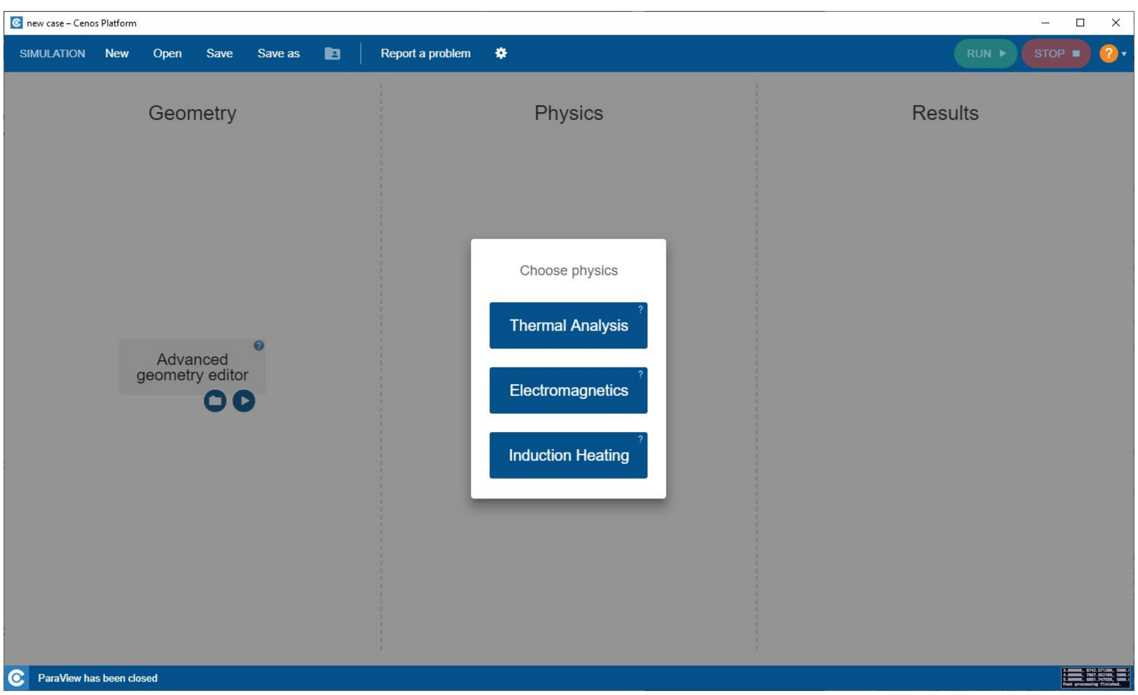
1. Open pre-processor

1.1 Choose pre-processing method

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



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



Click the **Play** icon to open Salome.

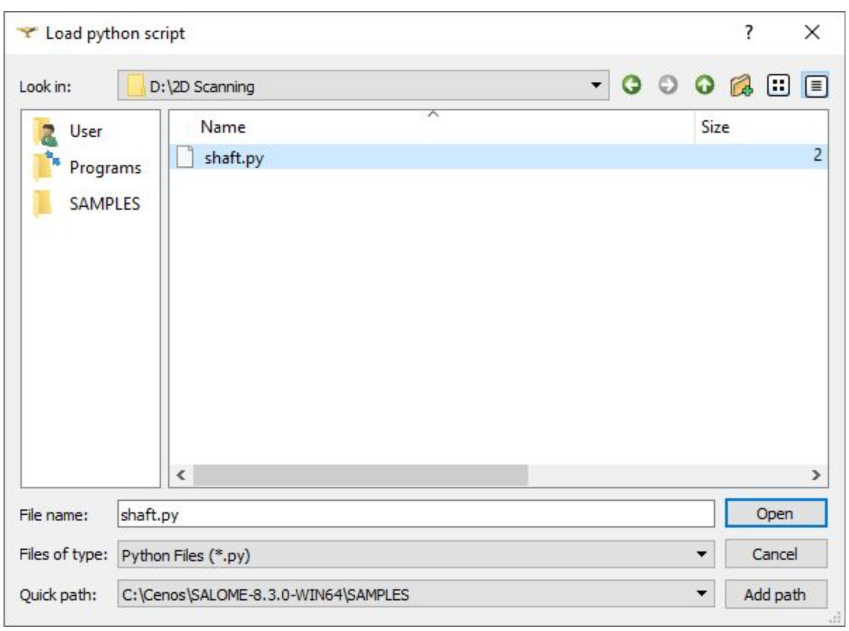
1.2 Load shaft geometry

In this tutorial **we will not create the shaft geometry from scratch**, but rather import it as Salome script.

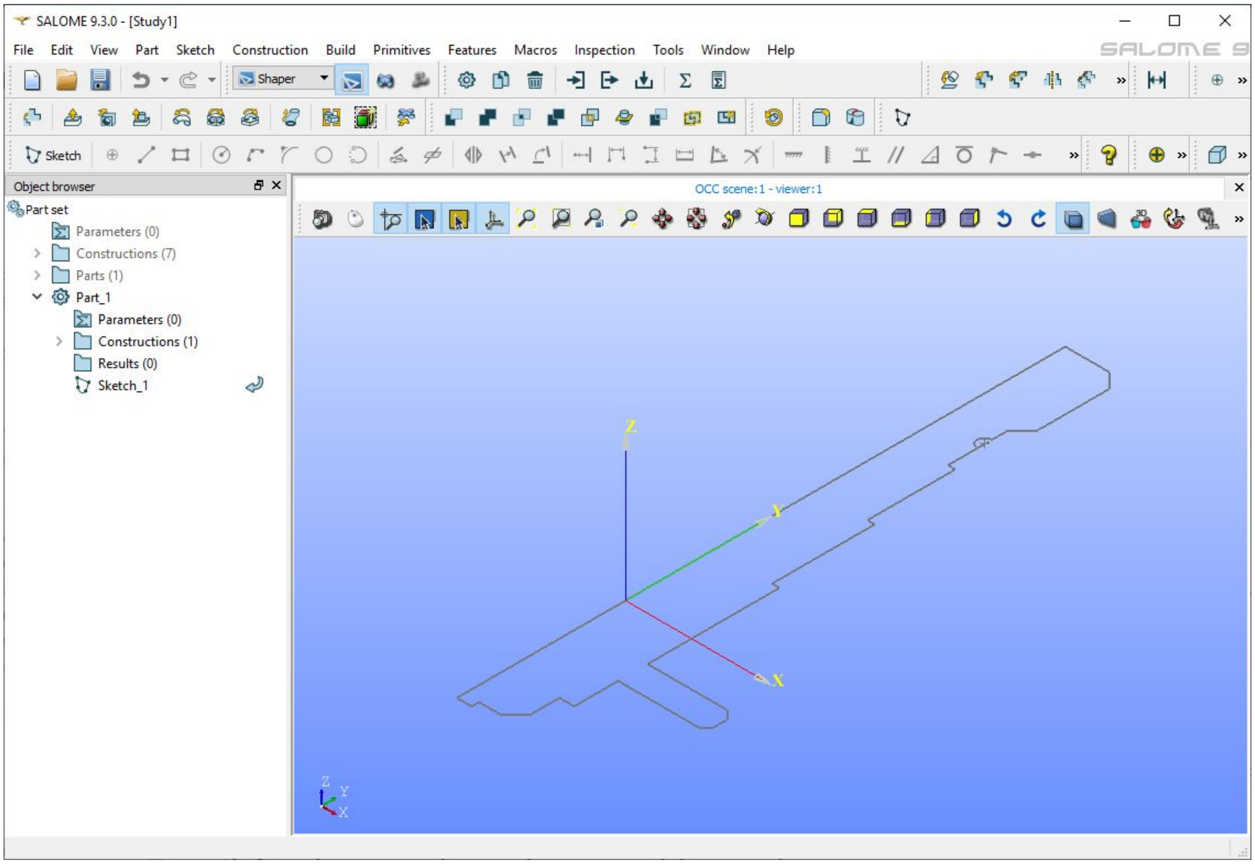
IMPORTANT: To follow this tutorial, download the **shaft.py** Salome geometry script from documentation website.

Click *File* → *Load script...*.

In the *Load python script* window select and open previously downloaded *shaft.py* script file.



Once loaded you should **see the geometry in the 3D viewer**:

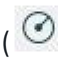


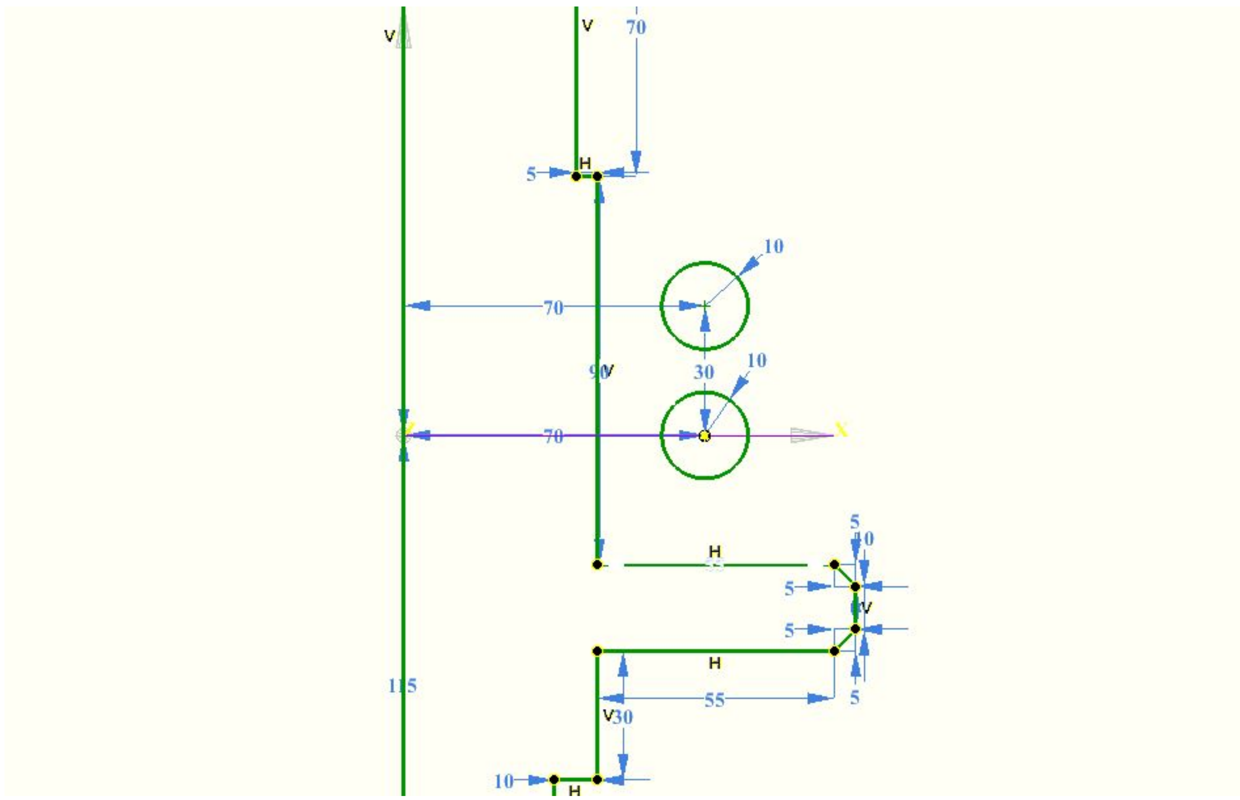
2. Create geometry and prepare it for meshing

Even though we imported the shaft geometry, **we still need to create the inductor windings and air domain.**

2.1 Create the inductor windings

In *Object Browser* right click on 

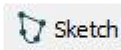
 and click *Edit*. By using *Circle* () tool **create two circles**, both **70 mm** far from the Y axis and with **30 mm** distance between them.

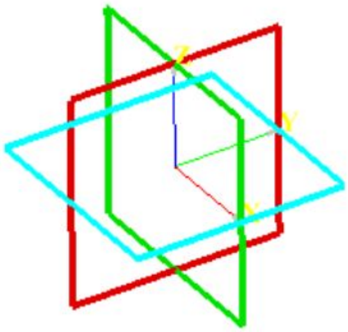


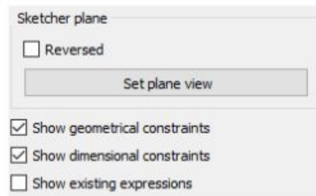
Use the constraints to set the dimensions:

If you cannot draw objects, you must disable camera control ().

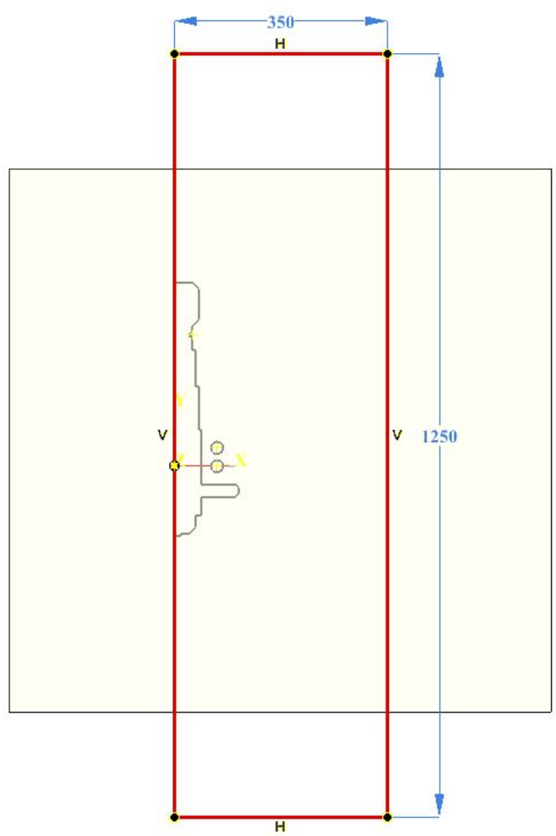
2.2 Create the air box

To create an air box, we first need to create a new sketch. Create a new *Sketch* by clicking the *Sketch* () icon. Select the **XY** plane and click **Set plane view**



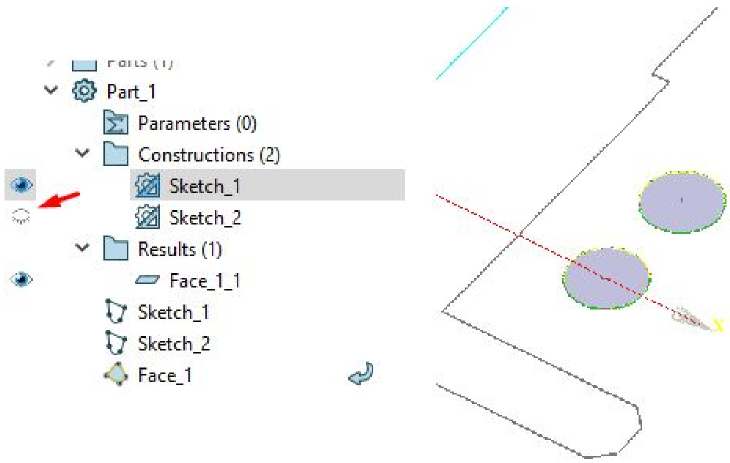


Create a rectangle (**1250mm x 350mm**) around the workpiece by clicking the *Rectangle* () icon. Add a *Coincidence* () constraint between the symmetry axis edge (left side of the air box) and the origin point.

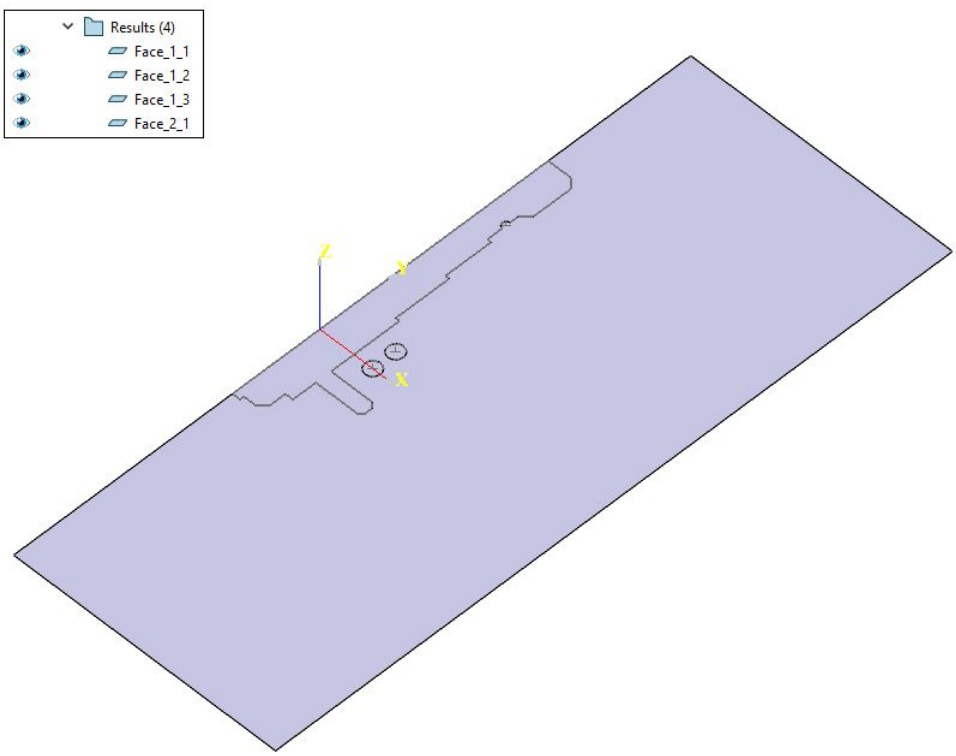


2.3 Create faces

Create a face by clicking *Face* () tool. Select the inductor windings and the workpiece. You might have to hide the air domain sketch to select the workpiece and inductor.



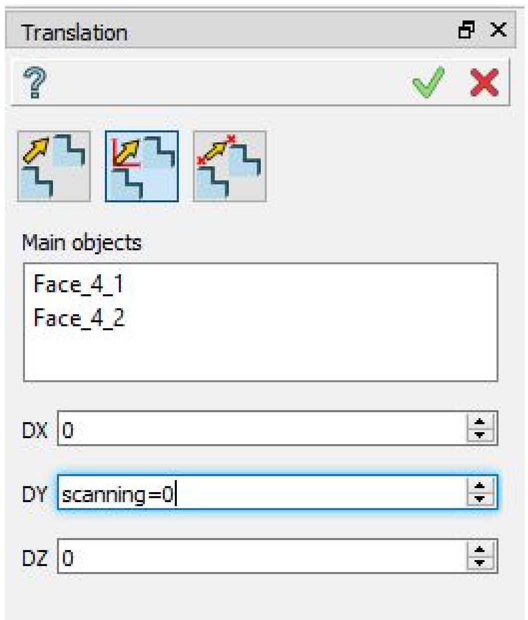
Now create the fourth air domain face and you should have all the faces as shown here:



2.4 Translate inductor windings

Click *Translation* () tool. Select the **middle option** for the translation system, and select the two inductor faces we created.

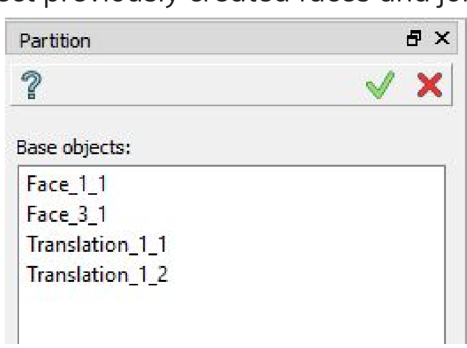
For *DX* and *DZ* enter 0. For *DY* enter **scanning=0**.



By entering "**scanning=0**", we automatically created a **new parameter** called "scanning" with a value of 0. You can use any name or value (within limits).

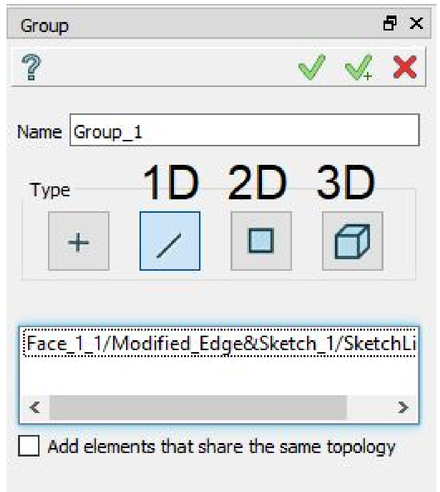
2.5 Create partition and groups

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.

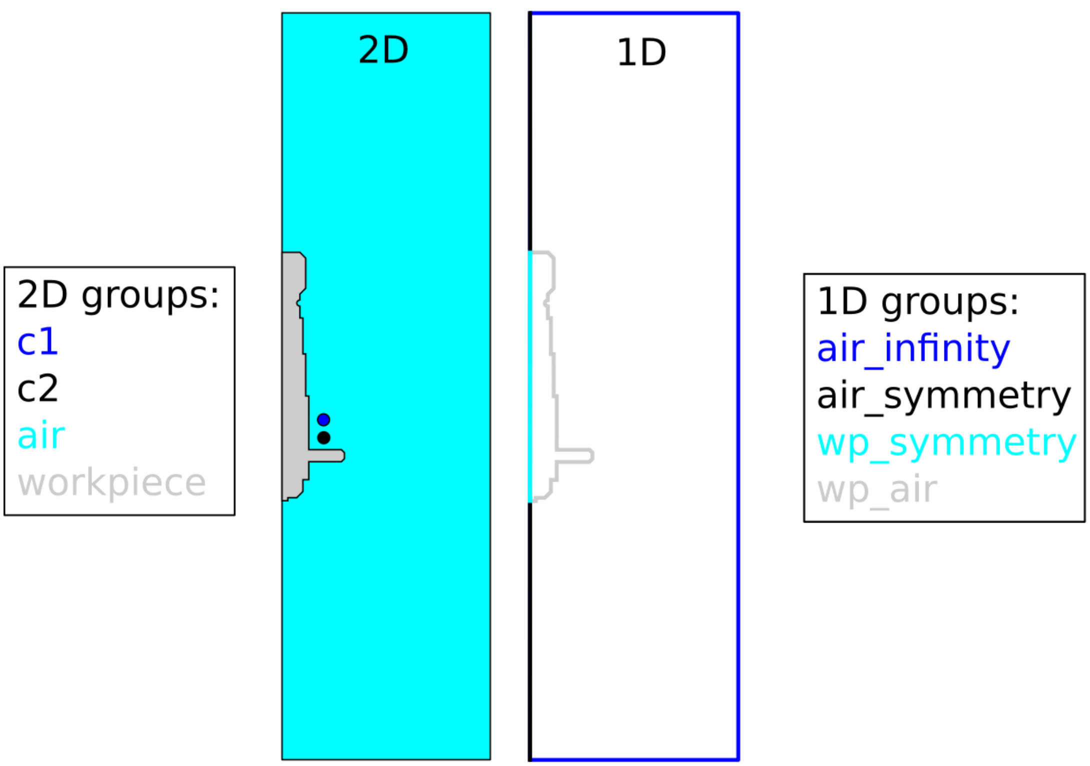
Select *Group* () tool and choose the *Shape Type*. Select one or more shapes from the screen, name the group and click the *Apply and continue* ( **).**



.

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

A detailed breakdown of these groups is as follows:



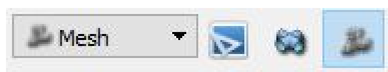
2.7 Export to GEOM

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.

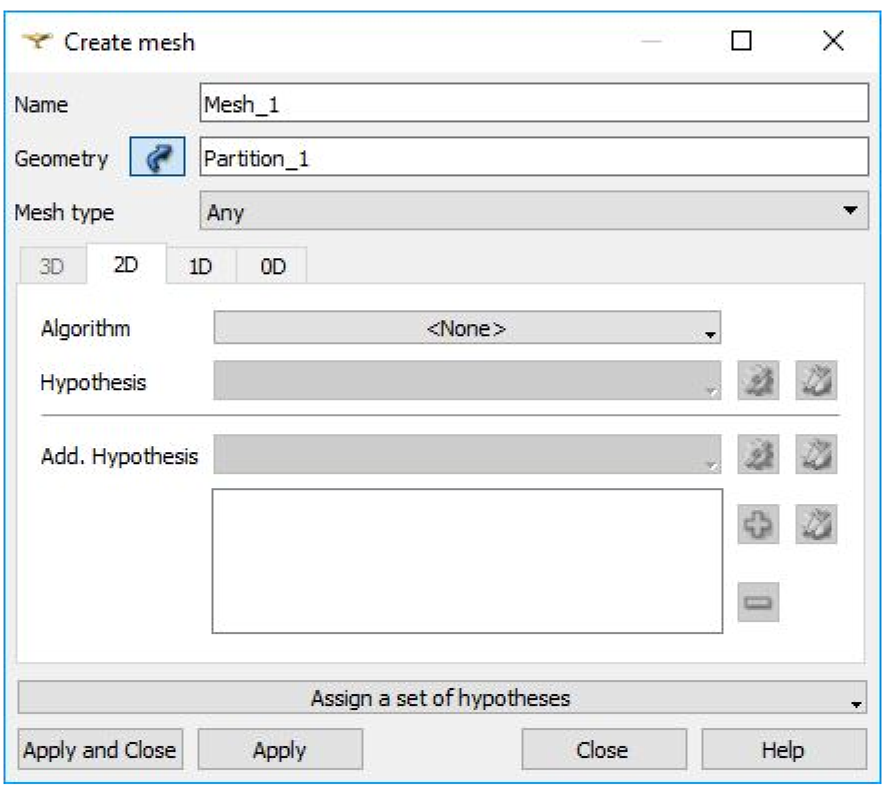
3. Create mesh and export it to CENOS

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.



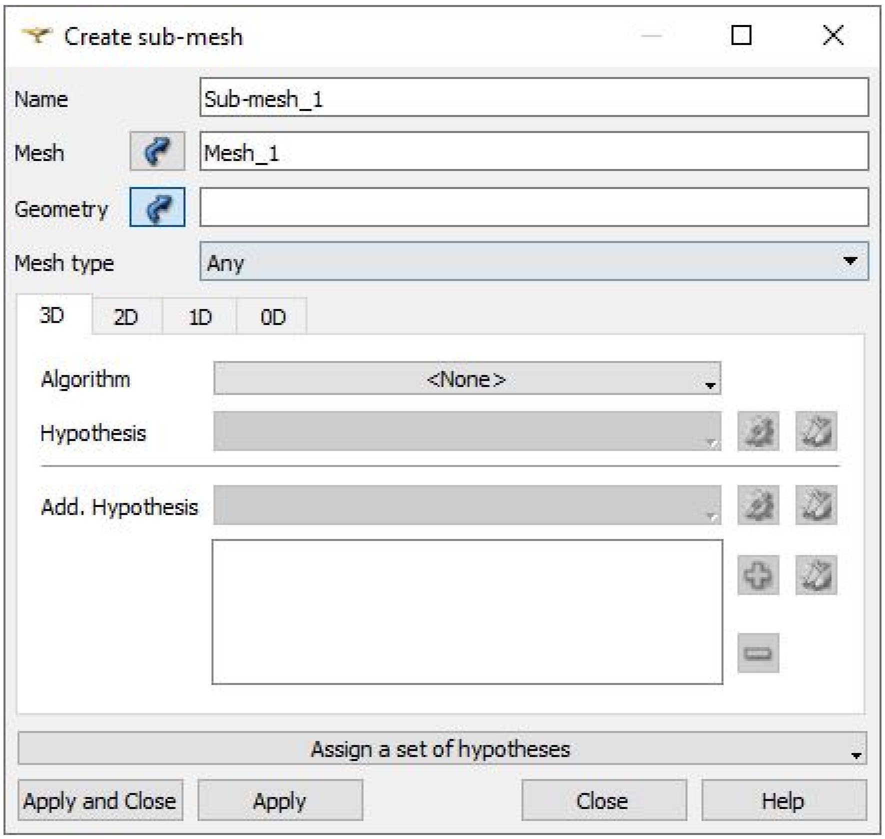
In *Object Browser* from *Geometry* dropdown menu select the previously created *Partition_1_1* and click *Create Mesh* ().



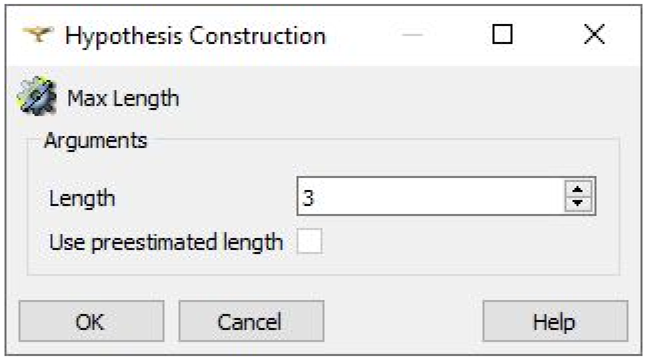
From the *Assign a set of hypothesis* dropdown menu select *2D: Automatic Triangulation* - leave the *Max Length* value default and click *Apply and Close*.

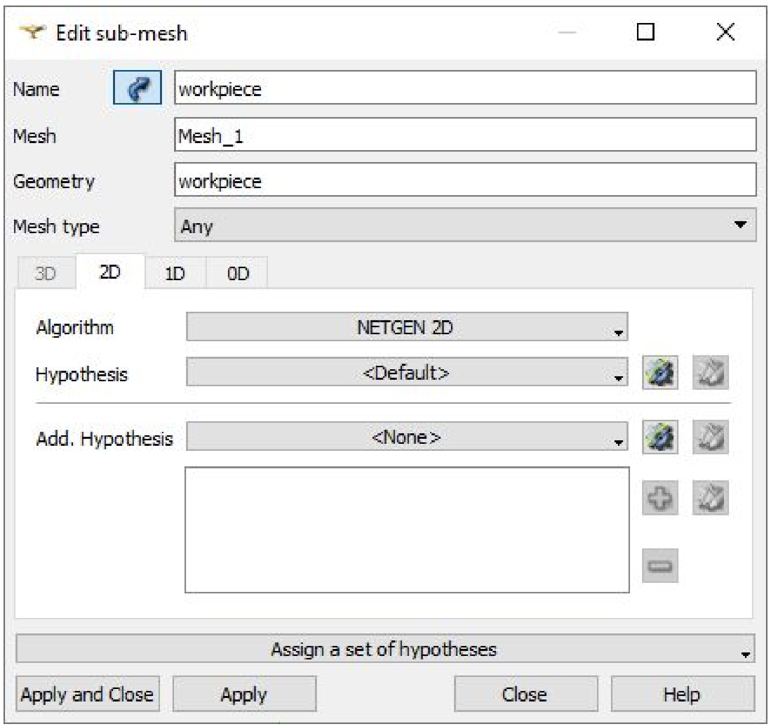
3.2 Create a sub-mesh for the workpiece

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



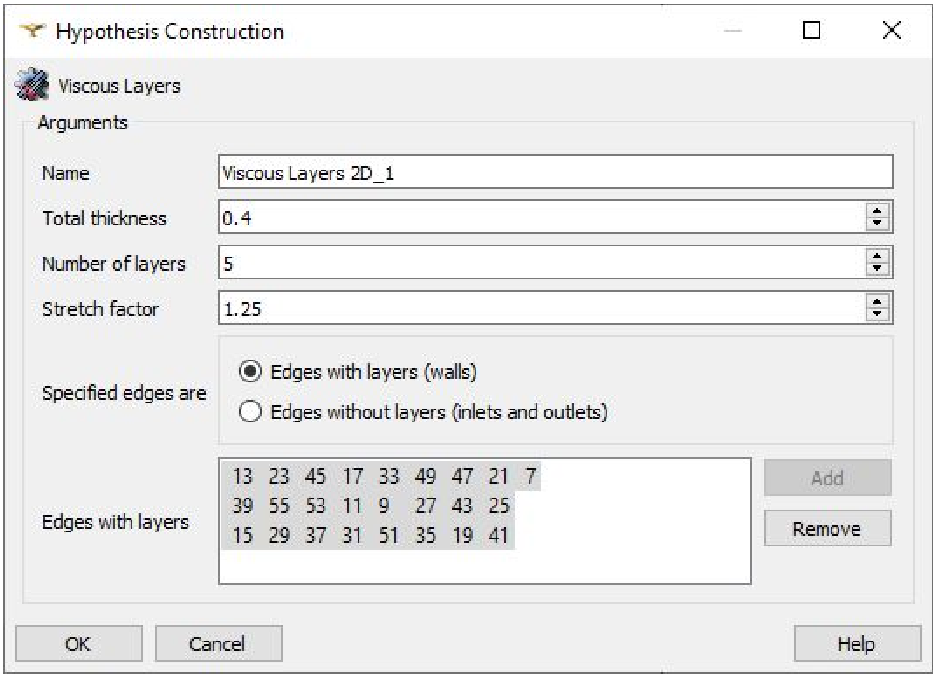
Choose **workpiece** group from the *Partition_1_1* dropdown menu as *Geometry*. From the *Assign a set of hypothesis* dropdown menu choose *2D: Automatic Triangulation*. In the *Hypothesis Construction* window enter **3** for *Max Length* and change the 2D algorithm from *Triangle: Mefisto* to *NETGEN 2D*.





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 *wp_air* from the *Partition_1_1* dropdown menu and click *Add*. Enter **0.4** for *Total thickness*, **5** for *Number of layers*, **1.25** for *Stretch factor* and check the *Edges with layers (walls)* box.



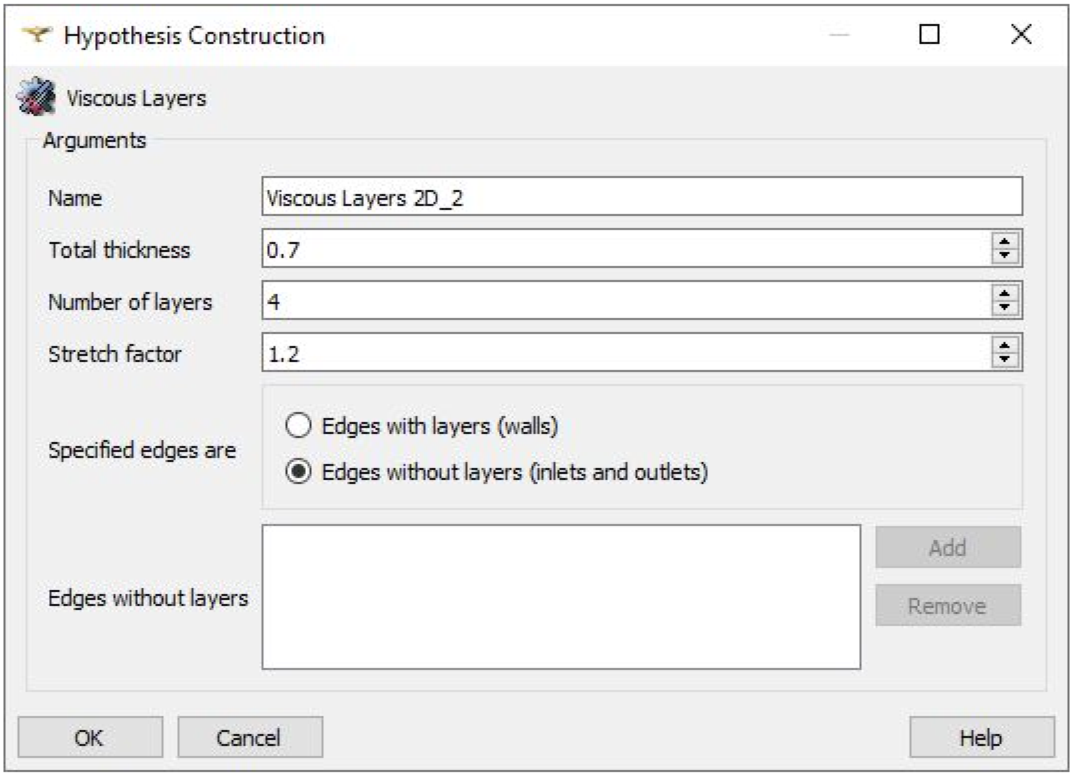
When all is set, click *Apply and Close*.

3.3 Create a sub-mesh for the inductor

Create a sub-mesh and select *c1* and *c2* groups from the *Partition_1_1* dropdown menu as *Geometry*. From the *Assign a set of hypothesis* dropdown menu choose *2D: Automatic Triangulation* and enter **3** 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*, **1.2** for *Stretch factor* and check the *Edges without layers (inlets and outlets)* box.



IMPORTANT: If you create a sub-mesh from multiple groups, Salome will auto-group them and create a new group, in this case named **Auto_group_for_Sub-mesh_2**.

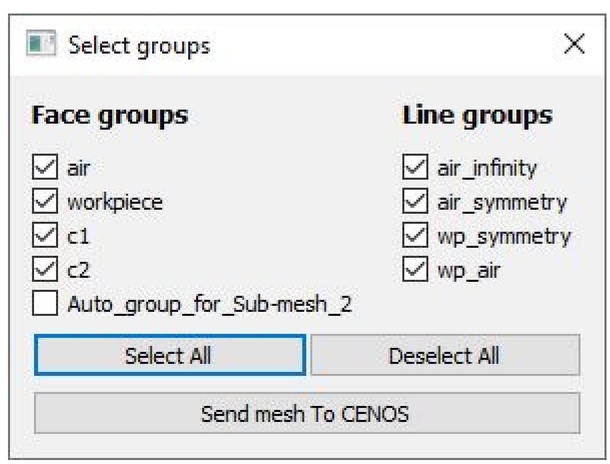
3.4 Calculate and export mesh to CENOS

Right-click on *Mesh_1* and click *Compute*. **Evaluate the final mesh and export it to CENOS.** To do that, select *Mesh to CENOS* 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 *Auto_group_for_Sub-mesh_2*.

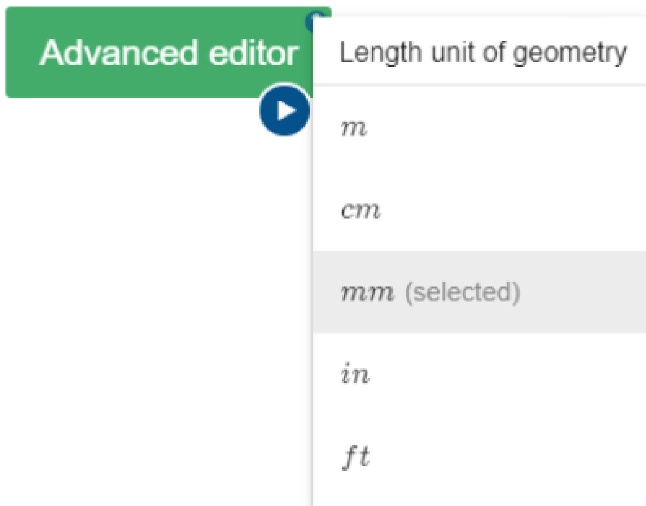
When selected, click *Send mesh to CENOS.*



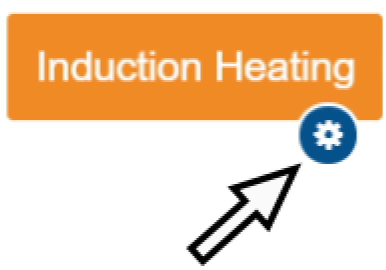
4. Define physics and boundary conditions

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)**.

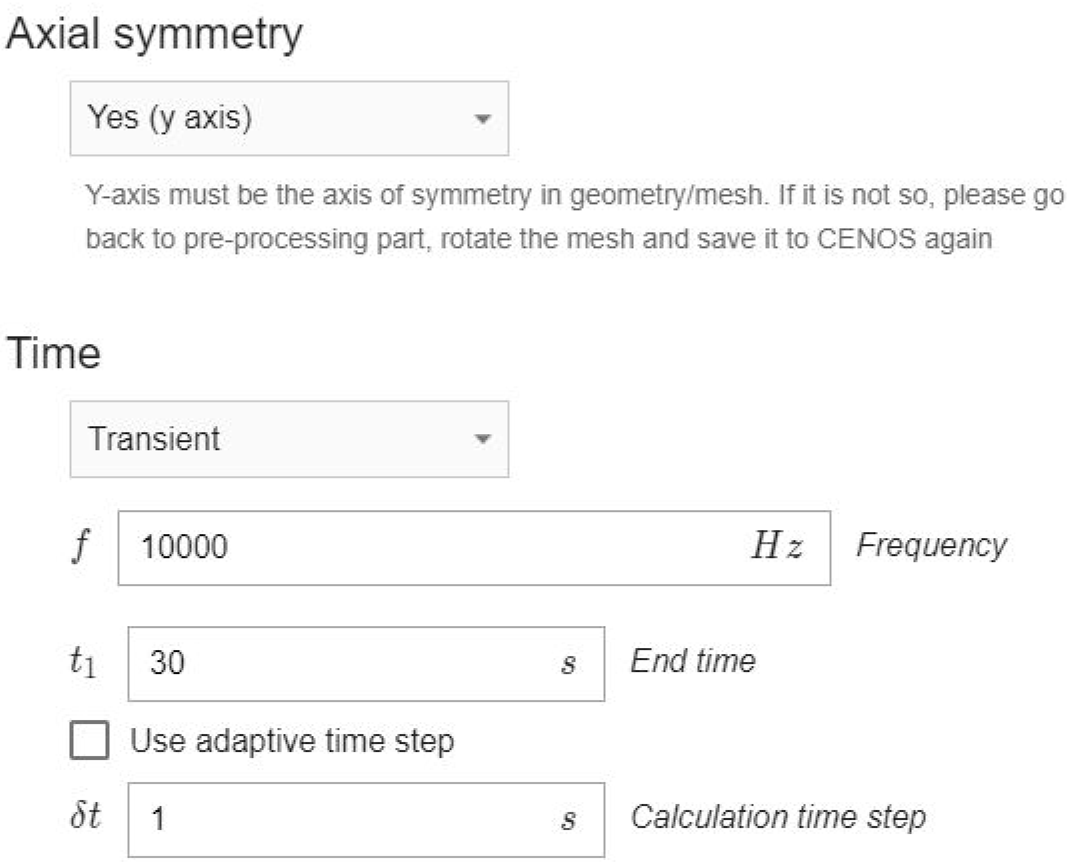


Click the **gear icon** under *Induction Heating* block to enter the physics setup.

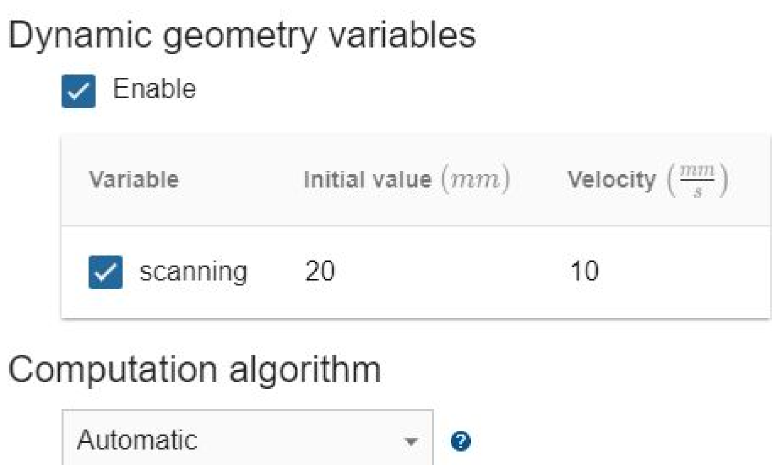


4.2 Simulation control

In SIMULATION CONTROL window define the simulation as *axial symmetric* and *transient* with **10 kHz** *frequency*, **30 s** *End time* and **1 s** *time step*.

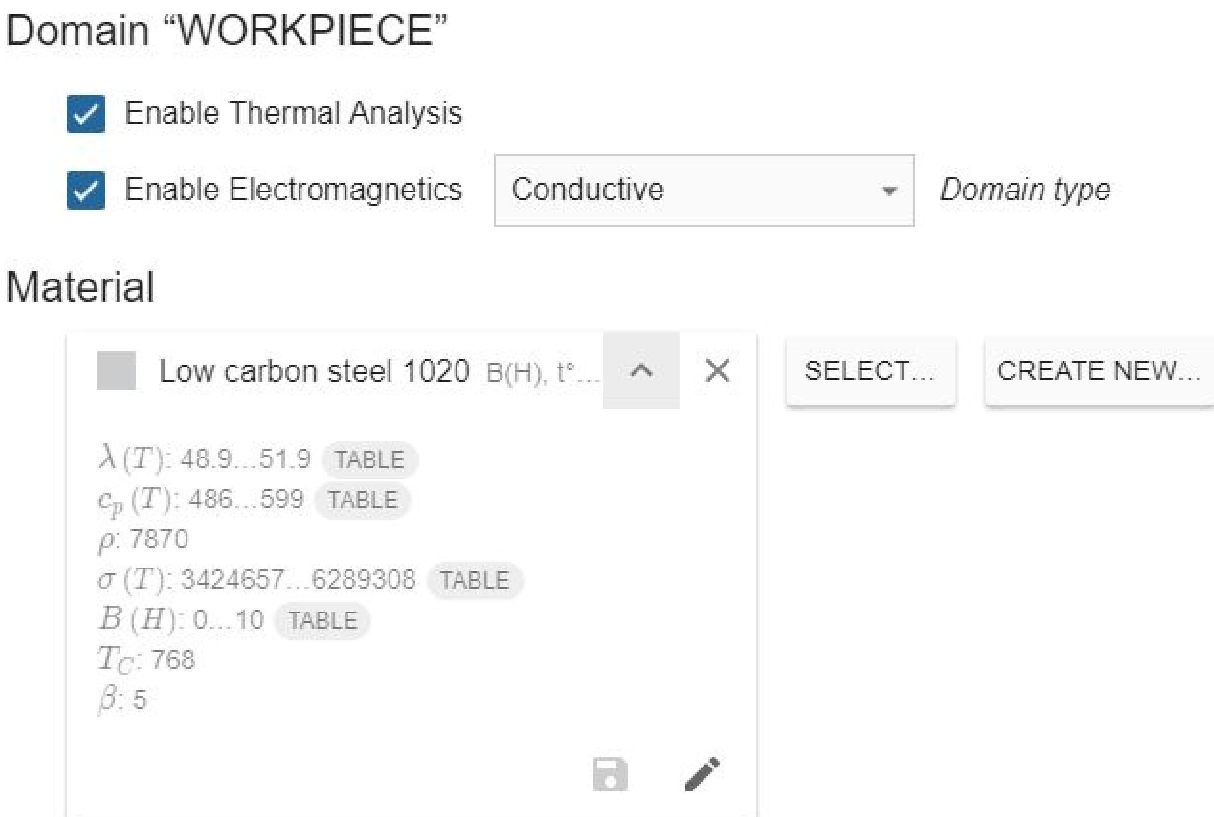


Check the **Dynamic geometry variables** check box. Leave the *Initial value* **20 mm** and enter **10** for *Velocity*. For *Computation algorithm* leave **Automatic**.

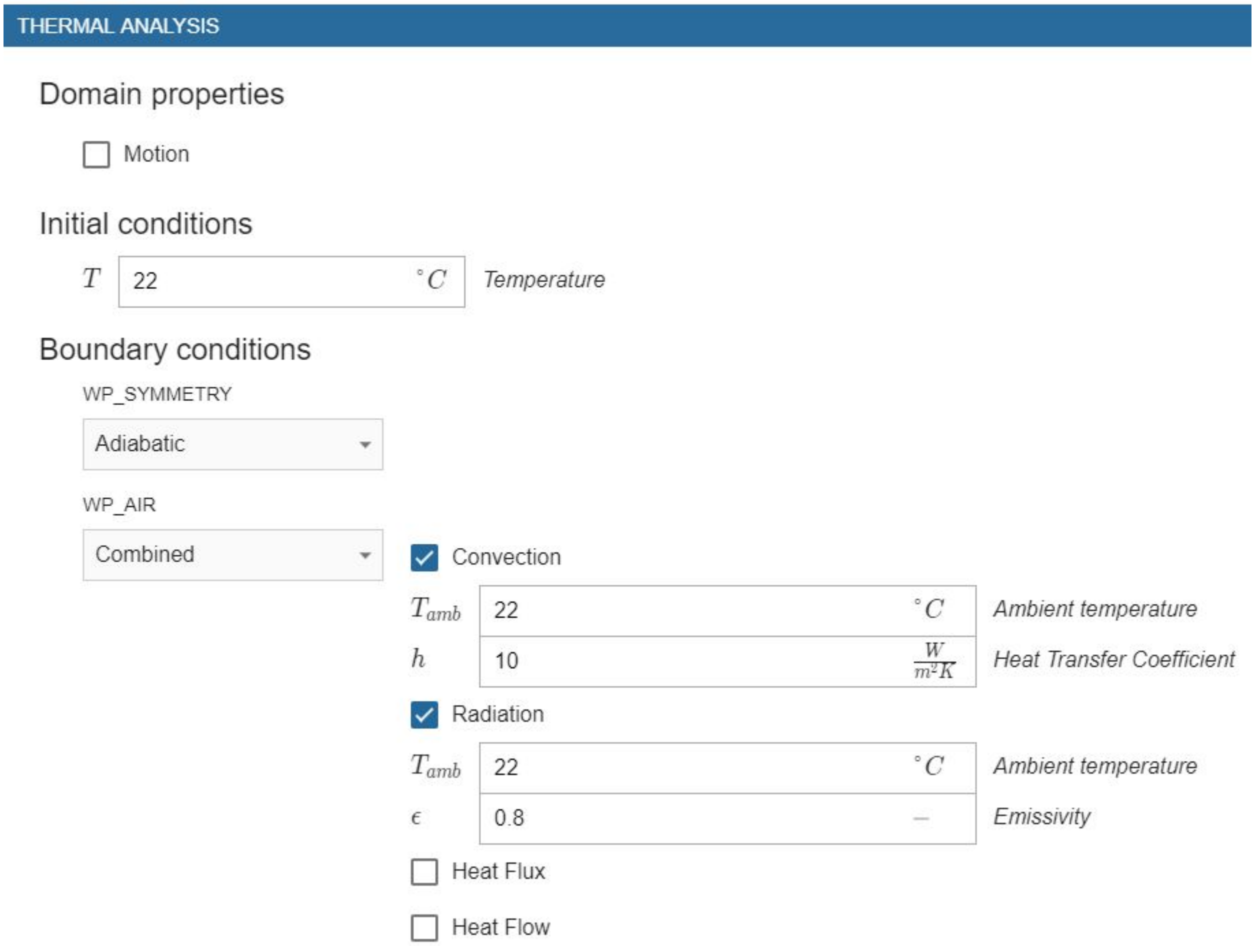


4.3 Workpiece definition

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**.



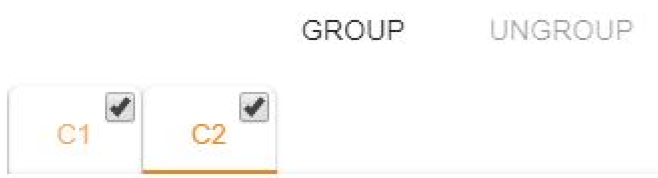
Under THERMAL ANALYSIS for boundary conditions choose *Combined* for WP_AIR – check the *Convection* and *Radiation* boxes and enter **10** for *Heat Transfer Coefficient* and **0.8** for *Emissivity*. Choose *Adiabatic* for WP_SYMMETRY.



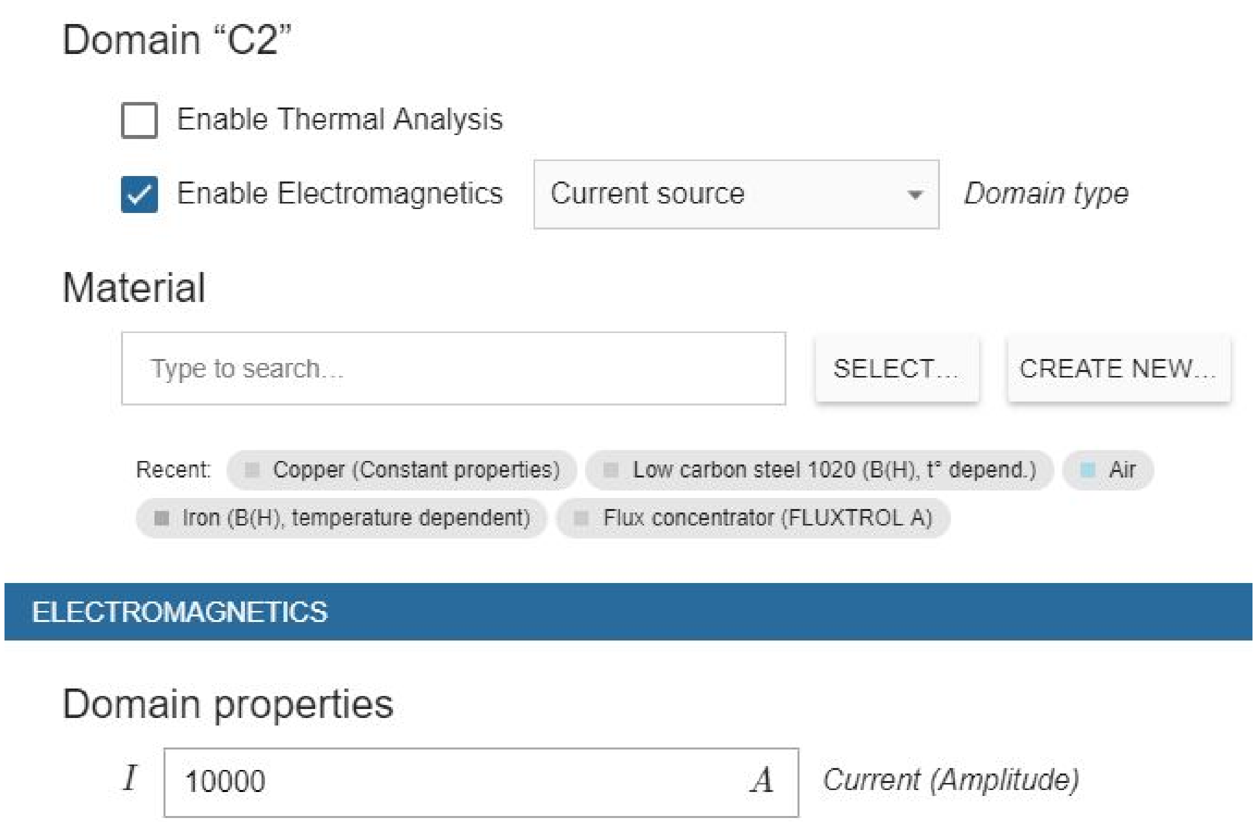
Under ELECTROMAGNETICS choose *Interface* for WP_AIR and *Symmetry axis* for WP_SYMMETRY.

4.4 Coil definition

We created 2 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.

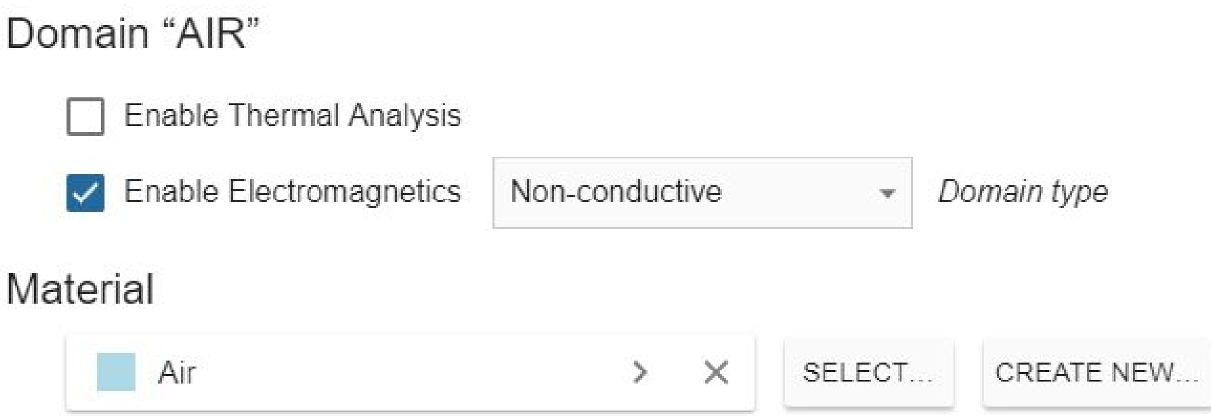


Disable *Thermal analysis* and select *Current source* for *Domain type*. For *Material* choose *Copper Constant properties* and enter **10000 A** for *Current (Amplitude)*.

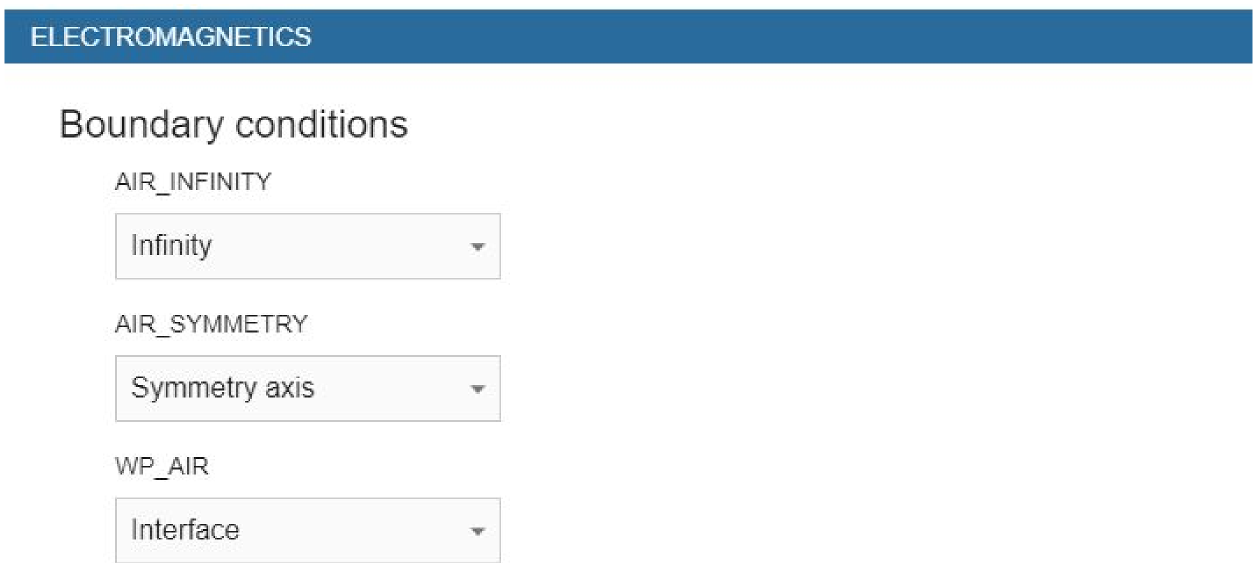


4.5 Air definition

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



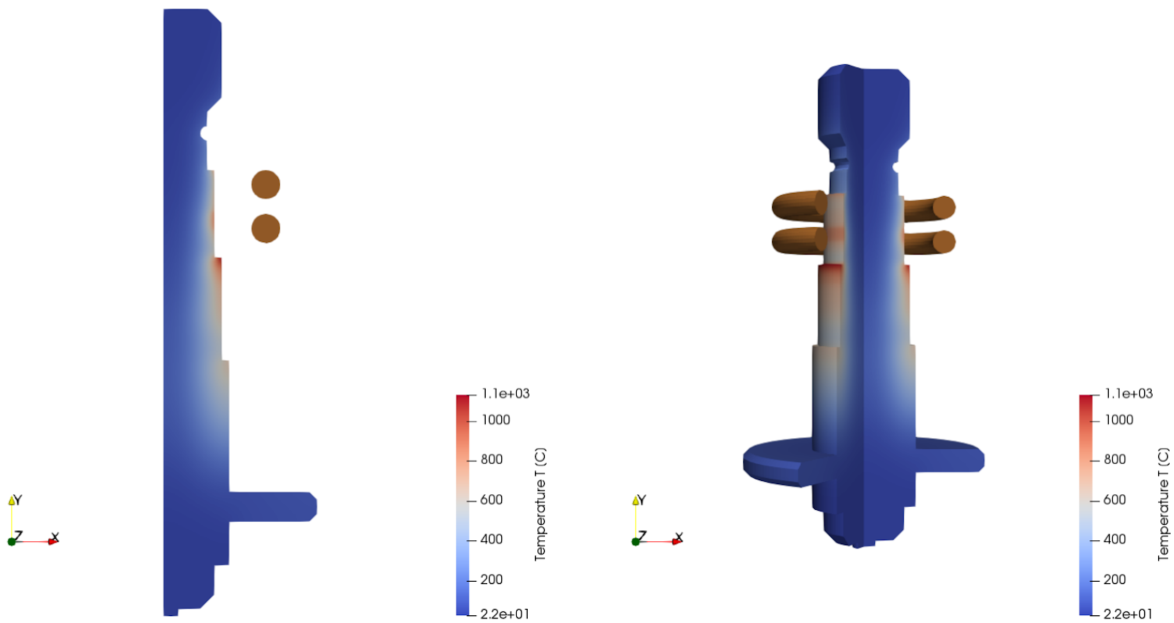
Under ELECTROMAGNETICS choose *Infinity* for AIR_INFINITY, *Symmetry axis* for AIR_SYMMETRY and *Interface* for WP_AIR.



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 Transmission Shaft Scanning tutorial. For any recommendations or questions contact our support.