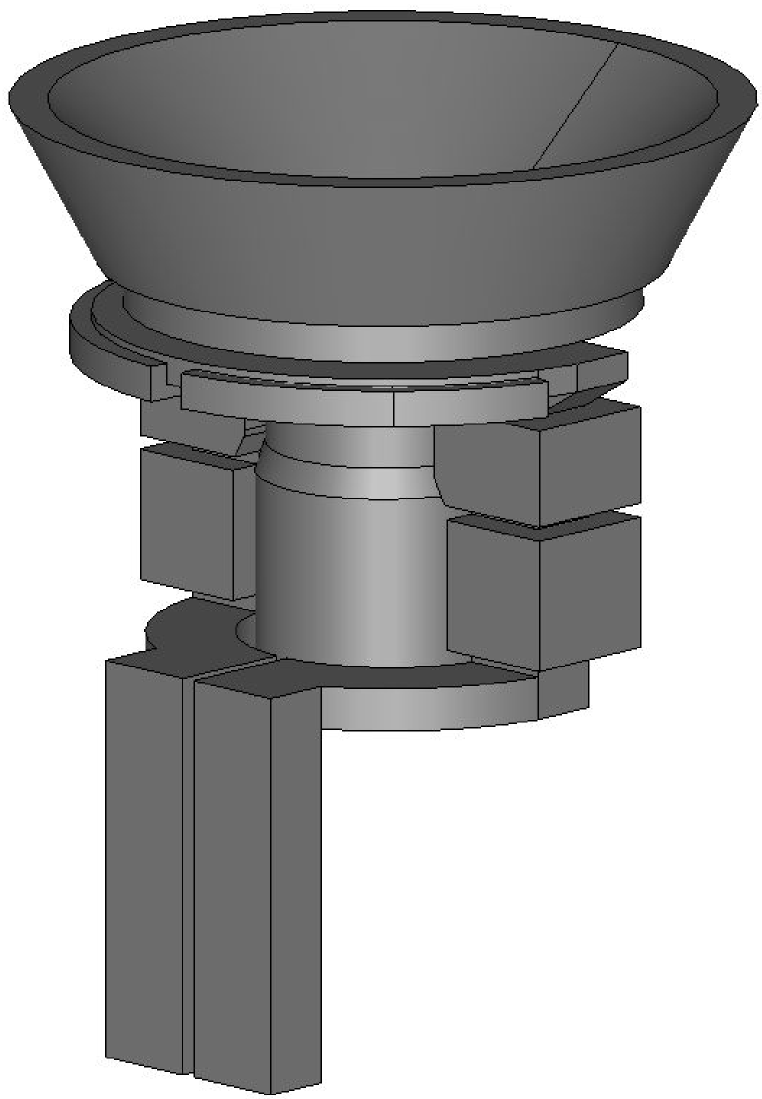
3D Single Shot

With CENOS platform it is possible to build and simulate full 3D induction hardening cases from imported CAD files. In this tutorial we will show you how to prepare and run an induction heating simulation from elsewhere created CAD file.

We will upload the **pre-made CAD** file into the CENOS platform, prepare it for meshing and mesh it while paying attention to the mesh quality and element count.

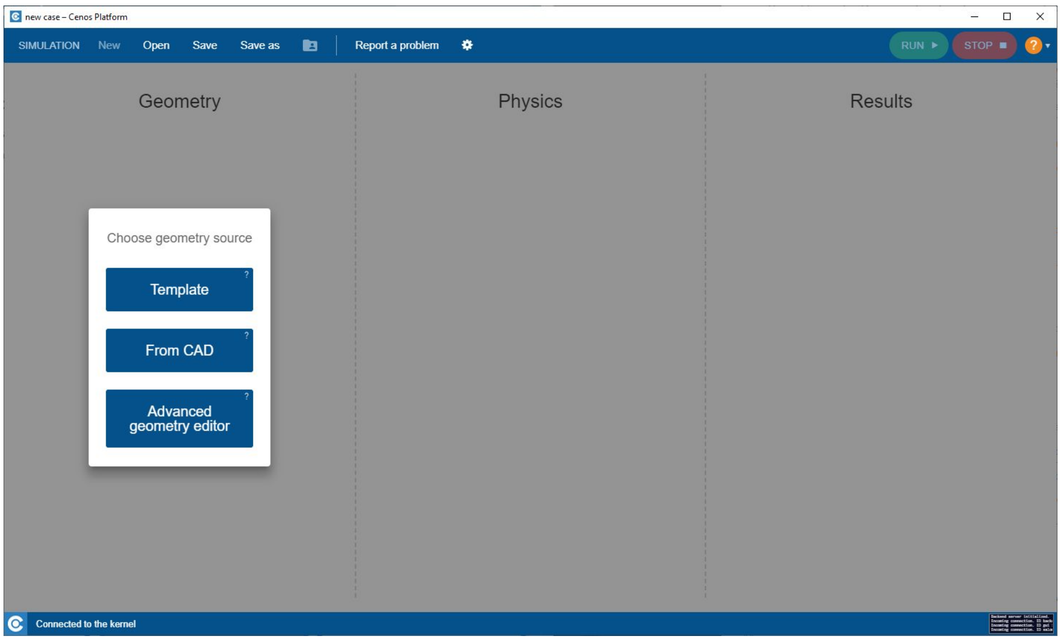
In the next pages a **2.5 s** long induction hardening example of a **rotating AISI 1020 workpiece** at **5 kHz** and **10 kA** with **custom shape conductor** and **FLUXTROL A** yoke is presented.



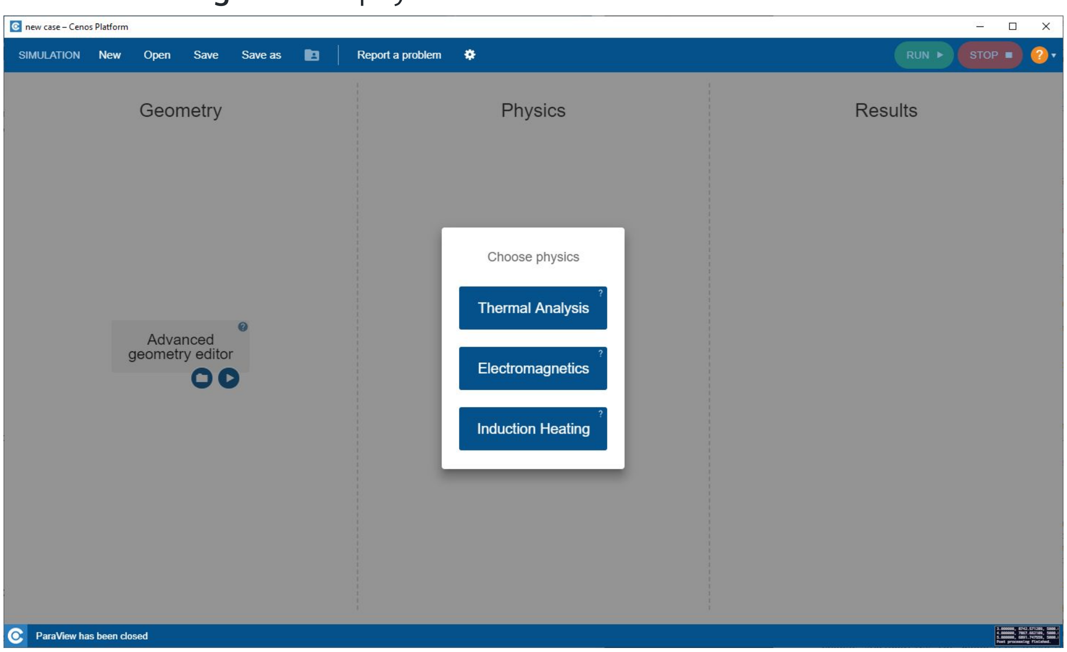
1. Open pre-processor

1.1 Choose pre-processing method and import CAD

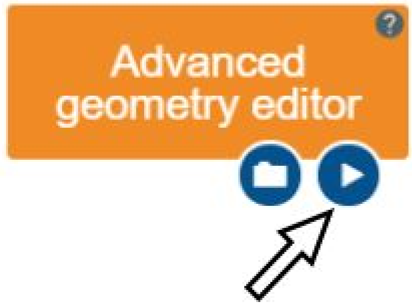
To manually create geometry and mesh, in CENOS home window click **Advanced geometry editor.**



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



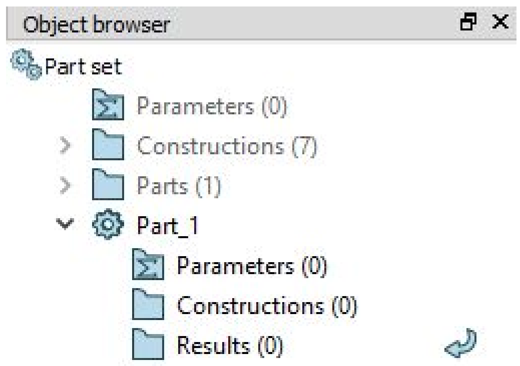
Click the **Run** icon to select and import STEP files.



2. Create geometry and prepare it for meshing

2.1 Import geometry

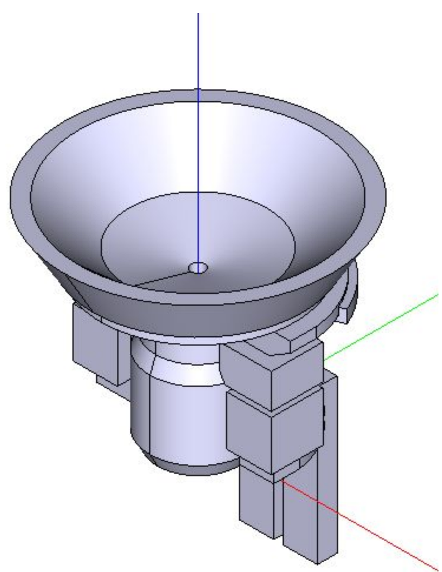
Create a new *Part* by clicking the *New part* ( ) tool. A new part will be added to *Object browser*.



Click *Import* ( ) and select the step file provided earlier.

To see the geometry, click *Fit All* ( ).

The imported geometry should look like this:



2.2 Create the air box

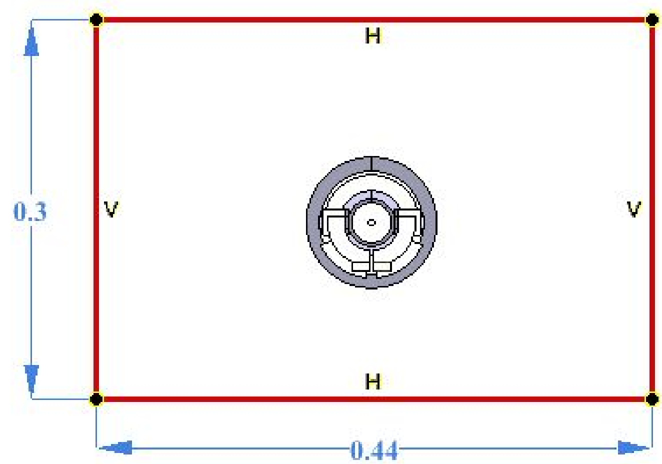
To correctly define inductor domain, **one air box side must be in the same plane as the inductor terminals.**

To achieve this, we will create a sketch on one of the inductor terminals. Click *Sketch* (Sketch) tool and **select one of the inductor terminals.**

IMPORTANT: To rotate and move geometry around, toggle the *Interaction style switch* ( ).

Reposition the geometry by clicking +OZ ( ) and disable *Interaction style switch* ( ).

Select *Rectangle* ( ) tool and draw a rectangle around the geometry. Click *Length* ( ) tool and **define the size of the rectangle** (0.3m x 0.44m). Center it by eye:

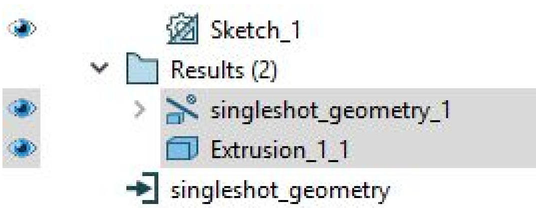


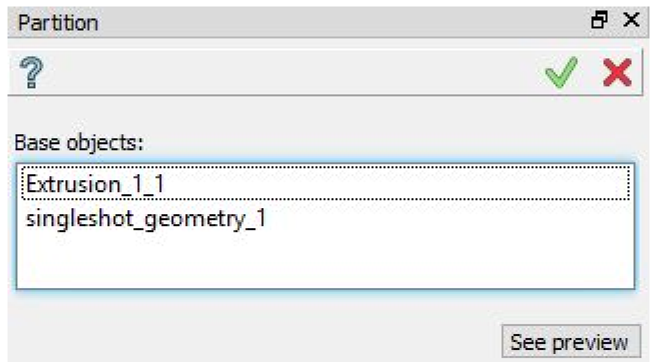
Once completed, click *Apply* ( ) and **close the sketch.**

Now we need to extrude the rectangle we created. Click *Extrusion* ( ) tool and select the rectangle. In the extrusion parameters enter **0.2m** in the reverse direction:

2.3 Create Partition and Groups

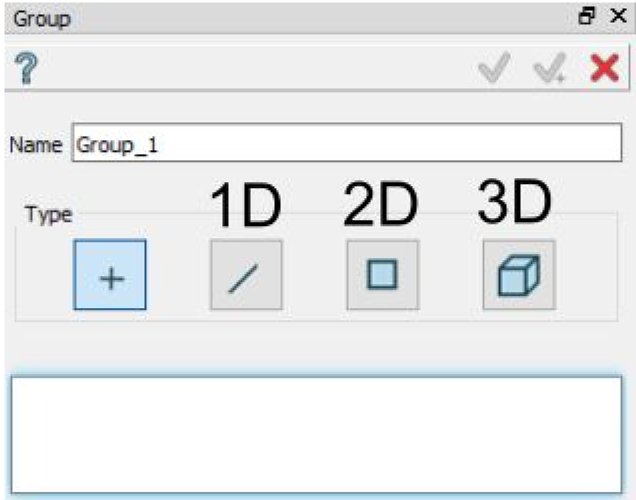
Click *Partition* () tool. From *Object Browser* select the imported geometry and created air box under *Results* and join them in one partition.





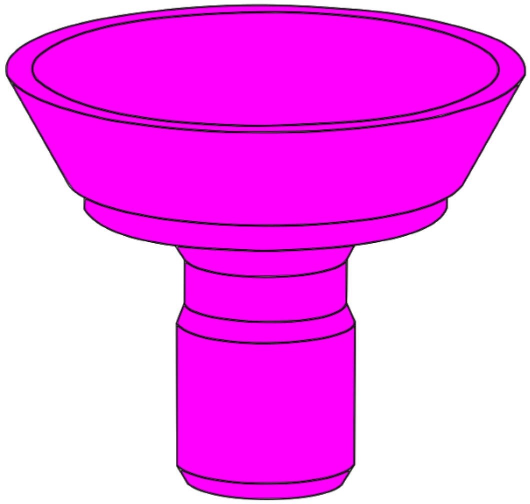
IMPORTANT: Partition and Groups are vital for simulation setup with CENOS, because mesh creation as well as physics and boundary condition definitions are based on the groups created in this part.

Click *Group* () tool and select the partition we just created (*Partition_1_1*). Choose the *Shape Type*, select one or more shapes from the screen, name the group and click *Apply and continue* ().

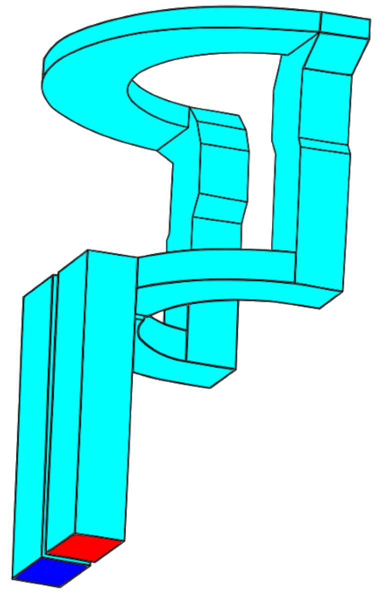


For this tutorial we will create four 3D groups for domains and four 2D groups for boundary conditions. The *workpiece_air* group includes full workpiece surface, while *air_infinity* includes all 6 sides of the air box, but without both terminals.

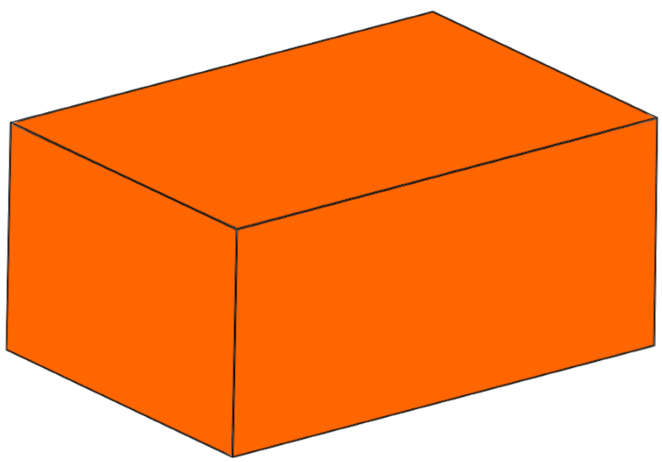
Detailed breakdown of these groups is as follows:



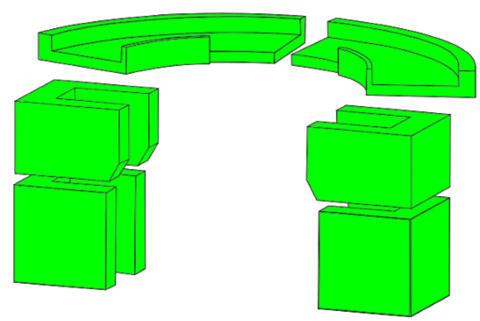
Workpiece groups:
3D - workpiece
2D - workpiece_air



Conductor groups:
3D - conductor
2D - terminal1
2D - terminal2

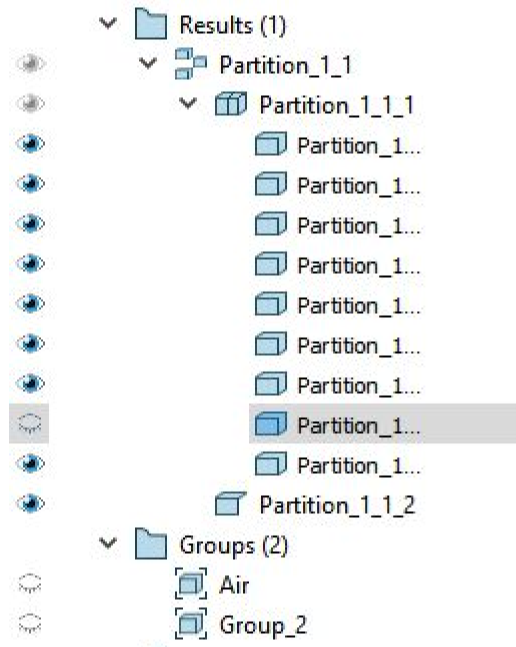


Air groups:
3D - air
2D - air_infinity



Yoke group:
3D - yoke

IMPORTANT: To select faces or volumes that are located within the Partition behind other objects and cannot be simply selected, you have to hide the **partition and group** in the object tab. Use the eye icon to toggle them.



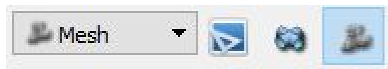
2.4 Export to GEOM

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.

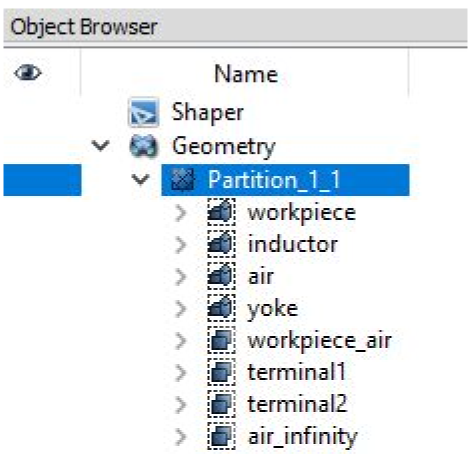
3. Create mesh and export it to CENOS

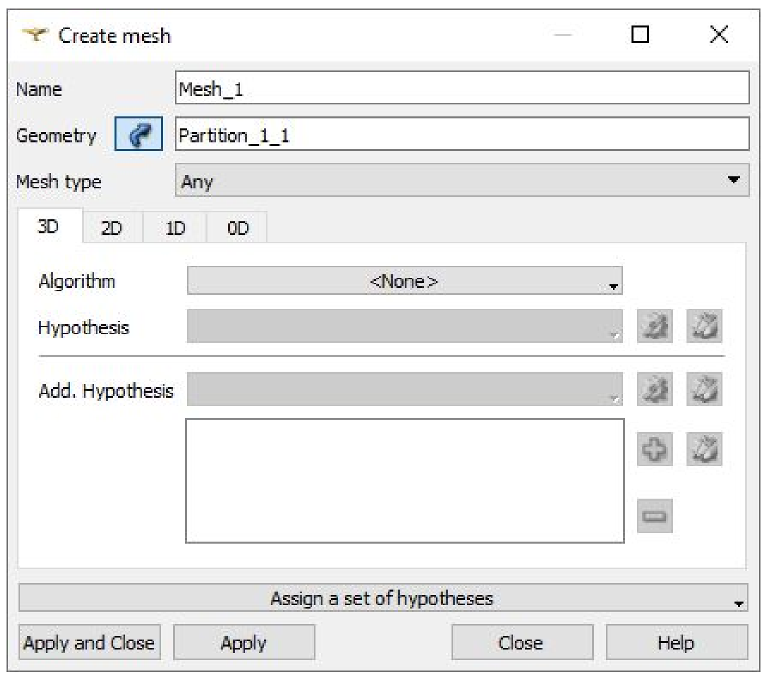
3.1 Switch to the Mesh module and create Mesh

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* ().

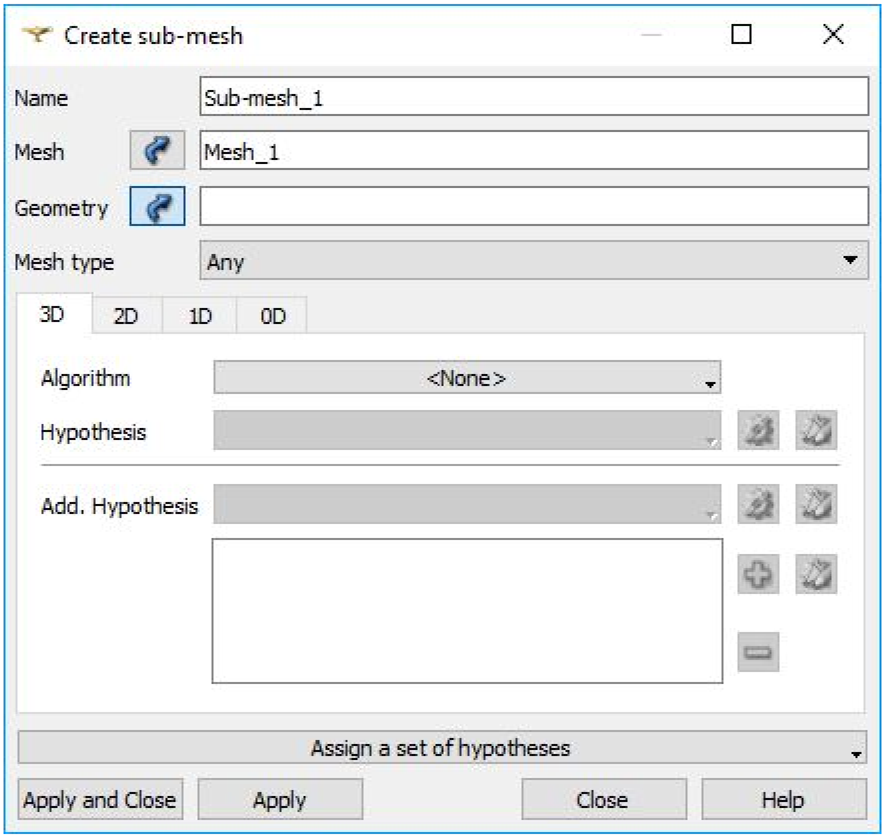




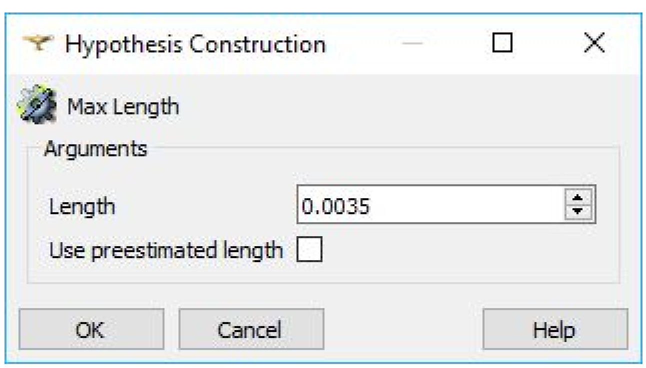
From the *Assign a set of hypothesis* dropdown menu select *3D: Automatic Tetrahedralization* – leave the *Max Length* value default and click *Apply and Close*.

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.

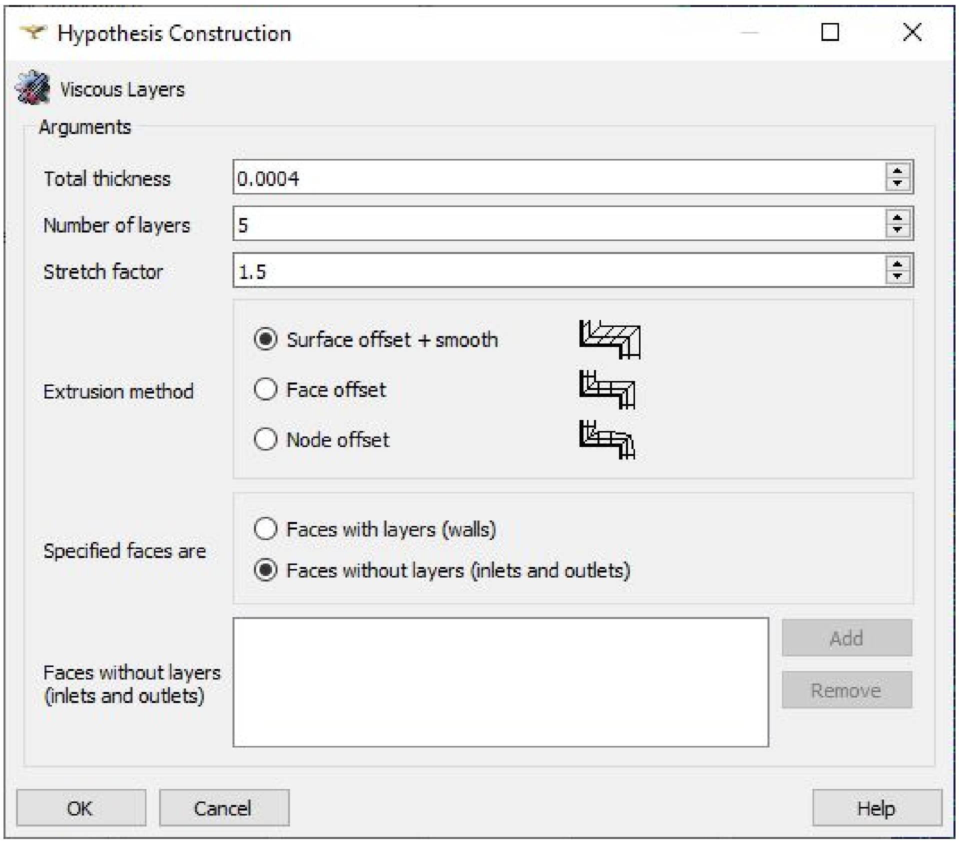


Select the *workpiece* group from the *Partition_1* dropdown menu as *Geometry*. From the *Assign a set of hypothesis* dropdown menu select *3D: Automatic Tetrahedralization*. In the *Hypothesis Construction* window enter **0.0035** for *Max Length*.

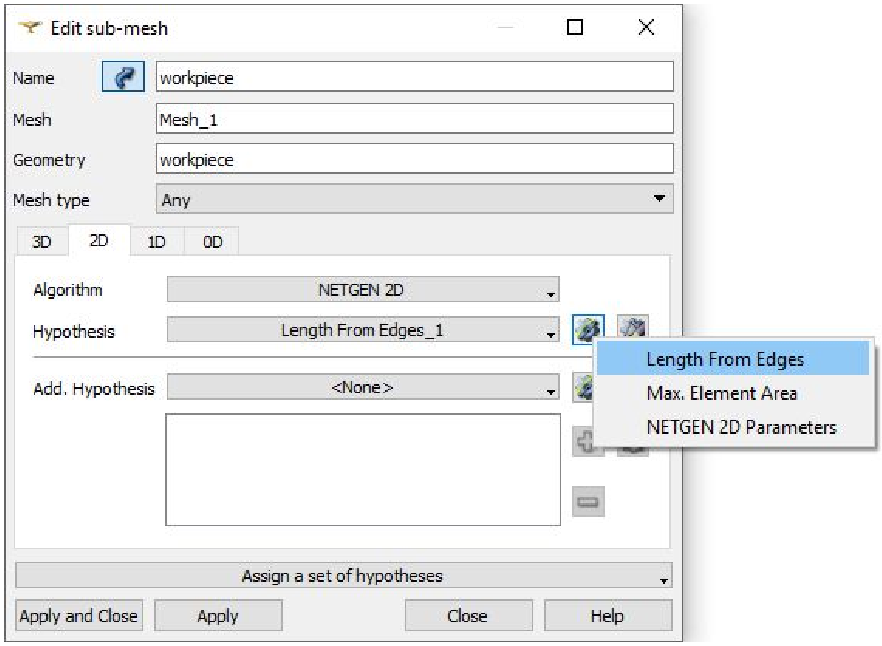


Resolve the skin layer on the surface of the workpiece by creating *Viscous Layers*. Under *3D algorithm* section click the gear icon ( ) next to *Add. Hypotheses* and select **Viscous Layers 2D**.

Enter **0.0004** for *Total thickness*, **5** for *Number of layers*, **1.5** for *Stretch factor* and check the *Faces without layers (inlets and outlets)* box.

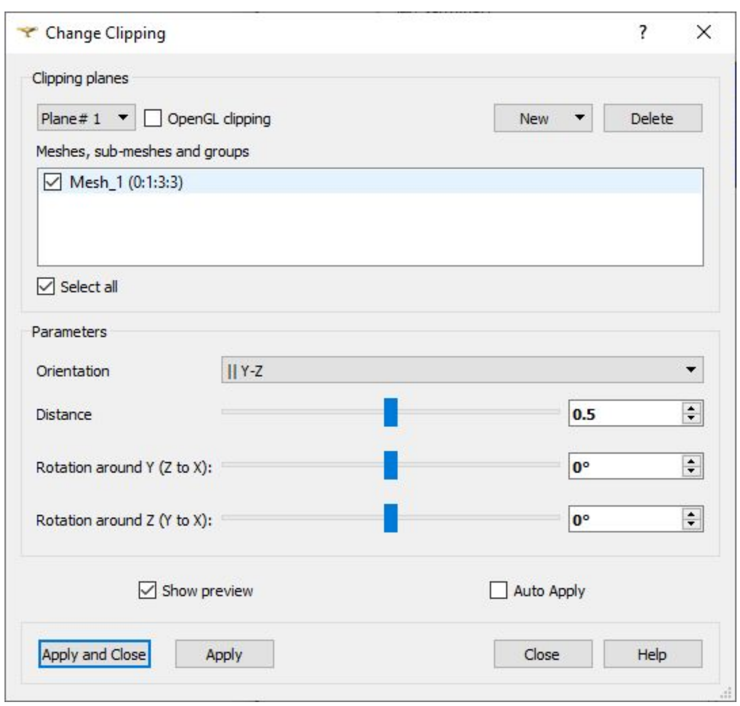


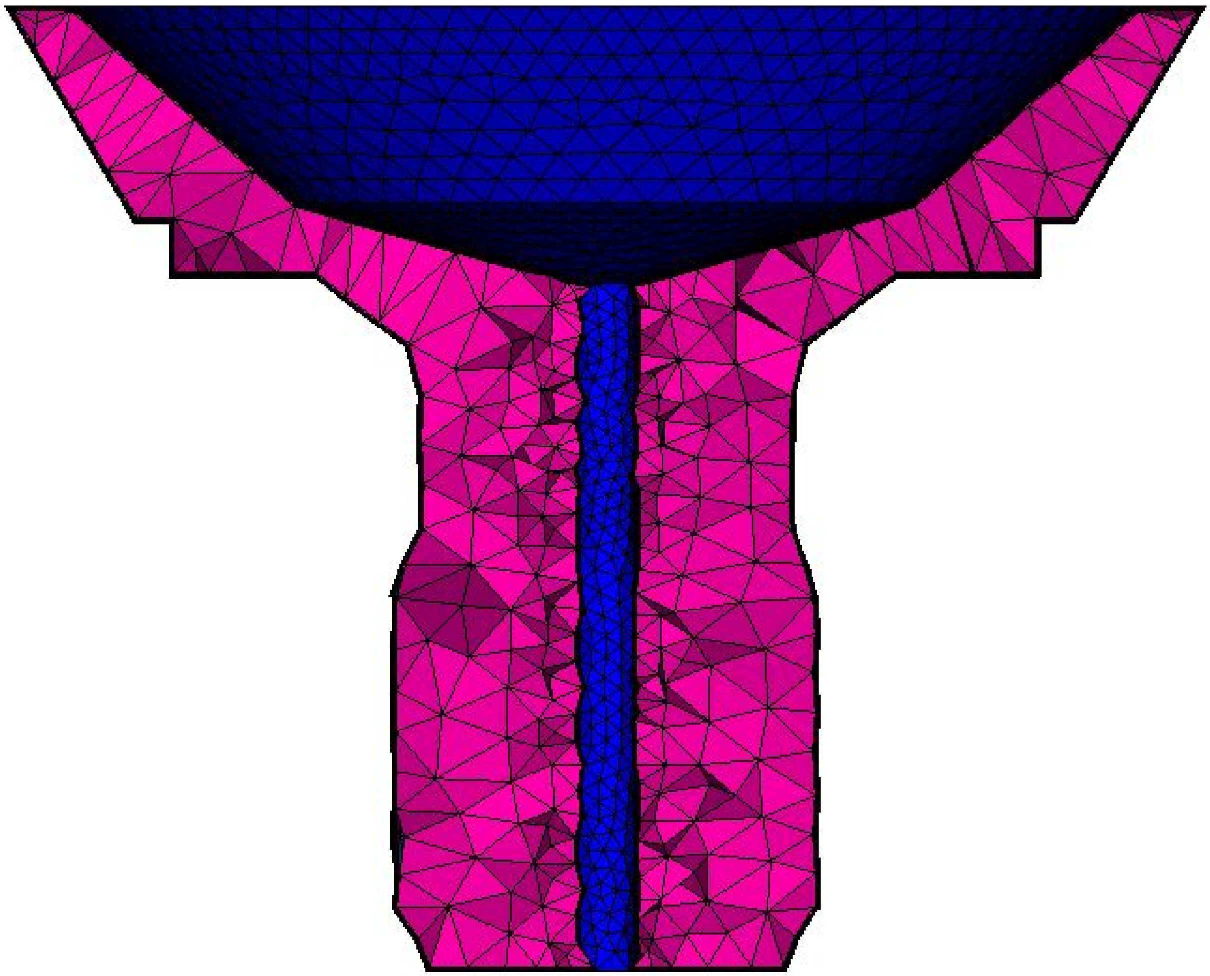
The **2D** meshing algorithm will have to be changed to **NETGEN 2D** for it to mesh correctly. Under 2D change the algorithm and then click the gear icon next to **Hypothesis** and select **Length From Edges.**



When all is set, click *Apply and Close*. Then right-click on just created sub-mesh and click *Compute Sub-mesh* to calculate and evaluate it.

To see what the mesh looks like inside of an object such as workpiece, right-click on the previously calculated mesh in the *VTK scene viewer* window and select *Clipping*. In the *Change Clipping* window select *Relative* from the *New* dropdown menu, then select orientation of the clipping plane and click *Apply and Close*.





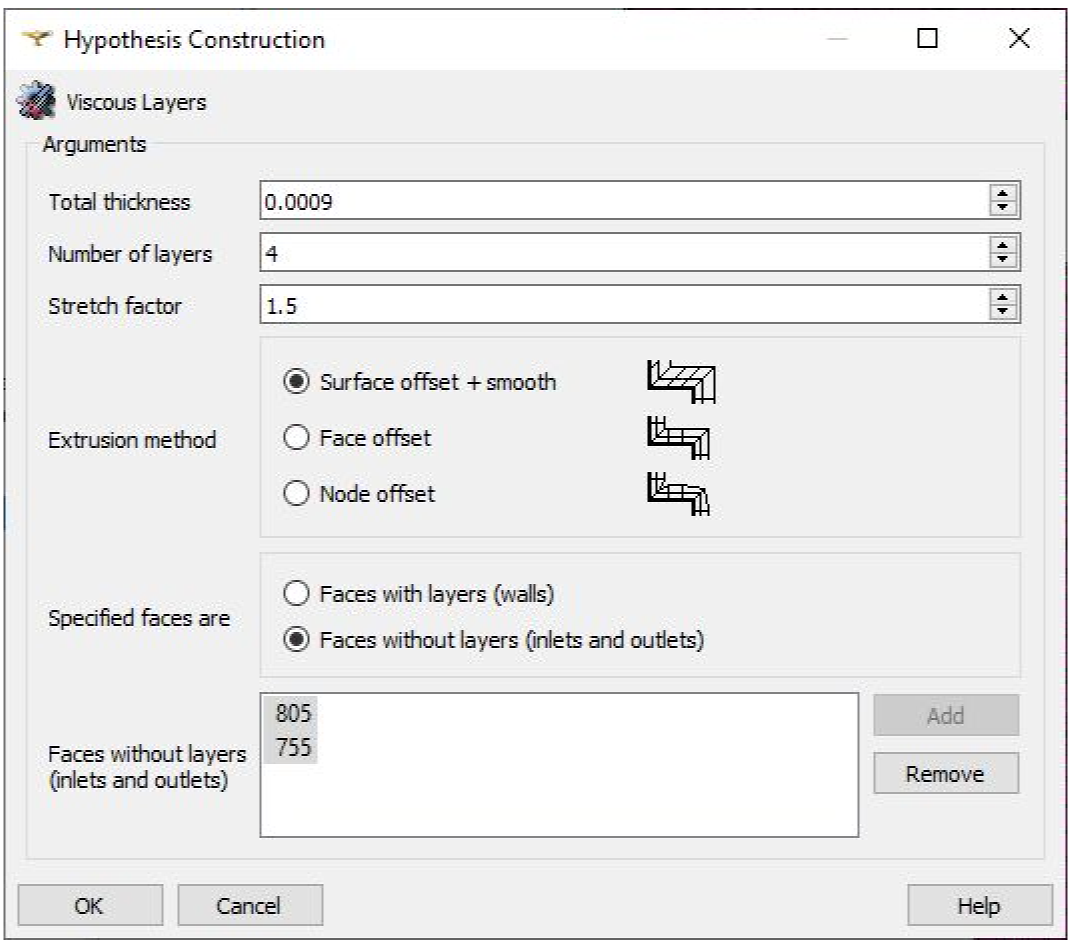
To return mesh to the unclipped state, enter the *Change Clipping* window and un-check the *Mesh_1(0:1:2:3)* box under *Meshes, sub-meshes and groups*.

3.3 Create a sub-mesh for the inductor

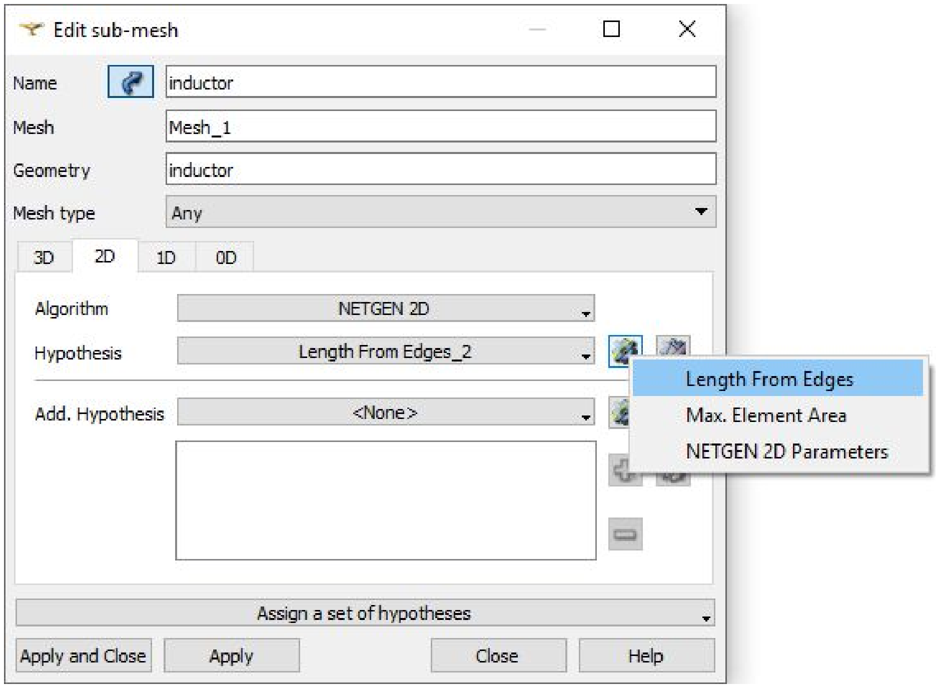
Create a sub-mesh and select the inductor group from the *Partition_1_1* dropdown menu as *Geometry*. From the *Assign a set of hypothesis* dropdown menu choose *3D: Automatic Tetrahedralization* and enter **0.003** for *Max Length*.

Resolve the skin layer on the surface of the workpiece by creating *Viscous Layers*. Under *3D algorithm* section click the gear icon () next to *Add. Hypotheses* and select **Viscous Layers 2D**.

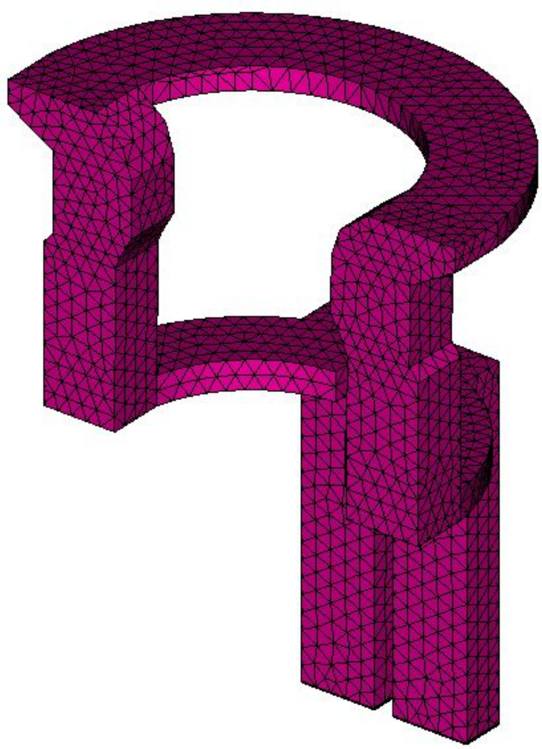
Select the groups *terminal1* and *terminal2* from the *Partition_1_1* dropdown menu and click *Add*. Enter **0.0009** for *Total thickness*, **4** for *Number of layers*, **1.5** for *Stretch factor* and check the *Faces without layers (inlets and outlets)* box.



Again, the **2D** meshing algorithm will have to be changed to **NETGEN 2D.** And the ***Hypothesis*** set to ***Length From Edges***.



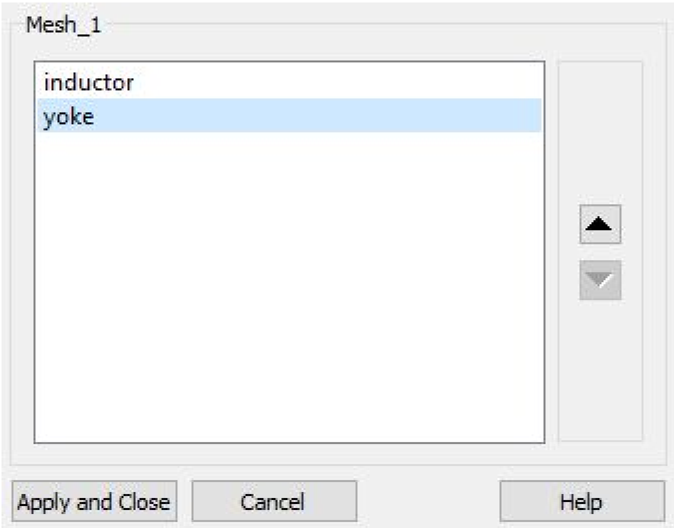
Calculate and evaluate the mesh.



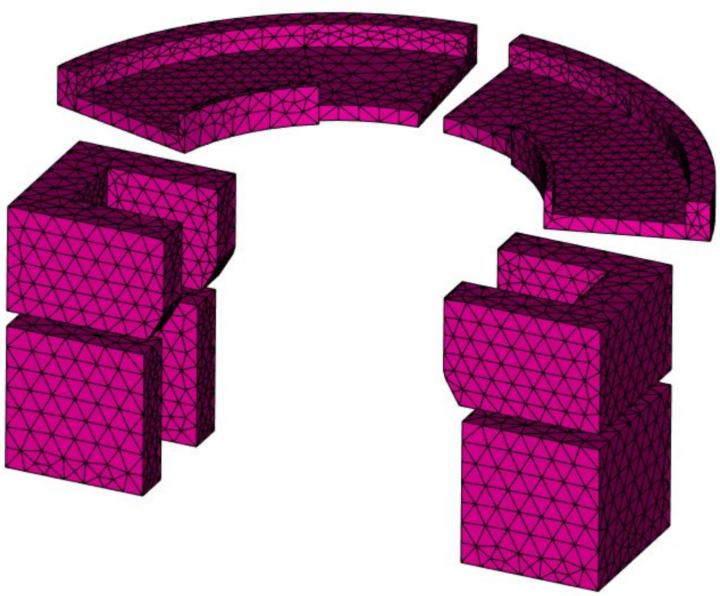
3.4 Create a sub-mesh for the yoke

Create a sub-mesh and select the *yoke* group from the *Partition_1_1* dropdown menu as *Geometry*. From the *Assign a set of hypothesis* dropdown menu choose *3D: Automatic Tetrahedralization* and enter **0.003** for *Max Length*.

Before mesh calculation Salome will ask to **set sub-mesh priorities**, because the inductor and yoke geometries share common surfaces. Click *Yes* when the *Warning* window shows. Make sure that the *conductor* is above *yoke*, then click *Apply and Close*.



Calculate and evaluate the mesh.

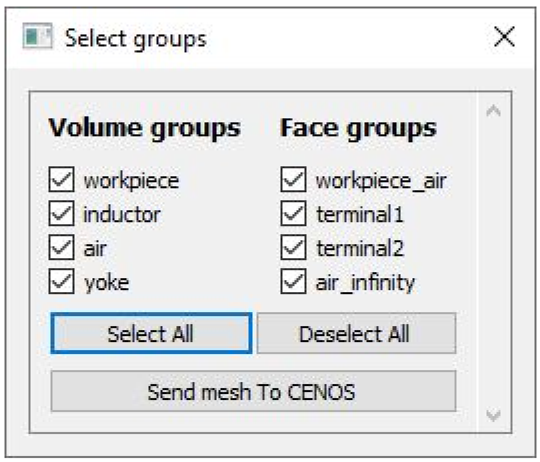


3.5 Calculate and export the mesh to CENOS

Right-click on *Mesh_1* and click *Compute*. Evaluate the final mesh and export it to CENOS by selecting *Mesh to CENOS* from the dropdown menu under *Tools* ⟶ *Plugins* ⟶ *Mesh to CENOS*.

Before exporting the 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 the groups relevant for the physics setup, i.e. those who will be defined as domains or boundary conditions.

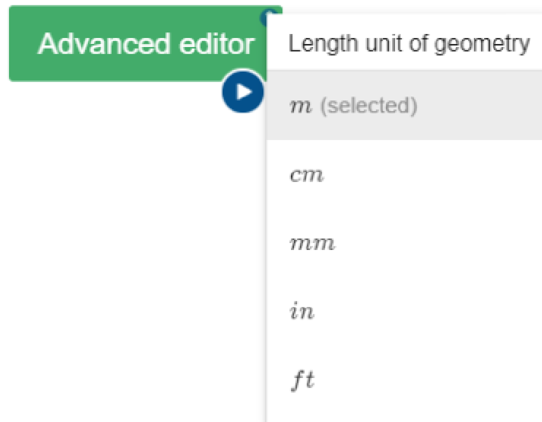


When selected, click *Send mesh to CENOS*.

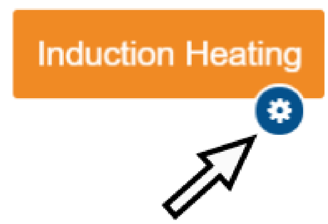
4. Define physics and boundary conditions

4.1 Set the units and enter the 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 **meters (m)**.

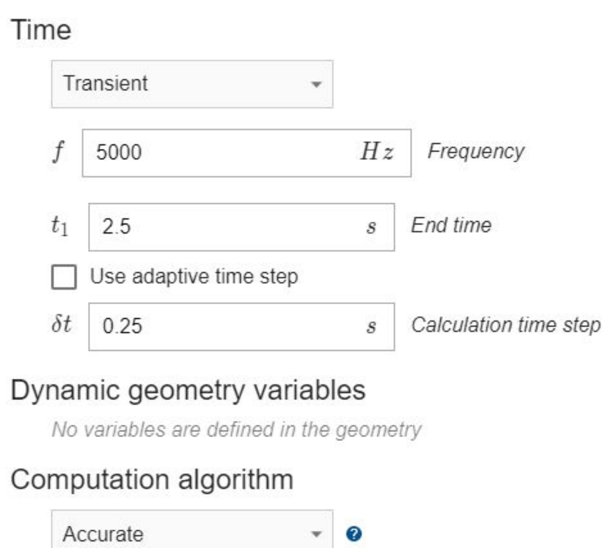


Click the **gear icon** under *Induction Heating* block to enter the physics setup.



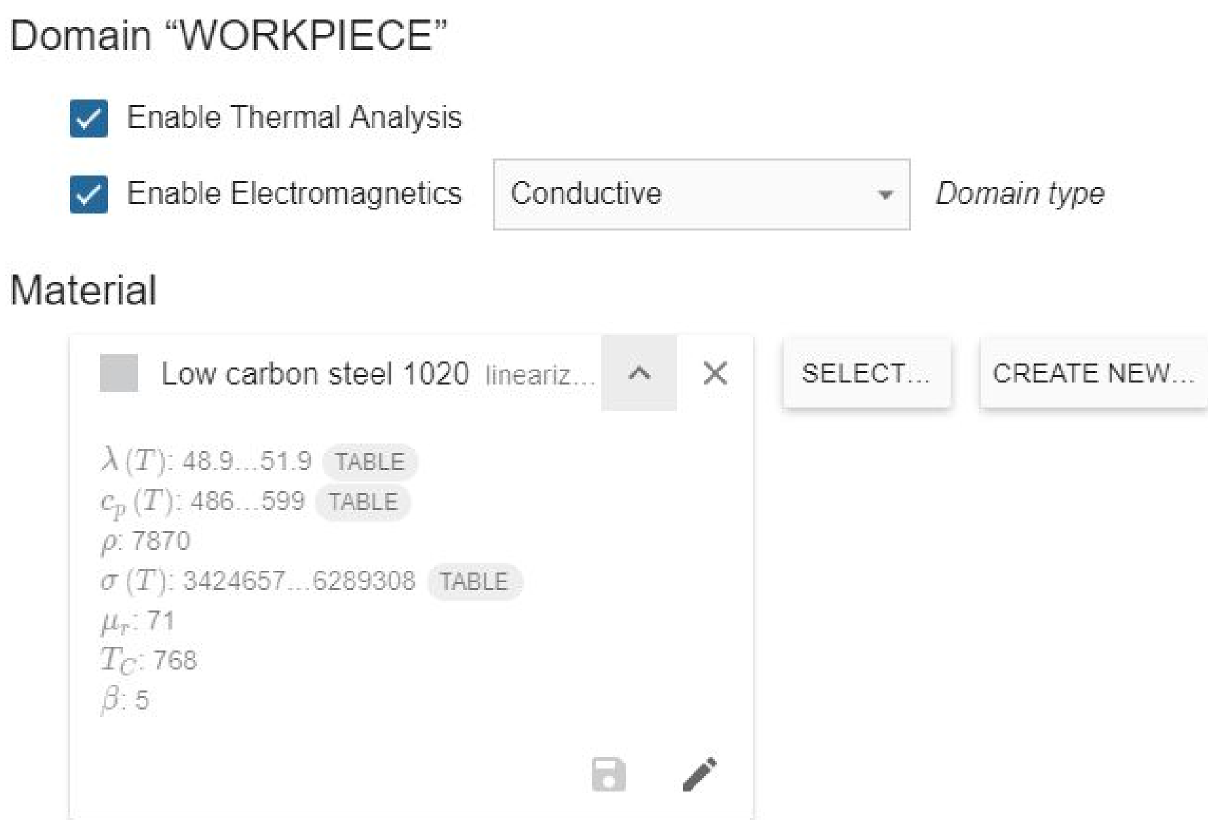
4.2 Simulation control

In the SIMULATION CONTROL window define the simulation as *transient* with **5 kHz** *frequency*, **2.5 s** *End time* and **0.25 s** *time step*. For *Computation algorithm* choose **Accurate.**

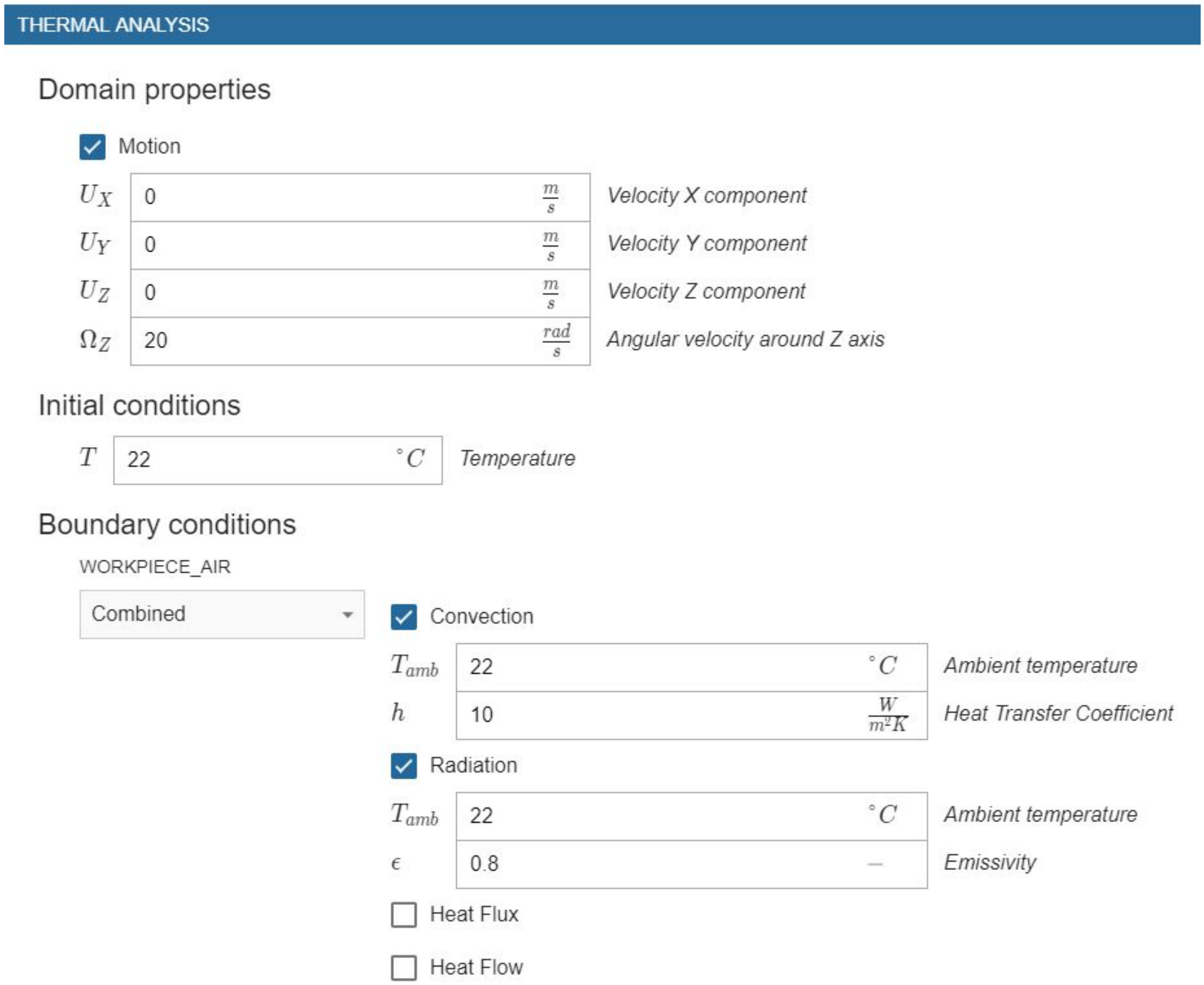


4.3 Workpiece definition

Select WORKPIECE from *Domain bar*. Leave *Enable Thermal Analysis* and *Enable Electromagnetics* boxes checked under Domain “WORKPIECE”. Choose **Conductive** as the domain type. For *Material* click SELECT... and choose **Low carbon steel 1020 linearized (H=100000A/m), t depend.**



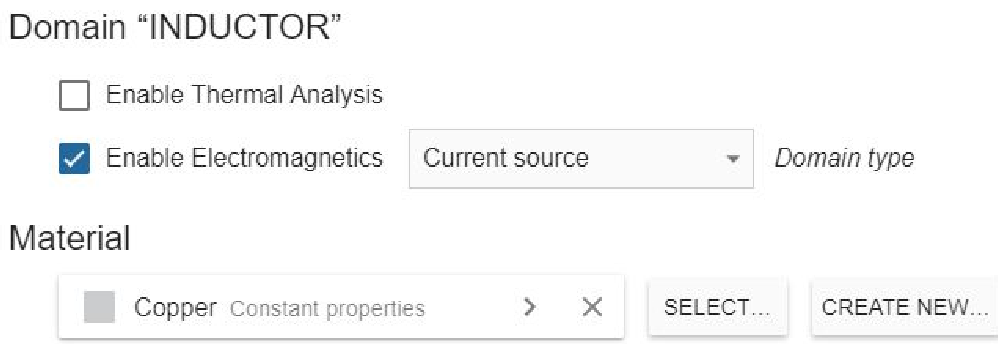
Under THERMAL ANALYSIS check the *Motion* box and enter **20** for *Angular velocity around the Z axis*. For WORKPIECE_AIR choose **Combined** – check the **Convection** and **Radiation** boxes and enter **10** for *Heat Transfer Coefficient* and **0.8** for *Emissivity*.



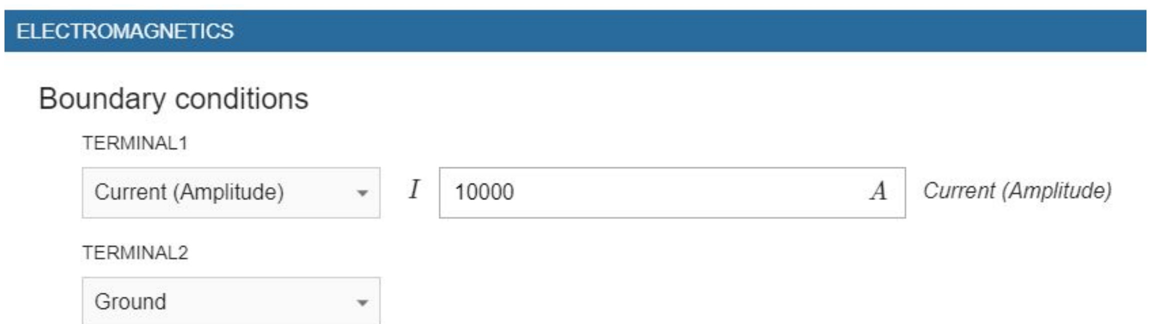
Under ELECTROMAGNETICS choose **Interface** for WORKPIECE_AIR.

4.4 Inductor definition

Switch to INDUCTOR in *Domain bar*. Disable *Thermal analysis* and select *Current source* as *Domain type*. For *Material* choose **Copper Constant properties.**

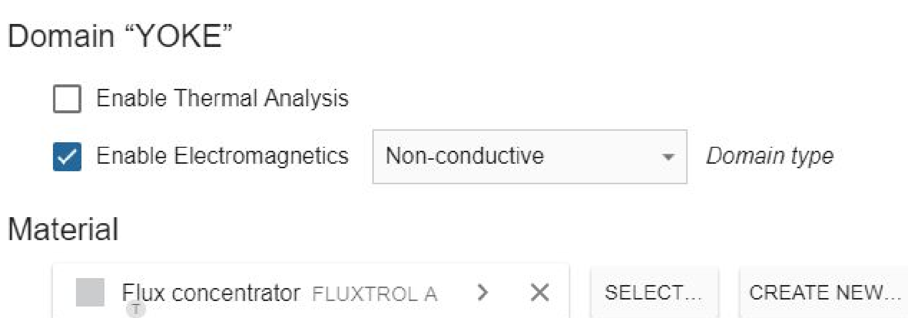


Under ELECTROMAGNETICS choose **Current (Amplitude)** for TERMINAL1, Ground for TERMINAL2 and enter **10000 A** as *Current (Amplitude)*.



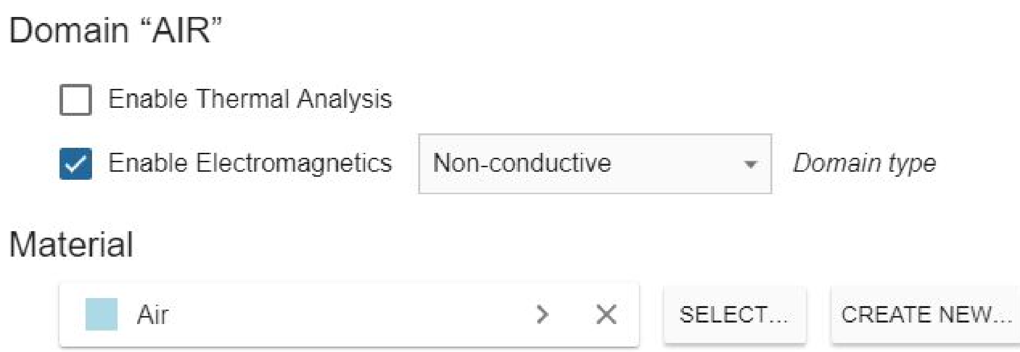
4.5 Yoke definition

Switch to YOKE in *Domain bar*. For the yoke domain disable *Thermal analysis* and select *Non-conductive* as *Domain type*. For *Material* choose **Flux concentrator FLUXTROL A.**

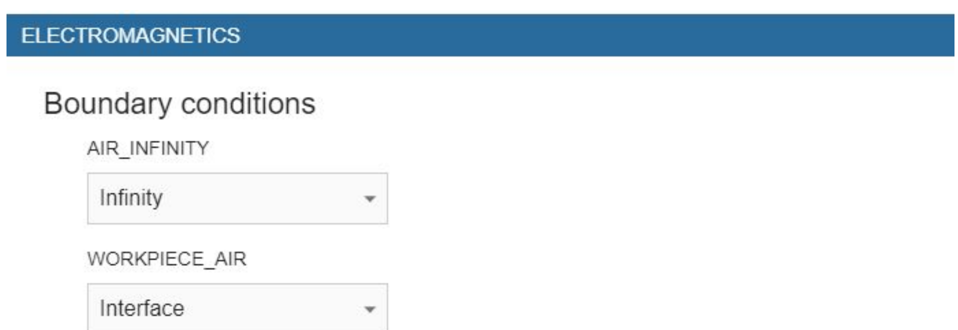


4.6 Air definition

For air domain disable *Thermal analysis* and select **Non-conductive** for *Domain type*. For *Material* choose **Air**.



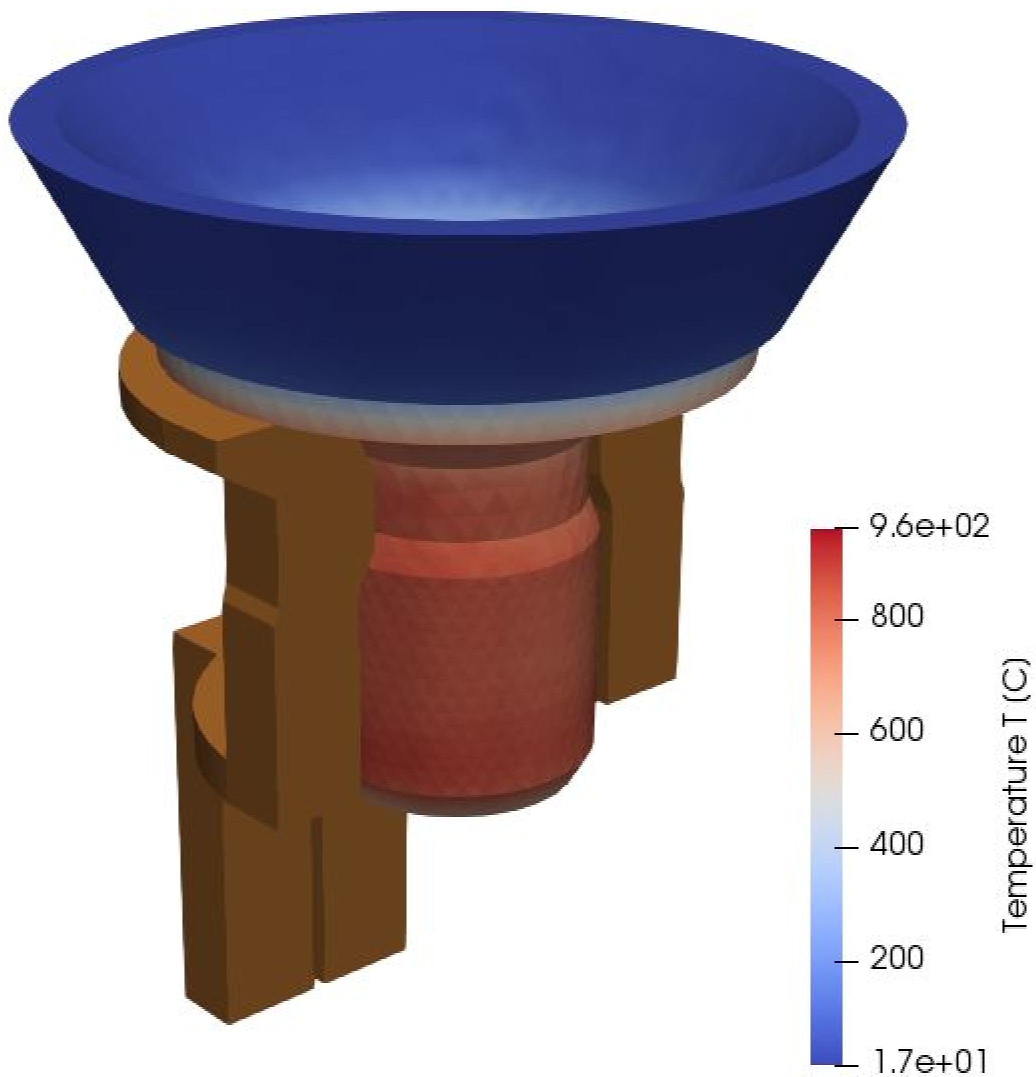
Under ELECTROMAGNETICS choose **Infinity** for AIR_INFINITY and **Interface** for WORKPIECE_AIR.



When everything is set, click RUN.

5. Evaluate results

When CENOS finishes calculation, a ParaView window with pre-set temperature result state will open and you will be able to see the temperature field distribution in the workpiece in the last time step.



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

This concludes our 3D single shot induction heating simulation tutorial. For any recommendations or questions contact our support.