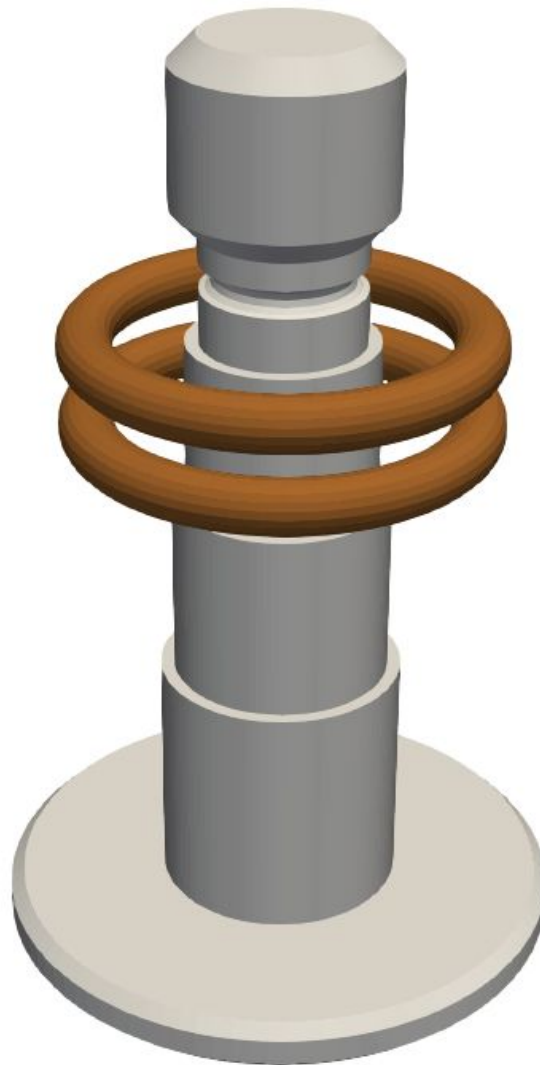


2D Scanning of Transmission Shaft

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**.

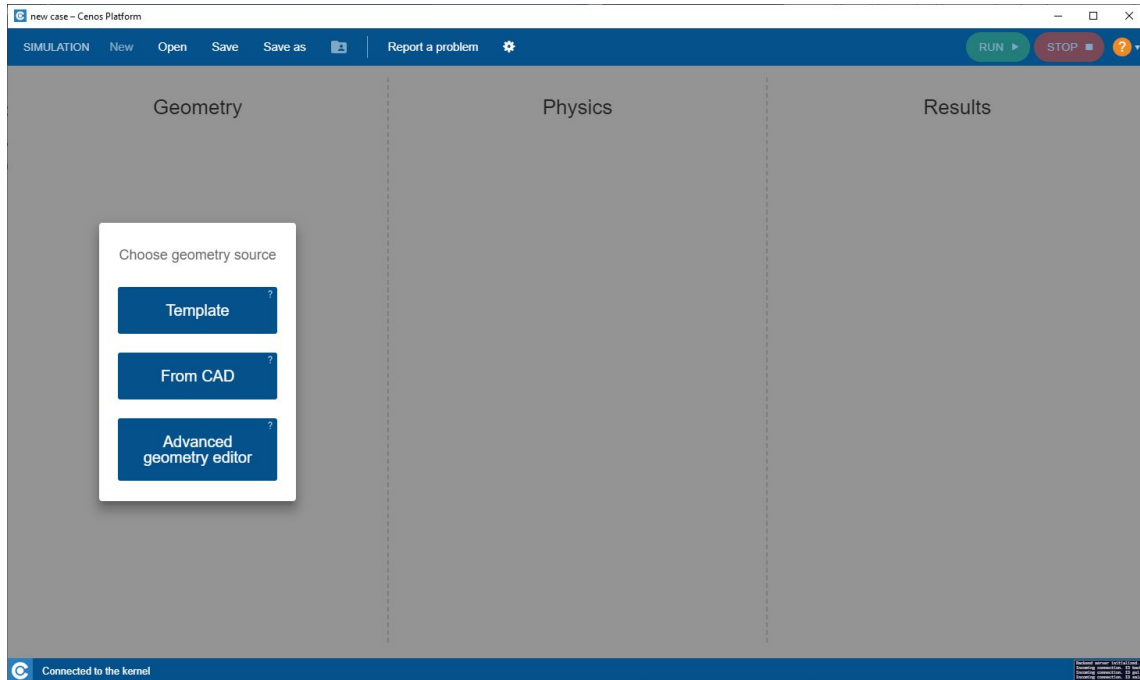
In this tutorial we will learn how to create and set up the simulation of a **30s** long **transmission shaft scanning** at **10 kHz** and **10 kA** with two winding inductor.



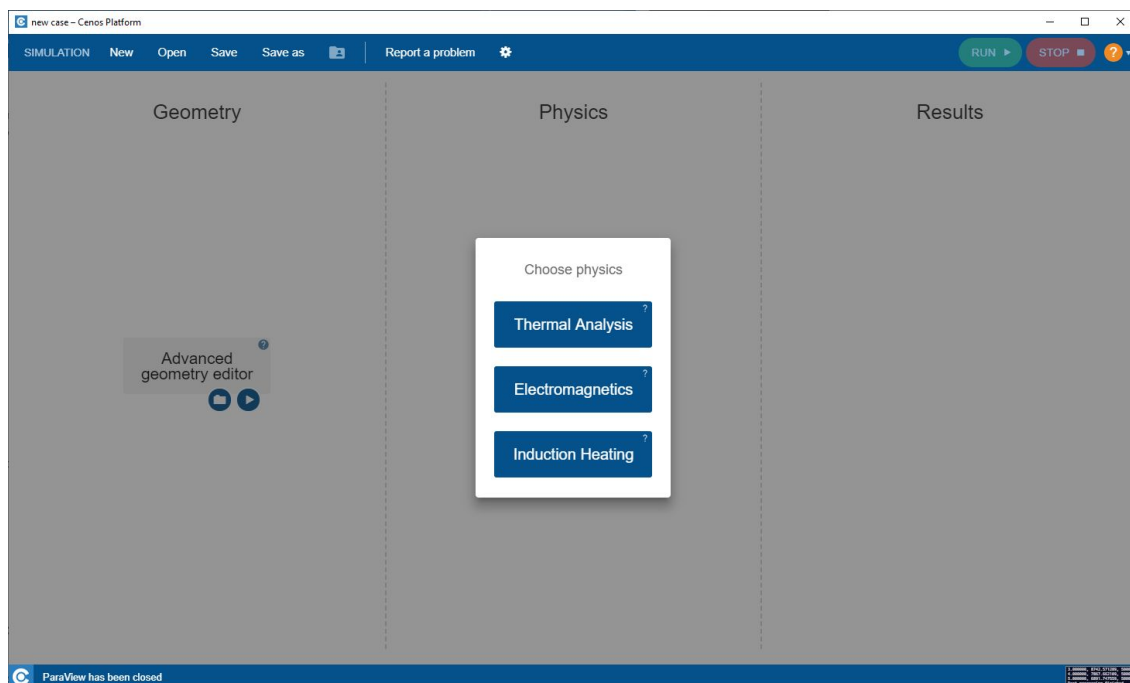
1. Open pre-processor

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.



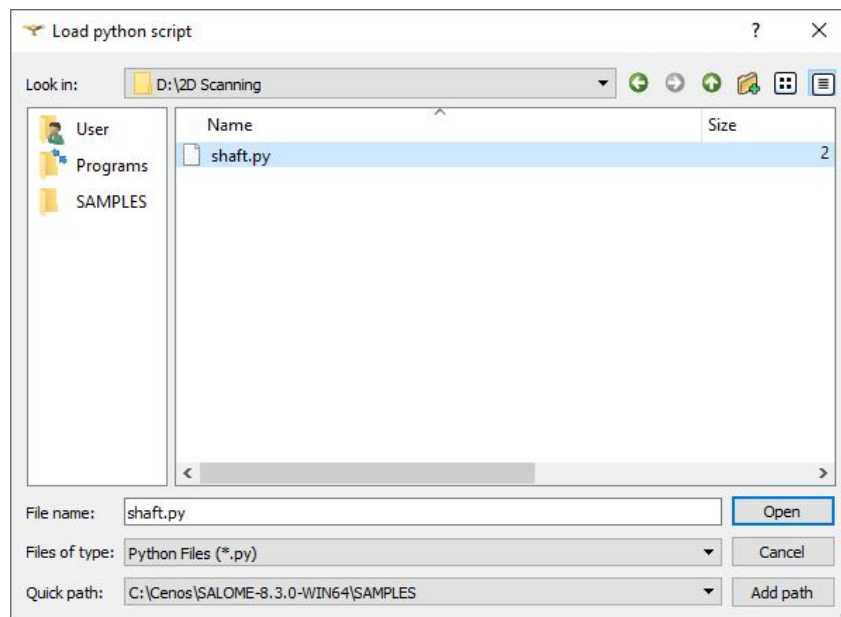
1.2 Load shaft geometry

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

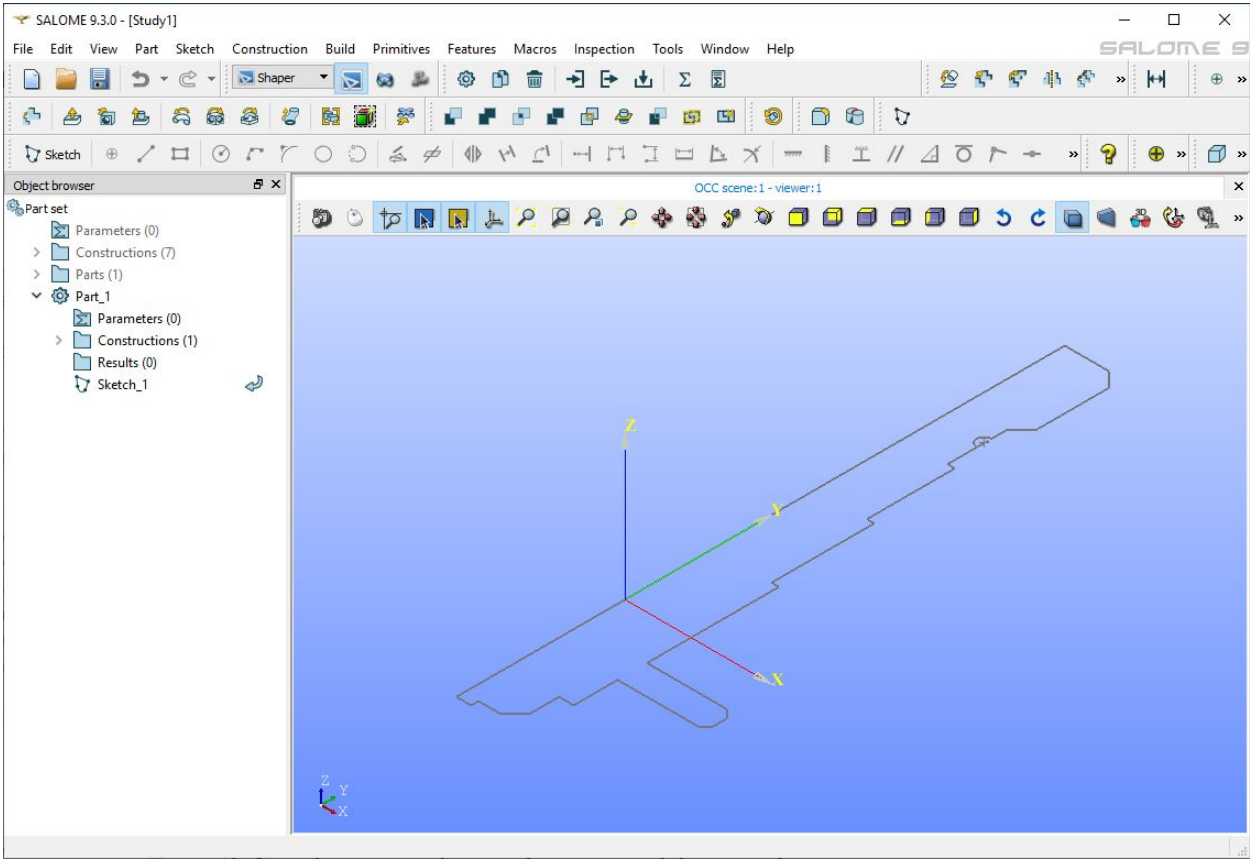
IMPORTANT: To follow this tutorial, download the **shaft.py** Salome geometry script from documentation website.

Click **X** → **x** .

In the **x** . window select and open previously downloaded script file.




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

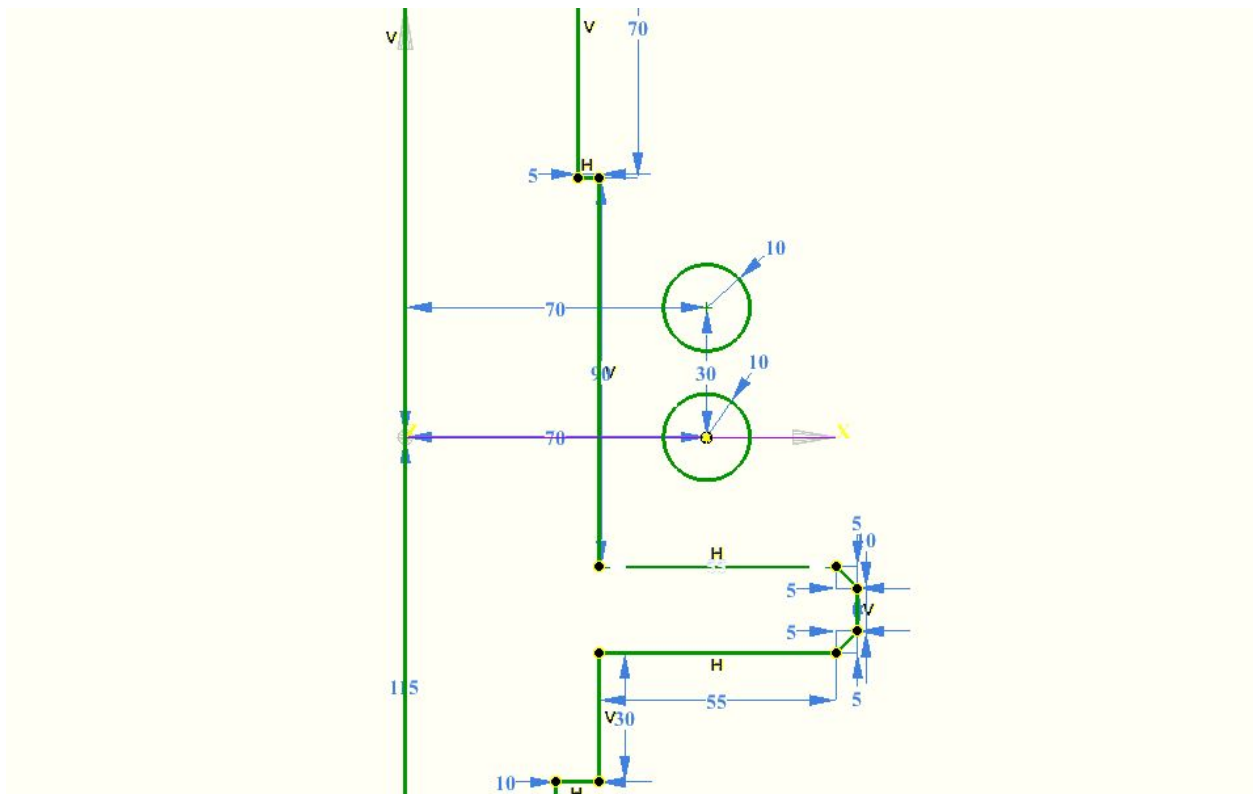


2. Create geometry and prepare it for meshing

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 **Sketch_1** right click on **Sketch_1** and click **@**. By using **1** () tool create **two circles**, both **70 mm** far from the Y axis and with **30 mm** distance between them.

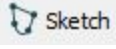


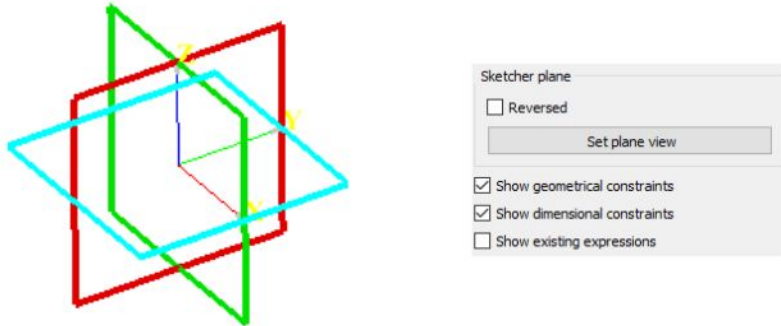
Use the constraints to set the dimensions:





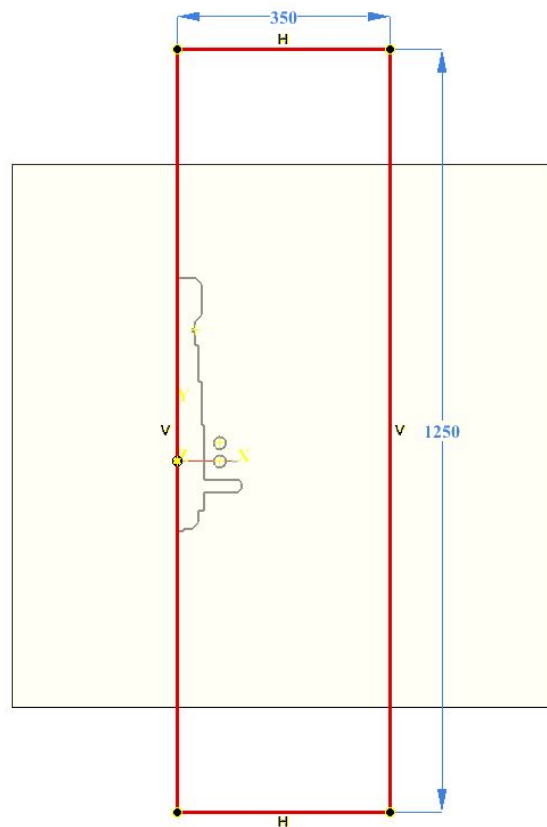
If you cannot draw objects, you must disable camera control ().

2.2 Create the air box


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

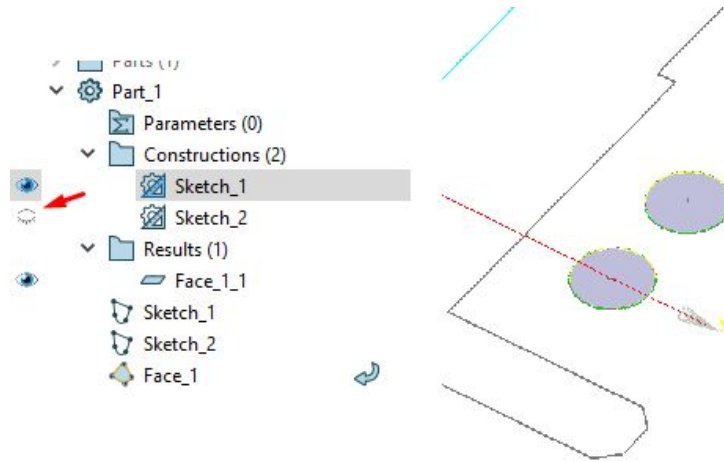


Create a rectangle (1250mm x 350mm) around the workpiece by clicking the  icon. Add a **1** () constraint between the symmetry axis edge (left side of the air box) and the origin point.

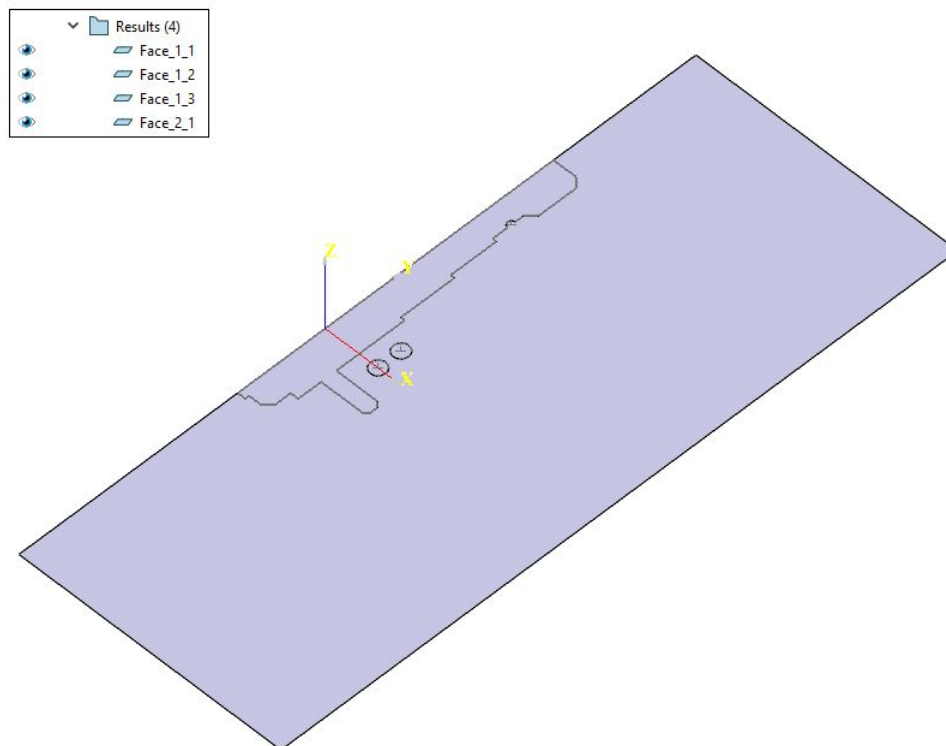


2.3 Create faces


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



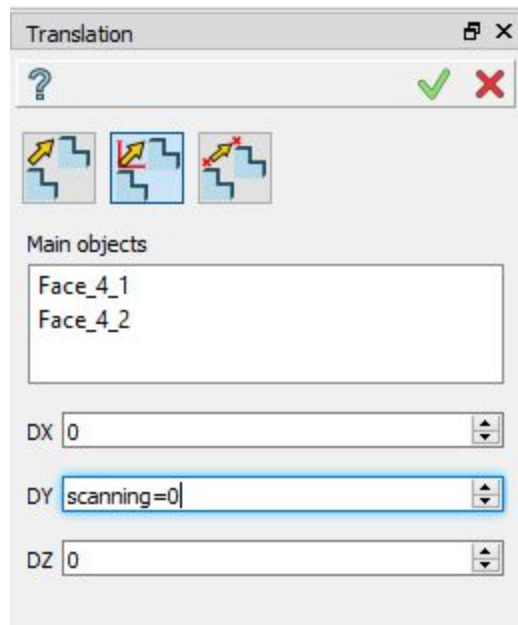
Now create the fourth air domain face and you should have all the faces as shown here:



2.4 Translate inductor windings


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

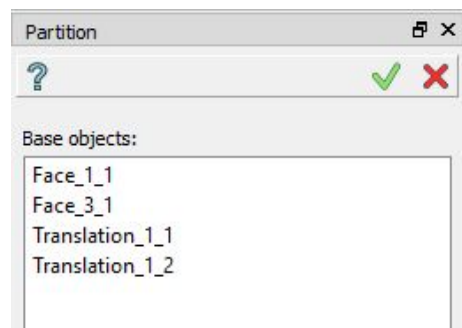
For δx and δz enter 0. For δy enter **scanning=0**.







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).

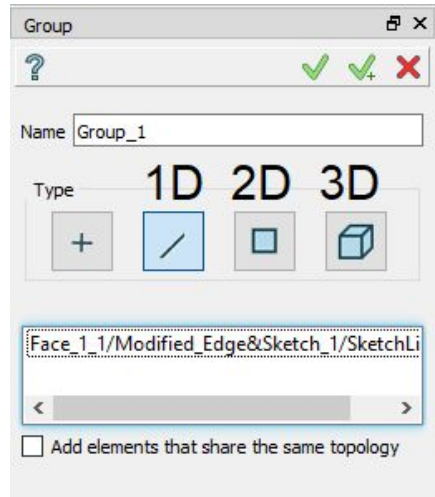
2.5 Create partition and groups

Click  tool, select previously created faces and join them in one partition.



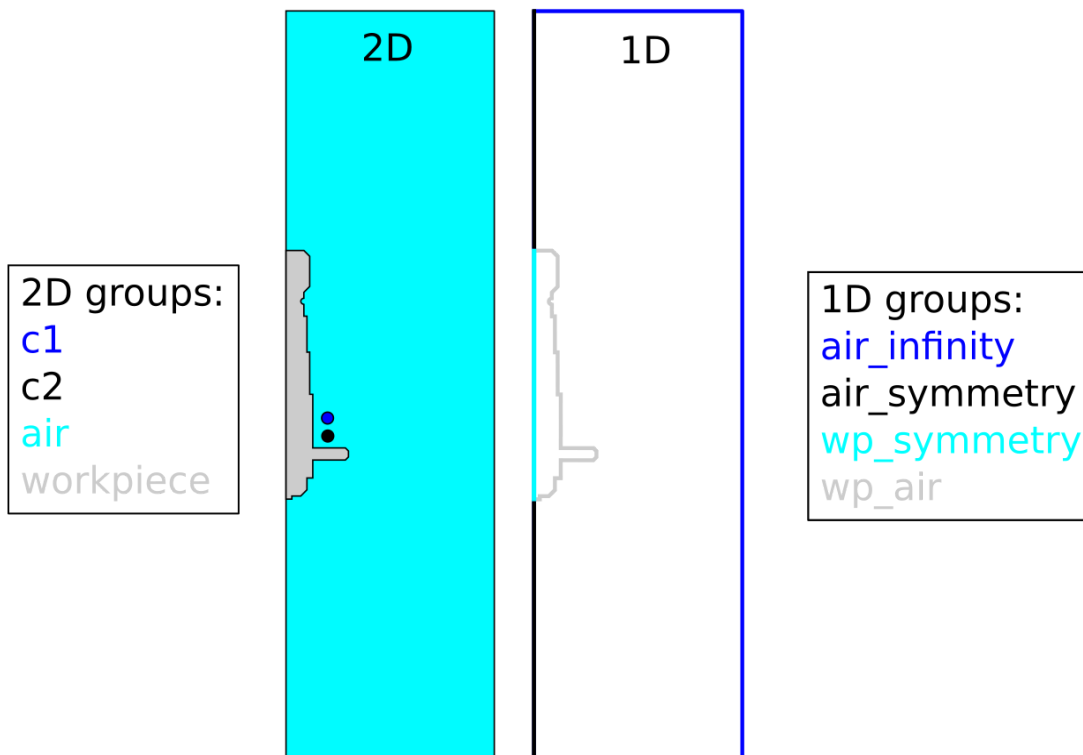
IMPORTANT: Partition and Groups are vital for simulation setup with CENOS, because mesh creation as well as physics and boundary condition definitions are based on groups created in this part.

Select **Y** () tool and choose the **AE**  . Select one or more shapes from the screen, name the group and click the  ().





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 @  (). This will export the » and Y to GEOM module, which is needed to proceed with mesh creation.

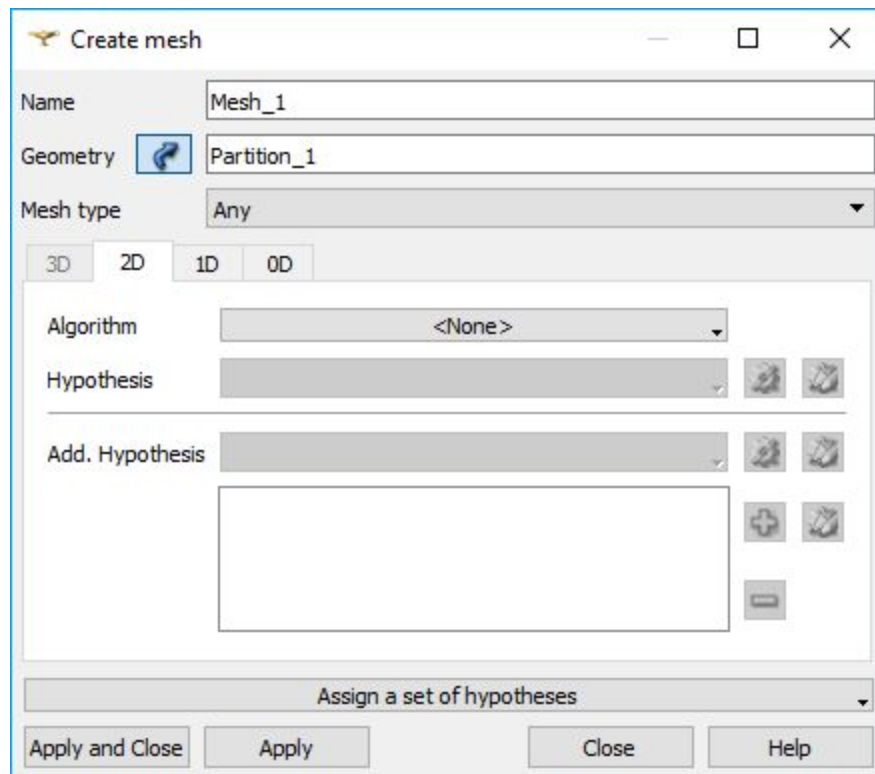
3. Create mesh and export it to CENOS

3.1 Switch to Mesh module and create Mesh

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



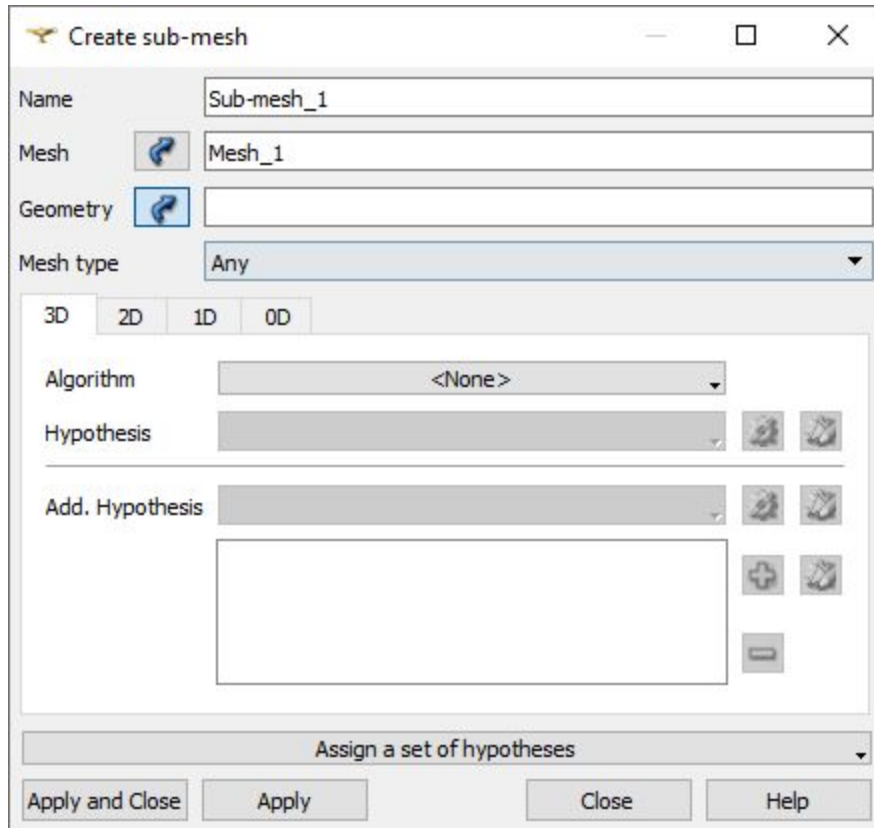
In •  from Y  dropdown menu select the previously created » and click  ().



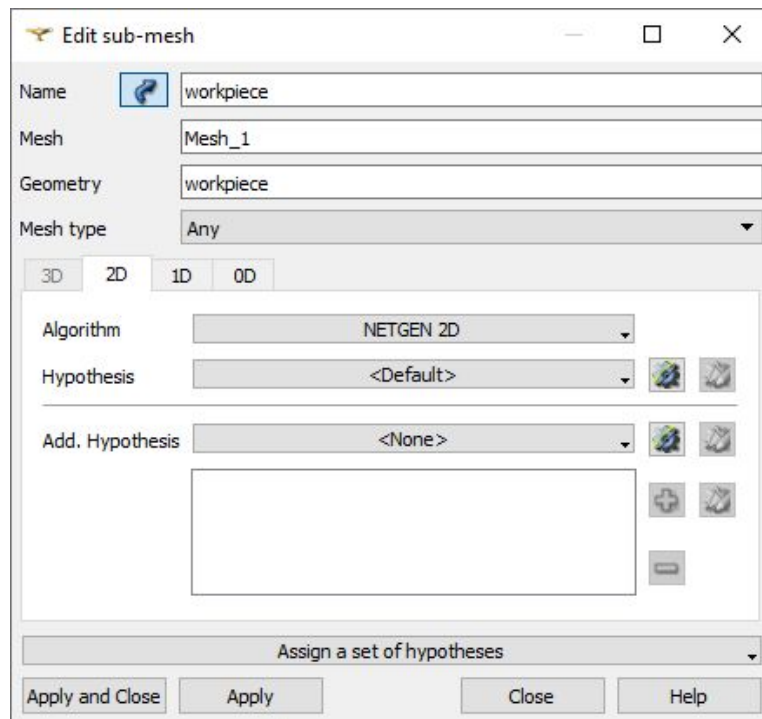
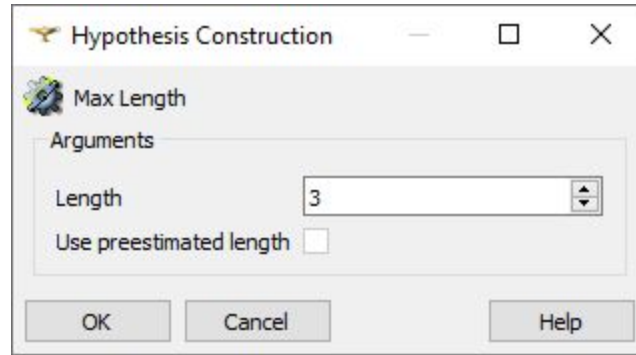
From the **Mesh** dropdown menu select **Hexahedron**.
 leave the **Order** value default and click **OK**.


3.2 Create a sub-mesh for the workpiece

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

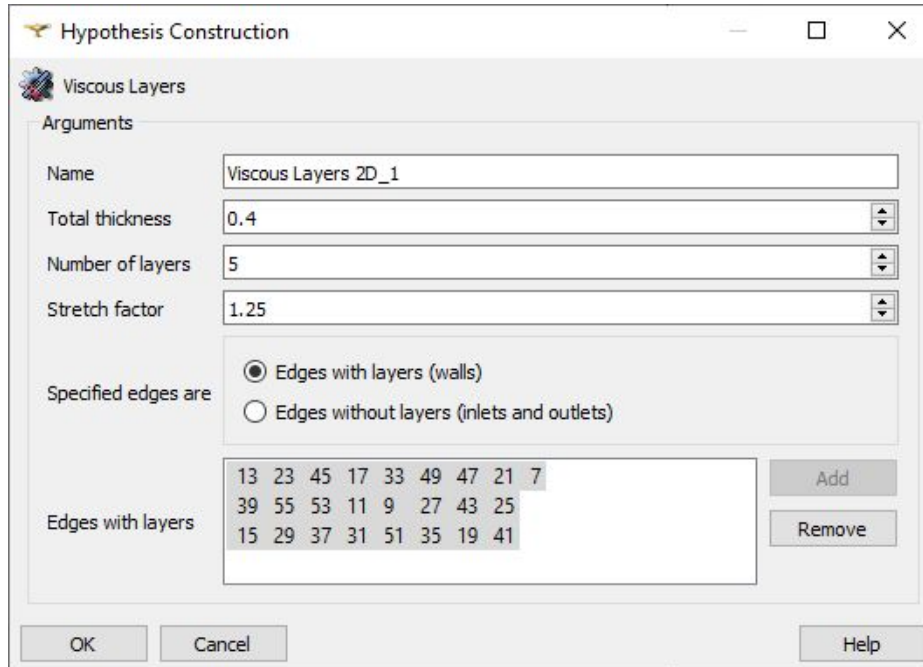


Choose **workpiece** group from the **Mesh** dropdown menu as **Hexahedron**. From the **Order** dropdown menu choose **3**. In the **Order** window enter **3** for **Order** and change the 2D algorithm from **Hexahedron** to **tetrahedron**.







Resolve the skin layer on the surface of the workpiece by creating \bar{n} \times \bar{x} . Click the gear icon () next to \bar{n} and select **Viscous Layers 2D**.




Select the group \bar{t} from the \bar{x} dropdown menu and click \bar{n} . Enter **0.4** for \bar{t} , **5** for \bar{t} , **1.25** for \bar{t} and check the \bar{t} .







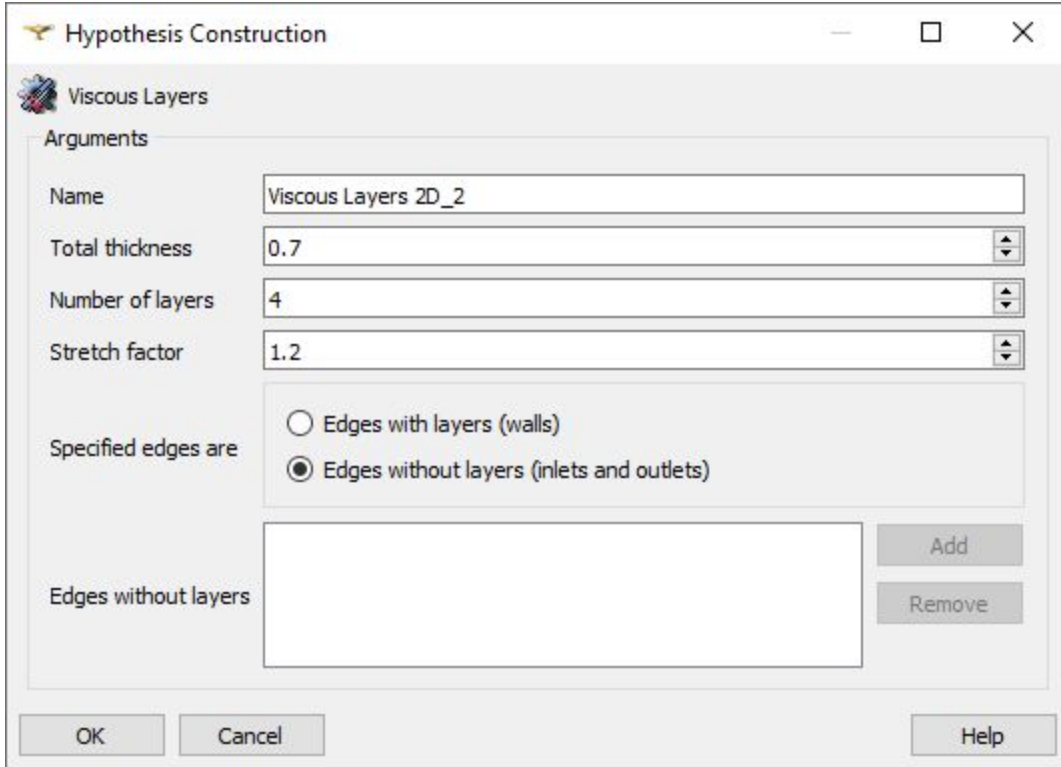
When all is set, click  .

3.3 Create a sub-mesh for the inductor

Create a sub-mesh and select  and  groups from the  dropdown menu as **Y** . From the  dropdown menu choose **8** .

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

Enter **0.7** for  , **4** for  , **1.2** for  and check the  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**.

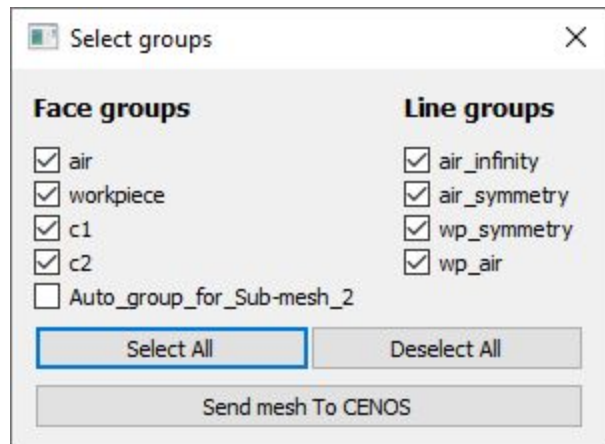
3.4 Calculate and export mesh to CENOS

Right-click on **Mesh** and click **1**. Evaluate the final mesh and export it to CENOS. To do that, select **1@1** from the dropdown menu under **1@1** to export your mesh to CENOS.

Before exporting mesh to CENOS, the **1@1** 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 **1@1**.

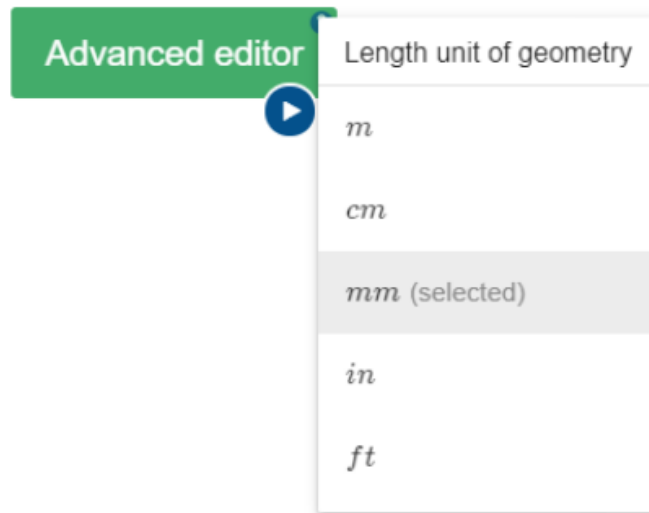
When selected, click **1@1**.



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 **Transient** and with **10 kHz**, **30 s @** and **1 s**.

Axial symmetry

Yes (y axis) ▼

Y-axis must be the axis of symmetry in geometry/mesh. If it is not so, please go back to pre-processing part, rotate the mesh and save it to CENOS again

Time

Transient ▼

f 10000 Hz Frequency

t_1 30 s End time

Use adaptive time step

δt 1 s Calculation time step

Check the **Dynamic geometry variables** check box. Leave the **e** **20 mm** and enter **10** for **\dot{n}** . For **1** leave **Automatic**.

Dynamic geometry variables

Enable

Variable	Initial value (mm)	Velocity ($\frac{mm}{s}$)
<input checked="" type="checkbox"/> scanning	20	10

Computation algorithm

Automatic ▼ ?

4.3 Workpiece definition

Switch to WORKPIECE in . Leave @ and @ boxes checked under the Domain "WORKPIECE". Choose **Conductive** as the domain type. For click SELECT... and choose **Low carbon steel 1020 B(H), t depend**.

Domain "WORKPIECE"

Enable Thermal Analysis

Enable Electromagnetics

Conductive Domain type

Material

Low carbon steel 1020 B(H), t°...
^
×

$\lambda(T)$: 48.9...51.9 TABLE

$c_p(T)$: 486...599 TABLE

ρ : 7870

$\sigma(T)$: 3424657...6289308 TABLE

$B(H)$: 0...10 TABLE

T_C : 768

β : 5

SELECT...
CREATE NEW...

Under THERMAL ANALYSIS for boundary conditions choose **1** for WP_AIR – check the **1** and **3/4** boxes and enter **10** for and **0.8** for @ . Choose for WP_SYMMETRY.

THERMAL ANALYSIS

Domain properties

Motion

Initial conditions

T °C *Temperature*

Boundary conditions

WP_SYMMETRY

WP_AIR

Convection

T_{amb}	<input type="text" value="22"/>	°C	<i>Ambient temperature</i>
h	<input type="text" value="10"/>	$\frac{W}{m^2K}$	<i>Heat Transfer Coefficient</i>

Radiation

T_{amb}	<input type="text" value="22"/>	°C	<i>Ambient temperature</i>
ϵ	<input type="text" value="0.8"/>	—	<i>Emissivity</i>

Heat Flux

Heat Flow

Under ELECTROMAGNETICS choose **e** for WP_AIR and **AE** for WP_SYMMETRY.

ELECTROMAGNETICS

Boundary conditions

WP_SYMMETRY

WP_AIR

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 **AE** . To do that, select all winding domains and click GROUP.



Disable **O** and select **1** for **8**. For „ choose **1** and enter **10000 A** for **1**.

Domain “C2”

Enable Thermal Analysis

Enable Electromagnetics *Domain type*

Material

Recent: Copper (Constant properties) Low carbon steel 1020 (B(H), t° depend.) Air Iron (B(H), temperature dependent) Flux concentrator (FLUXTROL A)

ELECTROMAGNETICS

Domain properties

I **A** *Current (Amplitude)*

4.5 Air definition

Switch to AIR in **8** . Disable **0** and select **f** as **8** . For „ choose ° .

Domain “AIR”

Enable Thermal Analysis

Enable Electromagnetics Non-conductive *Domain type*

Material

Air > X SELECT... CREATE NEW...

Under ELECTROMAGNETICS choose **e** for AIR_INFINITY, **AE** for AIR_SYMMETRY and **e** for WP_AIR.

ELECTROMAGNETICS

Boundary conditions

AIR_INFINITY

Infinity

AIR_SYMMETRY

Symmetry axis

WP_AIR

Interface

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

