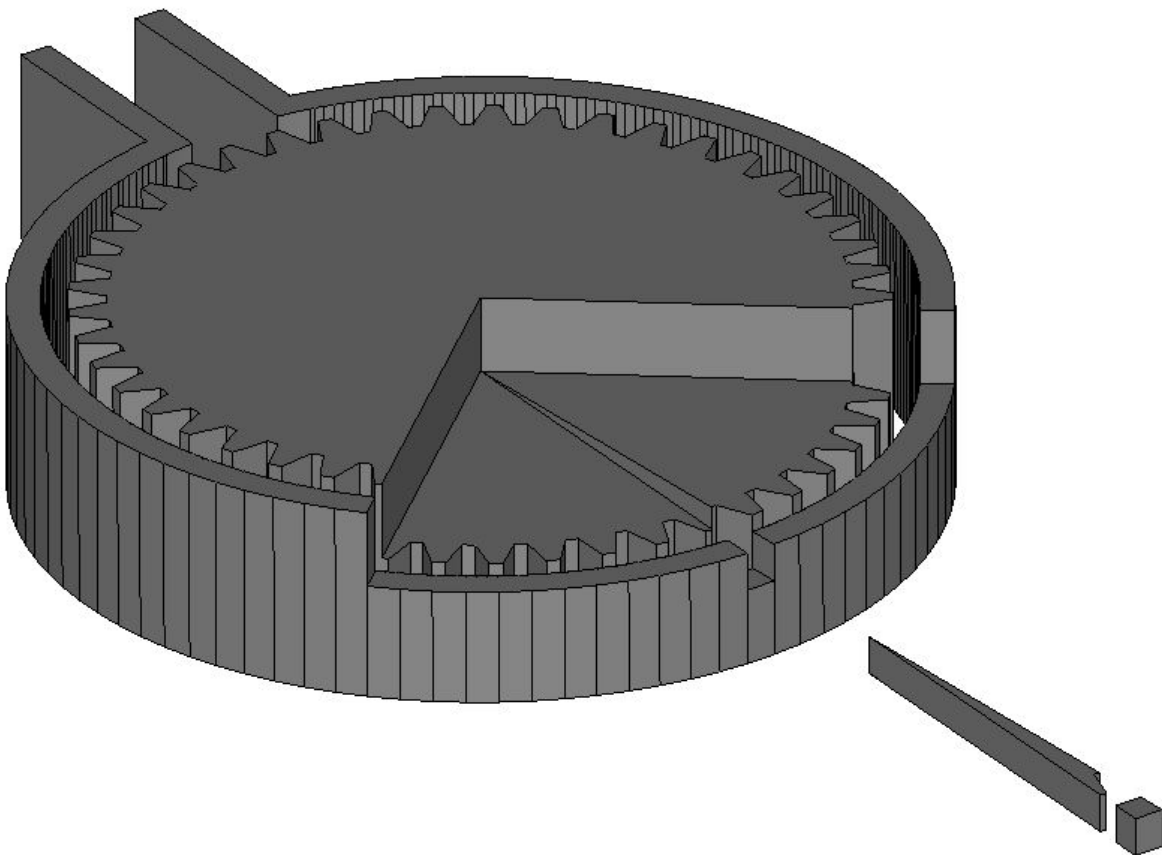


# 3D Spur Gear Hardening

Induction heating is one of the milestones for the metal industry nowadays, and it has applications with no end. One of them is **metal hardening**. In this tutorial we will be looking into steps needed to create a simulation for the hardening of a spur gear through induction heating using CENOS platform.

We will not create the whole gear and conductor, because it is not necessary. We will only create half of the tooth of the gear and use the symmetry boundary conditions to simulate the heating for a full gear.

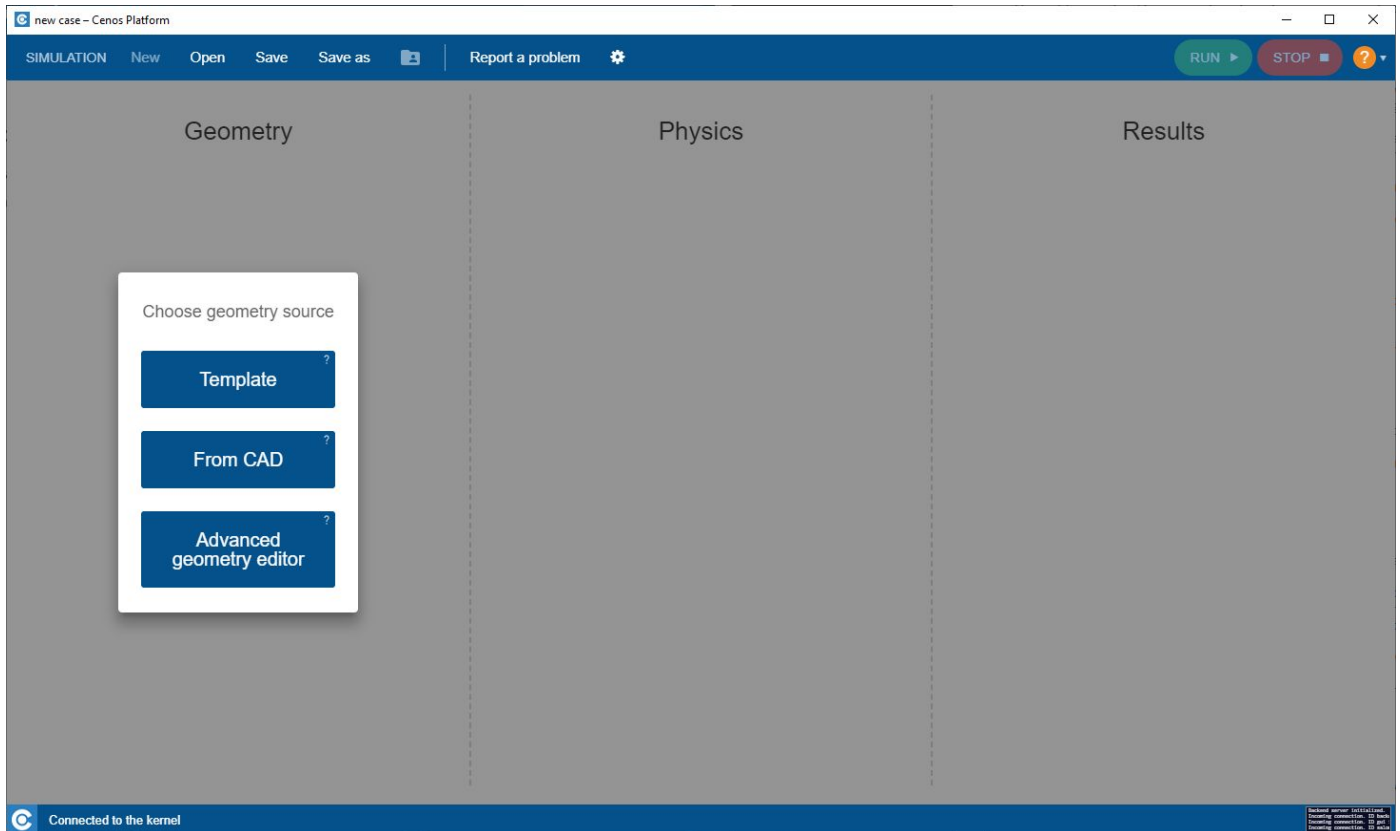
In the next pages a **100 ms** long induction heating example of an **AISI 1045** spur gear at **150 kHz** and **4000 A** with **symmetry boundary conditions** is presented.



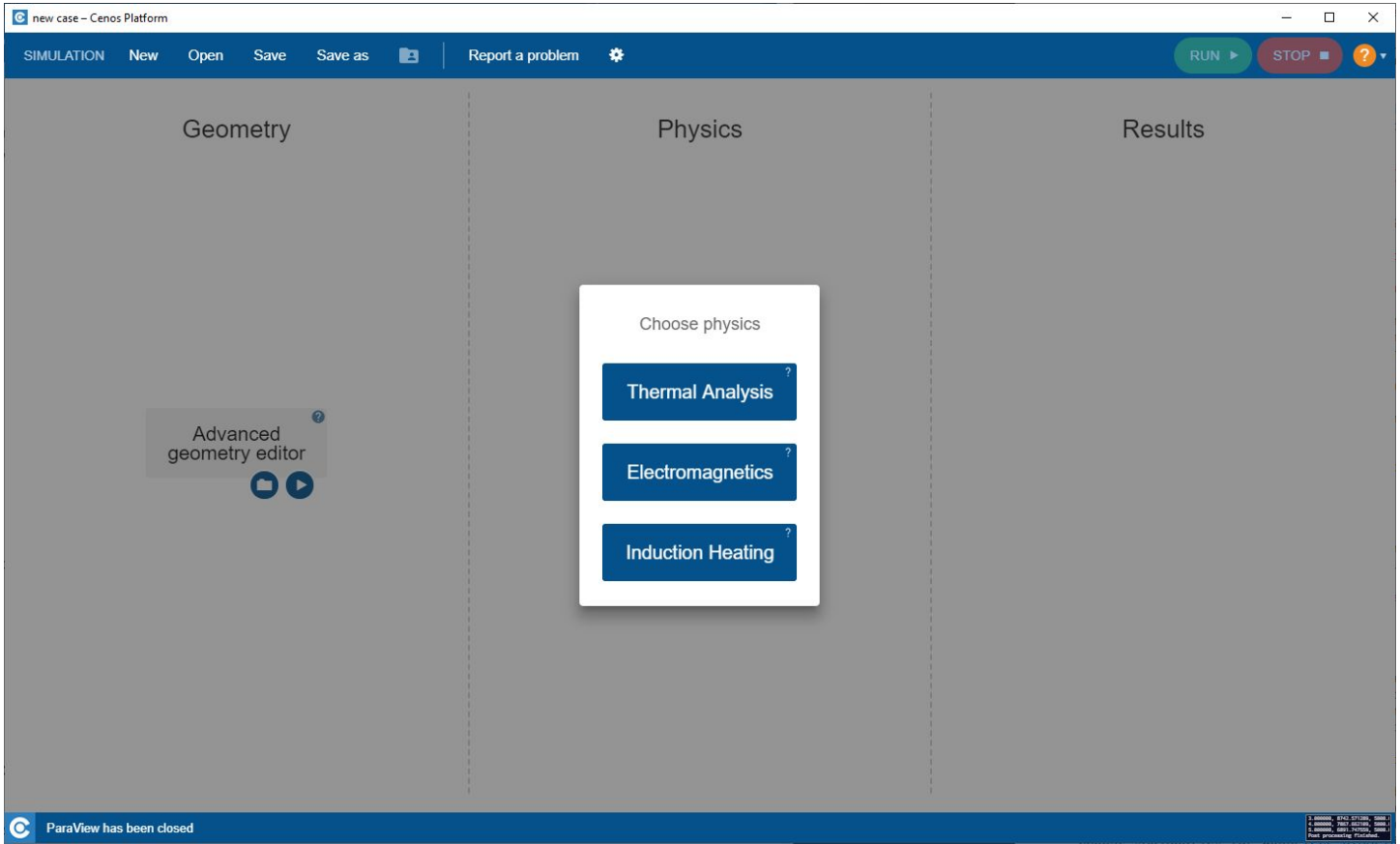
# 1. Open pre-processor

## 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.

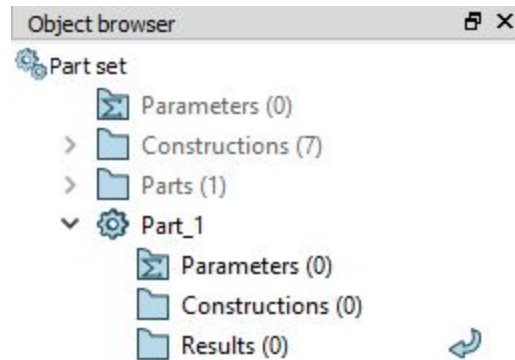


Salome window with already selected **Shaper module** will open.

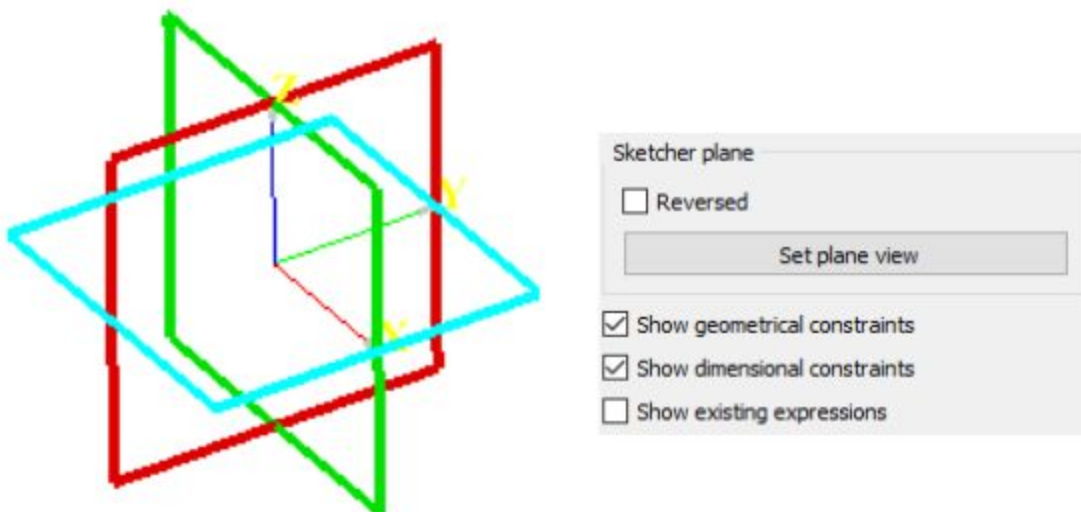
## 2. Create geometry and prepare it for meshing

### 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 (  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!**

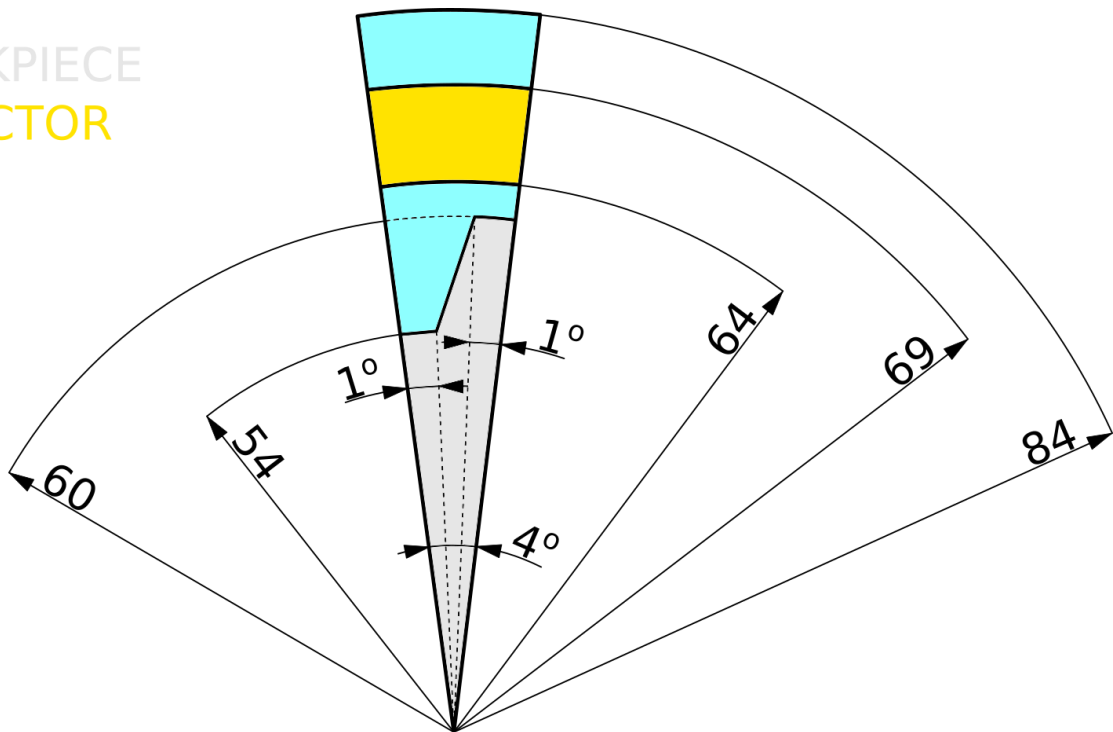
## 2.2 Create the base drawing

As we are creating the geometry from scratch, first we need to **draw the base** drawing which we will extrude to get the full 3D model of the gear.

We will draw the outline of the gear, inductor and air domains. Use the drawing and defining tools to create base from the blueprints provided here:




WORKPIECE  
INDUCTOR  
AIR

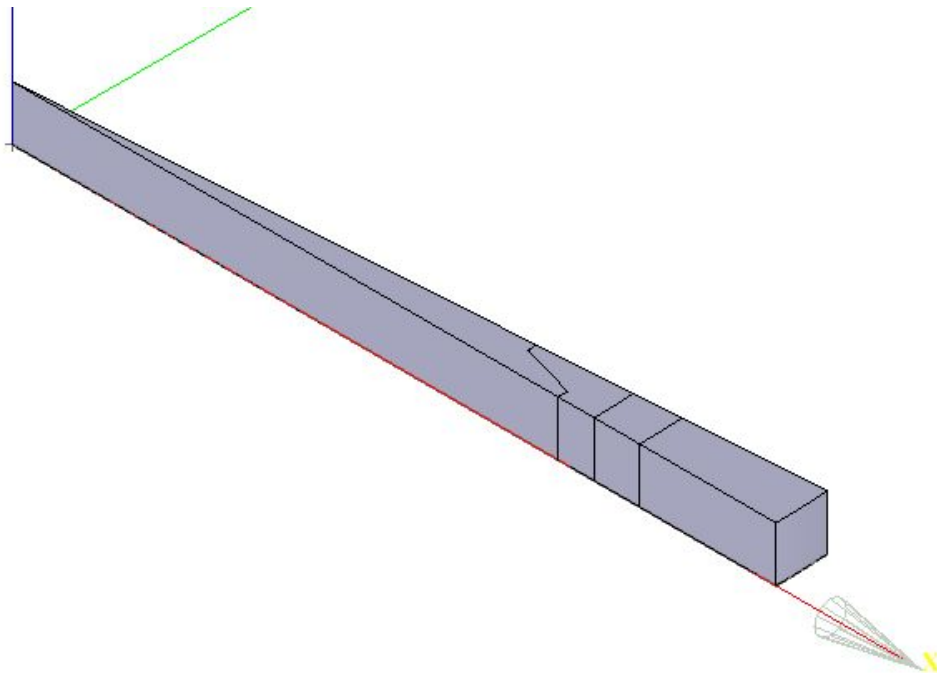
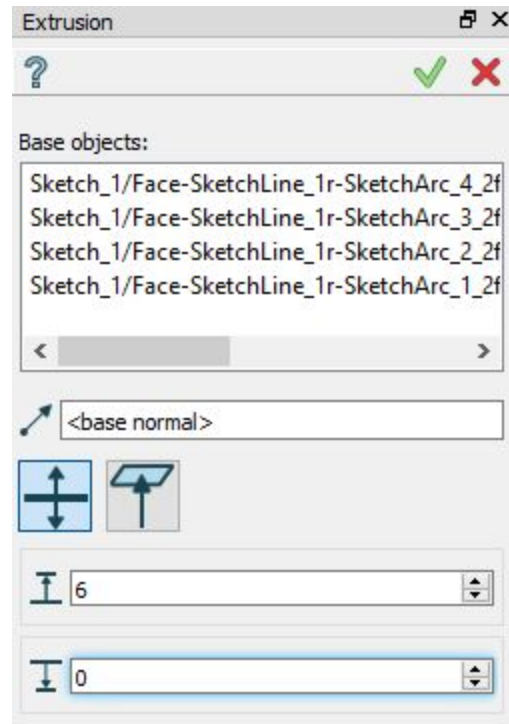


**IMPORTANT:** The drawing presented here is modified to more clearly explain the size parameters of the gear tooth half. In reality the geometry is much more thinner and longer.


Geometry Parameters:	Value:
Nb. of teeth:	45
Module (mm):	2.54
Pitch diameter (mm):	114
Face width (mm):	6
Pressure angle (mm):	22 deg 30'
Coil outer diameter (mm):	138
Airgap (mm):	4

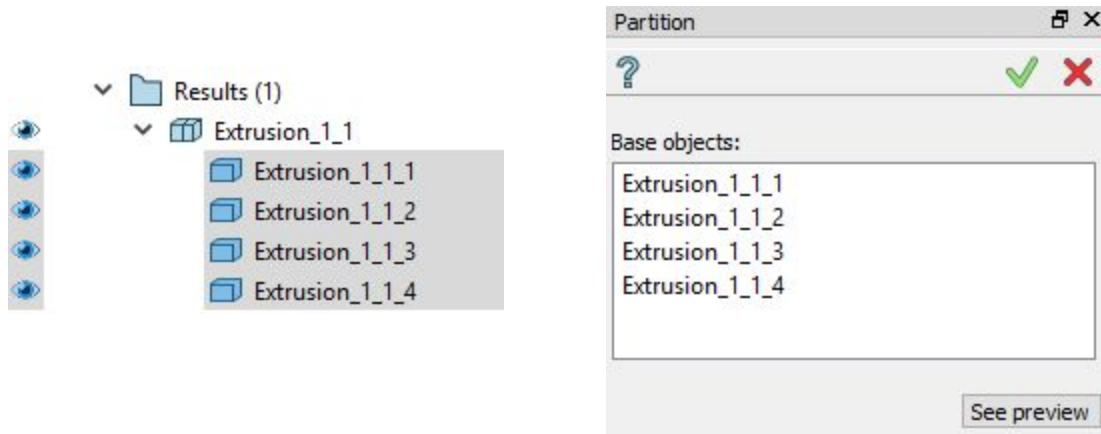
## 2.3 Extrude the base

Click the Extrude () tool, select the base outline parts (hold Shift to select multiple parts) and set the Extrusion height as 6.





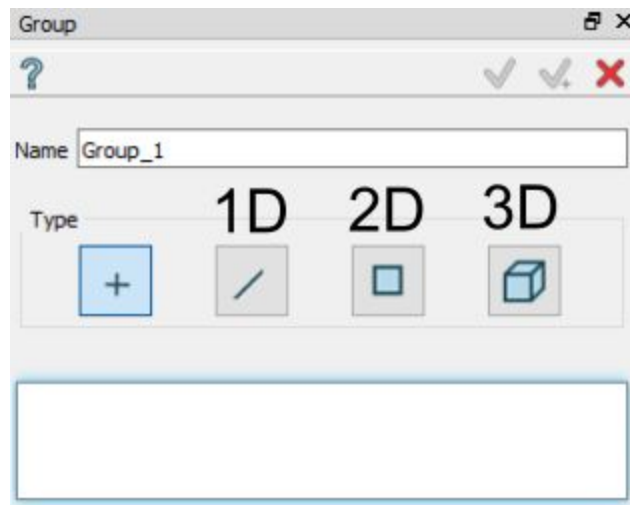
## 2.4 Create Partition and Groups

Click Partition (  ) tool. From Object Browser select all extrusions and join them into one partition.



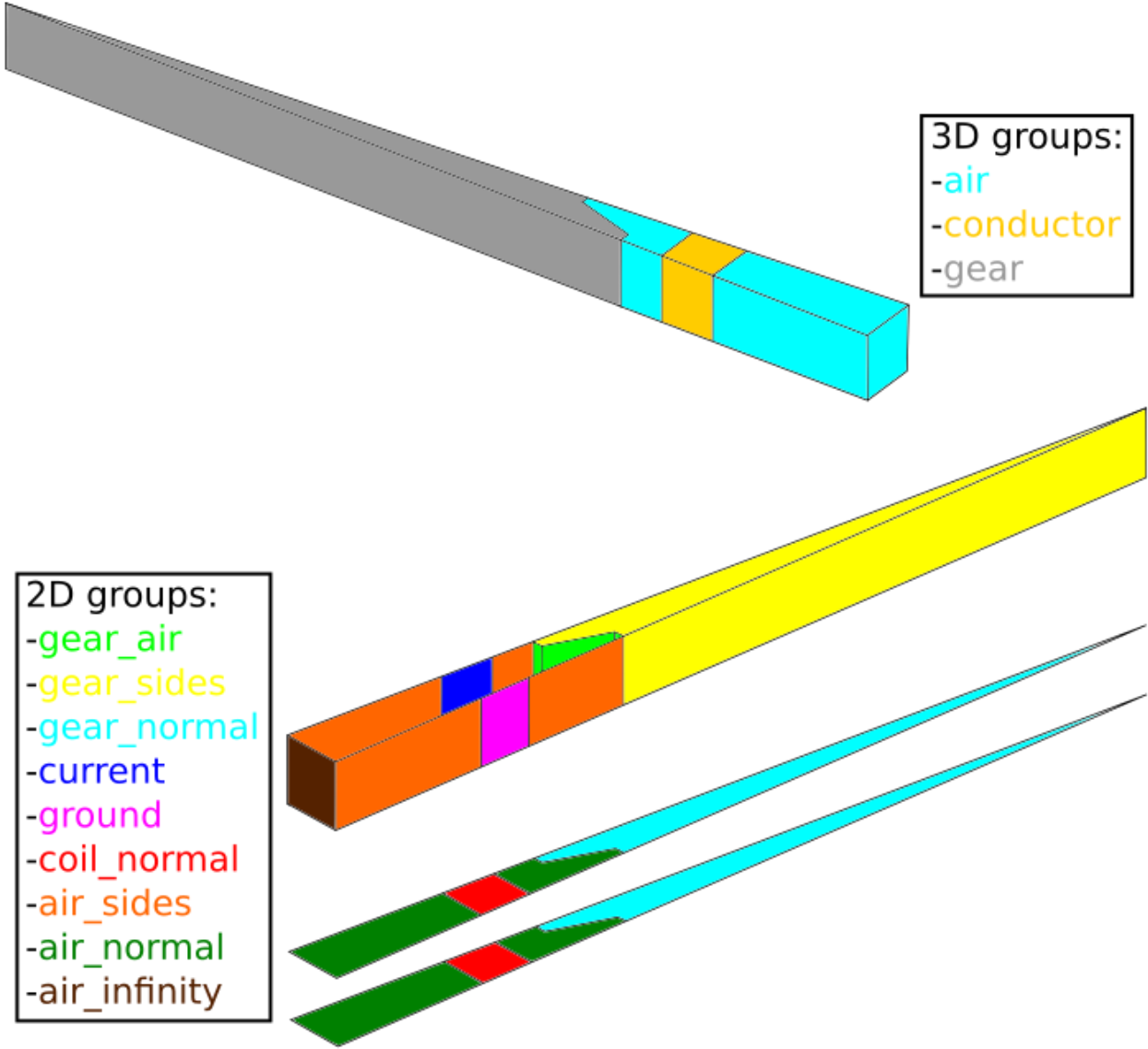
**IMPORTANT:** Partition and Groups are vital for the simulation setup with CENOS, because the mesh creation as well as the physics and boundary condition definitions are based on the groups created in this part.

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 (  ).



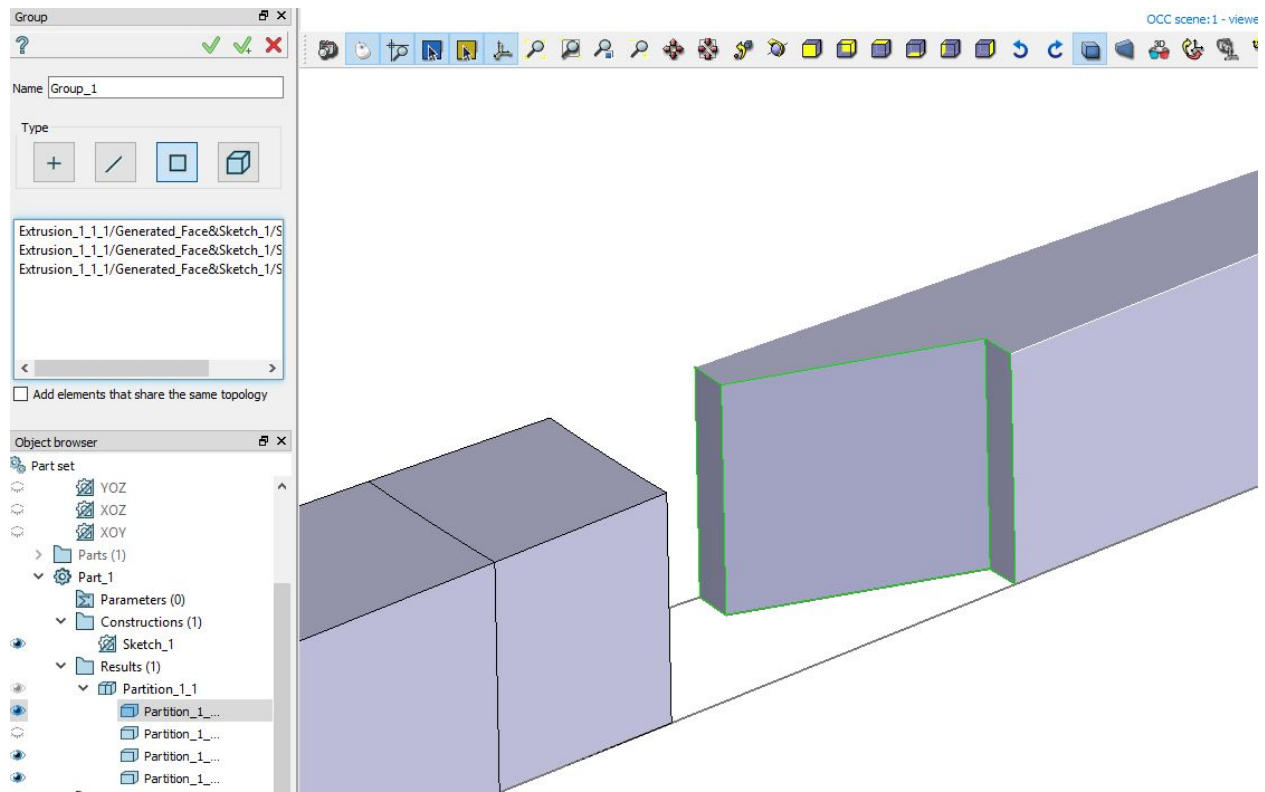
For this tutorial we will create three 3D groups for domains and nine 2D groups for boundary conditions.

Detailed breakdown of these groups is as follows:






**IMPORTANT:** You can hide separate parts of the Partition to gain access to faces within it.



## 2.5 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.

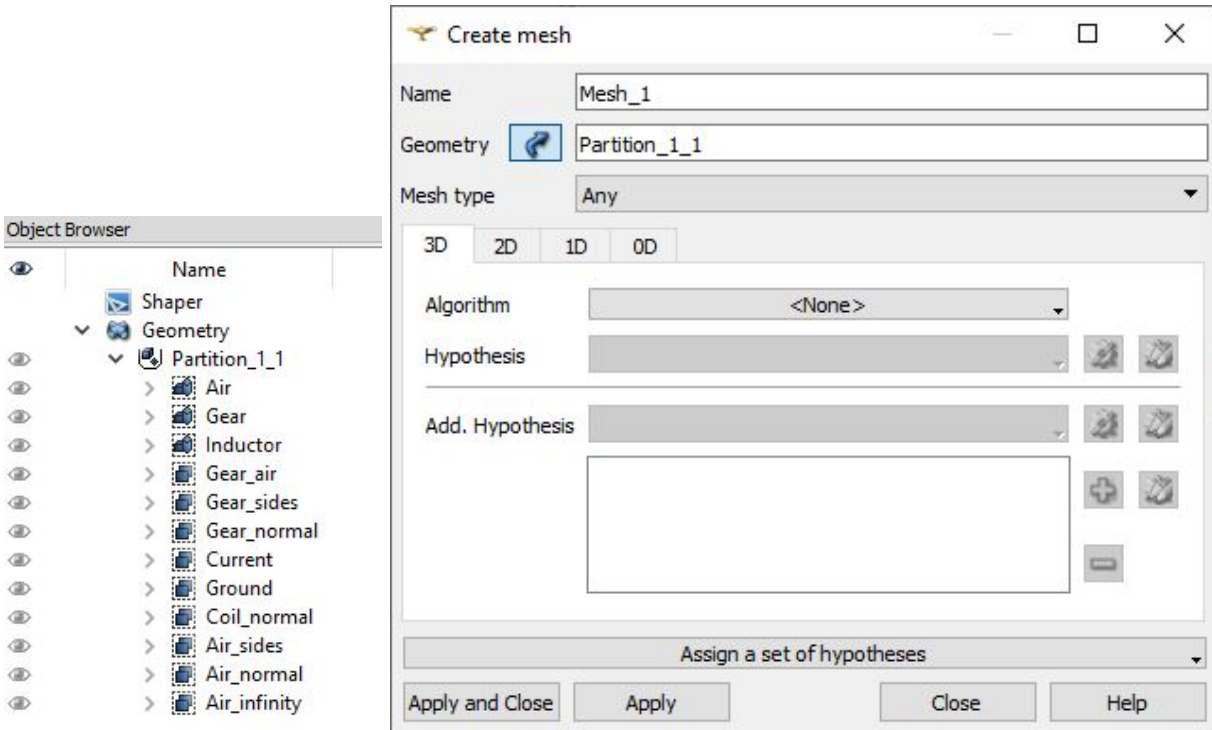
## 3. Create mesh and export it to CENOS

### 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 3D: Automatic Tetrahedralization. In the Hypothesis Construction window enter 3 for Max Length.

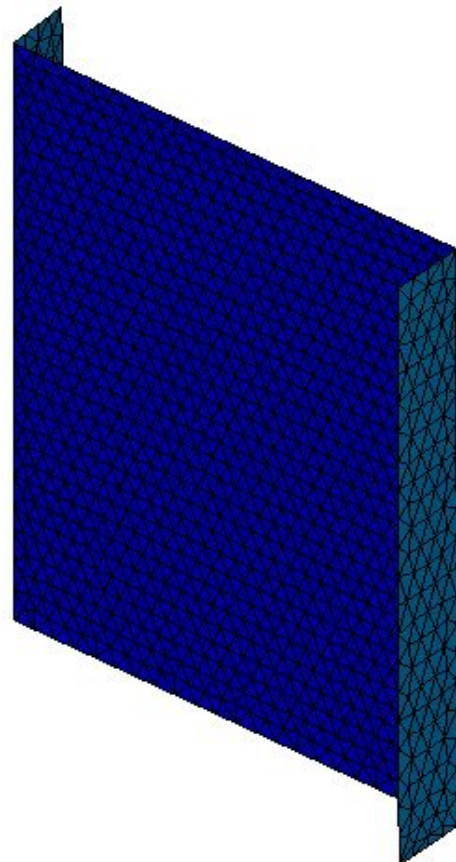
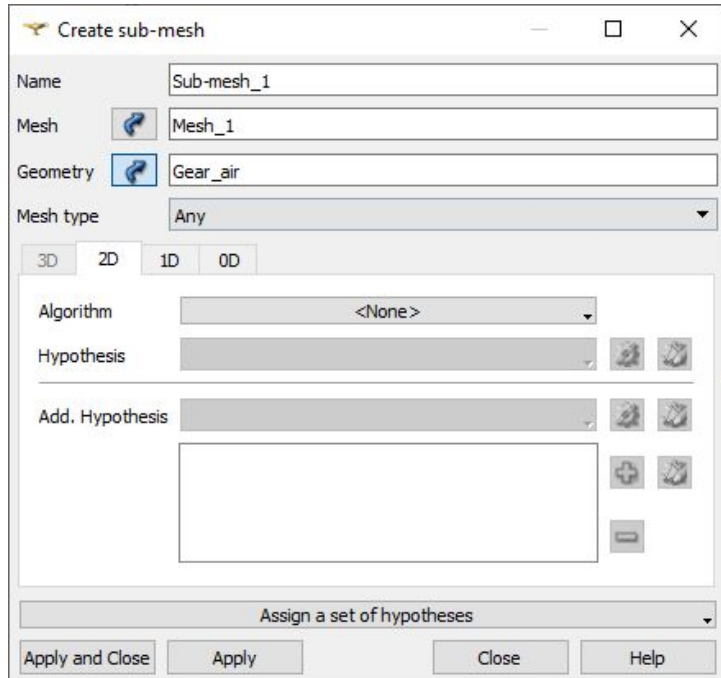
## 3.2 Create a sub-mesh for the gear-air surface

We will create a mesh for the gear using 2 sub-meshes - one for the surface between gear and air domains and other for the gear itself. The sub-mesh for the surface between the gear and air domains is necessary because we want to refine the outer part of the gear to increase the quality and accuracy of the results afterwards.


Right-click on Mesh\_1 and click Create Sub-Mesh or select Create Sub-mesh (  ) from the toolbar.

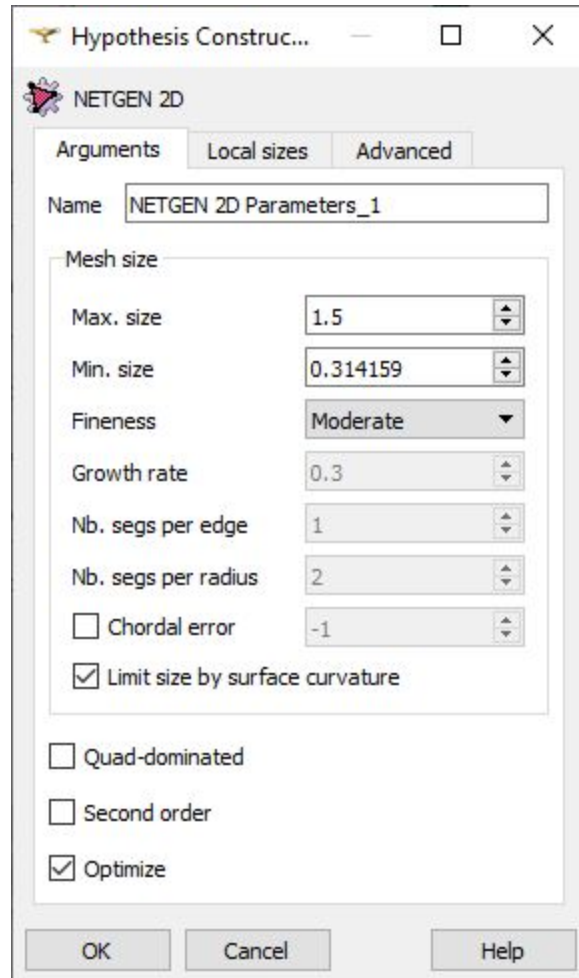
Select the Gear\_air group from the Partition\_1\_1 dropdown menu as Geometry. From the Assign a set of hypothesis dropdown menu select 2D: Automatic Triangulation. In the Hypothesis Construction window enter **0.3** for Max Length.


Right-click on the sub-mesh and select Compute Sub-mesh to calculate and evaluate it.



### 3.3 Create a sub-mesh for the gear sector

Create another sub-mesh and select the Gear group from the *Partition\_1\_1* dropdown menu as Geometry. For 3D algorithm choose *NETGEN 3D* from the *Algorithm* dropdown menu. For 2D algorithm choose *NETGEN 1D-2D* from the *Algorithm* dropdown menu, then click the gear icon (  ) next to *Hypothesis* and select *NETGEN 2D Parameters*. In *Hypothesis Construction* window enter **1.5** for Max. Size and click OK.

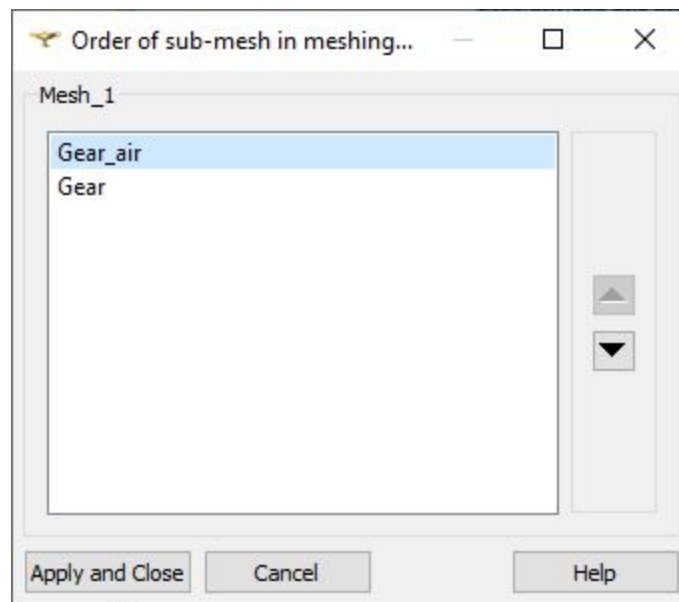
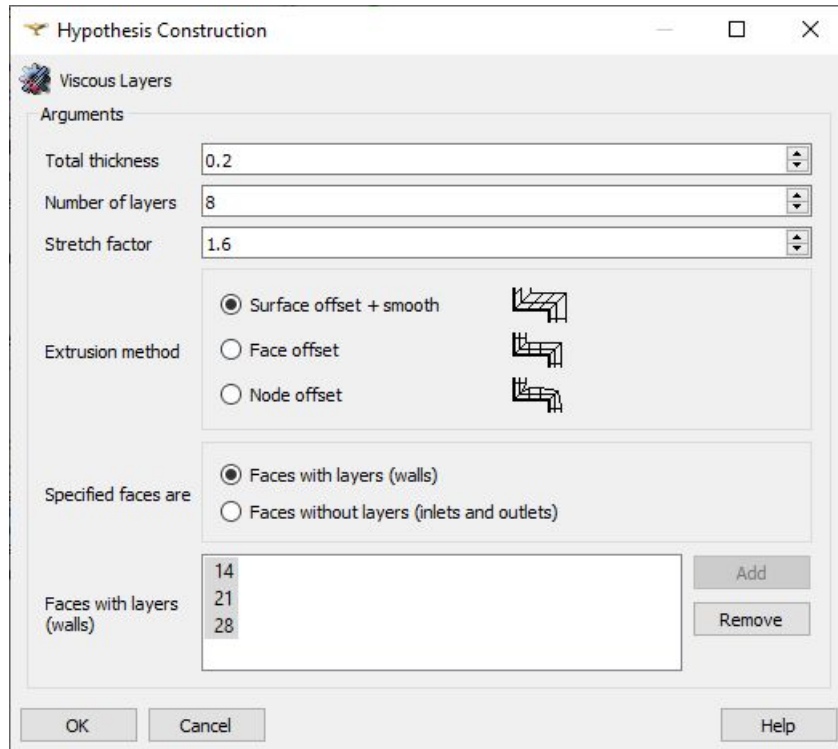


Resolve the skin layer on the surface of the workpiece by creating *Viscous Layers*. Under 3D algorithm section click the gear icon (  ) next to *Add. Hypotheses* and select **Viscous Layers**.

Select the *Gear\_air* group from the *Partition\_1\_1* dropdown menu and click *Add*.


Enter **0.2** for Total thickness, **8** for Number of layers, **1.6** for Stretch factor and check Faces with layers (walls) box.

**IMPORTANT:** You will be asked to set the mesh priority. Set it so that Gear<sub>air</sub> sub-mesh is above Gear sub-mesh (in that way Gear<sub>air</sub> will be calculated first).



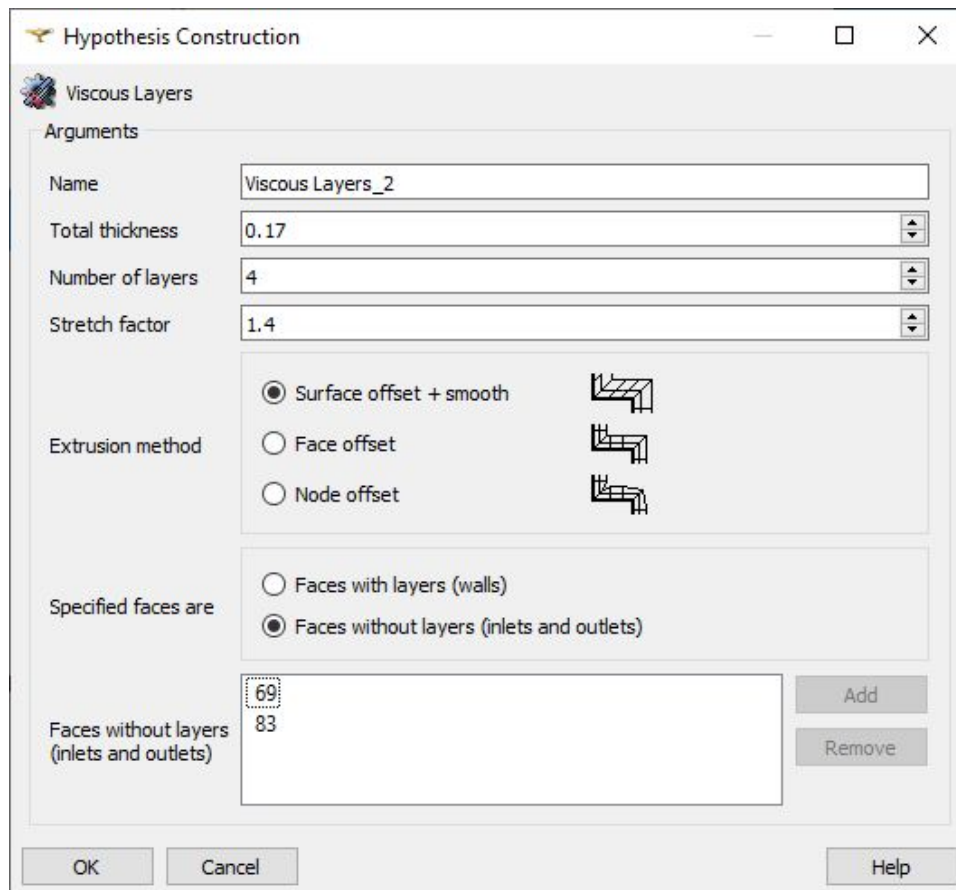
### 3.4 Create a sub-mesh for the inductor

Create a sub-mesh and select the *Inductor* group from the *Partition\_1* dropdown menu as *Geometry*. From the *Assign a set of hypothesis* dropdown menu choose *3D: Automatic Tetrahedralization* and enter **1** 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**.

Select the *Current* and *Ground* groups from the *Partition\_1\_1* dropdown menu and click *Add*.

Enter **0.17** for *Total thickness*, **4** for *Number of layers*, **1.4** for *Stretch factor* and check *Faces without layers (inlets and outlets)* box.

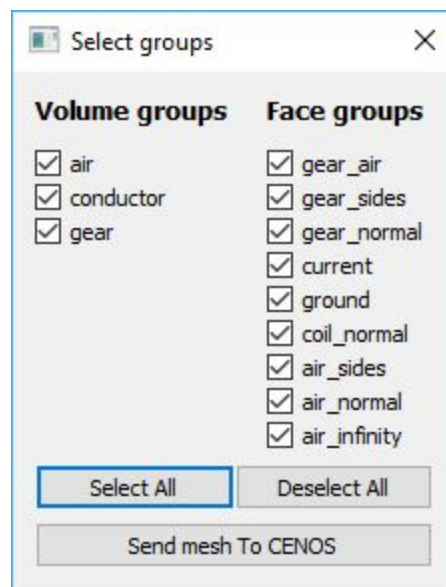


### 3.5 Calculate and export mesh to CENOS

Right-click on Mesh\_1 and click Compute. Evaluate the final mesh and export it to CENOS. 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 the domains or boundary conditions.

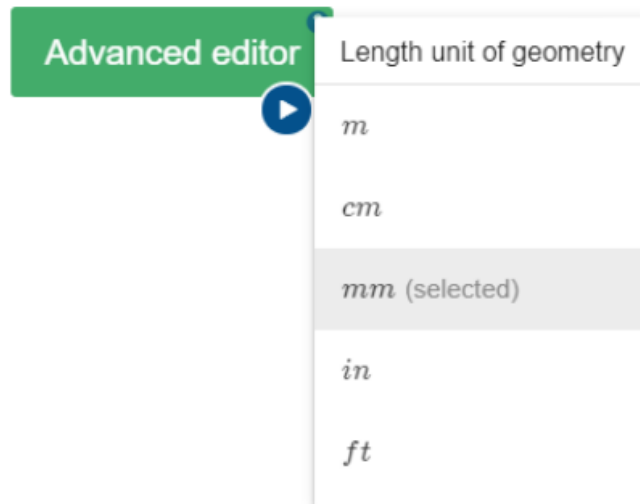


When selected, click *Send mesh to CENOS*.

## 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 **millimeters (mm)**.



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





## 4.2 Simulation control

In the SIMULATION CONTROL define the simulation as a slice with **4 degree** Slice angle. Set Transient with **15 kHz** frequency, **0.1 s** End time. Check the Use adaptive time step box and for Computation algorithm choose **Accurate**.

### Symmetry

$\alpha$   — *Slice angle*

### Time

$f$   *Hz* *Frequency*

$t_1$   *s* *End time*

Use adaptive time step

### Dynamic geometry variables

*No variables are defined in the geometry*

### Computation algorithm

 ?

### 4.3 Gear definition

Select GEAR from *Domain bar*. Leave *Enable Thermal Analysis* and *Enable Electromagnetics* boxes checked under Domain “GEAR”. Choose **Conductive** as the domain type. For Material click SELECT... and choose **Medium carbon steel 1045 B(H), t depend**.

#### Domain “GEAR”

Enable Thermal Analysis

Enable Electromagnetics

Conductive

Domain type

#### Material

Low carbon steel 1020 lineariz... ^ X

$\lambda(T)$ : 26...86 TABLE

$c_p(T)$ : 461...691 TABLE

$\rho$ : 7870

$\sigma(T)$ : 799361...6289308 TABLE

$\mu_r$ : 71

$T_C$ : 768

$\beta$ : 5

SELECT... CREATE NEW...

**IMPORTANT:** In this tutorial for every domain we will use [symmetry boundary conditions](#) such as *Flux normal* and *Flux parallel* to define the direction of magnetic field lines relative to symmetry planes.

Under THERMAL ANALYSIS for the boundary conditions choose **Combined** for GEAR\_AIR – check the **Convection** and **Radiation** boxes and enter **1.5** for Heat Transfer Coefficient and **0.8** for Emissivity. Choose **Adiabatic** for the GEAR\_SIDES and GEAR\_NORMAL groups.

## THERMAL ANALYSIS

### Domain properties

Motion

### Initial conditions

$T$   °C *Temperature*

### Boundary conditions

GEAR\_AIR

Convection

$T_{amb}$   °C *Ambient temperature*

$h$    $\frac{W}{m^2K}$  *Heat Transfer Coefficient*

Radiation

$T_{amb}$   °C *Ambient temperature*

$\epsilon$   — *Emissivity*

Heat Flux

Heat Flow

GEAR\_SIDES

GEAR\_NORMAL

Under ELECTROMAGNETICS choose **Interface** for GEAR\_AIR, **Flux parallel** for GEAR\_SIDES and **Flux normal** for GEAR\_NORMAL.

## ELECTROMAGNETICS

### Boundary conditions

GEAR\_AIR

GEAR\_SIDES

GEAR\_NORMAL

## 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**.

### Domain "INDUCTOR"

Enable Thermal Analysis

Enable Electromagnetics

Current source

*Domain type*

### Material

Copper Constant properties



SELECT...

CREATE NEW...

Under **ELECTROMAGNETICS** choose **Current (Amplitude)** for **CURRENT**, **Ground** for **GROUND** and **Flux normal** for **COIL\_NORMAL** and enter **4000 A** as *Current (Amplitude)*.

## ELECTROMAGNETICS

### Boundary conditions

COIL\_NORMAL

Flux normal

CURRENT

Current (Amplitude)

$I$

4000

$A$

*Current (Amplitude)*

GROUND

Ground

## 4.5 Air definition

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

### Domain "AIR"

Enable Thermal Analysis

Enable Electromagnetics

Non-conductive

*Domain type*

### Material

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

Under ELECTROMAGNETICS choose **Interface** for GEAR\_AIR, **Flux parallel** for AIR\_SIDES, **Flux normal** for AIR\_NORMAL and **Infinity** for AIR\_INFINITY.

## ELECTROMAGNETICS

### Boundary conditions

GEAR\_AIR

Interface

AIR\_SIDES

Flux parallel

AIR\_NORMAL

Flux normal

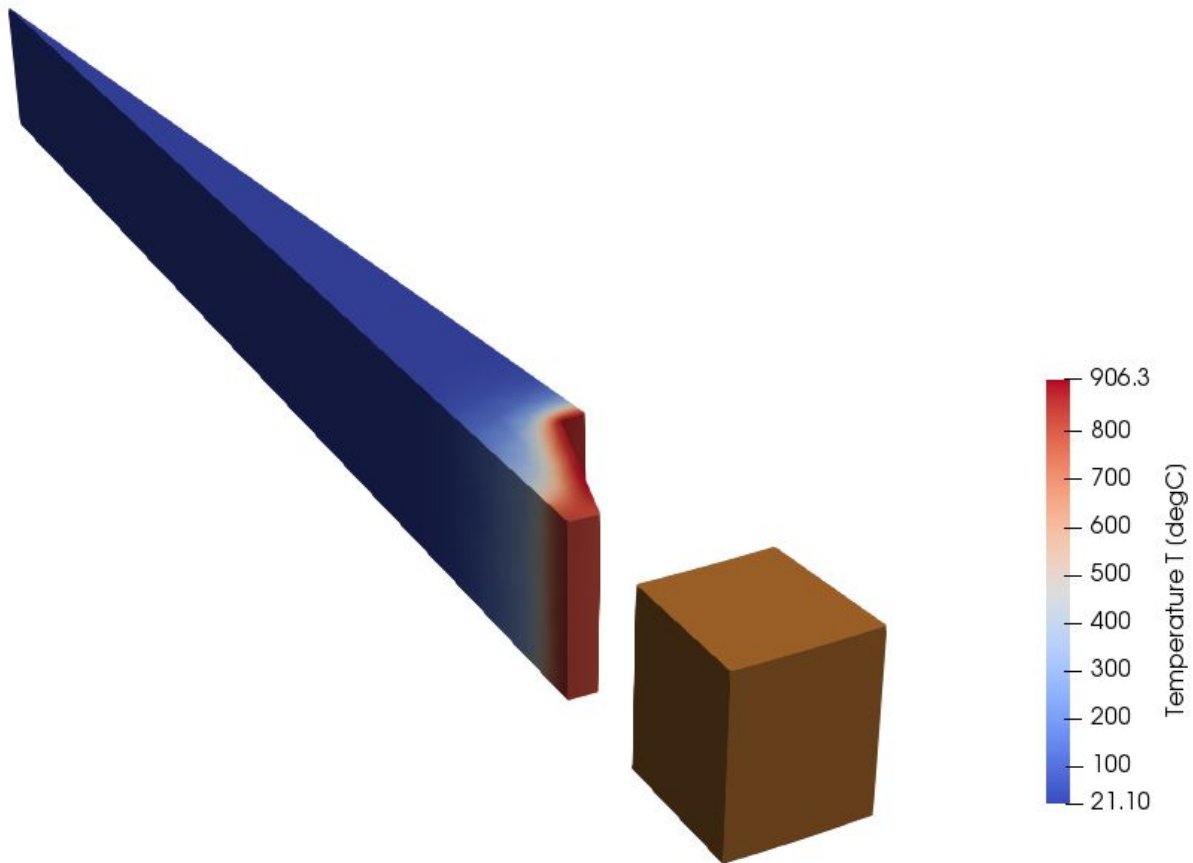
AIR\_INFINITY

Infinity

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

## 5. Evaluate results

When CENOS finishes calculation, ParaView window with the pre-set temperature result state will open automatically and you will be able to see the temperature field distribution in the gear sector 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 spur gear sector induction hardening simulation tutorial. For any recommendations or questions contact our support.