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.

C new case - Cenos Platform		– 🗆 X
SIMULATION New Open Save Save as 📭	Report a problem 🛛 🛱	
Geometry	Physics	Results
Choose geometry source Template Trom CAD		
Advanced geometry editor		
Connected to the kernel		

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 (^(D)) tool. A new part will be added to Object browser.



Click Import (→) and seled	t the step file	provided	earlier.
----------------	---------------	-----------------	----------	----------

To see the geometry, click Fit All ($\stackrel{\frown}{\sim}$).

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 (\bigcirc).

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

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



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

Now we need to extrude the rectangle we created. Click Extrusion (²) tool and select the rectangle. In the extrusion parameters enter **0.2m** in the reverse direction:

<u>T</u> o	÷
J 0.2	÷

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.

		Partition		₽×
	Sketch 1	?	~	×
2	Results (2)	Base objects:		
۲	> 💦 singleshot_geometry_1	Extrusion_1_1		
 Partition Sketch_1 Results (2) Singleshot_geometry_1 Extrusion_1_1 singleshot_geometry 				
	singleshot_geometry			
			10	
			See pre	view

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 (B) 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 (M).

Group			₽×
?			V V. 🗙
Name Group_1			
Type	1D	2D	3D
			A
T	/	-	

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:



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 ($\stackrel{56}{>}$). 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 ($\stackrel{\textcircled{\baselineskip}{\baselineskip}}$).

Object	t Brow	wser		🔷 💎 Create mesh				×
2	ľ	>	Name Shaper Geometry Partition_1_1 > M workpiece	Name Me Geometry Pa Mesh type Ar 3D 2D 1D	esh_1 rtition_1_1 IV OD			
			 inductor air yoke workpiece_air terminal1 terminal2 air_infinity 	Algorithm Hypothesis Add. Hypothesis		lone>		
					Assign a set of	fhypotheses		
				Apply and Close	Apply	Close	н	elp

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 (

Y Create sub-	mesh			l		×
Name	Sub-mesh_1					
Mesh 🧳	Mesh_1					
Geometry						
Mesh type	Any					-
3D 2D	1D 0D					
Algorithm		<none></none>		•		
Hypothesis				-	22	12
Add. Hypothesi	s			*	4	2
					÷	12
				5	-	
				-		
	Assign	n a set of hypo	theses			Ŧ
Apply and Close	Apply		Close		Hel	p

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.

✓ Hypothes	is Construction	
Max Leng Arguments	th	
Length Use preestir	0.0035	\$
ОК	Cancel	Help

Resolve the skin layer on the surface of the workpiece by creating Viscous Layers. Under 3D

algorithm section click the gear icon (20) 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.

Hypothesis Const	ruction		×
Viscous Layers			
Arguments			
Total thickness	0.0004		
Number of layers	5		\$
Stretch factor	1.5		•
Extrusion method	Image: Surface offset + smooth Image: Surface offset Image: Surface offset Image: Surface offset Image: Surface offset		
Specified faces are	 Faces with layers (walls) Faces without layers (inlets and outlets) 		
Eacon without laware		Add	
(inlets and outlets)		Remo	ve
OK Car	cel	 -	Help

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**.

🜱 Edit sub-me	sh			×
lame 🥐	workpiece			
1esh	Mesh_1			
eometry	workpiece			
lesh type	Any			•
3D 2D 1	ID OD			
Algorithm	NETGE	N 2D	-	
Hypothesis	Length Fro	n Edges_1	- 👰	198
Add. Hypothesis	s	ne>	- 3	Length From Edges Max. Element Area NETGEN 2D Parameters
	Assign a set of h	ypotheses		÷
Apply and Close	Apply	Close	Hel	p

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.

		?
penGL dipping	New	 Delete
and groups		
)		
Y-Z		•
).5 🗘
to X):)° 🗘
to X):)° 🗘
ow preview	🗌 Auto Appl	у
	penGL dipping and groups) []] Y-Z to X): to X): ow preview	penGL dipping



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 (2000)) 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.

 Hypothesis Const 	ruction			>
Viscous Layers				
Arguments				
Total thickness	0.0009			-
Number of layers	4			\$
Stretch factor	1.5			\$
Extrusion method	Surface offset + smooth Face offset Node offset			
Specified faces are	 Faces with layers (walls) Faces without layers (inlets and outlets) 			
	805		Add	
Faces without layers (inlets and outlets)	755	[Remov	/e
OK Can	cel		ŀ	Heln

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

🜱 Edit sub-me	sh	12 <u>_</u> 21		×	
Name	inductor				
Mesh	Mesh_1				
Geometry	inductor				
Mesh <mark>ty</mark> pe	Any			-	
3D 2D :	1D 0D				
Algorithm	NETGE	N 2D	-		
Hypothesis	Length From	n Edges_2	•	198	
Add. Hypothesis	s <nor< td=""><td>ne></td><td>• 😻</td><td>Length From Edges Max. Element Area NETGEN 2D Parame</td><td>ters</td></nor<>	ne>	• 😻	Length From Edges Max. Element Area NETGEN 2D Parame	ters
	Assign a set of h	ypotheses			
Apply and Close	Apply	Close	He	lp	

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.

madecor	
yoke	

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 \rightarrow Plugins \rightarrow 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.

Volume groups	Face groups	
workpiece	workpiece_air	
inductor	🗹 terminal 1	
🗹 air	terminal2	
✓ yoke	✓ air_infinity	
Select All	Deselect All	

When selected, click Send mesh to CENOS.

4. Define physics and boundary conditions

4.1 Set the units and enter the physics setup



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.

Domain "WORKPIECE"

Enable Thermal	Analysis		
Enable Electron	nagnetics Conductive	Э	✓ Domain type
Material			
Low carbon s	teel 1020 lineariz A	× SELE	ECT CREATE NEW
$\begin{array}{c} \lambda \left(T \right):48.951.9 \\ c_{p} \left(T \right):486599 \\ \tau \\ \rho:7870 \\ \sigma \left(T \right):342465762 \\ \mu_{r}:71 \\ T_{C}:768 \\ \beta:5 \end{array}$	TABLE ABLE 189308 TABLE		
		1	

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.

THERMAL ANALYSIS

Domain properties

1	I	Notion				
ŝ	U_X	0		$\frac{m}{s}$	Velocity X component	
	U_Y	0		$\frac{m}{s}$	Velocity Y component	
j.	U_Z	0		$\frac{m}{s}$	Velocity Z component	
	Ω_Z	20		$\frac{rad}{s}$	Angular velocity around Z axis	
Initi	al c	onditions				
	T	22	$^{\circ}C$	Temperature		
Βοι	unda wor	ary conditions				
	Cor	nbined -	Co	onvection		
			T_{amb}	22	$^{\circ}C$	Ambient temperature
			h	10	$\frac{W}{m^2K}$	Heat Transfer Coefficient
			🔽 Ra	adiation		
			T_{amb}	22	$^{\circ}C$	Ambient temperature
			ε	0.8	-	Emissivity
			🗌 He	eat Flux		
			🗌 He	eat Flow		

Under ELECTROMAGNETICS choose Interface for WORKPIECE_AIR.

ELECTROMAGNETICS				
Boundary condit	ions			
Interface	•			

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 so	urce	*	Domain type
Material				
Copper Constant propertie	es >	×	SELECT.	. CREATE NEW

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

ELECTROMAGNETICS					
Boundary conditions					
Current (Amplitude)	•	Ι	10000	A	Current (Amplitude)
TERMINAL2					
Ground	*				

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**.

Domain "YOKE"			
Enable Thermal Analysis			
Enable Electromagnetics	Non-conductive	•	Domain type
Material			
Flux concentrator FLUX	trol a 💙 🗙	SELECT.	CREATE NEW

4.6 Air definition

For air domain 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	> ×	SELECT.	CREATE NEW

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

ELECTROMAGNETICS			
Boundary conditions	5		
Infinity	•		
WORKPIECE_AIR			
Interface	•		

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.