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 \rightarrow Load script....

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

ኛ Load pyt	thon script					?		×
.ook in:	D:\2D	Scanning	-	00	0	G	::	
User Progra SAMP	PLES	Name] shaft.py			Siz	e		
	<						Onen	
lie name:	snart.py						open	-
iles of type:	Python Fil	es (*.py)			-	C	Cancel	



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 ^{Sketch_1} and click Edit. By using Circle (^O) 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 (Sketch) icon. Select the XY plane and click Set plane view



Create a rectangle (1250mm x 350mm) around the workpiece by clicking the Rectangle

(¹¹) icon. Add a Coincidence (¹) constraint between the symmetry axis edge (left side of the air box) and the origin point.



2.3 Create faces



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



2.4 Translate inductor windings

Click Translation (⁵) 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**.

Translation		8×
?	V	×
Main objects		
Face_4_1		
race_4_2		
DX 0		\$
DY scanning=0		
DZ 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

Partition	₽×	
?	 ✓ × 	
Base objects:		
Face_1_1 Face_3_1		
Translation_1_1		

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 (\checkmark).

		ć	7 ×
		 ✓ ✓ 	×
Jp_1			
1D	2D	3D	
1			
/Modified_E	dge&Sket	ch_1/Sketc	hLi
			>
	IP_1 1D	IP_1 1D 2D // D	IP_1 1D 2D 3D ID 2D 3D Modified_Edge&Sketch_1/Sketc

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 ($\stackrel{\$}{>}$). 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	(🔏).
		· /·

🜱 Create mesh					×
Name	Mesh_1				
Geometry	Partition_1				
Mesh type	Any				•
3D 2D 1	D 0D				
Algorithm		<none></none>		•	
Hypothesis				- 2	12
Add. Hypothesis				- 2	12
				÷	12
				-	
	Assign	a set of hypoth	eses		
Apply and Close	Apply		Close	н	elp

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

🜱 Create sub	-mesh				×
Name	Sub-mesh_1				
Mesh 🦿	Mesh_1				
Geometry 🧳					
Mesh type	Any				-
3D 2D	1D 0D				
Algorithm	14 19	<none></none>		•	
Hypothesis				. 2	12
Add. Hypothe	sis			- 4	12
				4	2
					-
	Assign	a set of hypot	theses		
Apply and Close	Apply		Close	He	ln

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.

Y Hypothe	sis Construction	
Max Leng	th	
Arguments		
Length	3	•
Use preesti	mated length	
OK	Canad	Usla

🜱 Edit sub-mes	sh		12_1		×
Name	workpiece				
Mesh	Mesh_1				
Geometry	workpiece				
Mesh <mark>ty</mark> pe	Any				•
3D 2D 1	D <mark>OD</mark>				
Algorithm		NETGEN 2D		•	
Hypothesis		<default></default>		- 🏄	12
Add. Hypothesis		<none></none>		-	12
				\$	12
				0	
	Assig	in a set of hypoth	eses		
Apply and Close	Apply		Close	He	lp

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.

Viscous Layers		
Arguments		
Name	Viscous Layers 2D_1	
Total thickness	0.4	÷
Number of layers	5	÷
Stretch factor	1.25	 \$
Specified edges are	 Edges with layers (walls) Edges without layers (inlets and outlets) 	
	13 23 45 17 33 49 47 21 7	Add
	20 55 52 44 0 27 12 25	

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.

Viscous Layers 2D_2	
0.7	\$
4	-
1.2	\$
 Edges with layers (walls) Edges without layers (inlets and outlets) 	
Add	
Remo	ve
Remo	Ve

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 \rightarrow Plugins \rightarrow 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.

Select groups	×
Face groups	Line groups
🗹 air	air_infinity
workpiece	air_symmetry
✓ c1	wp_symmetry
✓ c2	✓ wp_air
Auto_group_for_Sub-m	esh_2
Select All	Deselect All
Send mes	h 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.

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

		10.0	-
	- 1	m	വ
- 1			

Tr	ansient -			
f	10000		Hz	Frequency
t_1	30	8	End time	
	Use adaptive time step			
δt	1	s	Calculation tin	ne step

Check the **Dynamic geometry**

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

Dynamic geometry variables



Computation algorithm



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**.

Domain "WORKPIECE"

Enable Thermal AnalysisEnable Electromagnetics	Conductive	• D)omain type
Vaterial			
Low carbon steel 1020	B(H), t° ^ X	SELECT	CREATE NEW
$\begin{array}{l} \lambda \left(T \right): 48.951.9 \text{TABLE} \\ c_{p} \left(T \right): 486599 \text{TABLE} \\ \rho: 7870 \\ \sigma \left(T \right): 34246576289308 \text{TABLE} \\ B \left(H \right): 010 \text{TABLE} \\ T_{C}: 768 \\ \beta: 5 \end{array}$	LE		
	•		

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.

THERMAL ANALYSIS

Domain propertie	es
------------------	----

Motion

Initial conditions

	19-11-19-19-19-19-19-19-19-19-19-19-19-1		-2		
	T 22	$^{\circ}C$	Temperature		
Βοι	undary conditio	ons			
	Adiabatic	*			
	WP_AIR				
	Combined	· 🗸 (Convection		
		T_{amb}	22	$^{\circ}C$	Ambient temperature
		h	10	$\frac{W}{m^2K}$	Heat Transfer Coefficient
		F	Radiation		
		T_{amb}	22	$^{\circ}C$	Ambient temperature
		£	0.8	-	Emissivity
		□ H	Heat Flux		
		□ H	Heat Flow		

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

ROMAGNET	TICS	
undary c	condition	S
WP_SYMME	ETRY	
Symmetry	y axis	•
WP_AIR		
Interface		•
Interface		Ψ.

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).

Domain "C2"			
Enable Thermal Analysis			
Enable Electromagnetics	Current source	- De	omain type
Material			
Type to search		SELECT	CREATE NEW
Recent: Copper (Constant propert	ties) 🔲 Low carbon stee	l 1020 (B(H), t° depe	end.)
Iron (B(H), temperature dependent) Flux concentrator (F	FLUXTROL A)	
ELECTROMAGNETICS			
Domain properties			
<i>I</i> 10000	A	Current (Amp	olitude)

4.5 Air definition

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

Domain "AIR"

Enable Thermal Analysis				
Enable Electromagnetics	Non-conduct	ive	•	Domain type
Material				
Air	>	×	SELECT	CREATE NEW

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

ELECTROMAGNETICS			
Boundary condition	S		
Infinity	•		
AIR_SYMMETRY			
Symmetry axis	•		
WP_AIR			
Interface	•		

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.