



## TUTORIAL: how to import a modal basis coming from a third-party 3D FEA mechanical software in Manatee e-NVH software v2.2.5

### VERSIONING

Rev.	Date	Description
F	12/10/22	Update with Abaqus
G	31/05/23	Update on .RST compression option with Ansys
H	24/07/23	Update on node selection/ creation of a modal basis in Manatee format
I	27/09/23	Update on Ansys part

## SUMMARY

<b>1</b>	<b>Introduction.....</b>	<b>3</b>
<b>2</b>	<b>Mechanical FEA model adaptation (independent from FEA software) .....</b>	<b>3</b>
2.1	General recommendations.....	3
2.2	Mesh recommendations.....	3
2.3	Modal basis normalization.....	3
2.4	Manatee conventions.....	4
<b>3</b>	<b>Node selection (independent from FEA software) .....</b>	<b>7</b>
3.1	Introduction .....	7
3.2	Stator nodes for magnetic force application (optional for external rotor).....	7
3.3	Rotor nodes for magnetic force application (optional for internal rotor).....	8
3.4	Sets of nodes for acoustic noise radiation.....	8
3.5	Sets of nodes for vibration calculations .....	9
3.6	Sets of nodes for mode visualization (optional).....	10
<b>4</b>	<b>Mechanical FEA output files.....</b>	<b>11</b>
4.1	Introduction .....	11
4.2	Altair Optistruct / Hypermesh.....	11
4.3	Ansys Workbench.....	15
4.4	3DS Abaqus.....	19
<b>5</b>	<b>Import of modal basis in Manatee format.....</b>	<b>21</b>
<b>6</b>	<b>Use of Manatee modal basis format during e-NVH simulation .....</b>	<b>22</b>



## 2.4 Manatee conventions

### 2.4.1 Introduction

To ease the import of modal basis in Manatee, the FEA model should be adjusted in a specific way to be aligned with Manatee conventions. It is strongly recommended to proceed to these adjustments inside your own FEA mechanical software. If it is not the case, additional adjustments may be carried during the modal basis import GUI in Manatee as shown in 2.4.4.

### 2.4.2 Orientation

The middle of the stator lamination (active part where magnetic forces are applied) is centered in ( $x=0, y=0, z=0$ ). The stator is oriented along Z axis (from Drive End to Non Drive End). In case of several stators, the one the closest to Drive End side should be used.

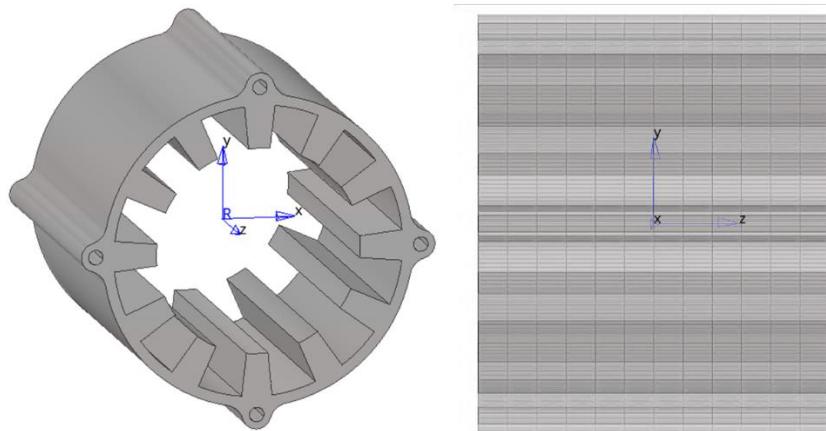


Figure 2 : Stator orientation

### 2.4.3 Angle of first tooth center

Angle between first stator tooth tip center and x-axis is  $\alpha=0^\circ$ . So that x-axis is aligned with the middle of one stator tooth (refer to figure below).

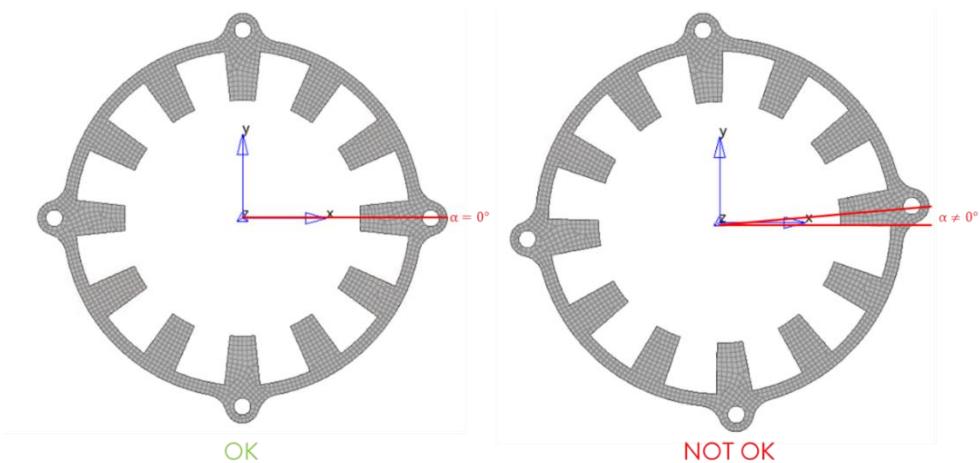
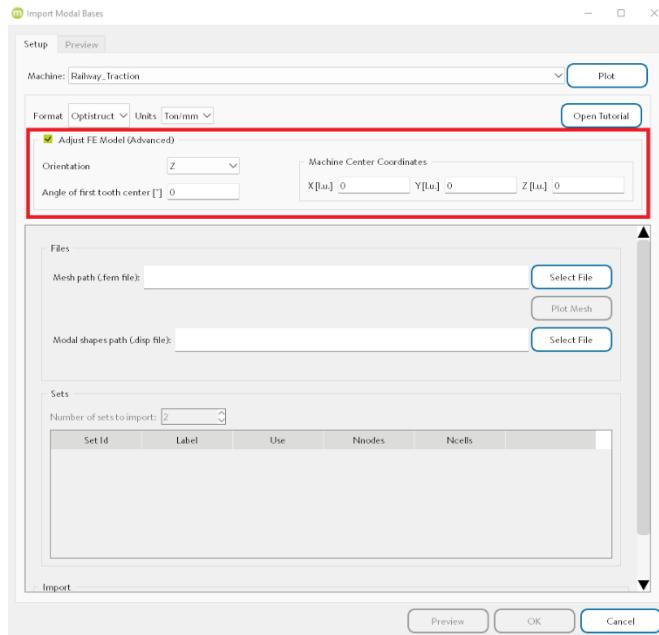


Figure 3 : Tooth orientation

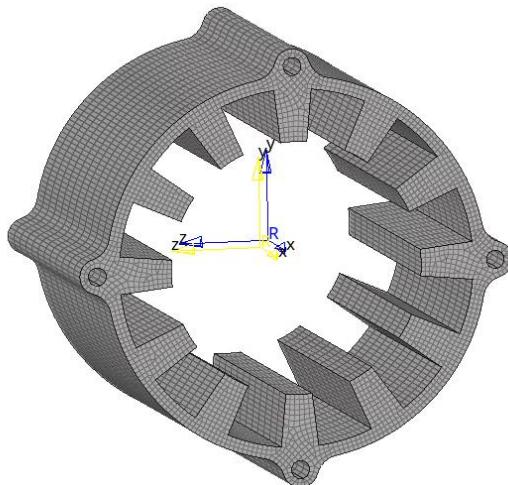
#### 2.4.4 Convention adjustments in Manatee GUI

If the FEA mechanical model does not follow Manatee conventions, it is possible to directly adjust the model in Manatee by clicking on the button “Adjust FE Model” during the modal basis import (refer to figure below).



*Figure 4 : Adjust FE model in manatee*

In the example of Figure 5, the yellow coordinate system is the center of the stator and the blue one is the global coordinate system. The stator is oriented along X. The center of the stator is (5,-2,4) in local units (l.u.). The first tooth is at 5°.



*Figure 5 : Stator orientation example*

The advanced parameters in Manatee should then be defined as shown below:

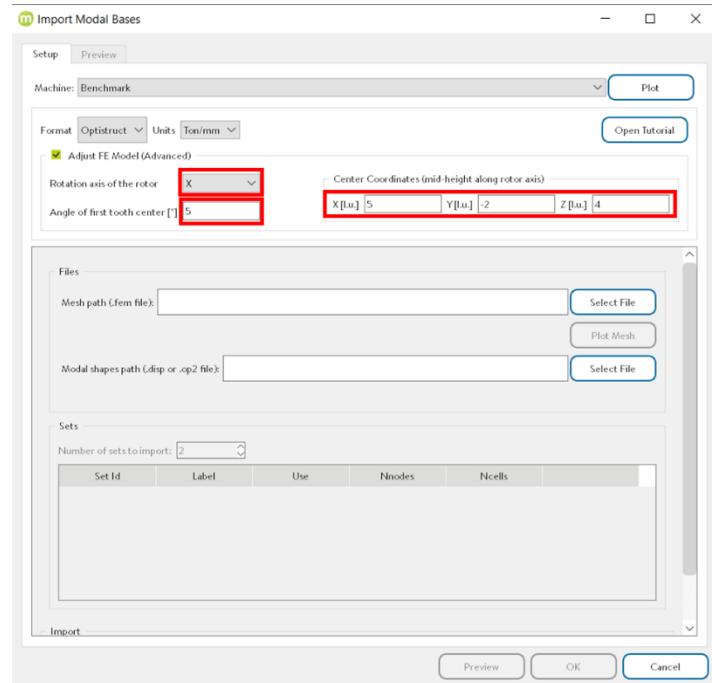


Figure 6 : Adjust FE model in manatee example

### 3 Node selection (independent from FEA software)

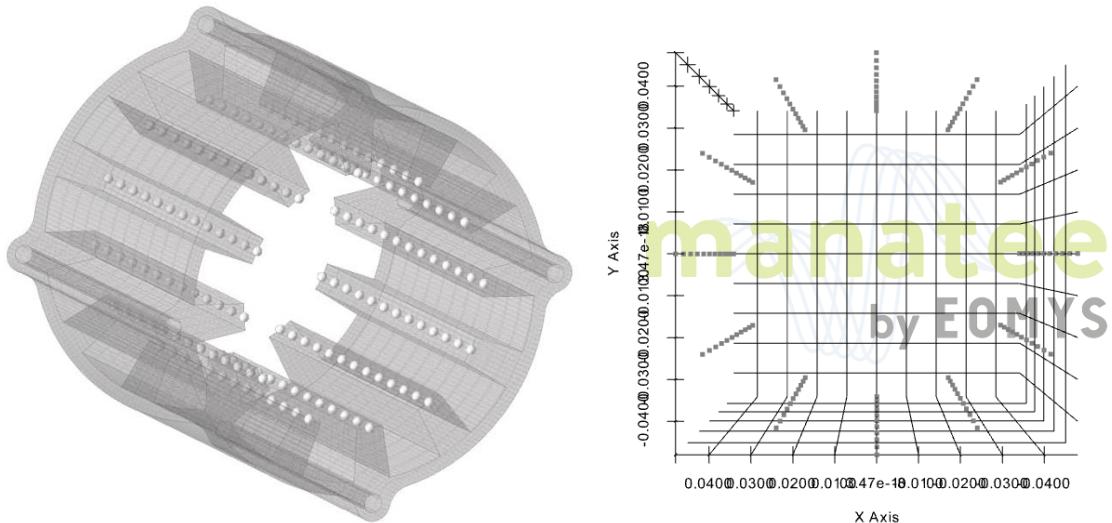
#### 3.1 Introduction

Several sets of nodes need to be selected before running the modal analysis and importing the files in Manatee. They can be used for

- magnetic force application (e.g. stator & rotor nodes at the interface between air & iron)
- acoustic radiation calculation (e.g. enveloped nodes)
- vibration / blocked force calculation (e.g. interface nodes)
- mode visualization (e.g. stator bore radius & yoke to better visualize stator modes)

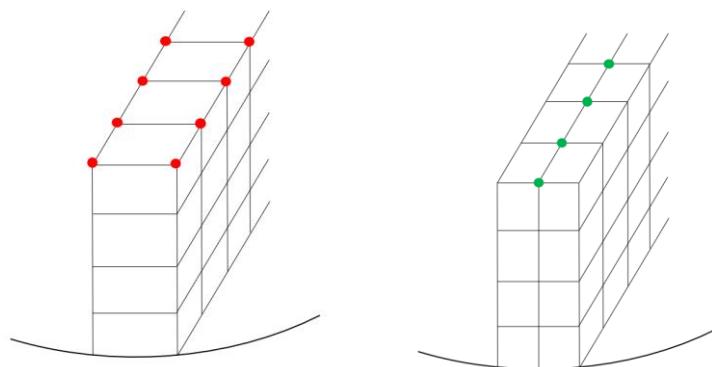
#### 3.2 Stator nodes for magnetic force application (optional for external rotor)

A specific set should be created selecting the middle nodes of each stator tooth's tip (in through stator length to apply forces on stator side).



*Figure 7 : Stator force nodes*

If the mesh is not suited to select “middle nodes”, then the mesh should be refined to create nodes in the middle of the tooth tip as shown below.



*Figure 8 : Teeth mesh*

The nodes must be evenly distributed along the machine axis.

### 3.3 Rotor nodes for magnetic force application (optional for internal rotor)

It is also possible to create a specific set of nodes on rotor component to apply forces on rotor side. Nodes from rotor lamination (and not magnets) should be selected. At least  $2p$  lines where  $p$  is the number of pole pair should be selected. In case of external rotor machine,  $4p$  lines should be selected.

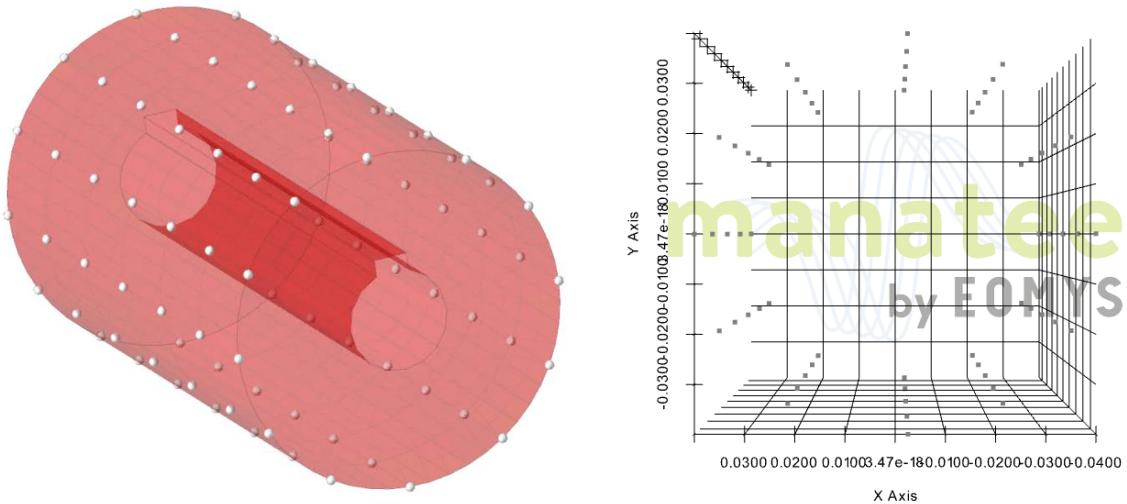


Figure 9 : Rotor force nodes

In case of skewed rotor, straight lines (parallel to rotor axis) are still required.  
The nodes must be evenly distributed along the machine axis.

### 3.4 Sets of nodes for acoustic noise radiation

Specific sets should contain the nodes of the external envelope on which vibration and acoustic radiation is calculated (this set of nodes need to create elements). Several sets can be defined to perform panel contribution analysis (split of sound power level radiated by different surfaces).

Isolated nodes should not be selected but nodes should be connected with other nodes inside the vibration set.

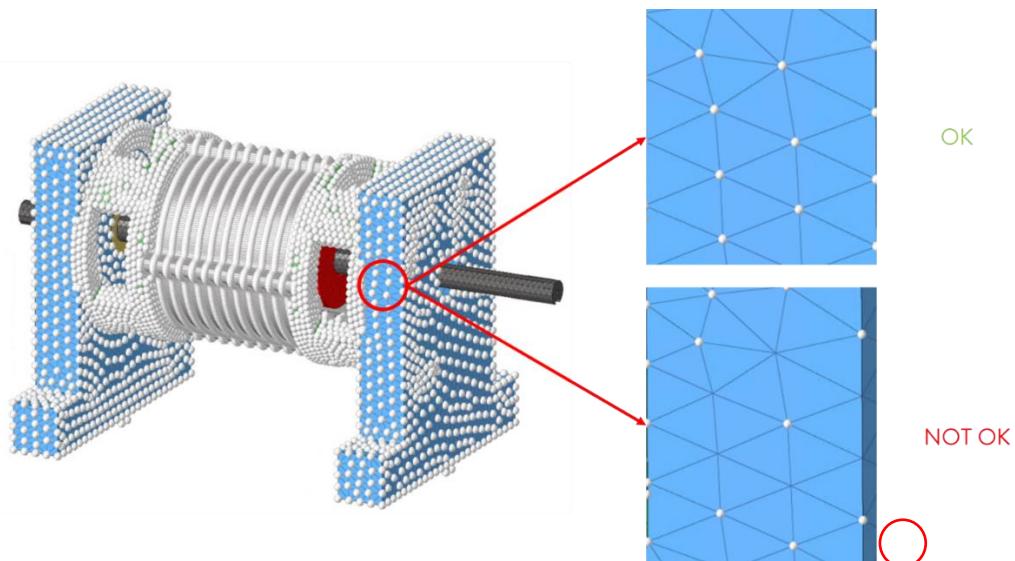
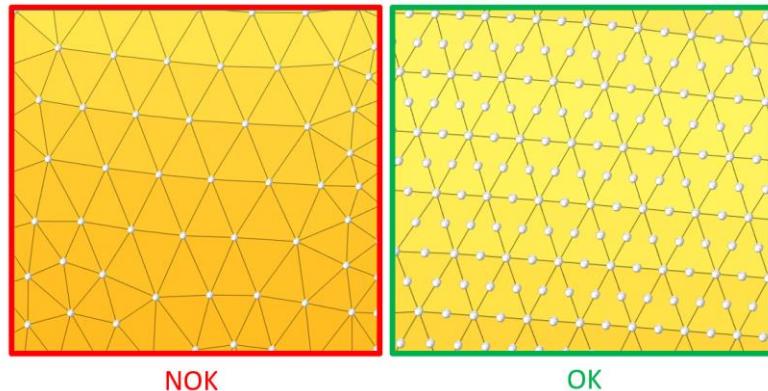


Figure 10 : Sets of nodes for acoustic radiation

Remark: Second order elements

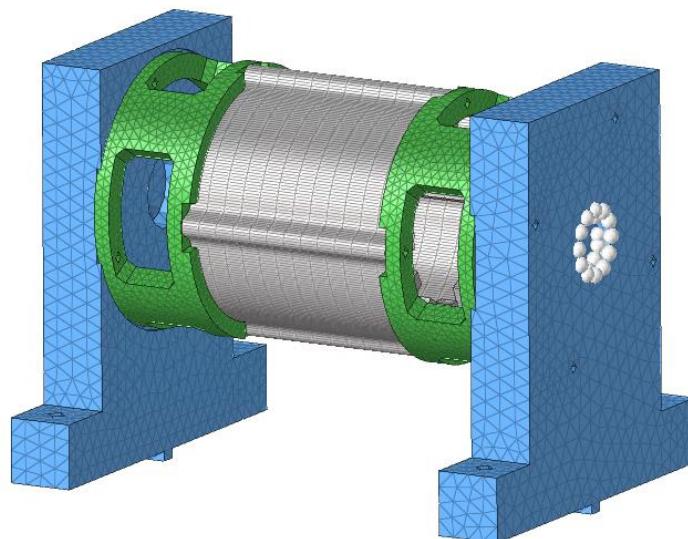
If second order elements are used, all the nodes composing the elements should be selected, not only the nodes on the vertices (see below).



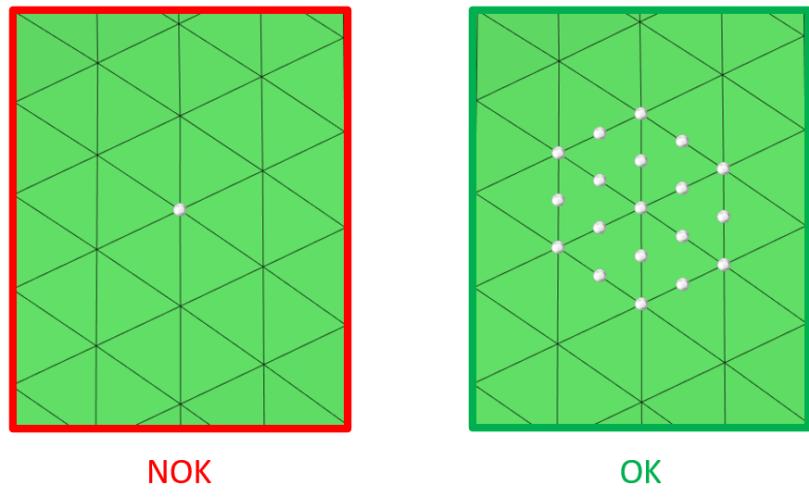
*Figure 11 : Second order nodes elements*

### 3.5 Sets of nodes for vibration calculations

Additional sets of nodes can be created to extract vibration on specific areas (e.g. an accelerometer location, a fixation point or a subassembly), called “sensor nodes”. The nodes defining at least one element should be selected.



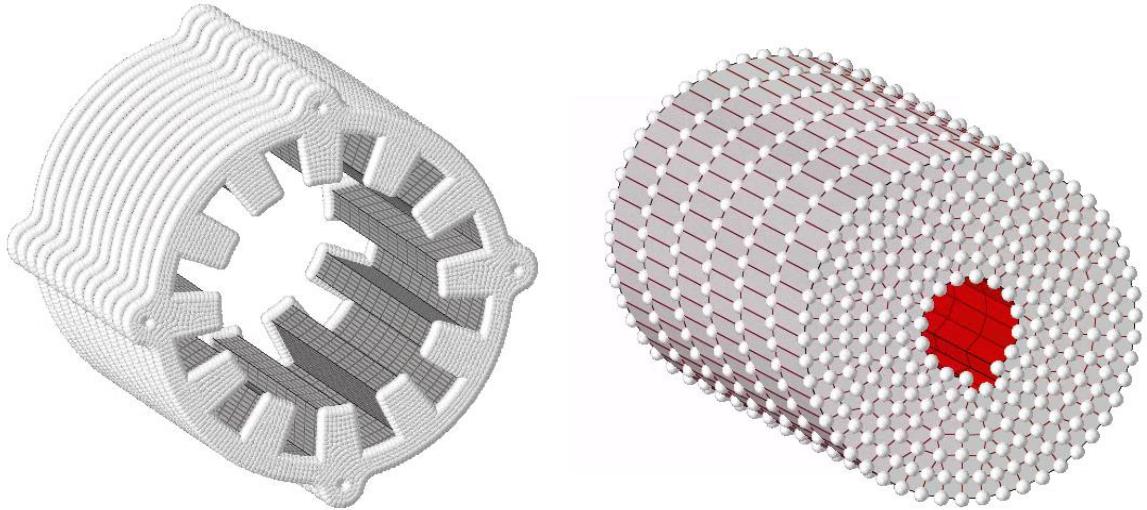
*Figure 12 : Sensor nodes*



*Figure 13 : Isolated nodes*

### 3.6 Sets of nodes for mode visualization (optional)

Additional sets can be created selecting all the surface nodes of the stator (or/and of the rotor) to more precisely visualize stator modes (or/and rotor modes). Figure 14 shows examples for stator and rotor visualization.



*Figure 14 : Visualization sets, left: stator visualization, right: rotor visualization*

## 4 Mechanical FEA output files

### 4.1 Introduction

This part presents how to define the sets and which modal basis output files should be used for modal basis import in Manatee.

### 4.2 Altair Optistruct / Hypermesh

#### 4.2.1 Set selection

The node sets are created following §3 as described below:

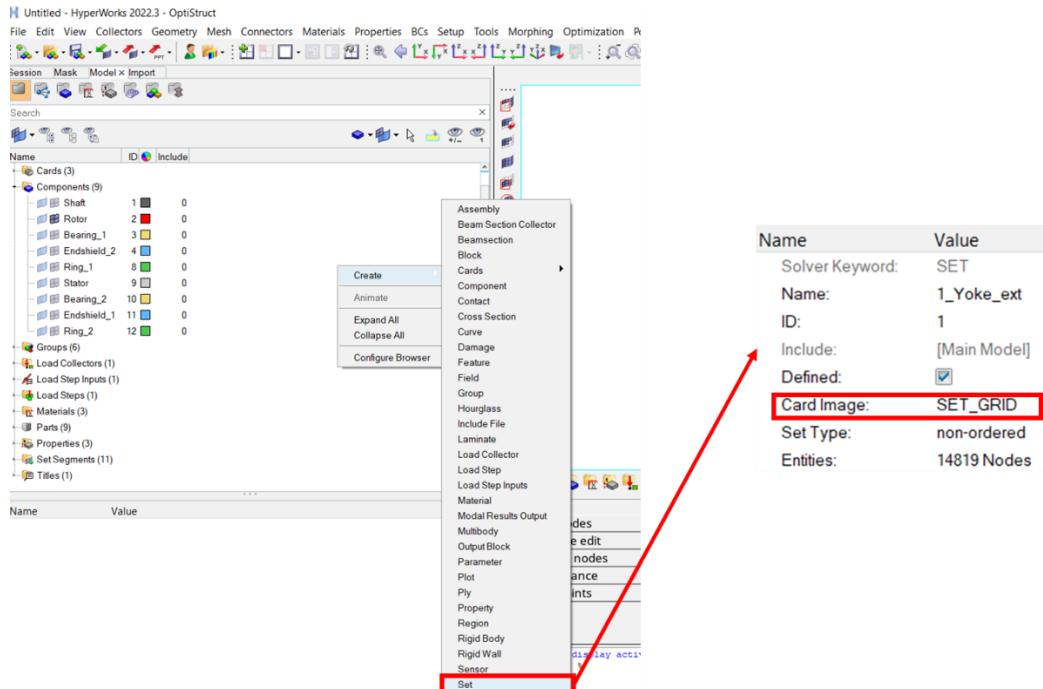
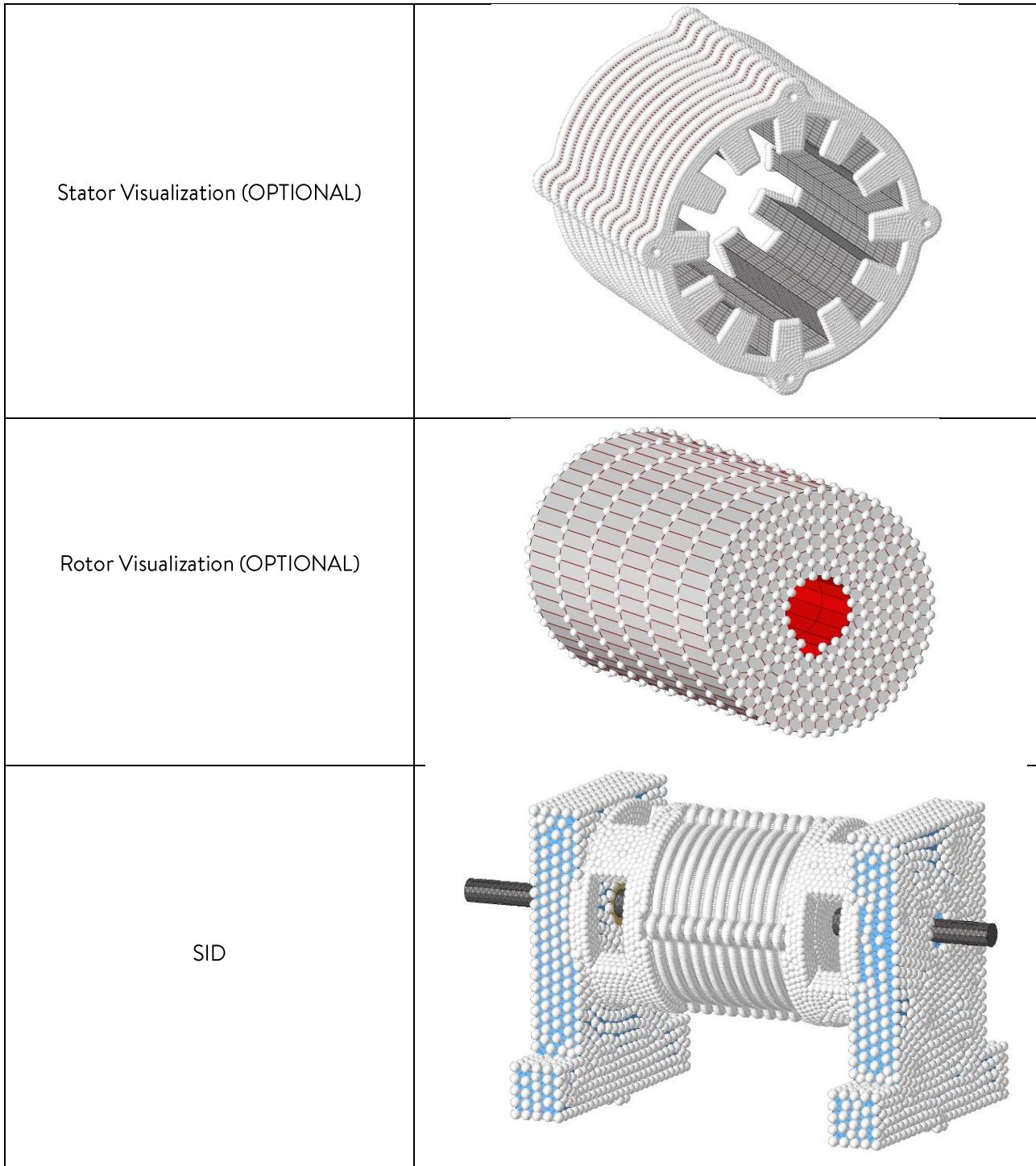


Figure 15 : Nodes set creation

Additionally, another set of nodes should be created to be able to extract the modal displacements of the nodes from all the sets identified in 3. The set usually named 'SID' should contain all the defined sets to extract the modal displacements.

All the different sets can be viewed below:

Set	Visualization
Stator Force	
Vibration	
Rotor force (OPTIONAL)	



#### 4.2.2 Result file format

The control card OUTPUT must be added with keyword OP2 to write results in op2 file.

#### 4.2.3 Output displacement

The output file can be very heavy if it contains the displacements of every node in the model. Thus, it is convenient to output only the nodes contained in the sets previously created. To do so, the control card GLOBAL OUTPUT REQUEST must be added, checking the box DISPLACEMENT, then the option SID, and specifying the SID set

previously created.

$SID = \text{Set1\_Stator\_forces} \cup \text{Set2\_Rotor\_forces} \cup \text{Set3\_Vibration} (\cup \text{Set4\_Vibration\_2} \cup \dots)$   
 (U for union)

The number of nodes contained in SID set should be equal to:

Number of nodes contained in Set1\_Stator\_forces + Number of nodes contained in Set2\_Rotor\_forces +  
 Number of nodes contained in Set3\_Vibration (+ Number of nodes contained in Set4\_Vibration...)

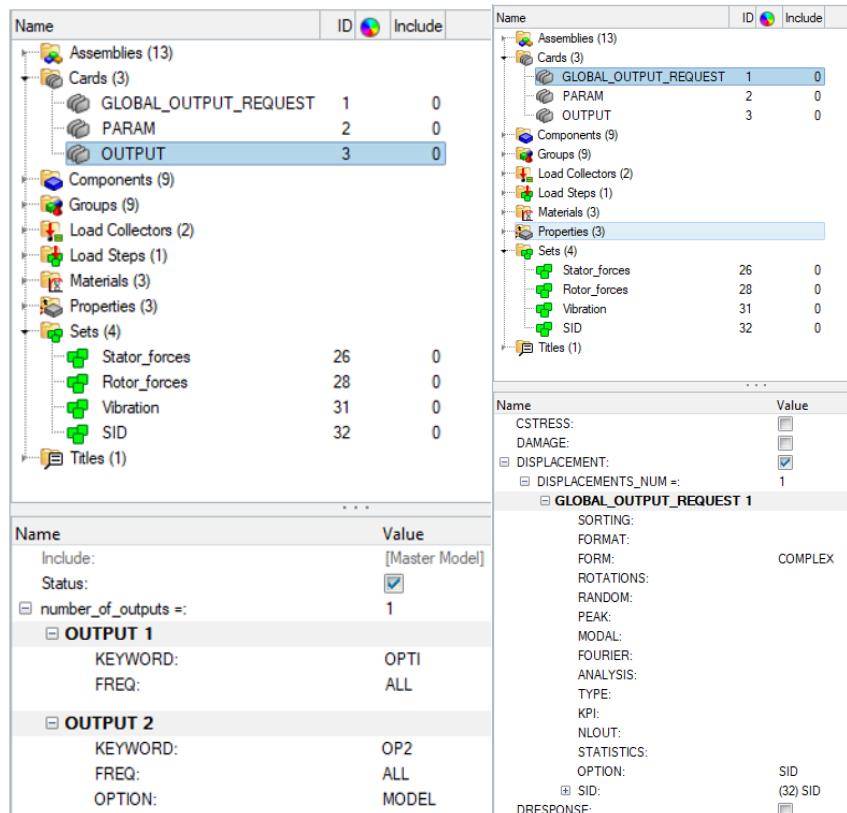
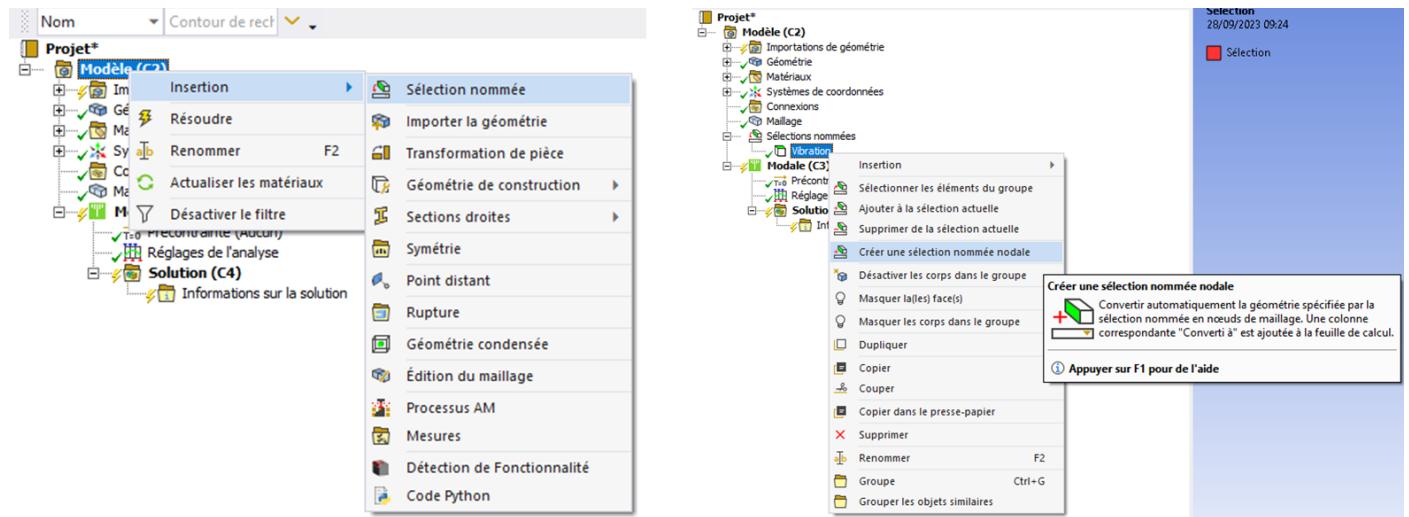


Figure 16 : Optistruct output

## 4.3 Ansys Workbench

### 4.3.1 Named selection

As described in 3, the following “named selections” are created as described below :



*Figure 17: Named selection creation*

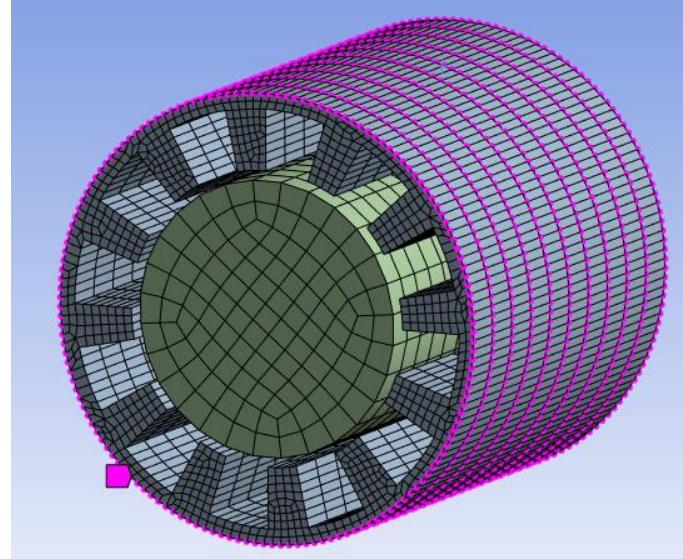
Named selection must be a nodal one.

Additionally, another set of nodes should be created to be able to extract the modal displacements of the nodes from all the sets identified in §3. The set usually named ‘SID’ should contain all the defined sets to extract the modal displacements.

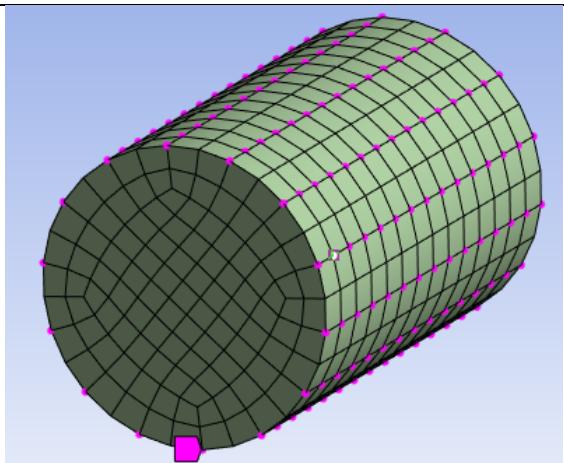
All the different sets can be viewed below:

Set	Visualization
Stator Force	

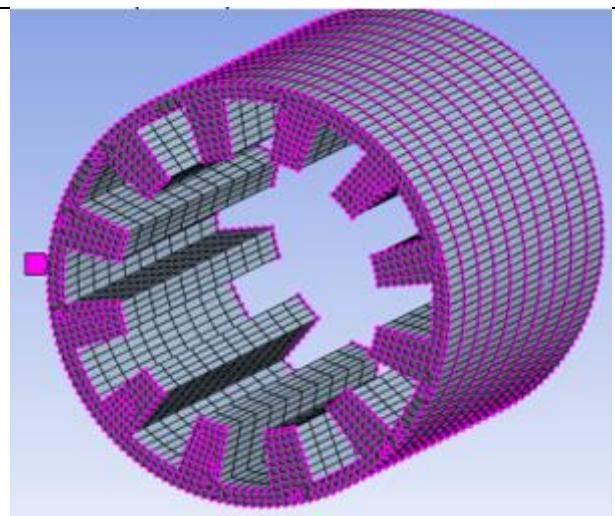
Vibration

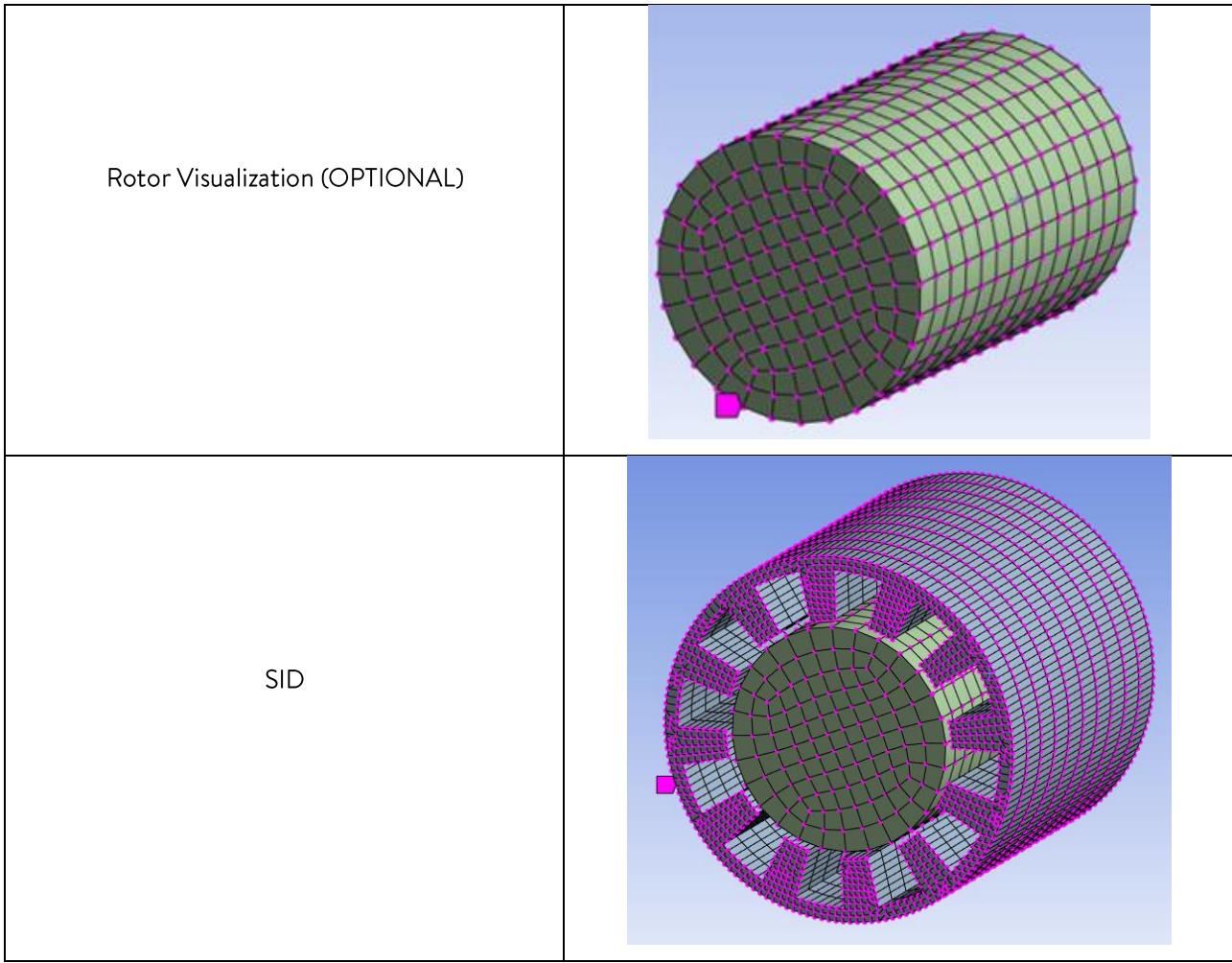


Rotor force (OPTIONAL)



Stator Visualization (OPTIONAL)





#### 4.3.2 Output displacement

The output file can be very heavy if it contains the displacements of every node in the model. Thus, it is convenient to output only the nodes contained in the sets previously created. To do so, an APDL script can be used (refer to the code below).

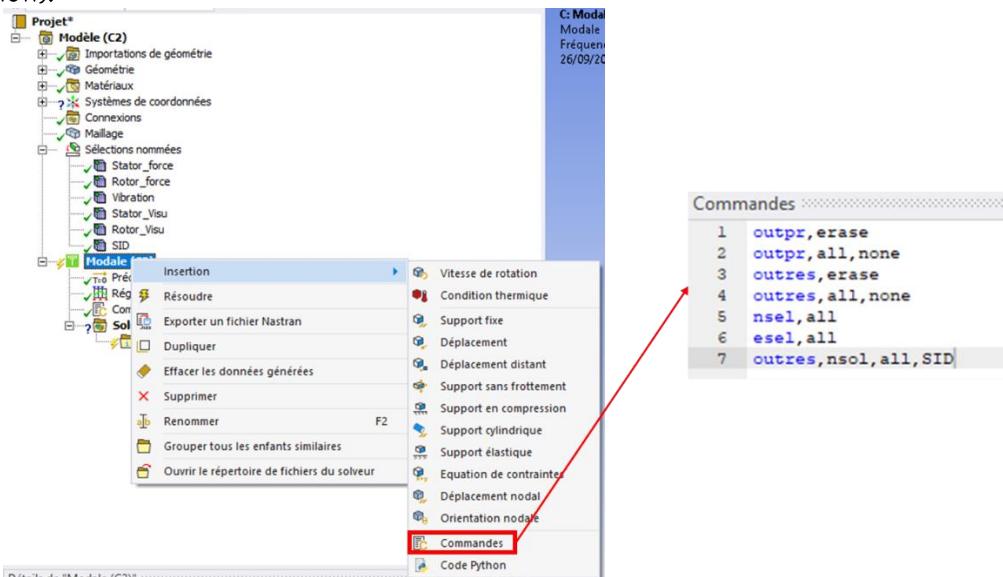


Figure 18 : Ansys output

#### 4.3.3 Result file compression

Result file compression should be turned off when running the modal analysis to ensure results in .rst file are not compressed.

Output Controls	
Stress	No
Surface Stress	No
Back Stress	No
Strain	No
Contact Data	No
Nodal Forces	No
Volume and Energy	No
Euler Angles	No
Calculate Reactions	No
General Miscellaneous	No
Cache Results in Mem...	Never
Combine Distributed ...	Program Controlled
Result File Compressi...	Program Controlled

Figure 19 : Ansys file compression

## 4.4 3DS Abaqus

### 4.4.1 Node selection

Several sets should be created to extract the modal displacements. Those sets are described in section 3.

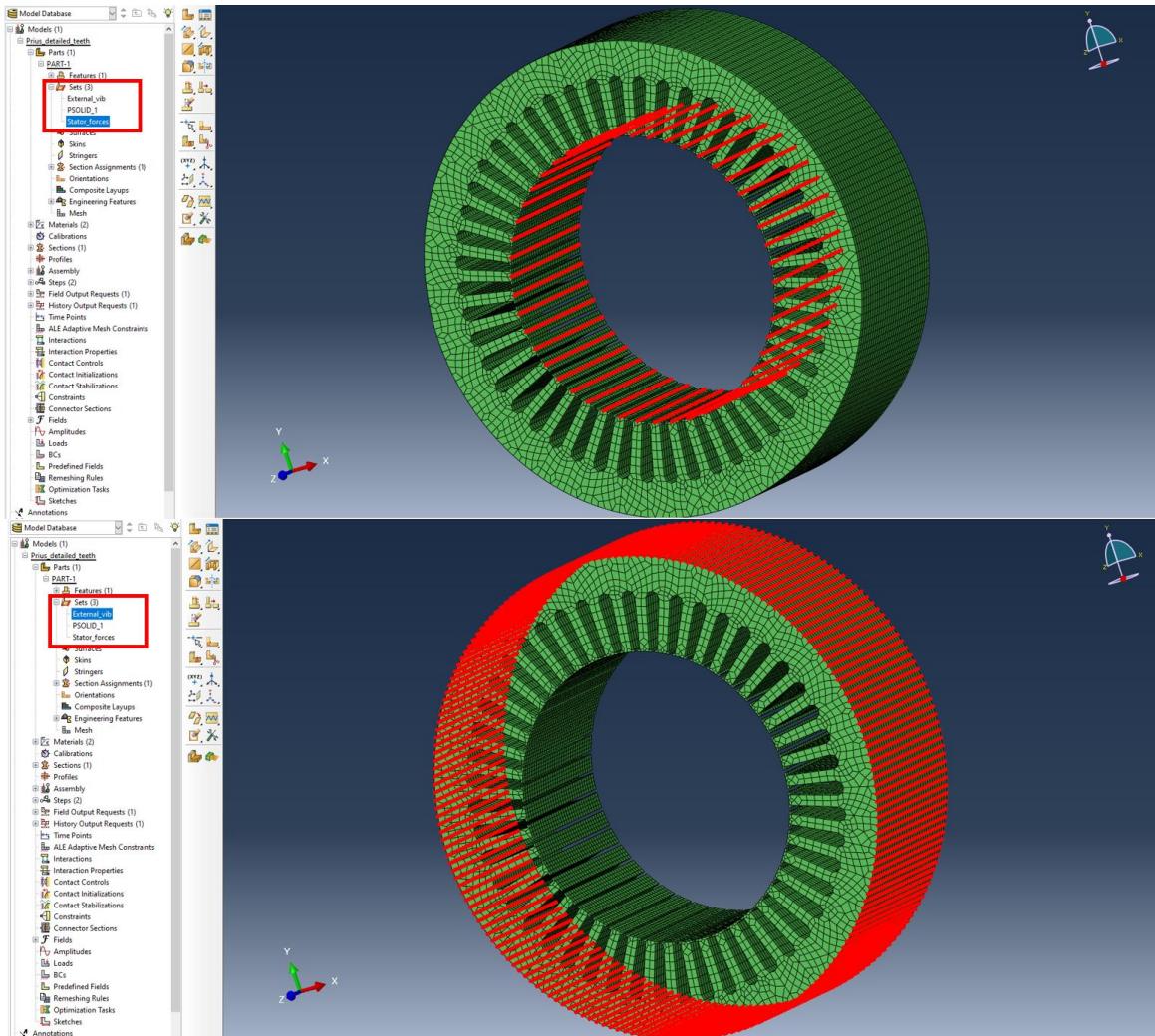
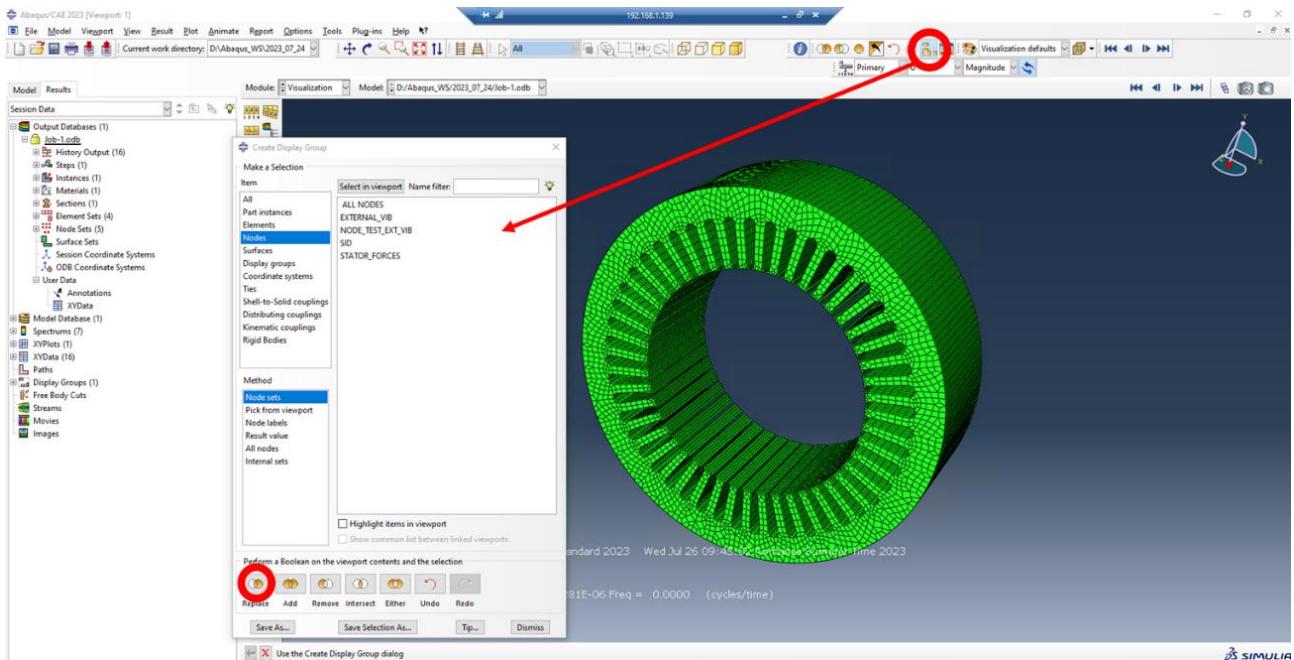


Figure 20 - Sets of nodes created respectively for forces application and vibration calculation.

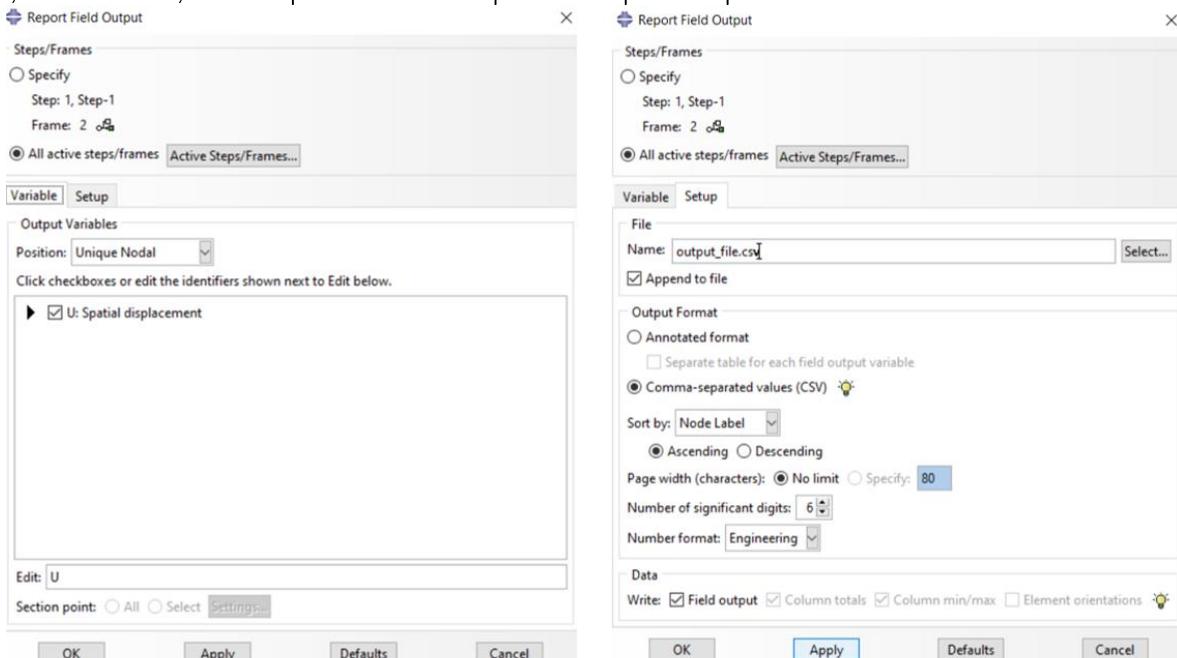
#### 4.4.2 Output displacement

The output file can be very heavy if it contains the displacements of every node in the model. Thus, it is convenient to output only the nodes contained in the sets previously created. To do so, click “Create Display Group” and select those sets. Click “Replace”. Elements are now hidden.



*Figure 21 – Display group of node sets*

Then, in the toolbar, click “Report” > “Field output” and export as .rpt file or as .csv file as indicated below.

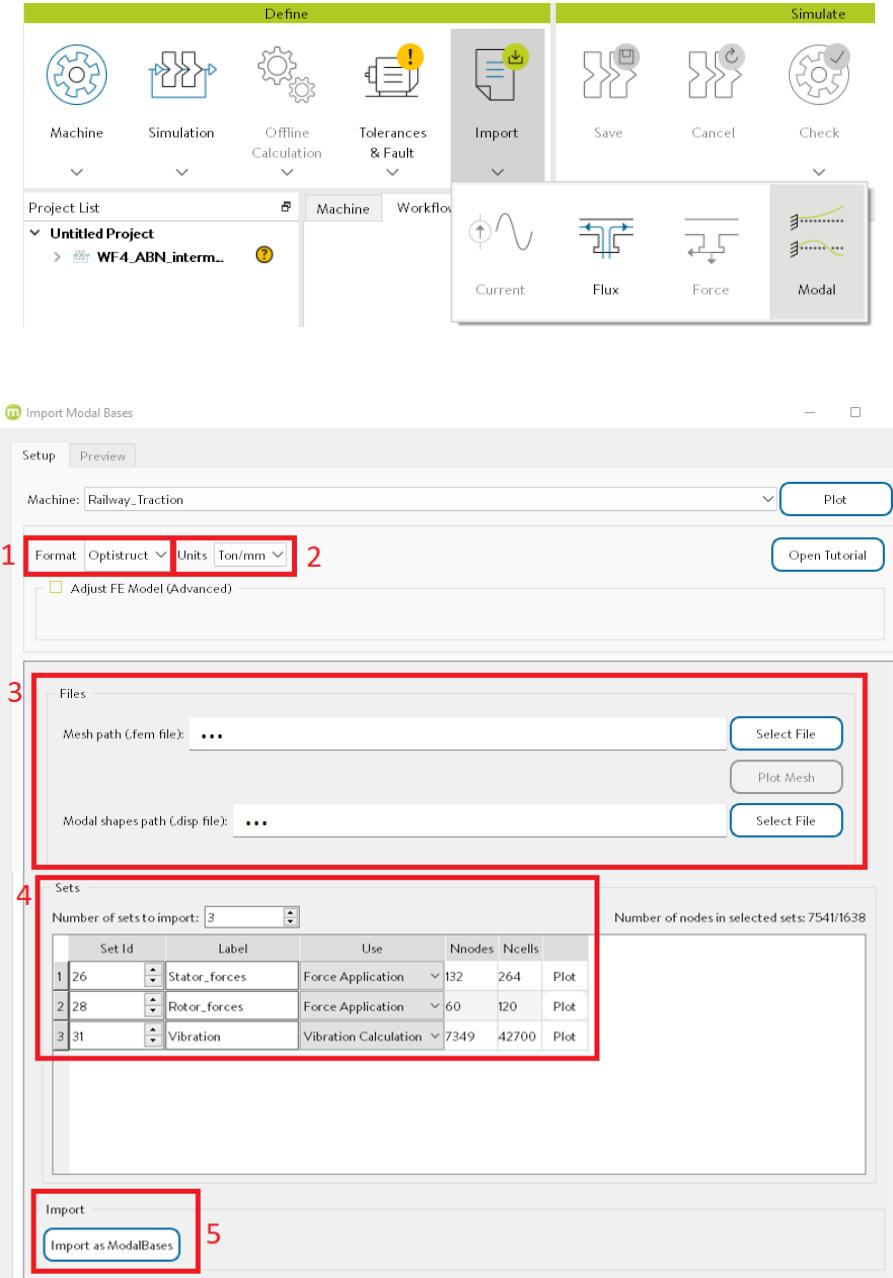


*Figure 22 – Export of nodal displacements*

Now that both \*.inp and \*.rpt/\*.csv files are available, the modal basis can be imported in Manatee. \*.inp file is automatically created in the workspace when a Job is submitted.

## 5 Import of modal basis in Manatee format

Click on Import>Modal under the Define tab.



*Figure 23 : Modal basis import in manatee*

In the contextual menu, follow the steps:

- 1- Choose the format (Optistruct, Ansys, Abaqus)
- 2- Choose the unit system.
- 3- Select the files to import (if quad mesh is in second order, they must be set in first order).
- 4- Select the sets previously created. Stator and rotor sets are for force application. Obviously, vibration set is for vibration calculation. At least one force application set, and one vibration calculation set must be selected.
- 5- Import the modal basis to Manatee format.

## 6 Use of Manatee modal basis format during e-NVH simulation

To use an imported modal basis for a simulation, click the “Modal Basis” icon.

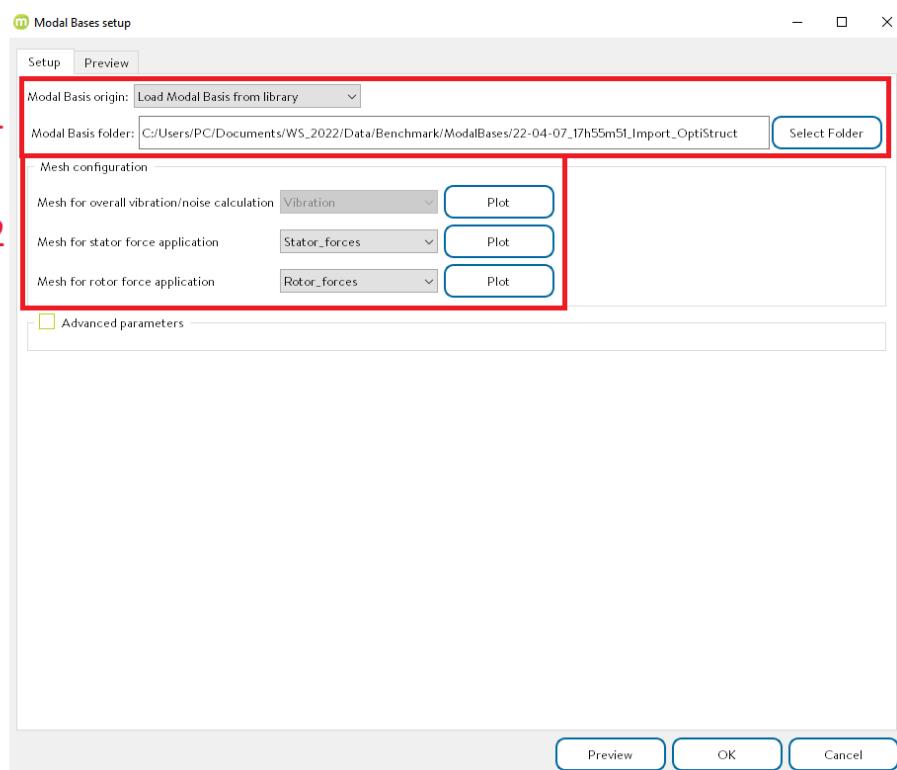
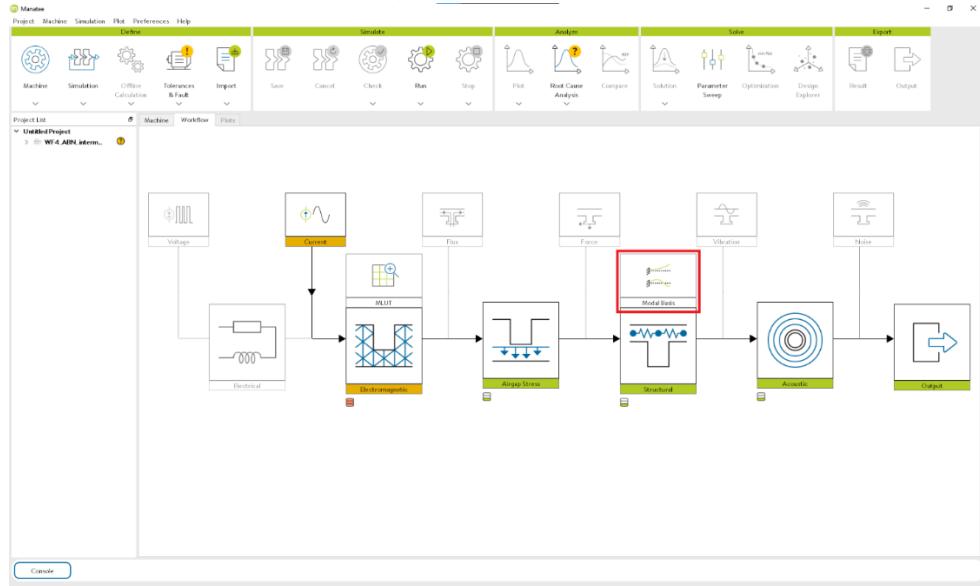


Figure 24 : Use of an imported modal basis in Manatee

In the contextual menu, follow the steps:

- 1- Choose “Load Modal Basis from library” and select the modal basis folder previously imported
- 2- Match the stator and rotor sets with stator and rotor meshes for force application

The modal basis can be previewed in Manatee.