

ALTAIR

ONLY FORWARD

Altair[®] Flux[®]

How to Make Flux-OptiStruct coupling
through Import/Export context ?

Updated: 05/23/2022

Contents

1 Introduction	3
2 Creation of the mechanical mesh in HyperMesh	5
3 Magnetic forces computation and export in Flux	17
4 NVH analysis in OptiStruct with data setting in HyperMesh	25

About

Flux-OptiStruct coupling allows making NVH analysis or static analysis in OptiStruct using the magnetic forces exported from Flux.

The various concerned applications according to the coupling are: Motor, actuator, busbar ...

This specific document gives the optimal workflow to follow in HyperMesh, Flux and OptiStruct to make the **NVH analysis on a motor**.

For other types of analysis (time/static forces, time/static heat sources, temperature), please refer to the online help documentation.

Concerning the software version to use: Flux2018.2 (mandatory), HyperMesh 2017.2 (strongly advised) and OptiStruct 2017 (advised).

Interest

Magnetic forces which exist in a motor on the interface between the airgap and ferromagnetic part (stator or rotor) can involve motor vibrations with annoying noise.

The Flux-OptiStruct coupling allows predicting and evaluating those vibrations.

This document

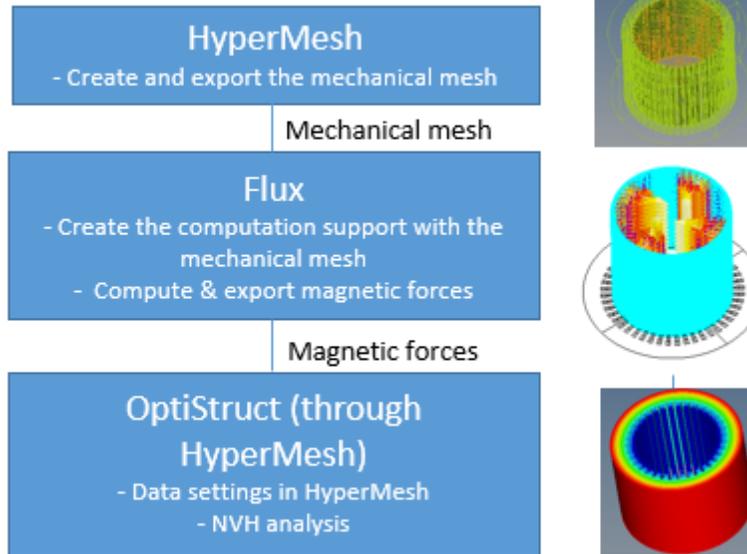
From Flux 2018.1, the Flux-OptiStruct coupling is available in a new context called **import/export context**. In future, this context will replace the **mechanical analysis context** to be the context dedicated to all software couplings.

For the moment, the **mechanical analysis context** is still available.

Another HowTo document explains the workflow to follow in this context (document written for Flux2018).

Steps

The following synoptic illustrates the different steps described in this workflow document:



Creation of the mechanical mesh in HyperMesh

Introduction

In this part, the steps in HyperMesh to create and export the “mechanical mesh” where the magnetic forces will be computed in Flux are presented. The advantage of this method is that the magnetic forces will be directly used in OptiStruct without any interpolation or projection which can generate errors.

Mechanical mesh

The mesh which will be used for forces computation in Flux must respect the following constraints:

- It must be cylindrical, centered on motor axis
- At the interface of two regions having different magnetic permeability value (stator/air or rotor/air)

 **Note:** The mesh should not contain superimposed nodes or unused free nodes

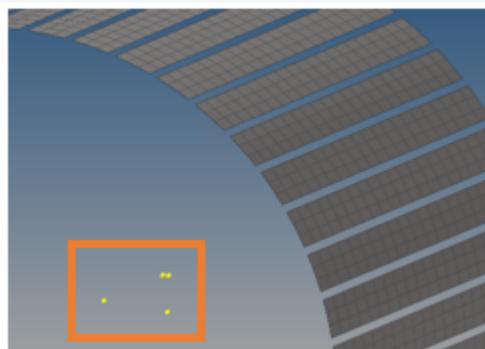


Figure 1: Temporary nodes in the model

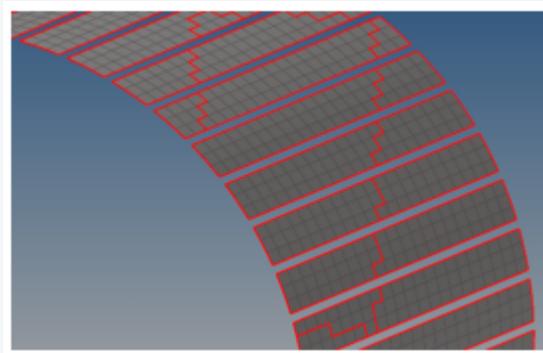


Figure 2: Elements not connected detected with the "free edges" tool in HyperMesh

Note: The normal vectors orientation is defined in HyperMesh and Flux respects this orientation. To visualize (and eventually change) the orientation, please use the following commands:

assemblies	find	translate	check elems	numbers	<input type="radio"/> Geom
organize	mask	rotate	edges	renumber	<input type="radio"/> 1D
color	delete	scale	faces	count	<input type="radio"/> 2D
rename		reflect	features	mass calc	<input type="radio"/> 3D
reorder		project	normals	tags	<input type="radio"/> Analysis
		position	dependency	HyperMorph	<input checked="" type="radio"/> Tool
		permute	penetration		<input type="radio"/> Post

elements
 surfs

displayed 2D elems
 orientation: auto
 use feature angle

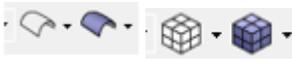
color display
 display adjusted only

display
 adjust
 reverse

HyperMesh graphic area manipulation

Some tips concerning graphic area manipulation (for first HyperMesh usage) are presented here:

- To Translate the geometry: use Ctrl + right click
- To rotate the geometry: use Ctrl + left click
- To zoom in/out the geometry: use control + roll scroll wheel
- To manage the transparency / opacity of geometry or mesh, use the following icons (below the

graphic area): 

- To manage the displaying of the model components geometry and mesh, use :



HyperMesh entities selection

Some tips concerning entities selection (for first HyperMesh usage) are presented here:

To select entities: for example, to select faces:

- Click on **surfs** 

Remarks:

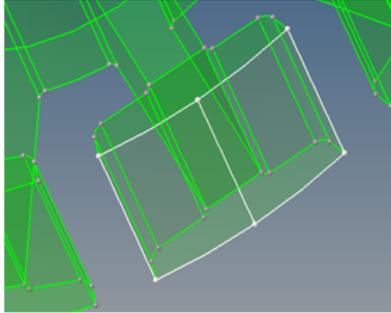
 allows to change the entity type to select.

 allows to reset the selected entities

- Option 1: click on the yellow field to open the temporary extended menu to get multiple selection methods. For example: « all / displayed / ... »

by window	on plane	by width	by geoms	by domains	by laminate
displayed	retrieve	by group	by adjacent	by handles	by path
all	save	duplicate	by attached	by morph vols	by include
reverse	by id	by config	by face	by block	
by collector	by assems	by sets	by outputblock	by ply	

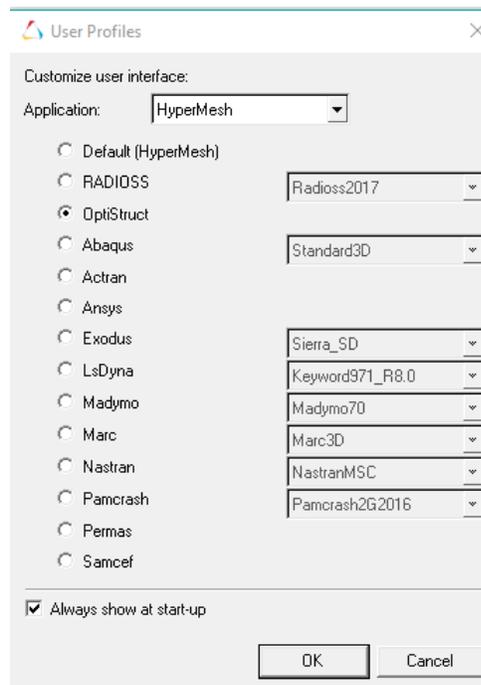
- Option 2: select face by face on the graphic with the mouse left click (without Ctrl button). The selected face is highlighted (in white). To deselect a face, use the mouse right click



Mechanical mesh in HyperMesh

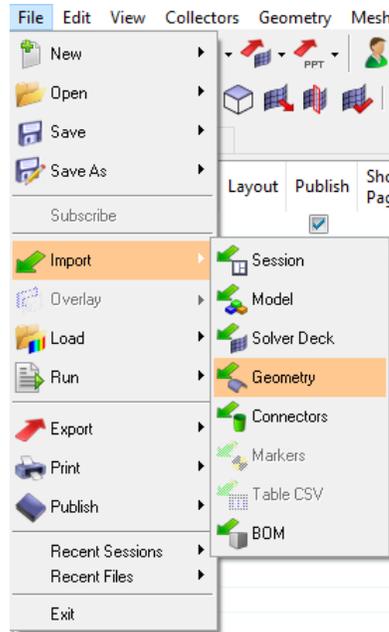
The steps in HyperMesh to create the mechanical mesh are described below. Screen shots of the different steps are available.

1. Make sure to select the OptiStruct user profile when opening HyperMesh:

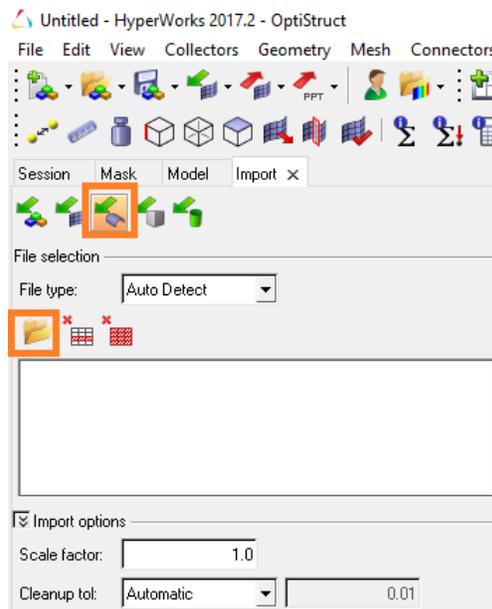


You can review this window by clicking on the  icon.

2. Import CAO file of the geometry to consider (stator / rotor / all / ...). Here we consider stator part:
 - Menu : **File > Import > Geometry**

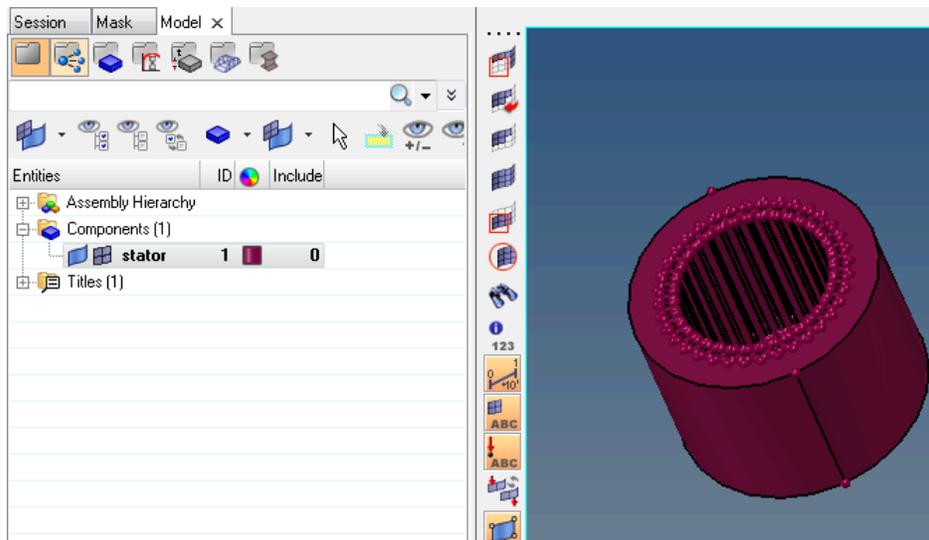
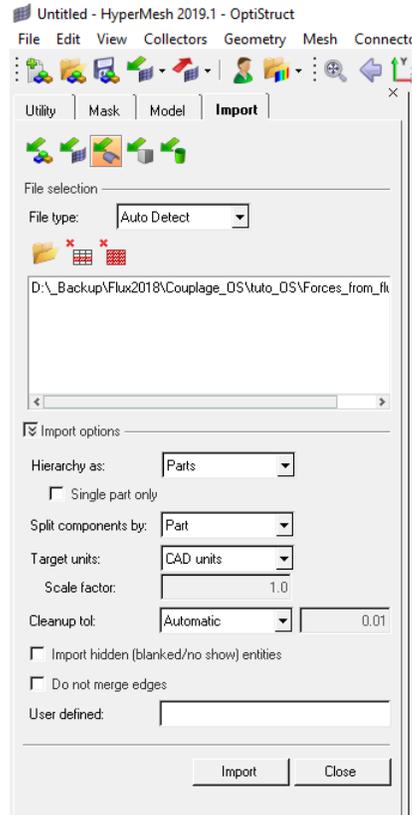


- Click on the following icon to select the CAO file



The path of the selected CAO file appears in the window below the icon

- Click on **import** to confirm the import



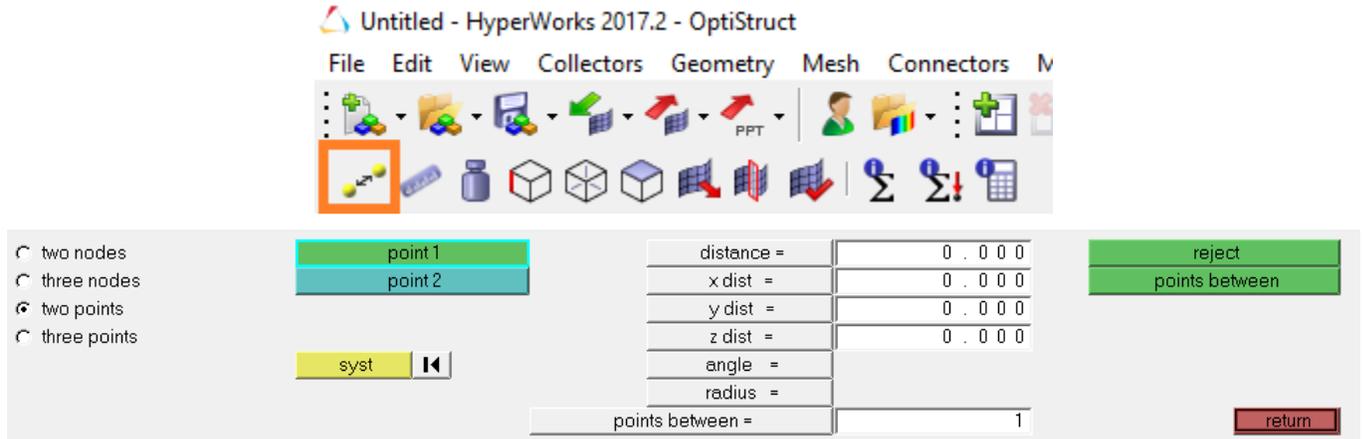
The geometry is imported and visible in the graphic area, and the **Model** tab contains the different geometry components

3. Mesh the solid components (volume mesh):

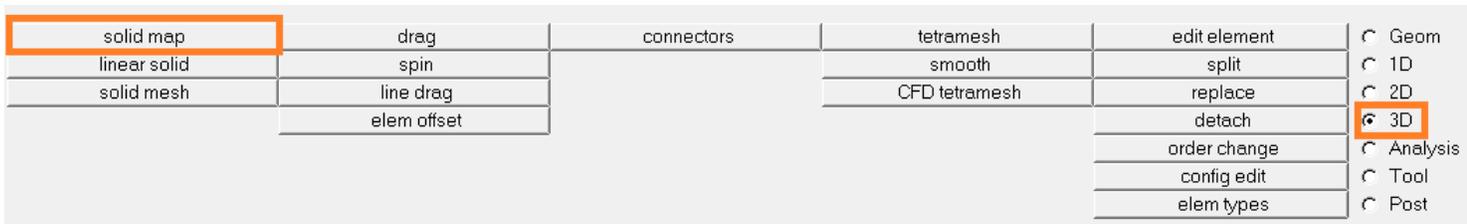
In our example case the **volume mesh** is the best tool to use.

This mesh step should be adapted according to the complexity of the input geometry.

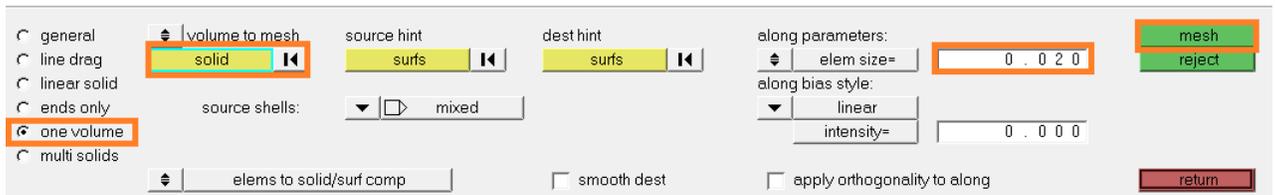
- Check the geometrical distance between two points to define the mesh element size:



- Open the **3D** panel and select the **solid map** tool:

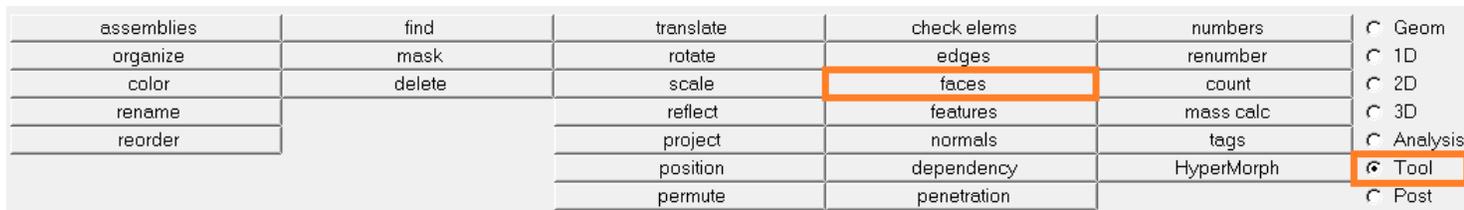


- To mesh, follow the steps below:
 - Select **one volume**
 - Click on **solid** (highlighted with blue line)
 - Select the components to mesh on the graphic
 - Fill the element size
 - Click on **mesh**
 - If the mesh is ok, click on **return**. Otherwise, click on **reject** and modify the mesh data.



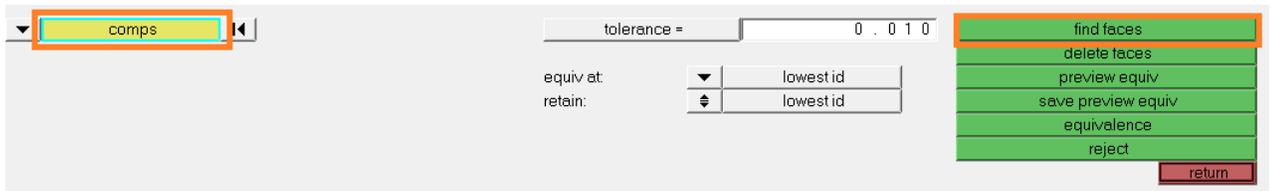
4. Create the faces of the solid component(s) containing the surface mesh support:

- Select the **Tool** panel and the tool **faces**

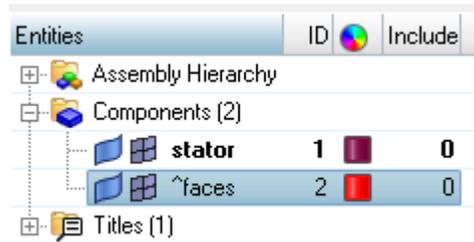


- To generate a component containing the external surface mesh:
 - Click on **comps**
 - select the component(s)

- click on **find faces**



The component **^faces** has been created. It contains the faces of all the solid component(s)



- Right click on this component and select **isolate** to display only the surface mesh
- 5.** In the **^faces** component, delete the surface mesh which should not be exported. This step can be fastidious, so the selection by relation and the **reverse** command can be used. On this example:

- Click on the following icon: 

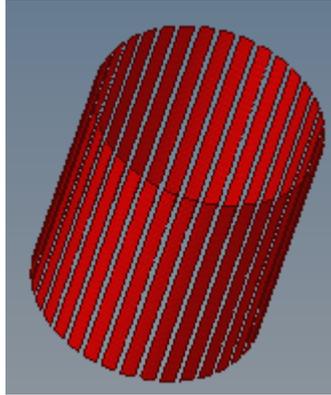
The following window is opened:



- Follow the below steps described:
 - Select one element by tooth as on the image

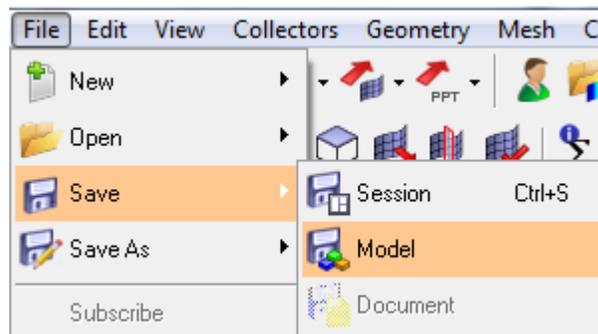


- Click twice on **elems** to activate the extended menu. At the first click select **by face**. All the meshed teeth faces are selected
 - At the second click on **elems** select **reverse**. All the other meshed faces are selected
 - Click on **delete entity**
- The final face mesh corresponds to:



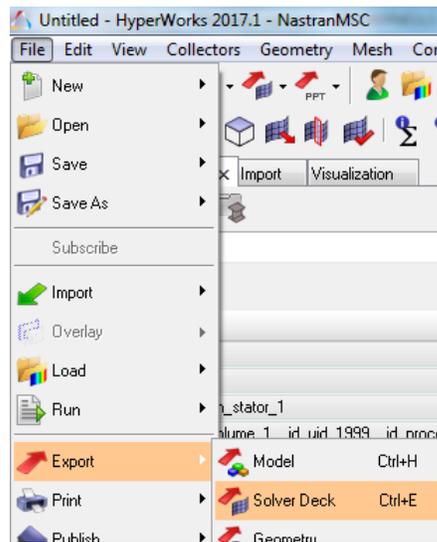
- Right click on ^**faces** component, **rename** and delete ^ to export it.

6. (optional) Save the HyperMesh model :

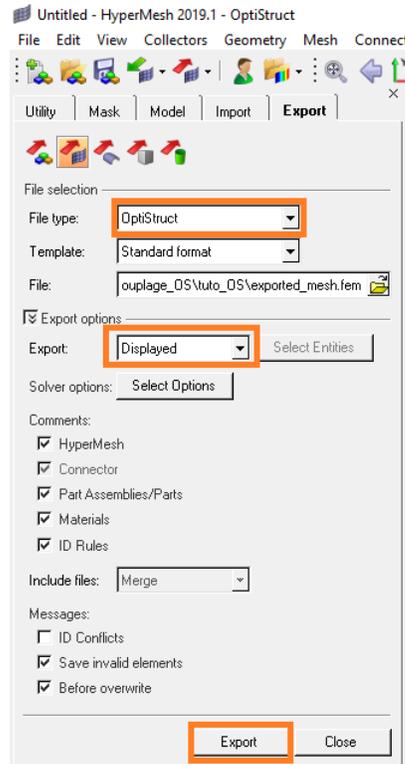


7. Export the displayed mesh :

- Menu : **File > export > Solver Deck:**



- Ensure to display only the meshed faces of the **faces** component
- Export the displayed mesh in **Optistruct** format:



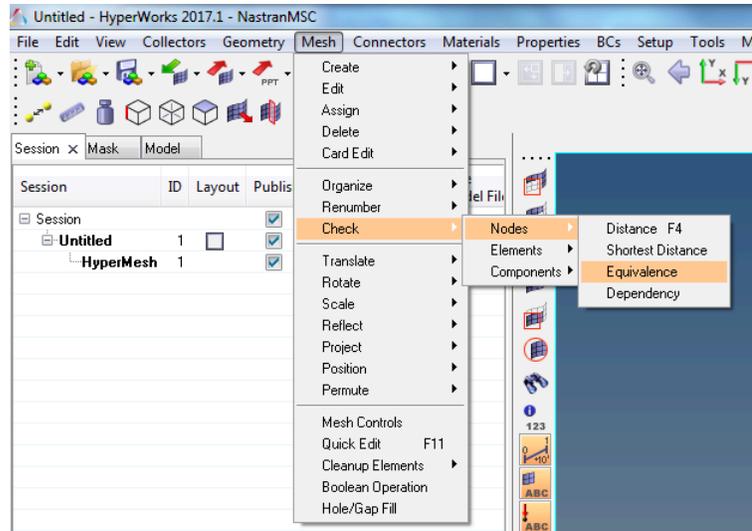
Superimposed nodes problem

HyperMesh, contrary to Flux, allows the creation of superimposed nodes / elements. Consequently, sometimes the exported mesh can be duplicated and superimposed (often because of a user manipulation error, for example by clicking two times on **mesh**). In Flux, the support creation using this mesh fails.

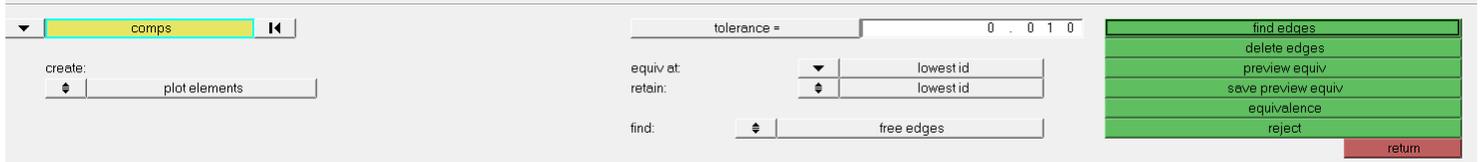
In this paragraph, the method to merge superimposed nodes is presented.

Attention: this command must be used carefully because some close points (under tolerance value) can be merged.

1. Open the HyperMesh meshed model
2. Launch the command **Equivalence** on nodes :



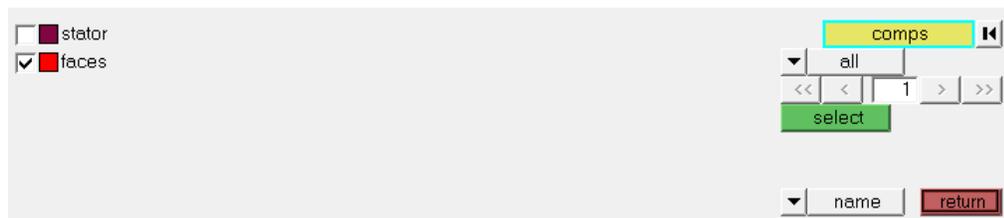
Following window opens :



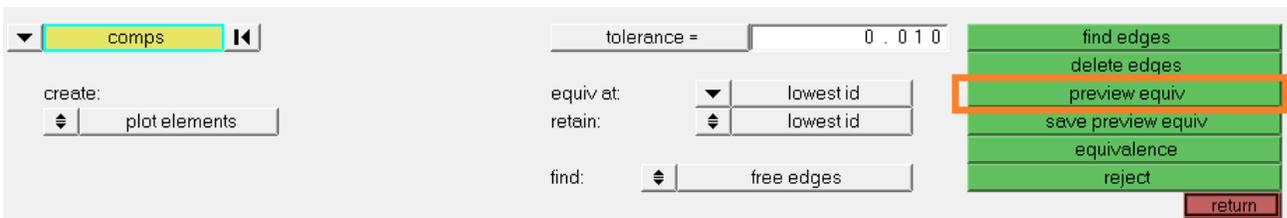
3. • Click on **comps** :



- Select **faces** and click on **select**



4. Verify the tolerance value and click on **preview equiv** .

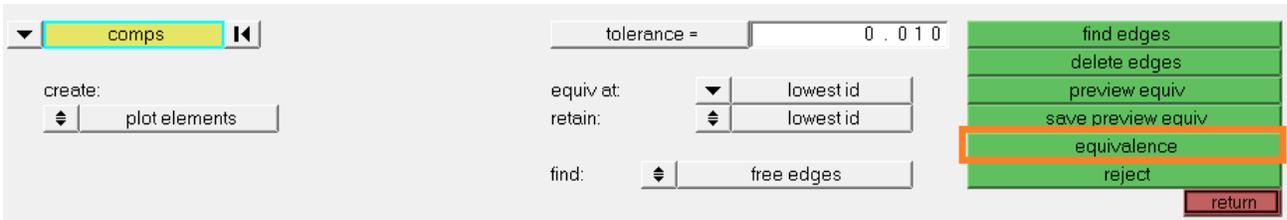


This step helps you to review the nodes that HyperMesh detects within the tolerance.

If you agree click on **equivalence**.

If not modify the tolerance and repeat the above steps.

Then, click on **return**



At the bottom left of HyperMesh window, the number of **equivalenced nodes** is indicated:

198 nodes were equivalenced.

- Export the mesh by following the step 5 of the previous step/action table.

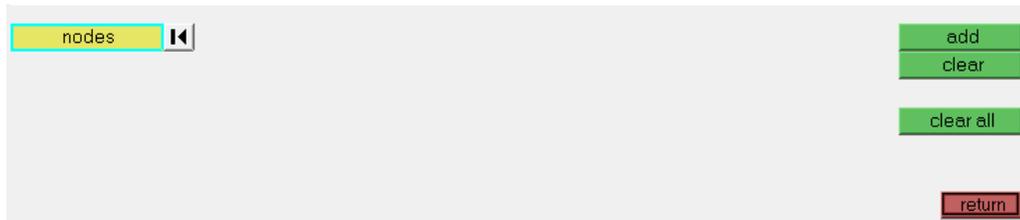
Free nodes problem

HyperMesh allows free nodes which are not attached to elements. If the mechanical mesh contains such nodes, the mesh import fails in Flux.

2 solutions in HyperMesh are proposed here.

The first solution:

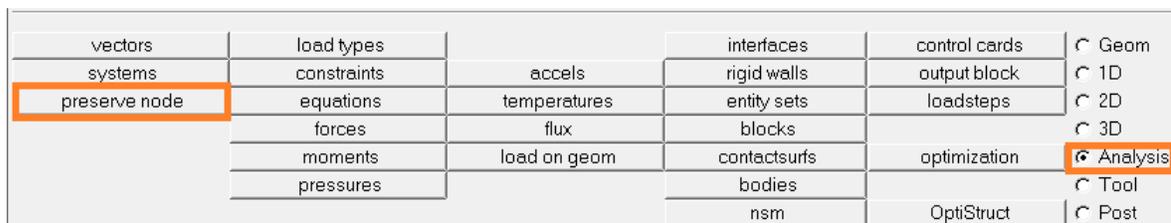
- Open the Geom panel below the graphic area and select the **temp nodes** tool:



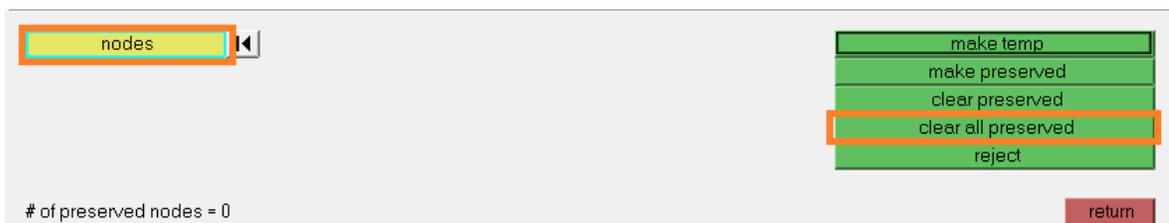
- Click on **nodes** and select **all**.
Click on **clear all**
- Export the mesh by following the step 5 of the previous step/action table

The second solution:

- On the window below the graphic area, select **Analysis** and **preserve node**:



- Click on **nodes** and select **all**.
Click on **clear all preserved**.



The preserved nodes will not be exported

3. Export the mesh by following the step 5 of the previous step/action table.

 **Note:** Please note that as you export the meshed support, you should not modify the mechanical mesh.

It will be used to setup the NVH analysis.

Magnetic forces computation and export in Flux

Introduction

This section presents the steps to follow in Flux to compute and export forces harmonics. The steps are as following:

- Create the computation support
- Compute the forces and forces harmonics
- Visualize the forces
- Export the forces

Flux project

The Flux project can be in 2D or 3D in transient magnetic application (for NVH analysis).

Import/Export context

The **Import/Export context** is dedicated to all couplings with other software, including the coupling with OptiStruct. In future, this context will replace all the specific coupling contexts including the **mechanical analysis context**.

To enter in the **Import/Export context** respectively after and before solving:

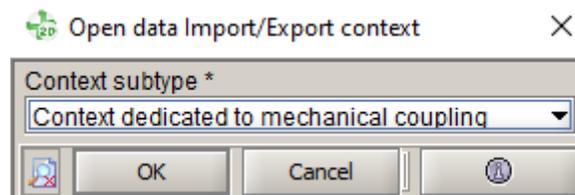
Data exchange > Open data import/export context

Parameter/Quantity > Open data import/export context

Mechanical context

When entering in the **Import/Export context**, user has to choose the dedicated context: Mechanical or thermal or generic.

Here the mechanical context is chosen:

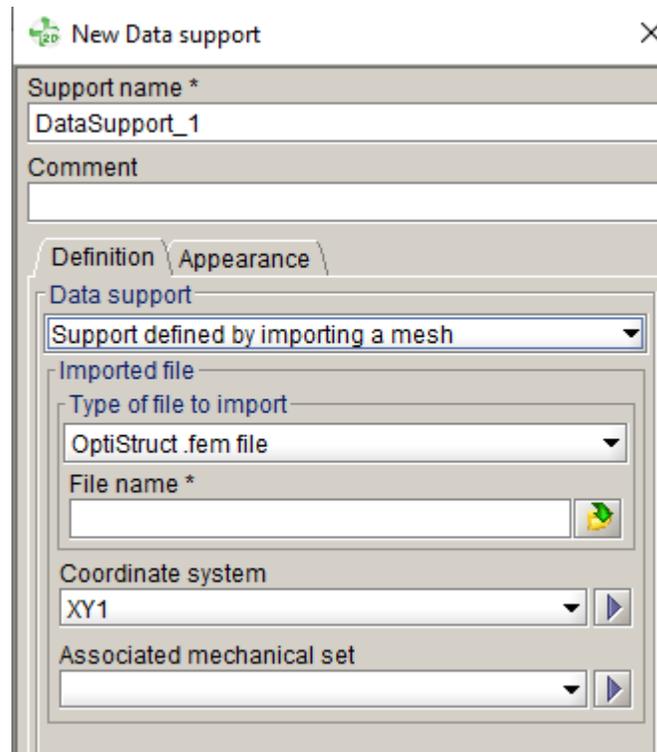


Creation of the support

In this part, the approach to create the support **defined by importing a mesh** is presented. The imported mesh corresponds to the mesh exported before which will be used for the forces computation in Flux and for the vibratory analysis in OptiStruct.

There is also in Flux a support defined by Flux entities, but this requires an interpolation step when importing forces into HyperMesh (for more information, please refer to the online help).

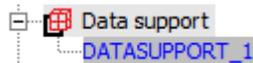
1. Open the box to create the support: **Data support > New**



2. Choose the data support **Support defined by importing a mesh**
3. Choose the type of file to import **OptiStruct .fem file**
4. Select the file to import
5. Select the coordinate system.

 **Note:** Depending on the length units, user may have to create and select a new global coordinate system with the right units.

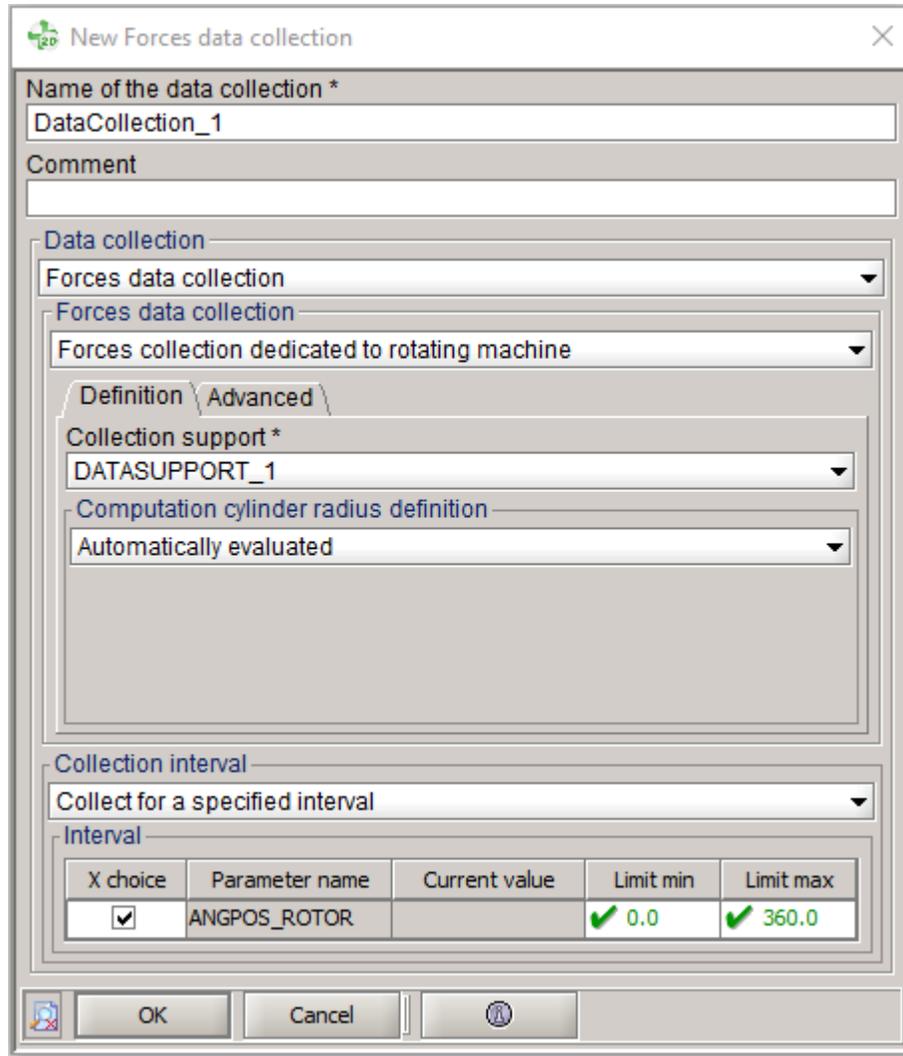
6. Select the mechanical set.
It is especially useful if the support must follow a moving mechanical set.
7. Click on **OK** to create the support.
The data support is created:



Compute the forces

The steps to compute the forces harmonics are described below.

1. Open the box to create the forces data collection: **Data collection > Forces data collection > New**



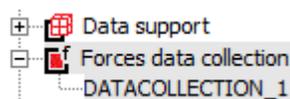
2. Choose the Forces data collection **Forces collection dedicated to rotating machine with mesh import**
3. Choose the data support created before
4. Choose the automatic computation of the radius in the airgap.

The forces will be computed on this radius before being projected on the imported mesh.

A value can be set by the user. In this case, it is advised to choose its value on $\frac{1}{4}$ distance of the airgap thickness, stator/rotor side, according to where the imported support is.

5. Choose the collection interval:
 - Collect for all steps of the scenario
 - Or collect for a specified interval. This choice allows reducing computation time
6. Click on **OK** to create the data collection

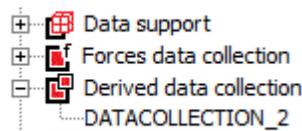
The data collection is created:



7. Open the box to create a derived data collection: **Data collection > Derived data collection > New**

8. Select the **FFT of a time collection**
9. Select the data collection **DATACOLLECTION_1** created before
10. Select the time interval for the FFT, corresponding to one period.
11. Select **Automatic duplication of forces for rotating machines**
12. Click on **OK** to create the derived data collection.

The derived data collection is created:



13. Evaluate the forces and forces harmonics values:

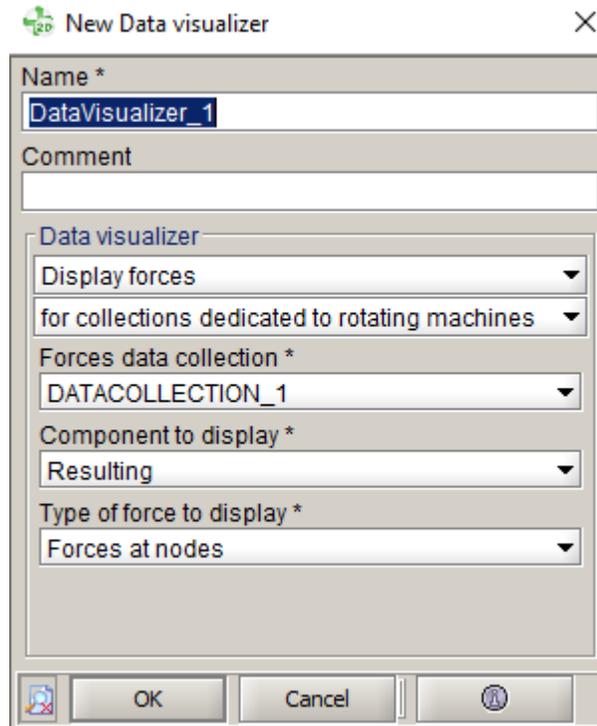
Data collection > collect data

Note: if the project is not solved, this command will not be available. In fact, the collection is done during the solving (it allows reducing computation time).

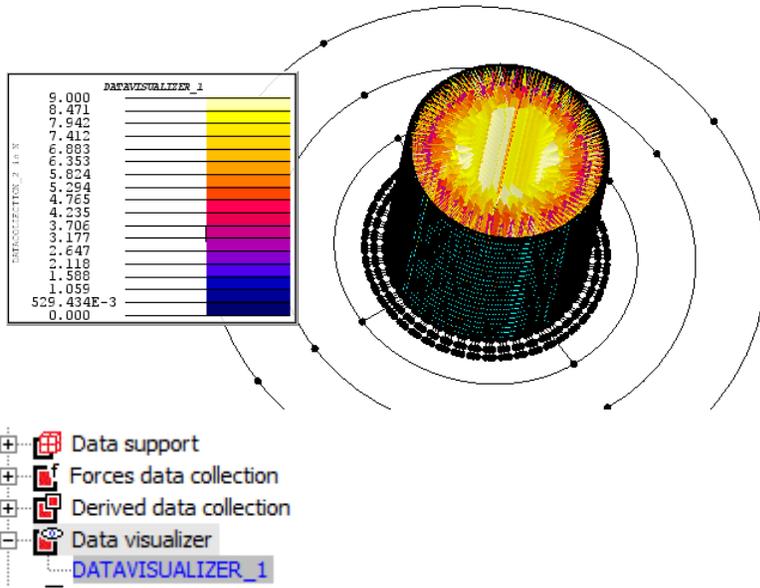
Visualize the forces

The steps to visualize the forces or forces harmonics arrows are described below.

1. Open the box to visualize the forces: **Data visualizer > Forces visualizer > New**



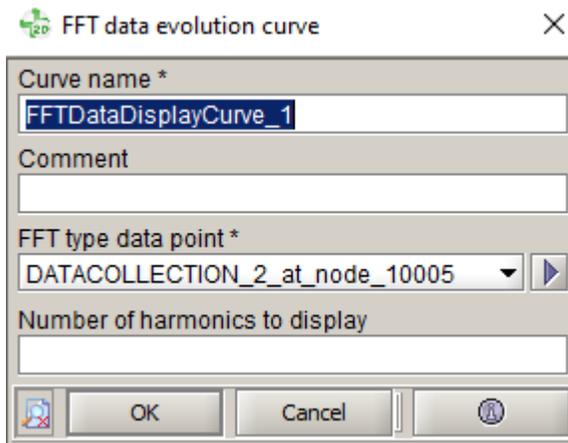
2. User can choose **display forces** or **display harmonic forces** and user has to select **for collections dedicated to rotating machines**
3. Choose the data collection to visualize (forces or forces harmonics)
4. It is possible to visualize the following components of the forces vectors:
 - **Resulting**
 - **Normal**
 - **Tangent**
5. It is possible to visualize:
 - **Forces at nodes**
 - **Global forces**. It corresponds to the resulting forces on one geometrical part (one stator teeth for example)
6. If user has selected the forces harmonics visualization, the rank of the harmonic to visualize must be chosen.
7. Click on **OK** to create the data visualizer entity and visualize the forces.
The data visualizer is created and the forces arrows displayed:



8. When the forces arrows are visible, it is possible to plot an evolutive curve:

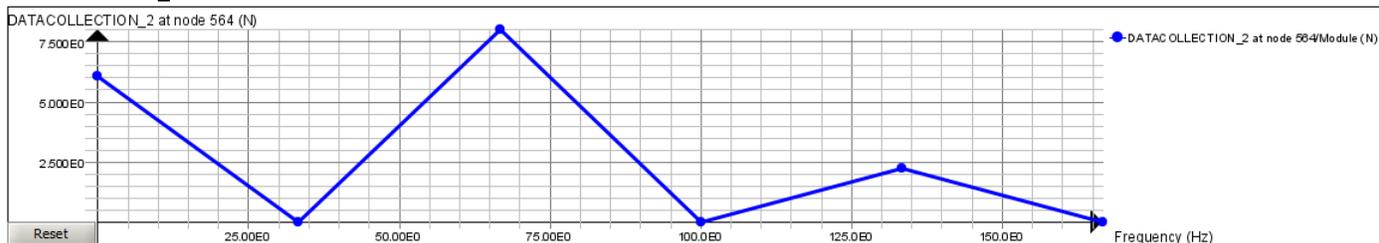
- Forces harmonics module depending on the frequency:
 - Make the harmonics arrows visible
 - Open the FFT data evolution curve:

Data visualizer > Results and curves on one point > FFT data evolution curve



- Choose the node in the graphic (FFT type data point)
- Select the number of harmonics to display

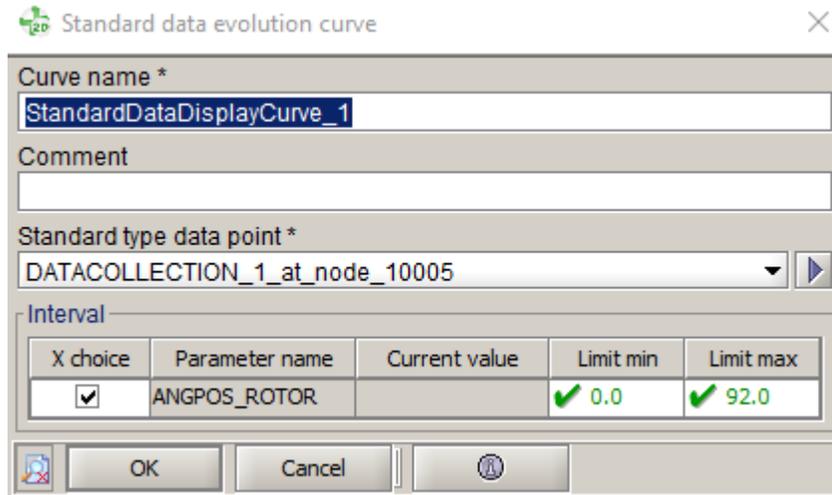
DATACOLLECTION_2 at node 564



- Forces module depending on the time:

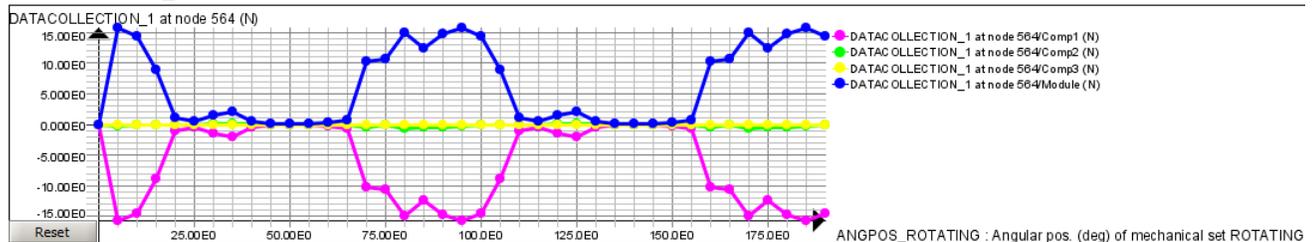
- Make the forces arrows visible
- Open the Standard data evolution curve

Data visualizer > Results and curves on one point > Standard data evolution curve



- Choose the node in the graphic (Standard type data point)
- Define the time/position interval

DATACOLLECTION_1 at node 564

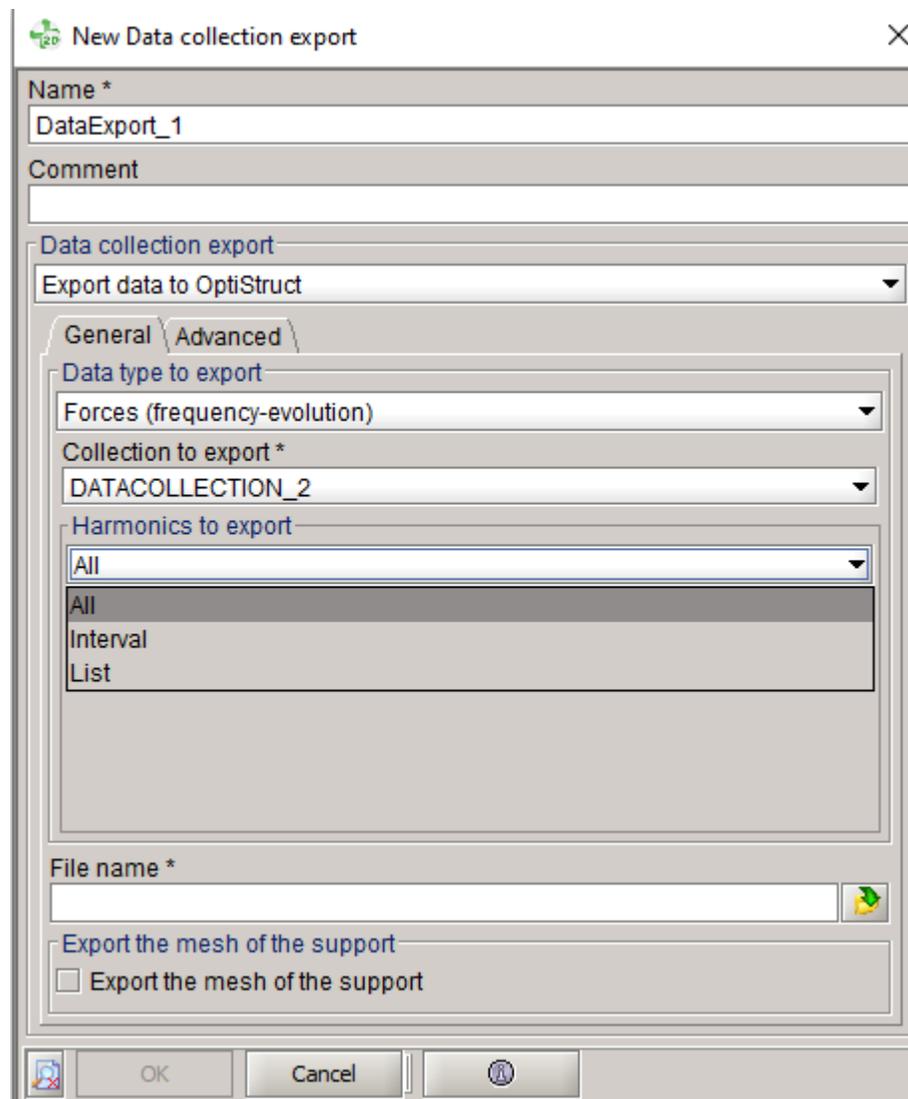


Export the forces

The steps to export the forces harmonics are described below.

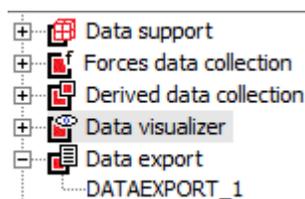
1. Open the box to export the forces:

Data export > Data export to OptiStruct > New



2. Select the **Forces (frequency-evolution)** to export
3. Select the derived data collection **DATACOLLECTION_2** to export
4. Select harmonics to export (the selection may be done by choosing either all, an interval or a list of harmonics).
5. Choose the file name
6. Click on **OK** to export the forces harmonics and create the data export entity:

The data export entity is created and the forces harmonics exported in a .fem file:



NVH analysis in OptiStruct with data setting in HyperMesh

Introduction

This section explains the following points:

- How to make the physical data setting in HyperMesh
- How to run a frequency response analysis in OptiStruct
- How to make the post-processing in HyperView

Data setting in HyperMesh for OptiStruct analysis

There are two possibilities to define the simulation data and settings:

- By manually modifying the files (mesh file, forces file) to import it directly into OptiStruct by using the **include** technology, without using HyperMesh
- By doing the data setting in HyperMesh

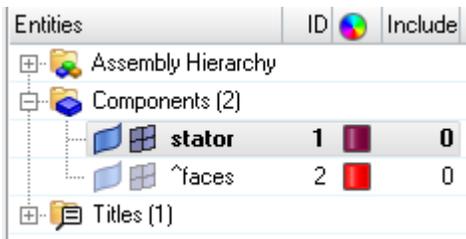
In this document, the following parts describe the second way which is more dedicated for beginner users.

The OptiStruct analysis which is done here is a frequency response analysis.

At the end, the post-processing in HyperView is presented.

1. Rename the component **faces** to **^faces** and turn it to hidden mode (to ignore it in the following steps).

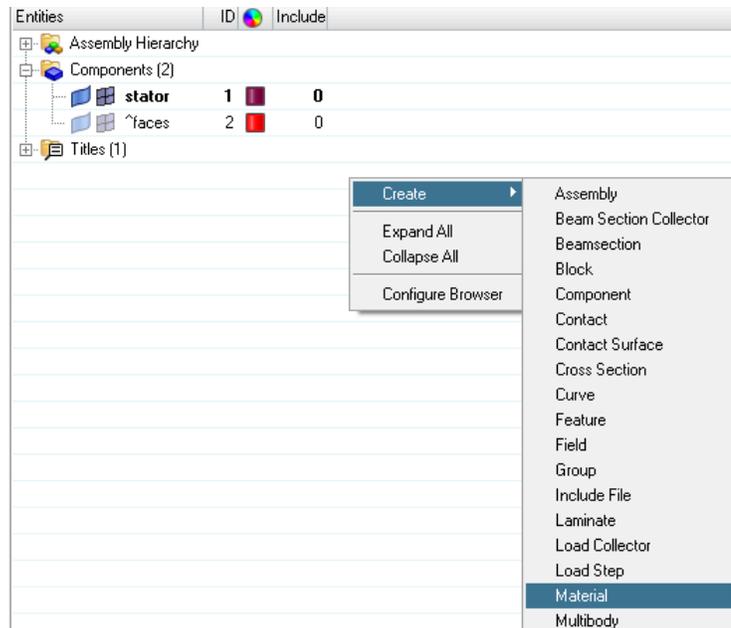
Make **stator** component visible.



2. Create the associated material to stator component.

Here **steel** material is created:

- On the model area: right click on the blank space / **Create > Material**



- The following 3 mechanical properties are defined for isotropic material (E=Young modulus, Nu=Poisson Ratio, Rho=density):

Name	Value
Include File	[Master Model]
Defined	<input checked="" type="checkbox"/>
Card Image	MAT1
User Comments	Hide In Menu/Export
E	210000000000
G	
NU	0.3
RHO	7850.0
A	
TREF	
GE	
ST	
SC	
SS	
MATS1	<input type="checkbox"/>

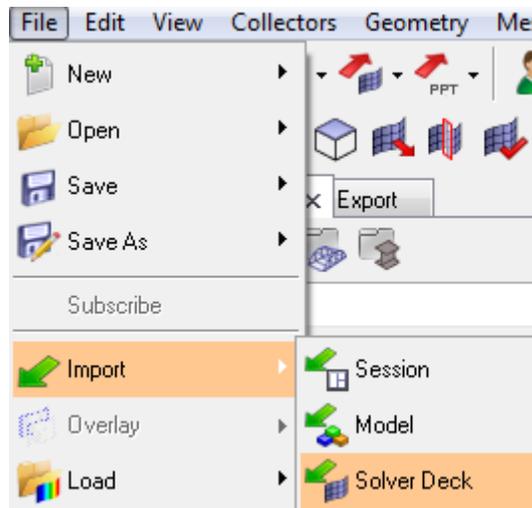
- Create a property to support 3D elements with a **PSOLID** card image and define the material as steel. Assign it to the component:
 - As for the material: right click in the model area / **Create** > **Property**.
 - Choose the following data:

Name	Value
Solver Keyword	PSOLID
Name	property1
ID	1
Color	
Include File	[Master Model]
Defined	<input checked="" type="checkbox"/>
Card Image	PSOLID
Material	steel (1)
User Comments	Hide In Menu/Export
CORDM options	BLANK
ISOP	
FCTN	
PSOLIDX	<input type="checkbox"/>

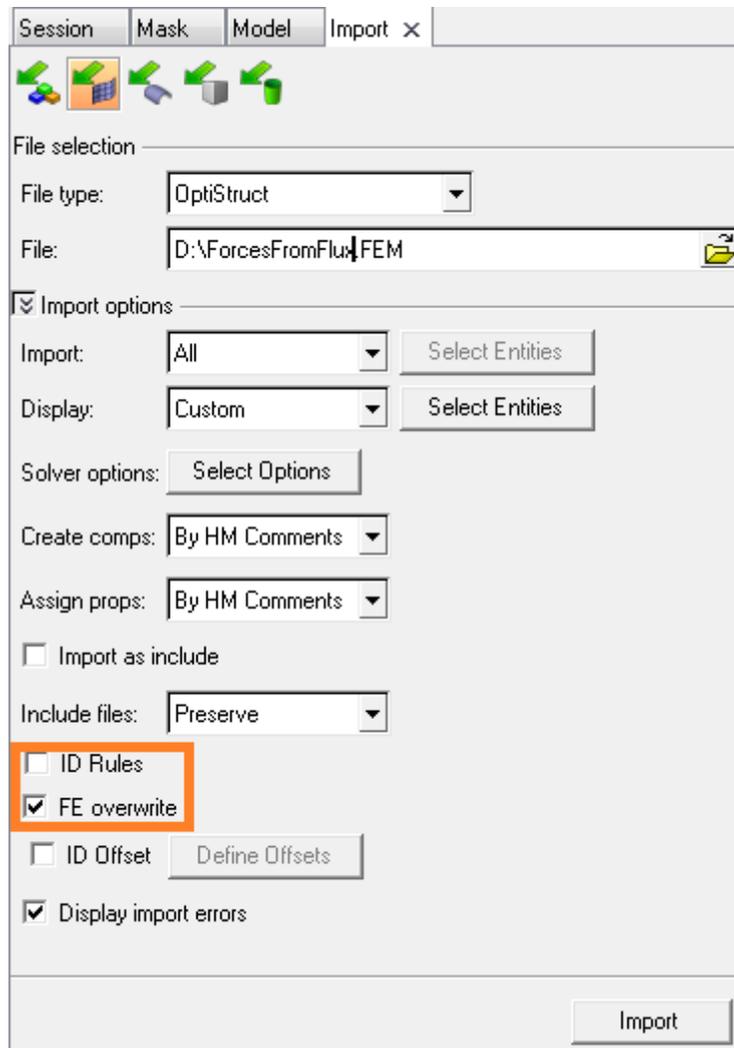
- Assign the property to the **stator** component: click on **stator** and choose **property 1** in the window at the bottom

4. Import forces file:

- Import **solver deck**:



Verify the two following options to avoid the IDs re-numbering:



All loads created in Flux are imported in HyperMesh

5. Create the Load step which defines the OptiStruct analysis parameters:

- As for material and property, right click on Model area / **Create > Load step**
- Rename the load step to **Freq_resp** for example



- Choose **Analysis type: Freq. resp (modal)**
- Choose **DLOAD** (the selection window opening can take a while):
 - **Global dynamic load case** or **First component dynamic load case** or **Second component dynamic load case**
- Create a new **METHOD(STRUCT)** by right click / **Create** on the corresponding field

Name	Value
Solver Keyword	SUBCASE
Name	Freq_resp
ID	1
Include File	[Master Model]
User Comments	Hide In Menu
Subcase Definition	
Analysis type	Freq. resp (modal)
SPC	<Unspecified>
SUPPORT1	<Unspecified>
DLOAD	Global case (2)
MPC	<Unspecified>
METHOD (STRUCT)	<Unspecified>

Create

Edit

Show

Hide

Isolate Only

XRef entities

Filter entities

Warn upon entity type change

- Choose the **Card Image: EIGRA** which allows performing eigenvalue analysis
- Choose **V2** value which corresponds to the maximum frequency to study. In general, the chosen value is 1.5 times the maximum frequency exported from Flux

Create Loadcols

Name	Value
Solver Keyword	EIGRA
Name	loadcol1
ID	80209073
Color	
Include File	[Master Model]
Card Image	EIGRA
User Comments	Hide In Menu/Export
V1	0.0
V2	4500.0
ND	
MSGLVL	
AMPFAC	
NORM	MASS

Close

- Choose **FREQ** (the selection window opening can take a while):
 - Choose **List of frequencies**
- It is advised to modify **RESVEC** parameter to **No** in order to limit the used memory and the solving time:

RESVEC	<input checked="" type="checkbox"/>
TYPE	UNITLOD
DAMPING	
VISC_OPTIONS	blank
OPTION	NO

Note: keeping the option to **YES** is more accurate but the computation time becomes non-industrial. Please refer to the residual vectors computation help section for more information

- Choose the **output** to display after solving. For example: the **DISPLACEMENT**, the **STRESS**, the imported forces which corresponds to **OLOAD** keyword, etc.

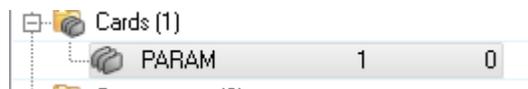
6. Damping Creation:

If you want to introduce global damping you can use the **PARAM,G** parameter:

The **PARAM** card can be access by the menu:

Tools > Create cards > p > Param

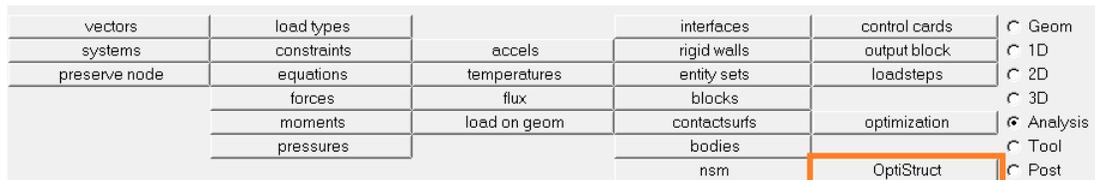
PARAM card appears in the model datatree:



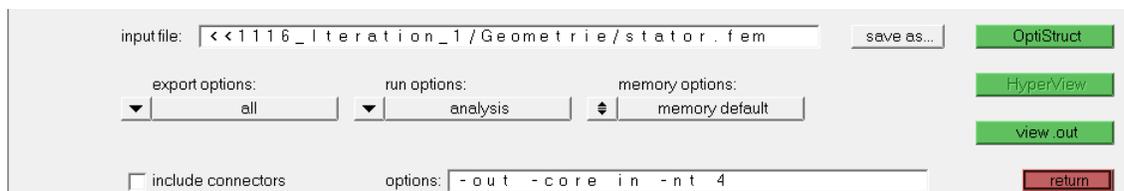
- Choose **G** which corresponds to the critical damping ration multiplied by 2. Here we choose $G=0.04$

7. Solve the OptiStruct problem:

- Open the **Analysis** panel below the graphic and click on **OptiStruct**



The following window is opened:



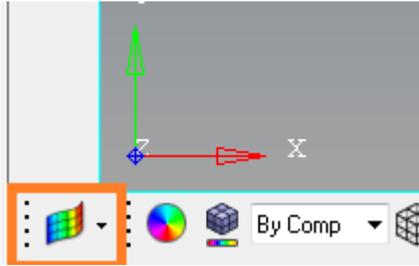
Note: All the set-up model will be saved in the file **.fem** specified in the **input file**

- The **export options** must be set to **all**
- The **run option** must be set to **analysis**
- In the **options**, enter **-out -core in -nt 4** (the spaces must be respected). Please refer to help section for solver options definition
- Click on OptiStruct to solve the problem

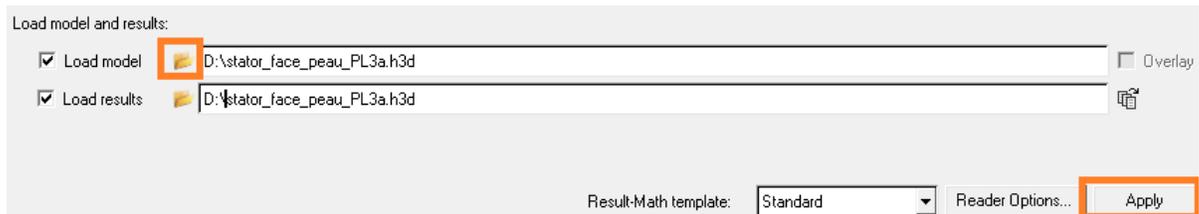
A window is opened after a short time. It displays the solving steps and the error messages if the problem can't be solved. When the solving finishes, at the end the window displays **Job completed**. As a remark, to stop the solving, click on **abort**.

8. Open HyperView to post-process.

In HyperMesh Desktop, choose the following icon:



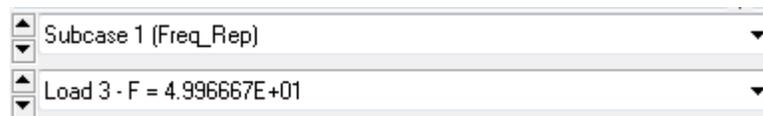
- Load the .h3d file which was automatically generated during the solving and included all the data results and click on **Apply**



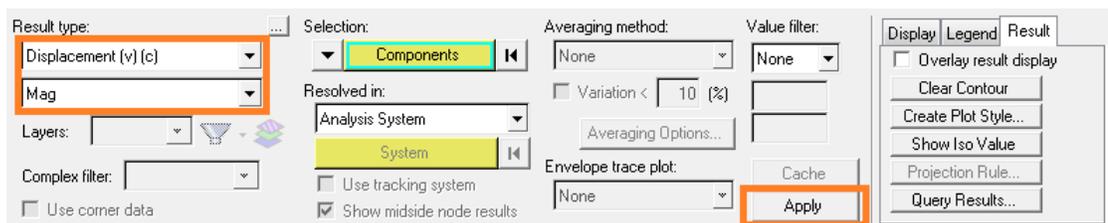
9. To post-process in HyperView, use the following bar below the graphic:



The subcase and the studied frequency can be chosen on the following fields on left side above the model tree:



- The icon  allows the user to review the contours requested:
- Select Displacement/ Magnitude to review the displacements:

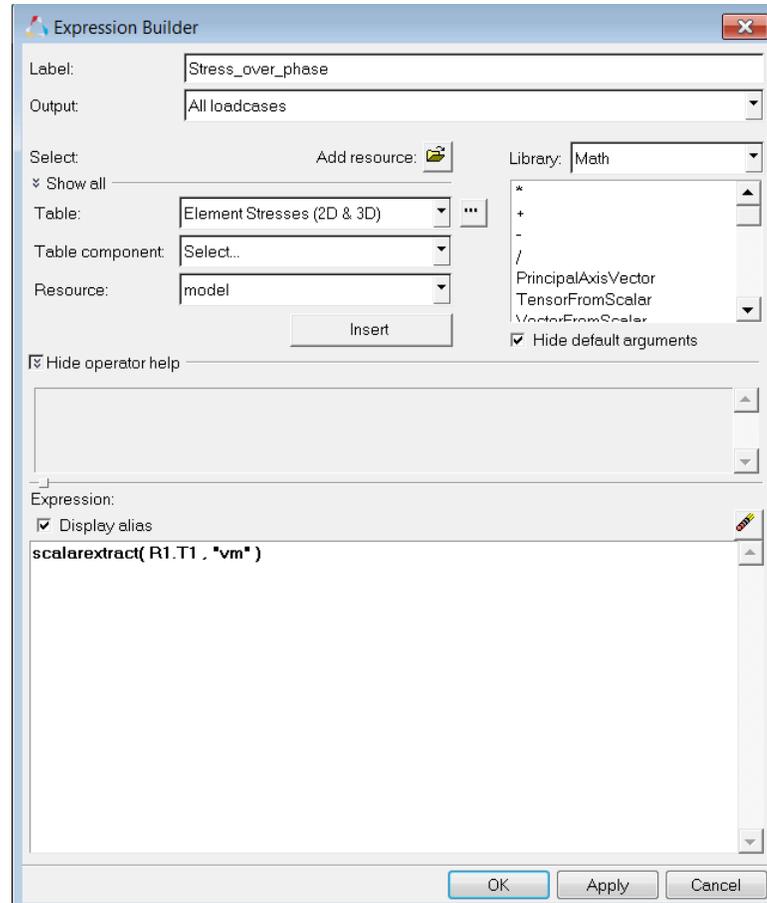


Select Stress VonMises to review the stresses computed using the invariant VonMises approach.

Please note that the stresses you are postprocessing are phase dependent.

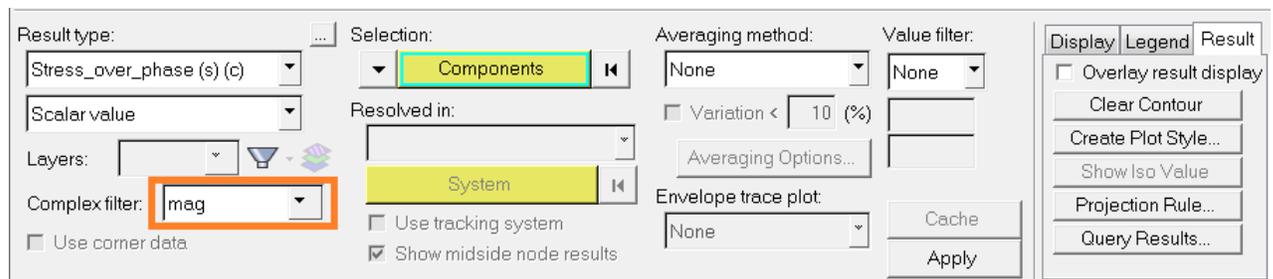
To get the maximum VonMises stress over the phase you can use the derived results tool with the operator **scalarextract**





The R1.T1 corresponds to the Stresses tensor.

To review the contour of these VonMises stress over phase please make sure to turn the complex filter to **mag**:



- Control of animation

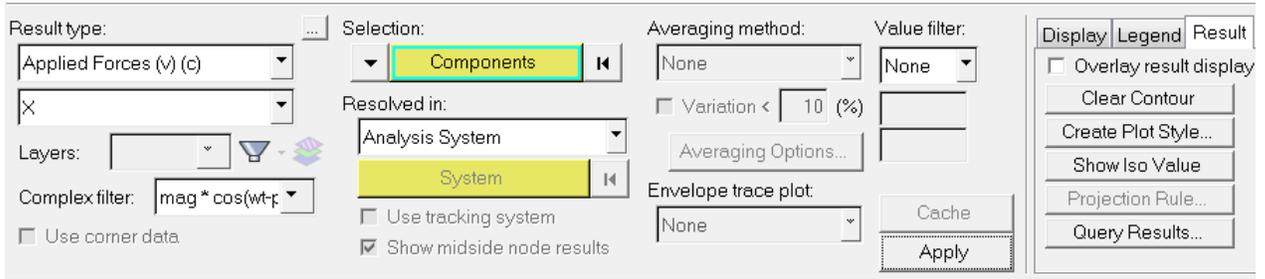


The second sliding bar allows you to slow down the animation.

The wheel icon on the right allows you to manage the animation steps thanks to the angular increment:

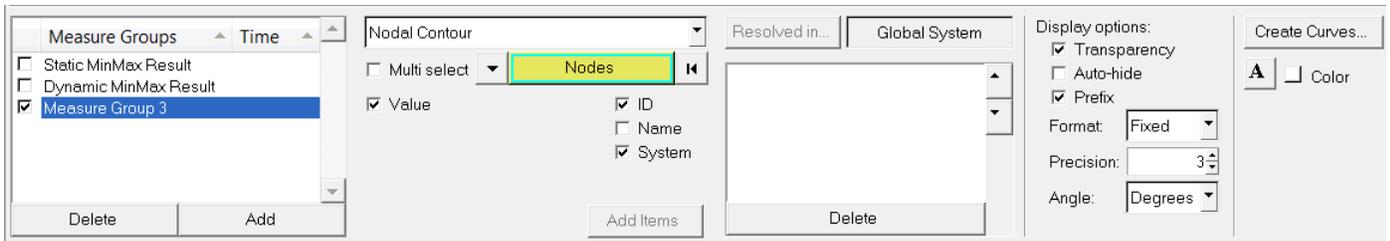


- You can also review the applied loads on 1 particular node:
 To do that, select the **Applied Loads** in the contour tool:

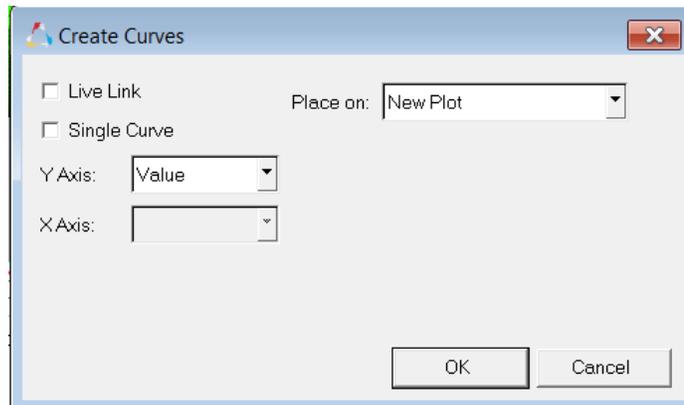


Set the angular increment to 5 degrees.
 Launch the animation over at least 1 phase.

Open the **Measures** tool 



Add a measure with the **Add** button and select the **nodal Contour** in the pulldown menu.
 Select the node of interest and click on **Create Curves**:



Click **OK** in this new popup window

A new window appears with the force vs phase graph:

