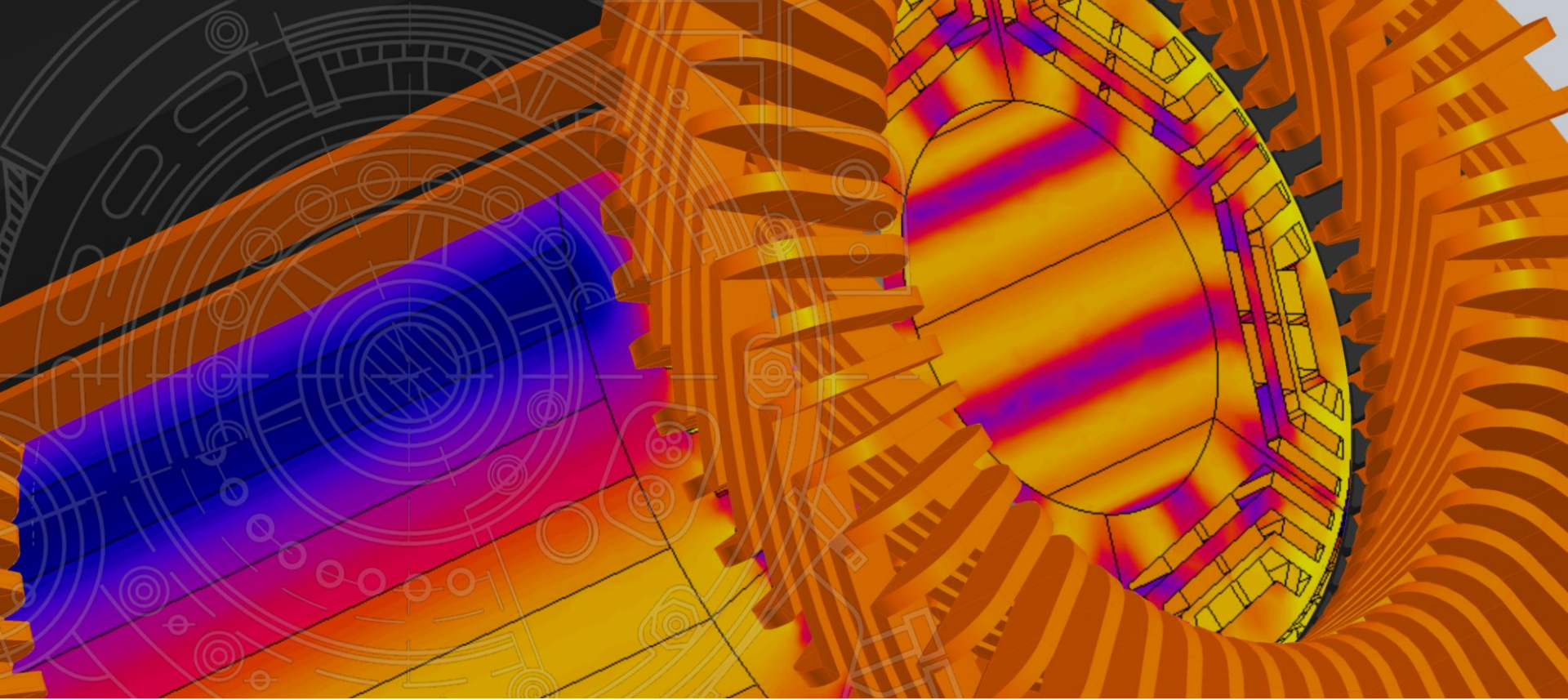


ALTAIR

APPLICATION OF MAGNETIC FORCE EXPORT PER TOOTH: VIBRATION ANALYSIS FOR ELECTRIC MOTORS AT MULTI-SPEED

FLUX 2D MULTIPHYSICS APPLICATION EXAMPLE

December 2020, Altair Valorization and Support Team

GENERAL PRESENTATION OUTLINE

- Part 1: Introduction
- Part 2: Motor pre-analysis in FluxMotor
- Part 3: Magnetic force export in Flux
- Part 4:Vibration analysis in SimLab
- Part 5: Conclusion

I. INTRODUCTION

INTRODUCTION

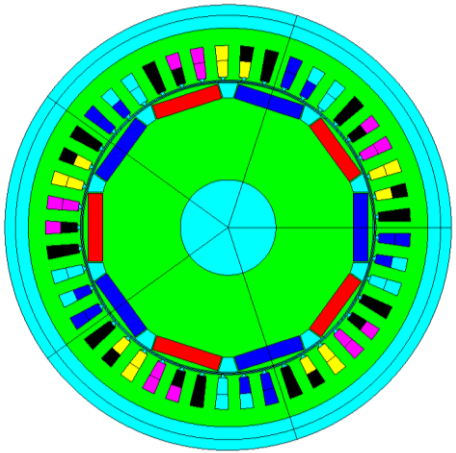
- Electric motor vibration analysis at multi-speeds: solution with Altair HyperWorks

Altair FluxMotor™	Step 1: Motor performance pre-analysis
Altair Flux™	Step 2: Electromagnetic modeling and analysis of the motor
Altair SimLab™	Step 3: Mechanical modeling of the motor
Altair OptiStruct™	Step 4: Mechanical analysis of the motor

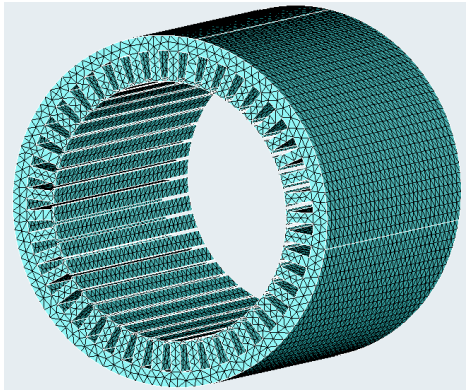
- Two excitation (magnetic forces) format are currently available from Flux to OptiStruct:
  - Magnetic force export via an imported mesh
  - Magnetic force export per tooth

INTRODUCTION

- Tutorial example
  - The studied device is an **inset permanent magnet synchronous motor (model: IkerMAQ).**
  - The comprehensive magnetic and thermal analysis of the motor can be found in the Flux 2D supervisor example / Application notes / **Magnetic and thermal analysis of a PMSM**

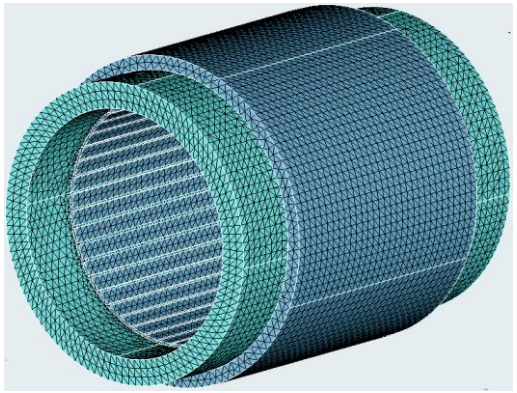


Electromagnetic model: Flux 2D project



a) Without windings

Mechanical model: SimLab projects



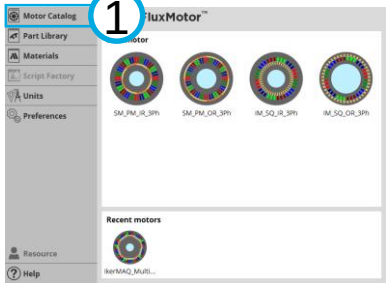
b) With windings

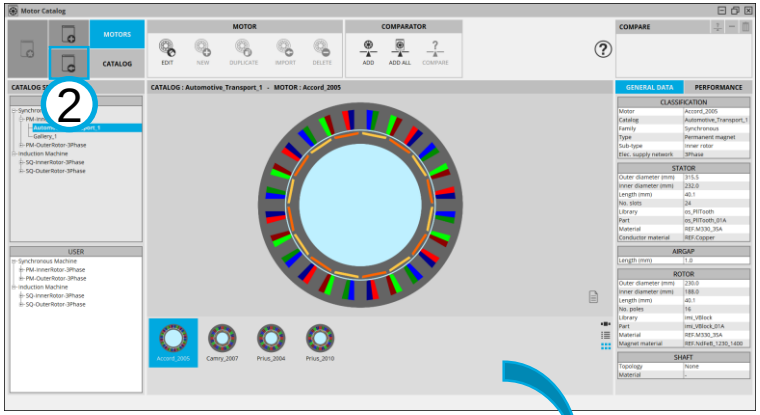
II. MOTOR PRE-ANALYSIS IN FLUXMOTOR

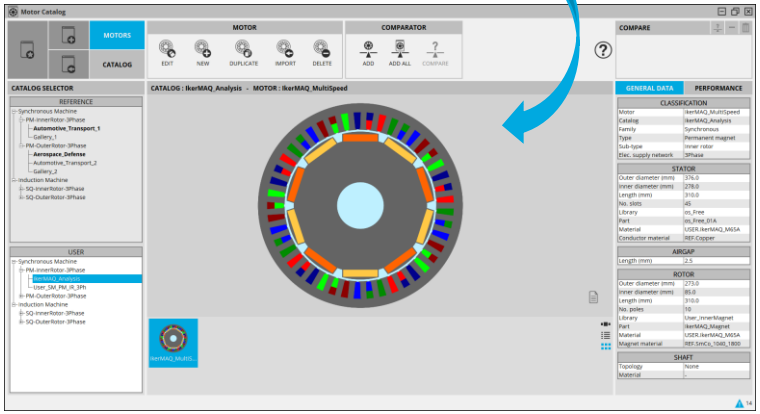
MOTOR PRE-ANALYSIS IN FLUXMOTOR

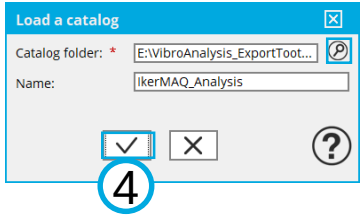
- FluxMotor project preparation
  - Import the Motor Catalog (“IkerMAQ_Analysis”)

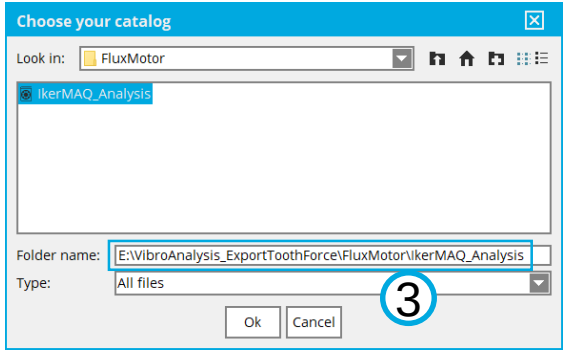
Step	Action
1	Click on [Motor Catalog]
2	Click on the icon 
3	Click on the icon  to select the provided Motor Catalog “IkerMAQ_Analysis” in the path “~/FluxMotor”, click on [OK]
4	Click on [√]









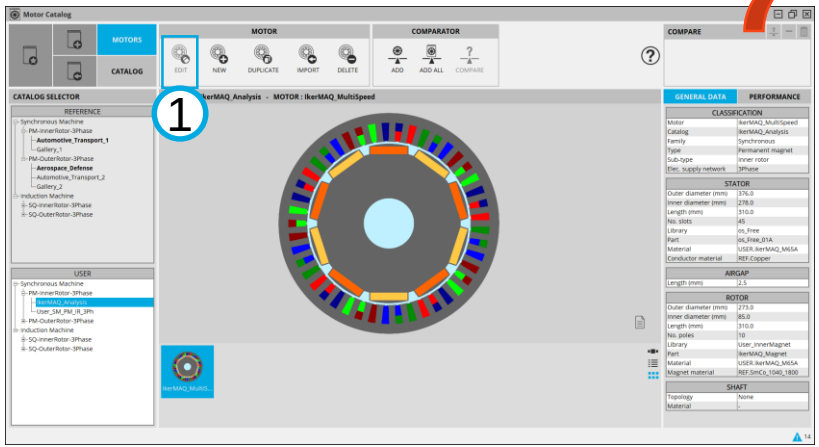


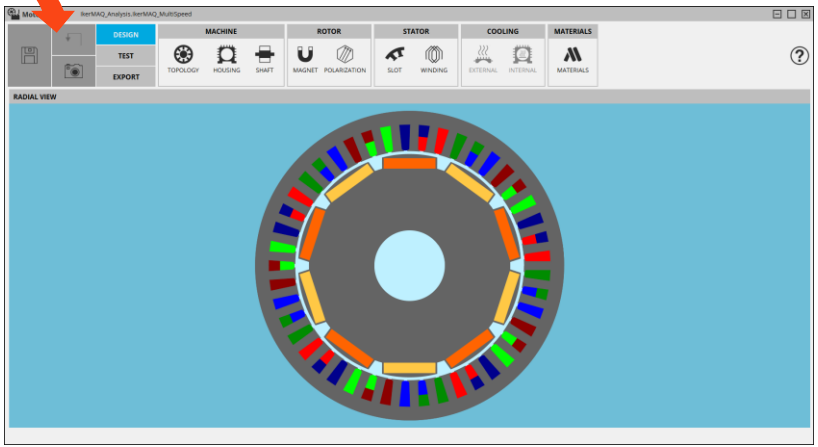
ALTAIR

MOTOR PRE-ANALYSIS IN FLUXMOTOR

- Performance mapping test creation
  - Open “Motor Factory”

Step	Action
1	Click on the icon  to open “Motor Factory” interface



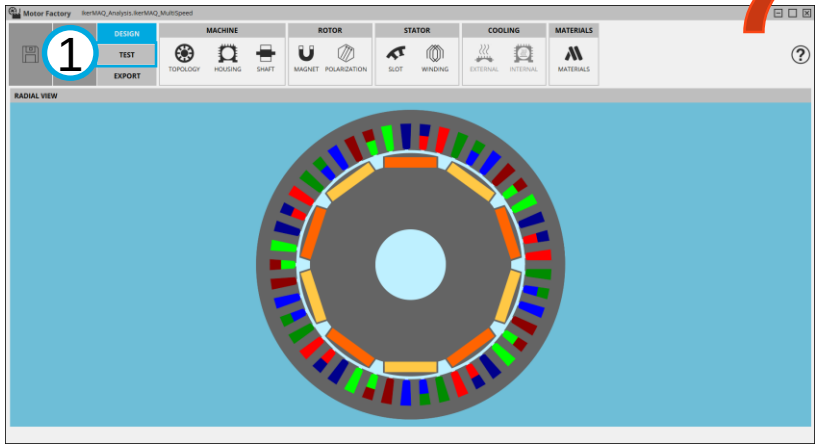


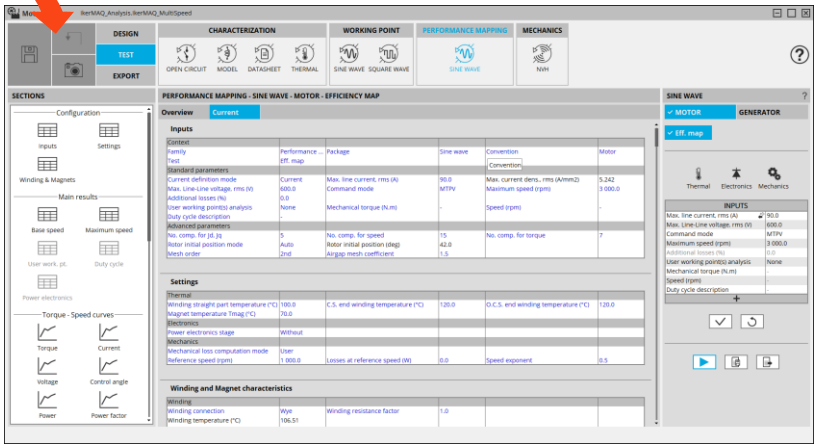
ALTAIR

MOTOR PRE-ANALYSIS IN FLUXMOTOR

- Performance mapping test
  - Access the performance mapping test results

Step	Action
1	Click on [TEST] to enter the TEST editing interface
2	Click on the icon SINE WAVE to access[1] the sinewave test results



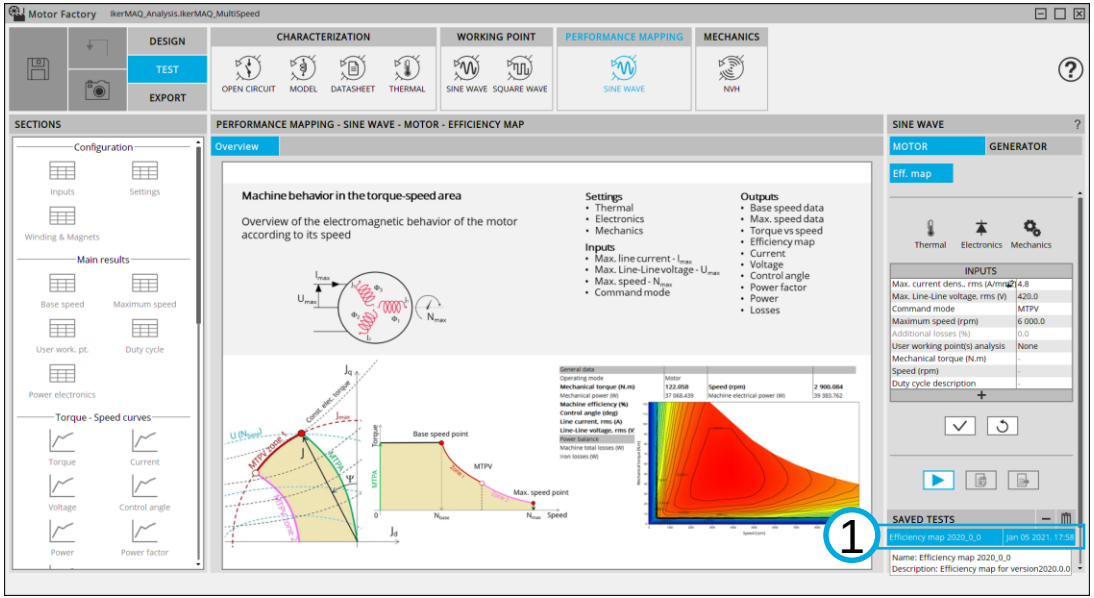


[1]The mapping test results are already available in this motor because it has been resolved.

MOTOR PRE-ANALYSIS IN FLUXMOTOR

- Performance mapping test
  - Access the performance mapping test results

Step	Action
1	Double Click on [Efficiency map 2020_0_0] to access[1] the performance mapping test results



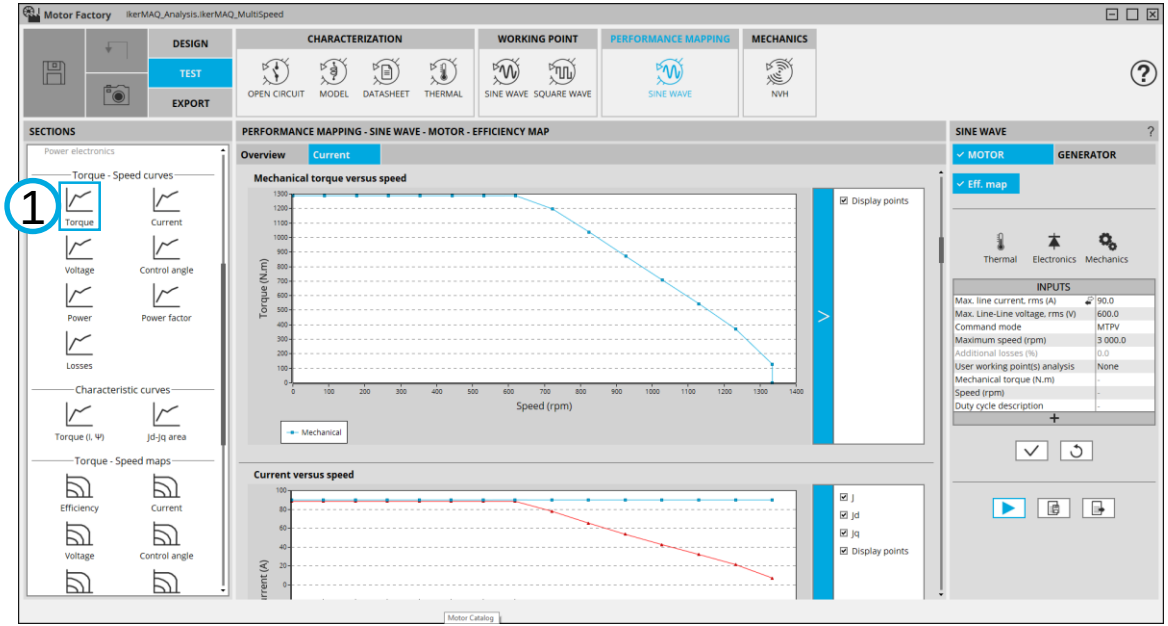
[1]The mapping test results are already available in this motor because it has been resolved.

ALTAIR

MOTOR PRE-ANALYSIS IN FLUXMOTOR

- Motor pre-analysis: sine wave
  - Mechanical torque results

Step	Action
1	In “SECTIONS”, click on [Torque] in [Torque - Speed curves] to plot the torque speed curve.



ALTAIR

MOTOR PRE-ANALYSIS IN FLUXMOTOR

- Motor pre-analysis: sine wave
  - Current results

Step	Action
1	In “SECTIONS”, click on [Current] in [Current - Speed curves] to plot the current - speed curve.
2	Select serval working points for the analysis

Current / A	Speed / RPM
90.0	88.597
90.0	265.792
90.0	442.986
90.0	620.181
90.0	824.113
90.0	1028.046
90.0	1231.978

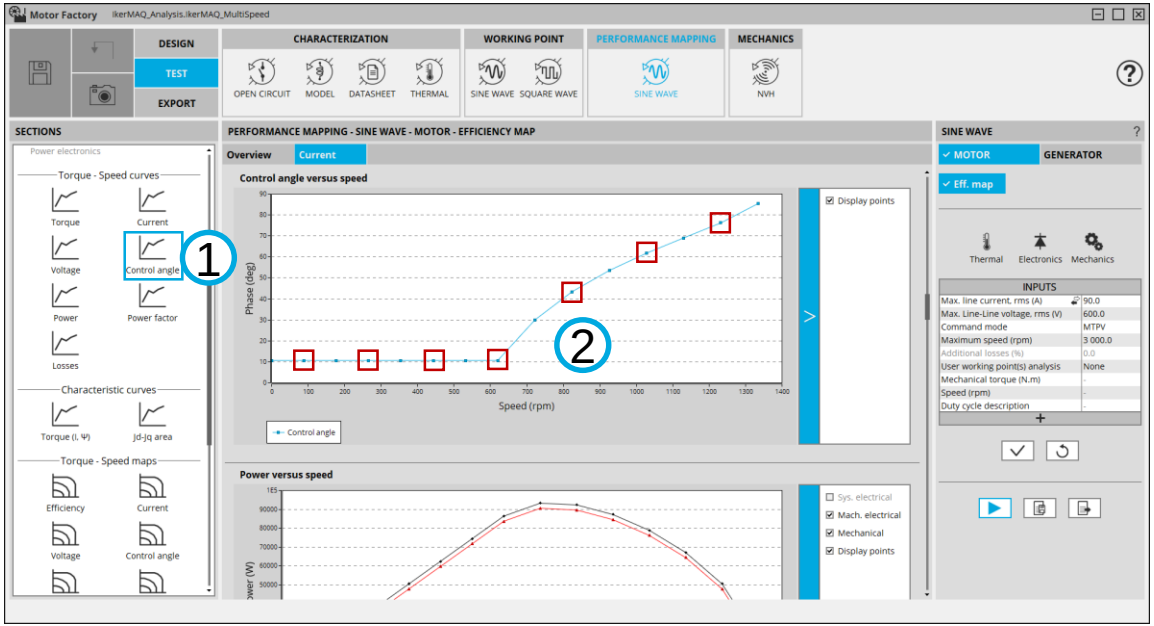
ALTAIR

MOTOR PRE-ANALYSIS IN FLUXMOTOR

- Motor pre-analysis: sine wave
  - Control angle results

Step	Action
1	In “SECTIONS”, click on [Control angle] in [Control angle - Speed curves] to plot the control angle-speed curve.
2	Select serval working points for the analysis

Control angle / deg	Speed / RPM
10.64	88.597
10.64	265.792
10.642	442.986
10.67	620.181
43.425	824.113
61.758	1028.046
76.19	1231.978

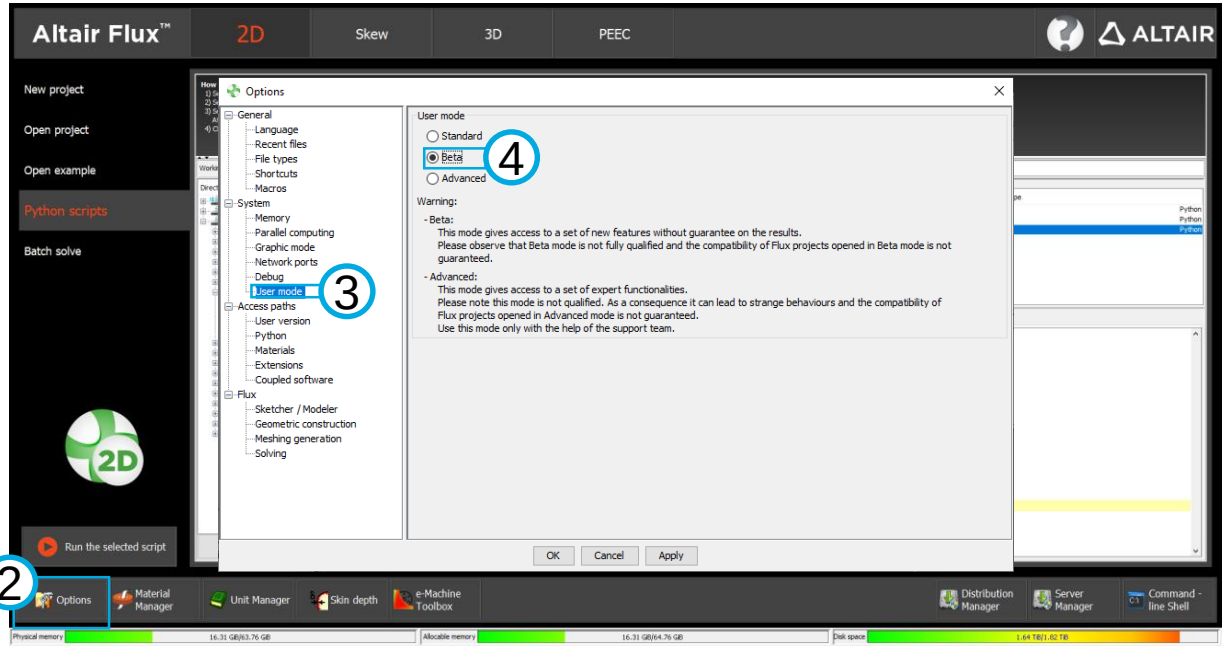


III. MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: initiation
  - Activate [Beta] mode (to access the menu for exporting global force)

Step	Action
1	Open Flux supervisor
2	Click on [Options]
3	Click on [User mode]
4	Select on [Beta], and click on [OK]

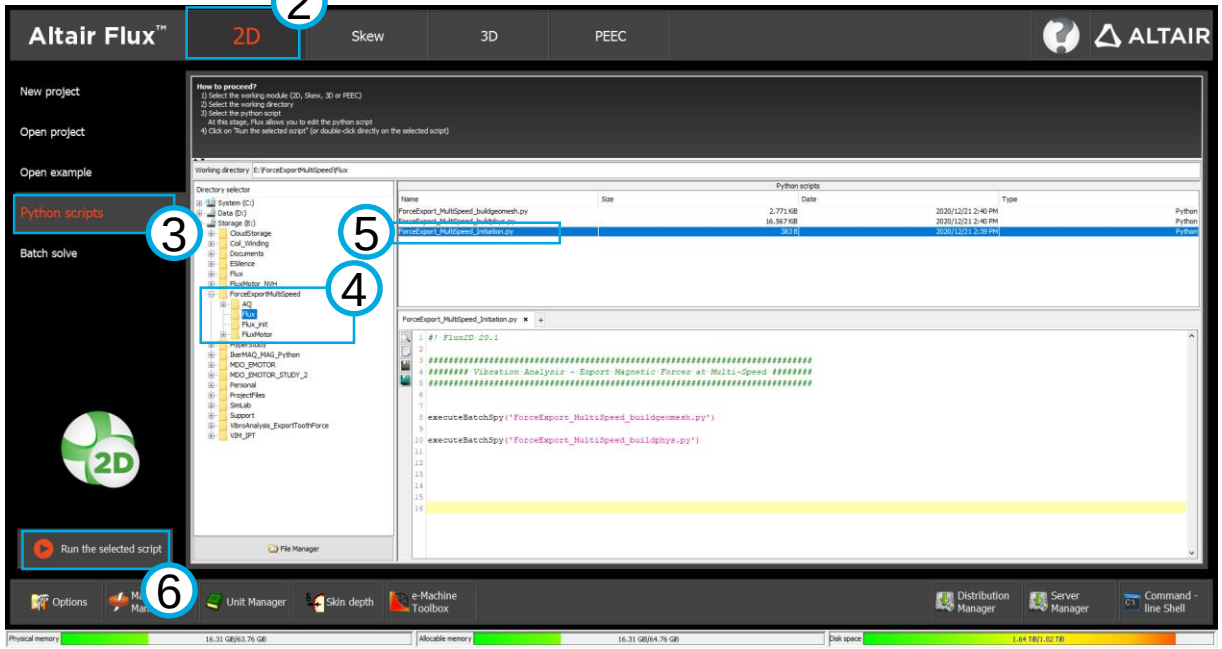


ALTAIR

FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: initiation
  - Load the physics-defined[1] Flux 2D project (driven by python script)

Step	Action
1	Open Flux supervisor
2	Select the [2D] simulation context
3	Click on [Python scripts]
4	Select the working path : “~/ForceExportMultiSpeed /Flux”
5	Select the python file “ForceExport_MultiSpeed_ Initiation.py”
6	Click on [Run the selected script]

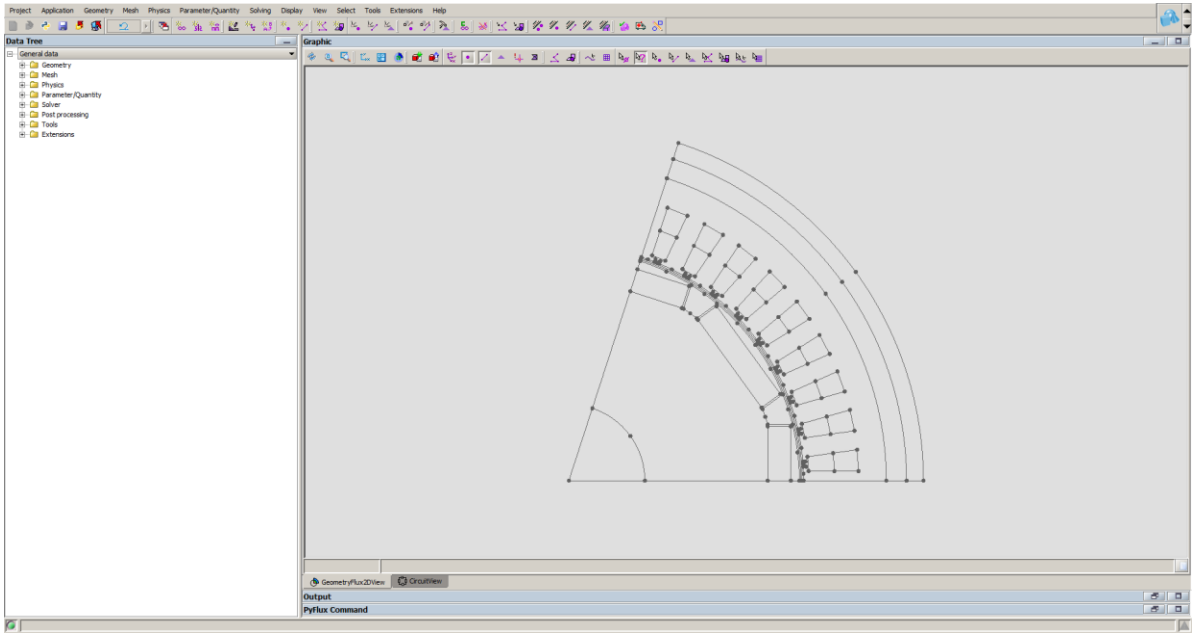


[1]The machine in the Flux project is set as generator mode by default.

ALTAIR

FLUX PROJECT PREPARATION (MULTI-SPEEDS)

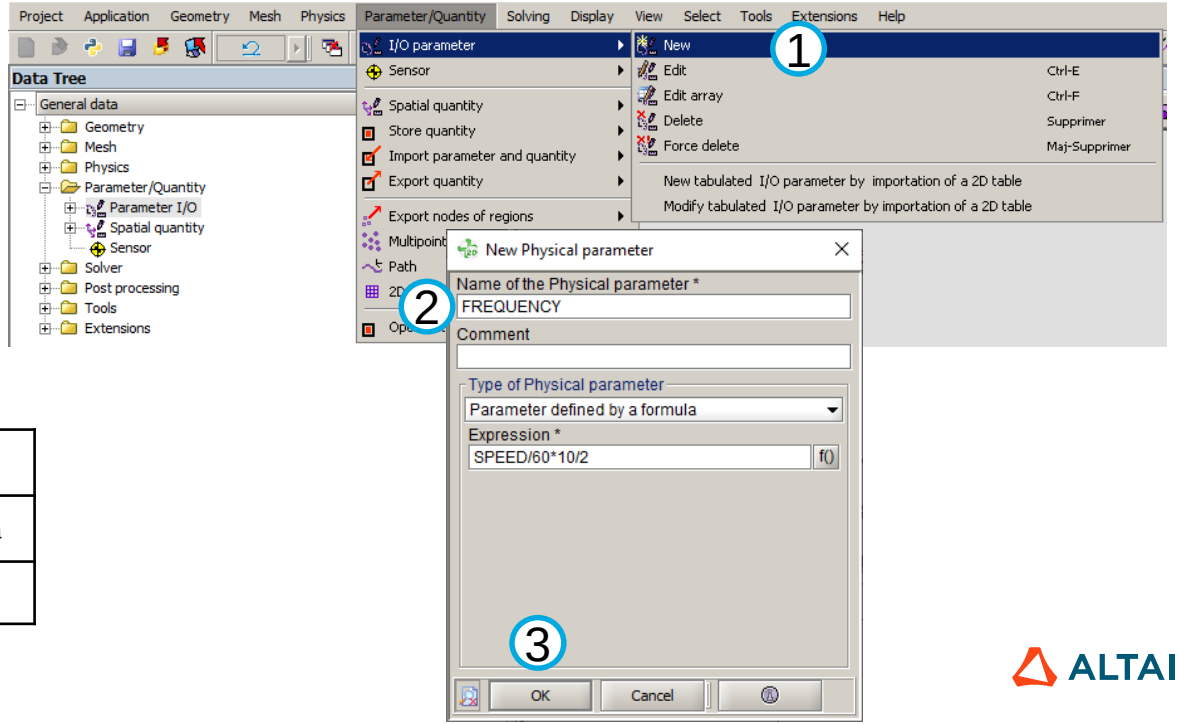
- Flux 2D project: initiation
  - Load the physics-defined Flux 2D project (driven by python script)



FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: physics modification
  - Create physics parameters: FREQUENCY

Step	Action
1	Click on [Parameter/Quantity] – [I/O parameter] – [New]
2	Define the parameter “FREQUENCY”
3	Click on [OK]



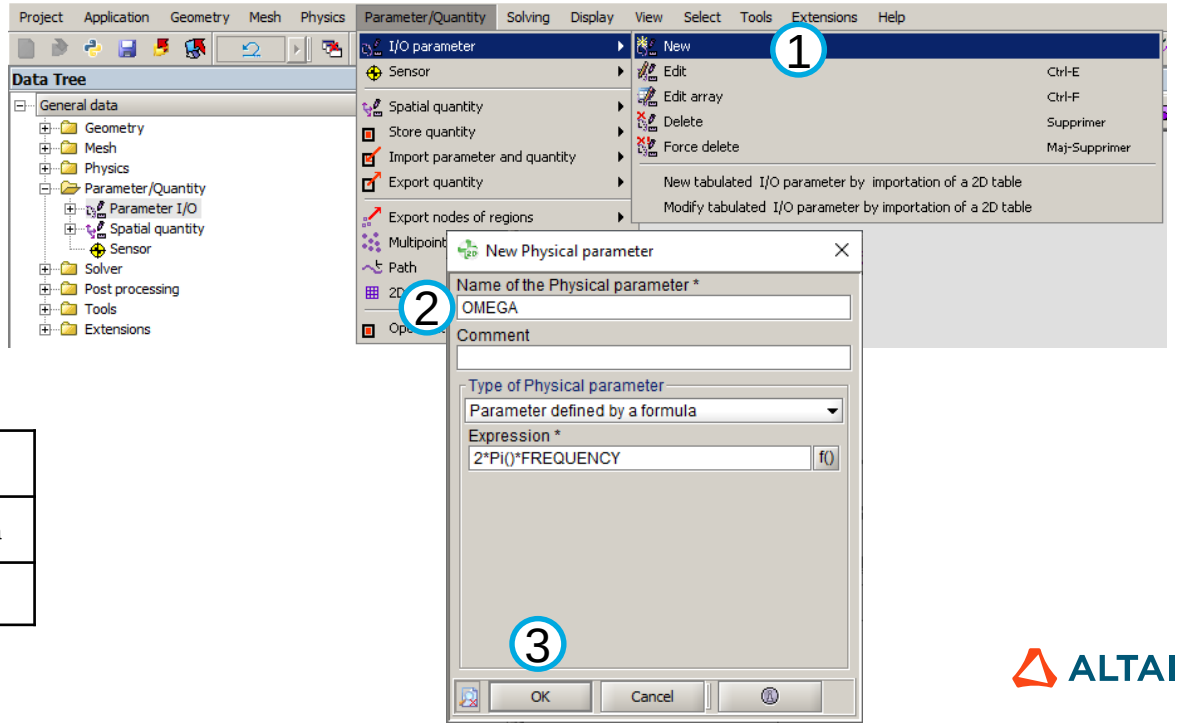
Parameter name	FREQUENCY
Parameter type	Parameter defined by a formula
Expression	SPEED/60*10/2

ALTAIR

FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: physics modification
  - Create physics parameters: OMEGA

Step	Action
1	Click on [Parameter/Quantity] – [I/O parameter] – [New]
2	Define the parameter “OMEGA”
3	Click on [OK]



Parameter name	OMEGA
Parameter type	Parameter defined by a formula
Expression	2*Pi()*FREQUENCY

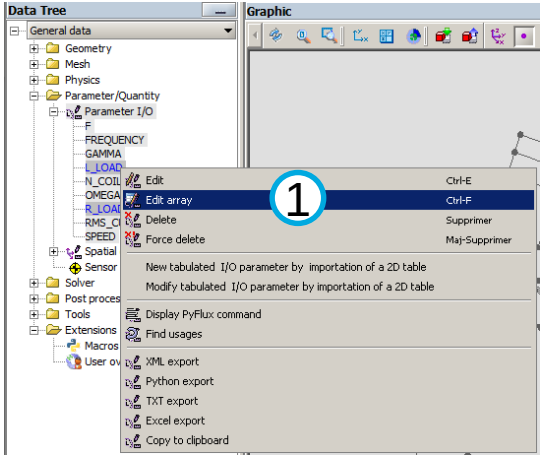
ALTAIR

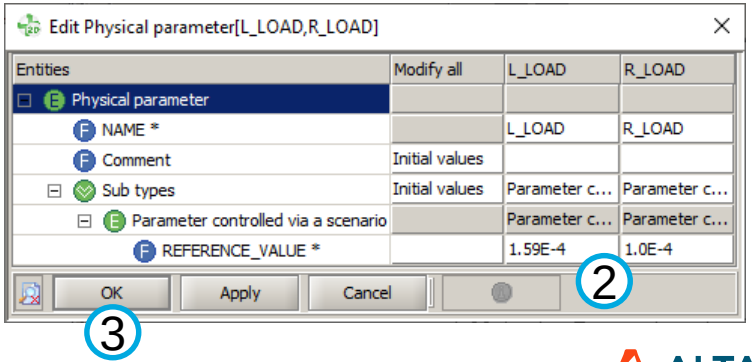
FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: physics modification
  - Modify physics parameters: L_LOAD + R_LOAD

Step	Action
1	Select from the Data Tree the two parameters “L_LOAD” and “R_LOAD”, right click and click on [Edit array]
2	Modify the values of the two parameters
3	Click on [OK]

Parameter name	Value
L_LOAD	1.59E-4
R_LOAD	1.0E-4



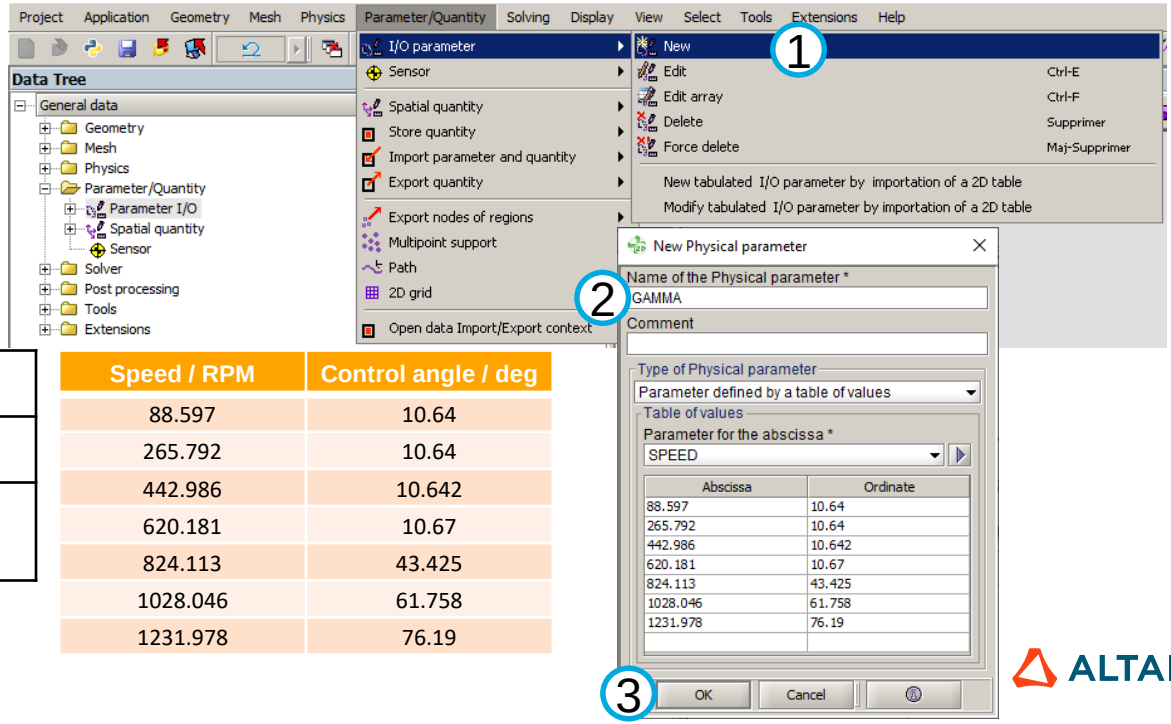


ALTAIR

FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: physics modification
  - Create physics parameters: GAMMA

Step	Action
1	Click on [Parameter/Quantity] – [I/O parameter] – [New]
2	Define the parameter “GAMMA”
3	Click on [OK]



Parameter name	GAMMA
Parameter type	Parameter defined by a table of values
Parameter for the abscissa	SPEED

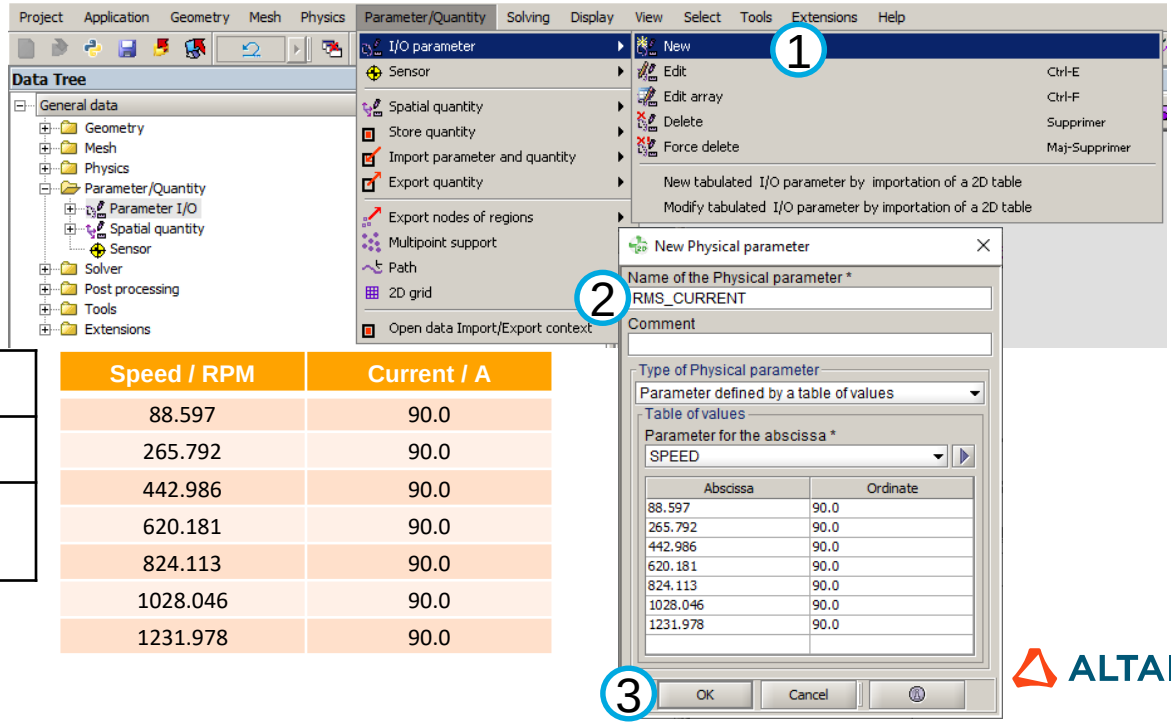
Speed / RPM	Control angle / deg
88.597	10.64
265.792	10.64
442.986	10.642
620.181	10.67
824.113	43.425
1028.046	61.758
1231.978	76.19

ALTAIR

FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: physics modification
  - Create physics parameters: RMS_CURRENT

Step	Action
1	Click on [Parameter/Quantity] – [I/O parameter] – [New]
2	Define the parameter “RMS_CURRENT”
3	Click on [OK]



Parameter name	RMS_CURRENT
Parameter type	Parameter defined by a table of values
Parameter for the abscissa	SPEED

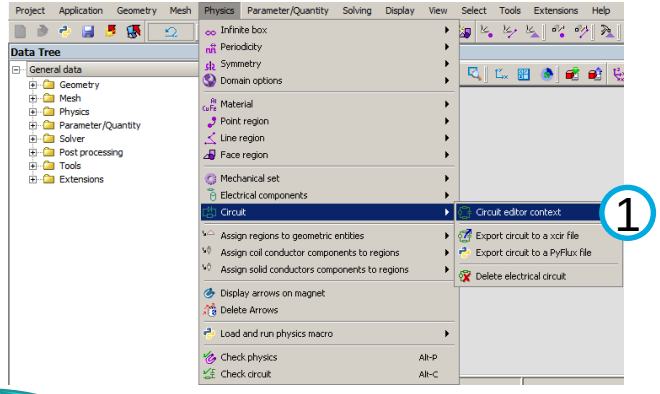
Speed / RPM	Current / A
88.597	90.0
265.792	90.0
442.986	90.0
620.181	90.0
824.113	90.0
1028.046	90.0
1231.978	90.0

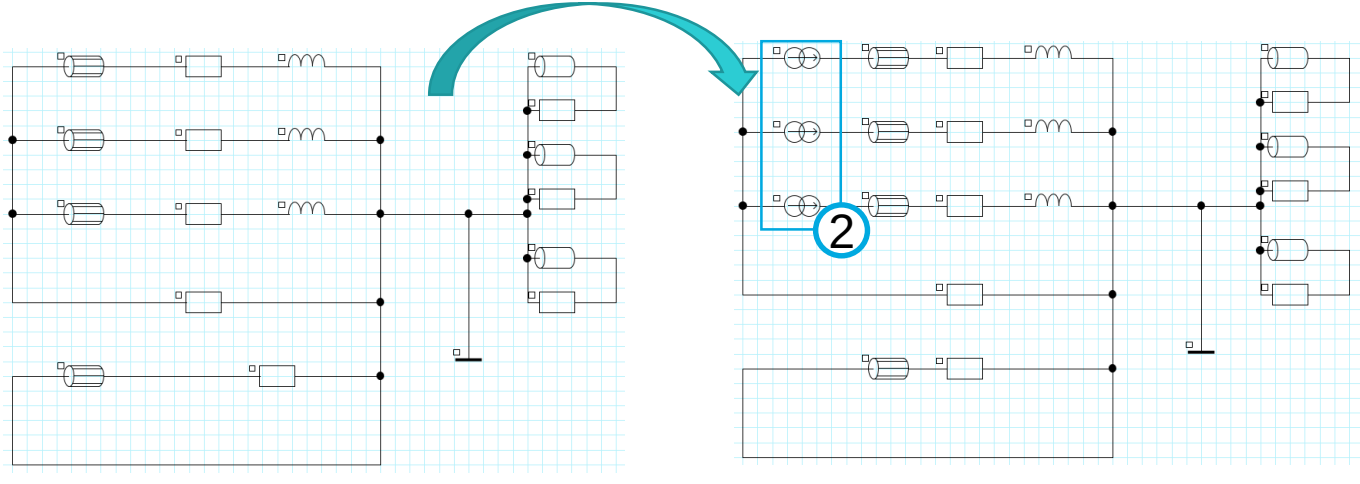
ALTAIR

FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: physics modification
  - Modify the coupling electric circuit

Step	Action
1	Click on [Physics] - [Circuit] - [Circuit editor context]
2	Add three current sources to the coupling electric circuit



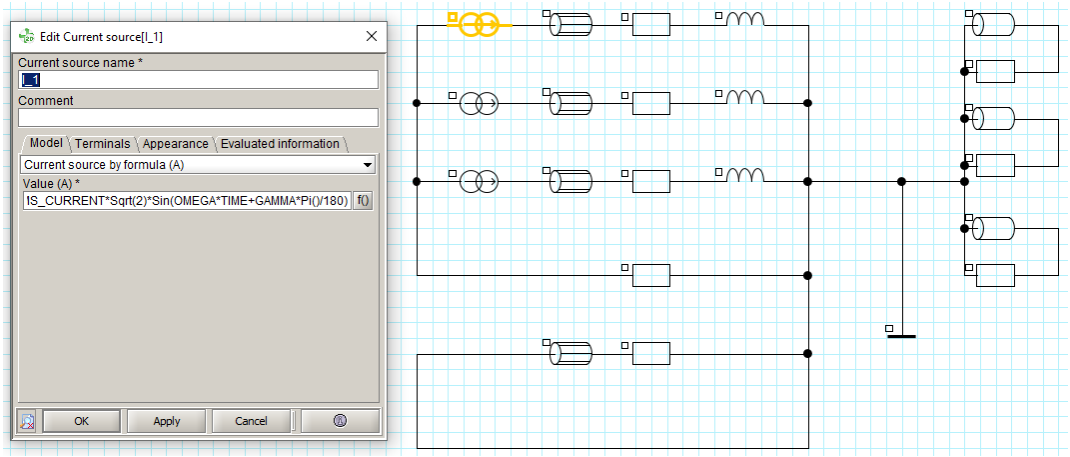


ALTAIR

FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: physics modification
  - Modify current values

Step	Action
1	Double click on each current sources, and modify the current value

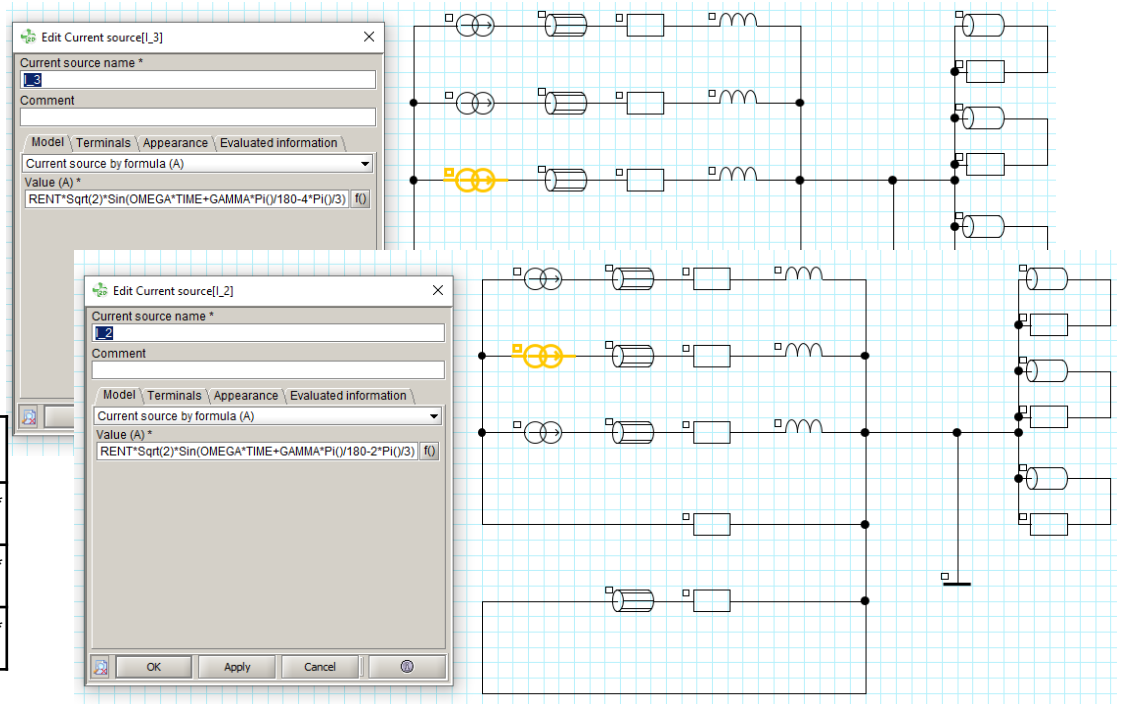


Parameter name	Value
I_1	RMS_CURRENT*Sqrt(2)*Sin(OMEGA*TIME+GAMMA*Pi()/180)
I_2	RMS_CURRENT*Sqrt(2)*Sin(OMEGA*TIME+GAMMA*Pi()/180-2*Pi()/3)
I_3	RMS_CURRENT*Sqrt(2)*Sin(OMEGA*TIME+GAMMA*Pi()/180-4*Pi()/3)

FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: physics modification
  - Modify current values

Step	Action
1	Double click on each current sources, and modify the current value

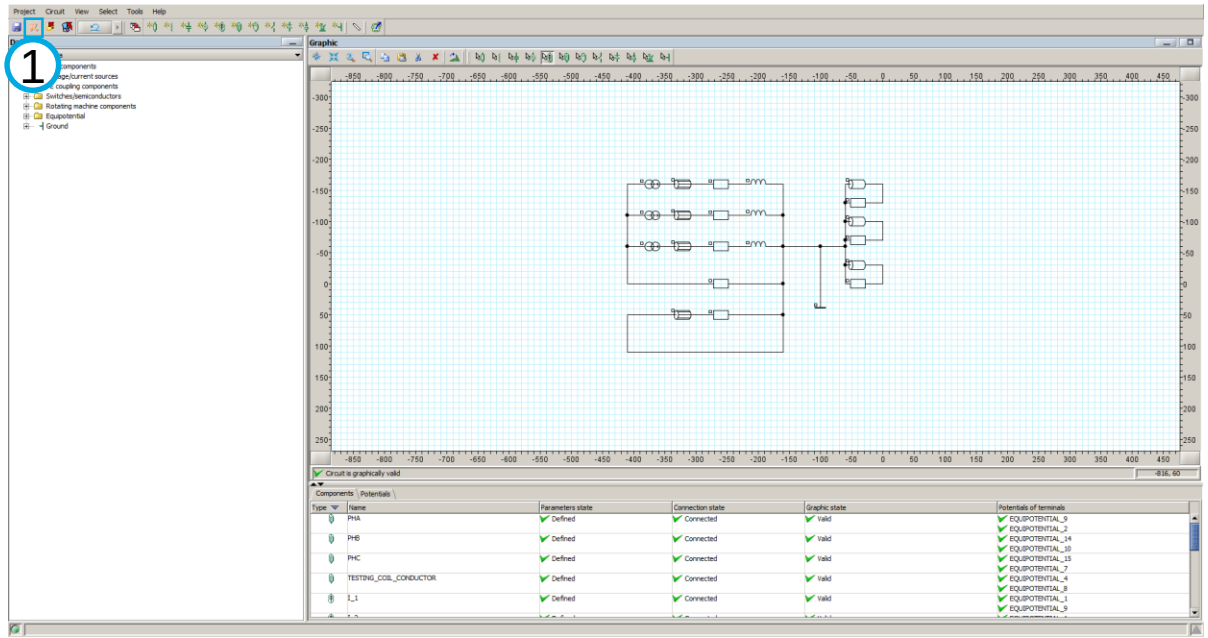


Parameter name	Value
I_1	RMS_CURRENT*Sqrt(2)*Sin(OMEGA*TIME+GAMMA*Pi()/180)
I_2	RMS_CURRENT*Sqrt(2)*Sin(OMEGA*TIME+GAMMA*Pi()/180-2*Pi()/3)
I_3	RMS_CURRENT*Sqrt(2)*Sin(OMEGA*TIME+GAMMA*Pi()/180-4*Pi()/3)

FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: physics modification
  - Close the electric circuit editor

Step	Action
1	Click on the icon to exit the circuit editor context



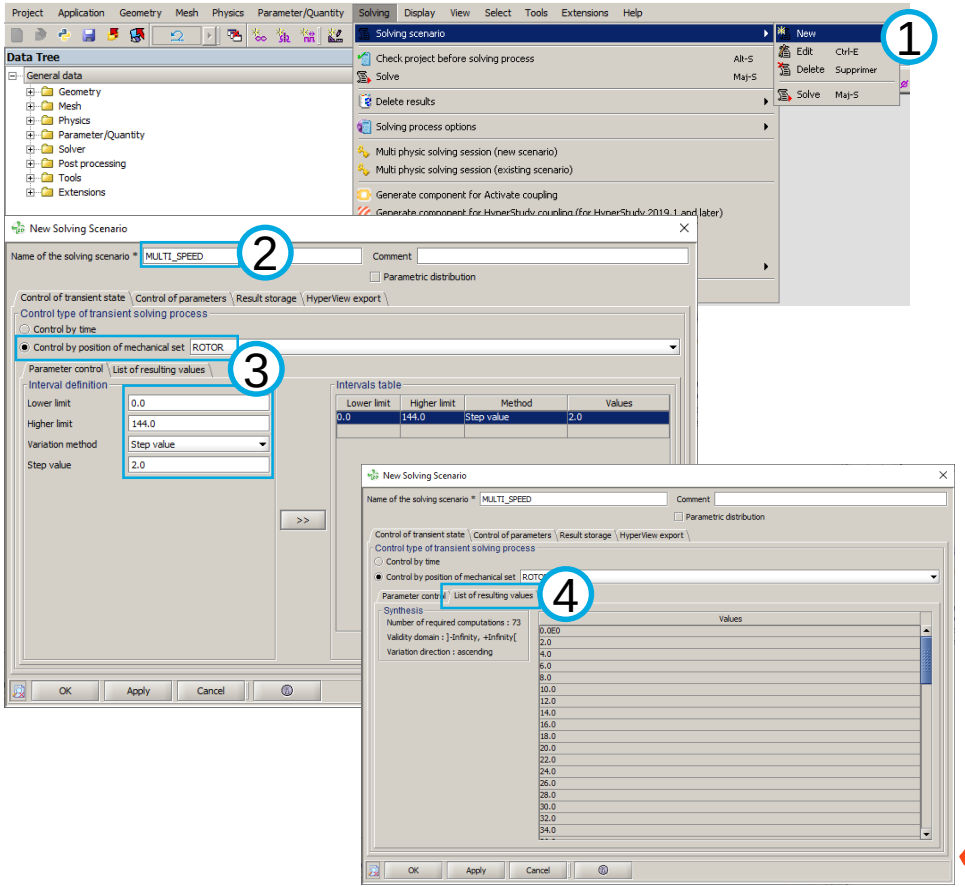
ALTAIR

FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: solving
  - Create solving scenario

Step	Action
1	Click on [Solving] – [Solving scenario] – [New]
2	Define the solving scenario “MULTI_SPEED”
3	Define the first control parameter “ROTOR”
4	Click on “List of resulting values” to verify the values

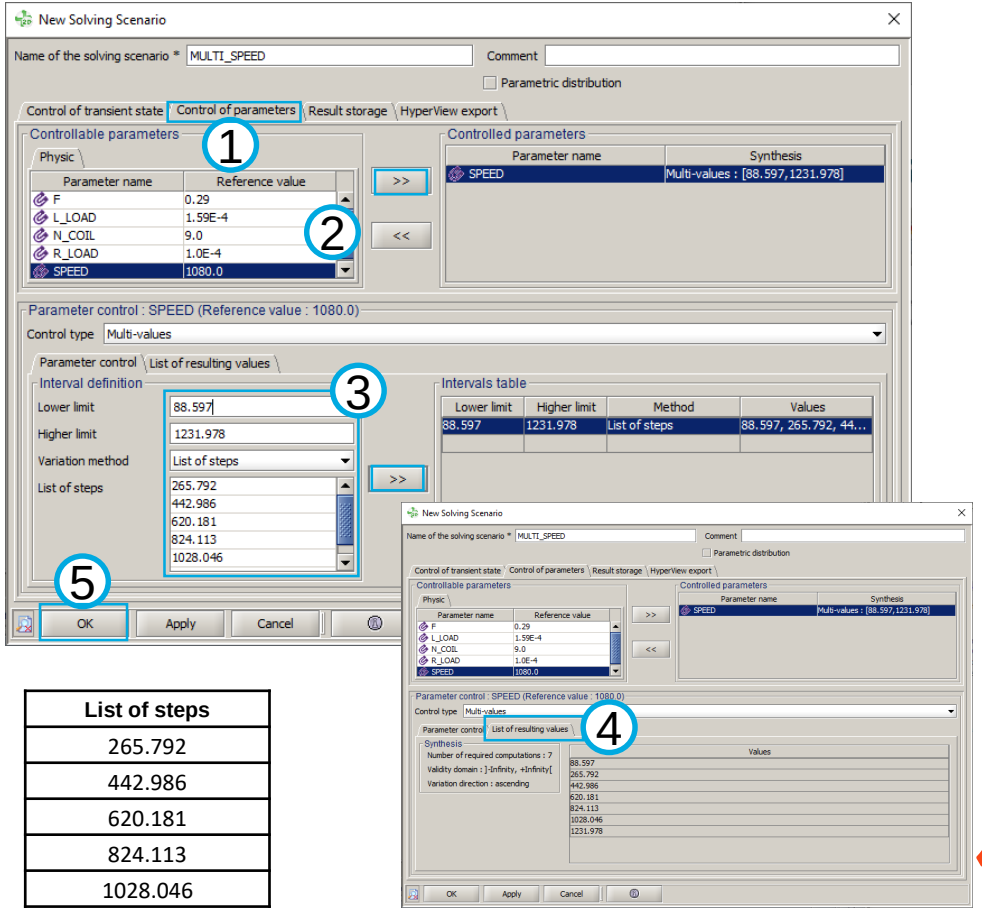
Parameter name	ROTOR
Lower limit	0.0
Higher limit	144.0
Variation method	Step value
Step value	2.0



FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: solving
  - Create solving scenario

Step	Action
1	Click on [Control of parameters]
2	Select the physic parameter “SPEED”, and click on [>>]
3	Define the second control parameter “SPEED” , and click on [>>]
4	Click on “List of resulting values” to verify the values
5	Click on [OK]



Parameter name	SPEED
Lower limit	88.597
Higher limit	1231.978
Variation method	List of steps

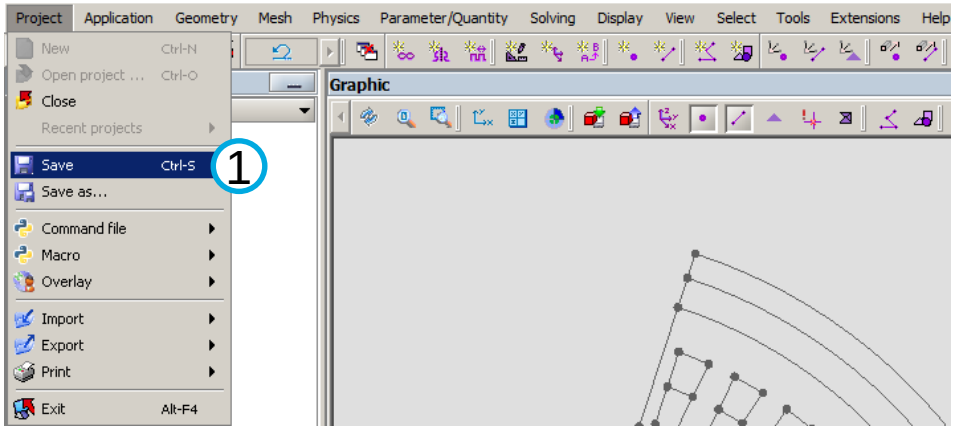
List of steps
265.792
442.986
620.181
824.113
1028.046

ALTAIR

FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: solving
  - Save the project

Step	Action
1	Click on [Project] – [Save]

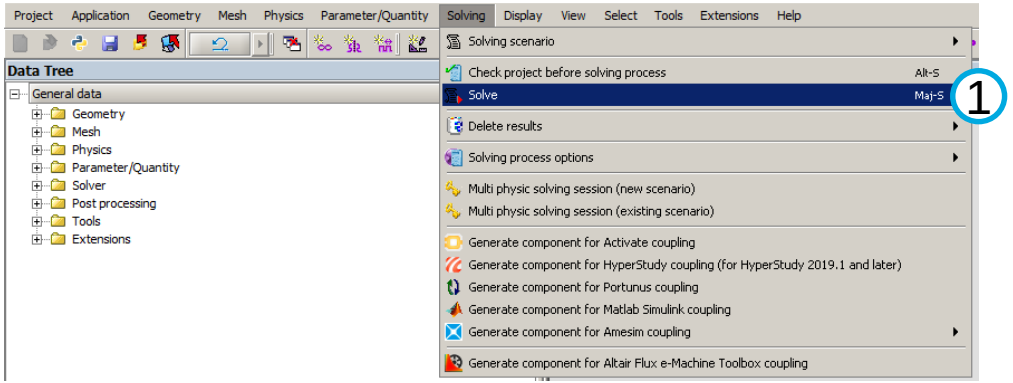


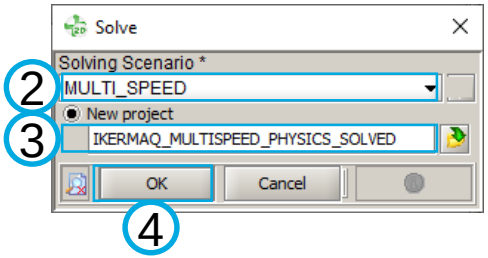
ALTAIR

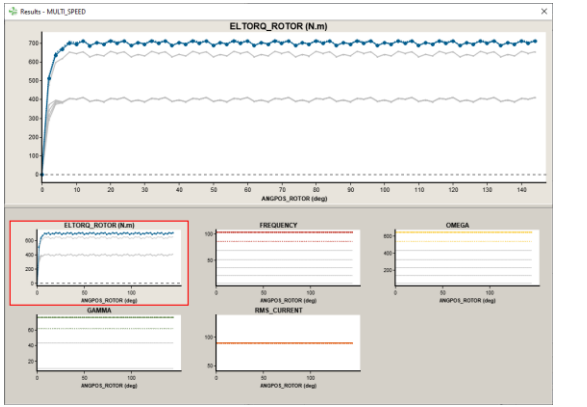
FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: solving
  - Solve the project

Step	Action
1	Click on [Solving] – [Solve]
2	Select the solving scenario “MULTI_SPEED”
3	Save the solved project as a new project “IKERMAQ_MULTISPEED_PHYSICS_SOLVED”
4	Click on [OK]





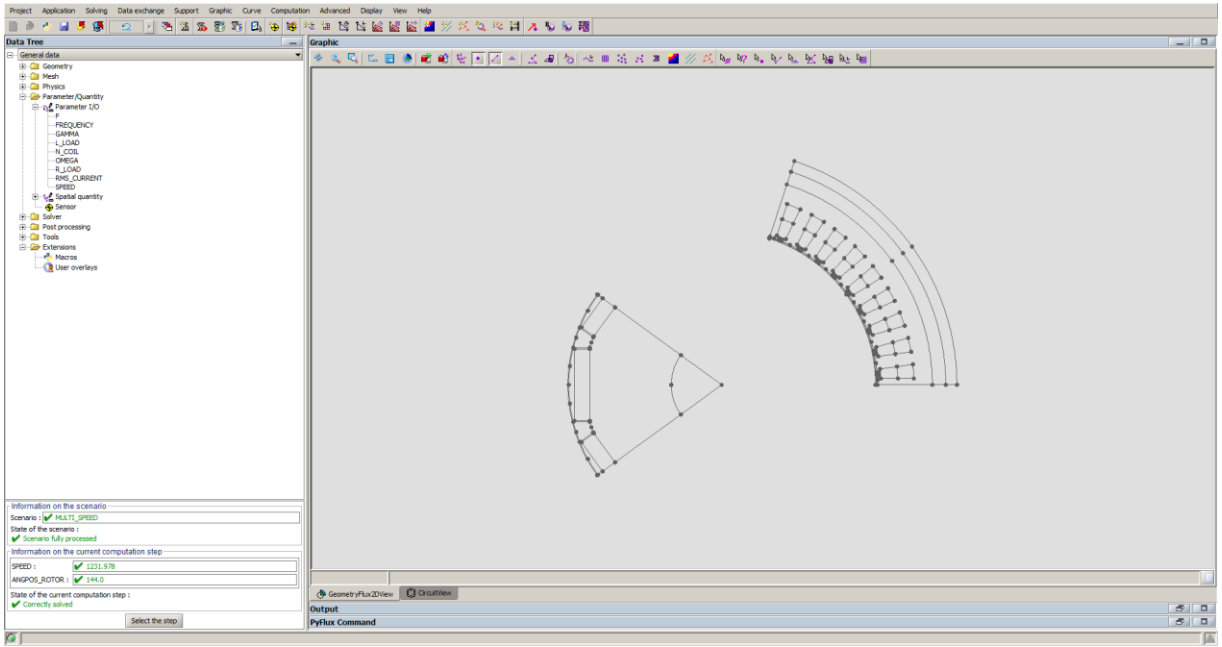


Computation time: 40 min

ALTAIR

FLUX PROJECT PREPARATION (MULTI-SPEEDS)

- Flux 2D project: solving
  - Solve the project



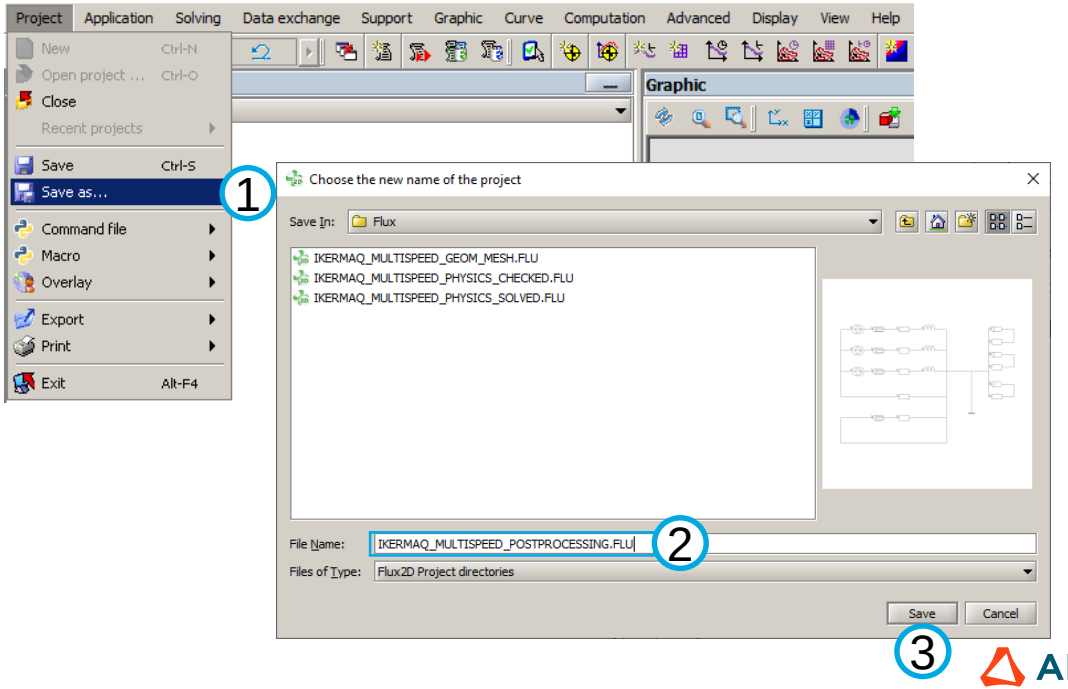
ALTAIR

MAGNETIC FORCE EXPORT (NODAL FORCES AND GLOBAL FORCES)

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Save the project

Step	Action
1	Click on [Project] – [Save as]
2	Define the name as “IKERMAQ_MULTISPEED_POSTPROCESSING.FLU”
3	Click on [Save]

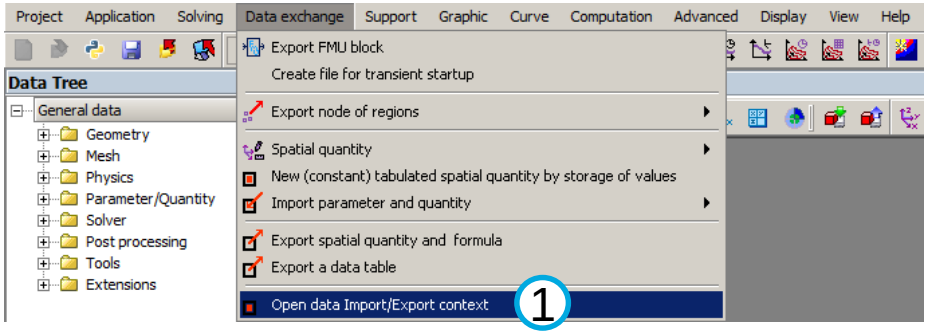


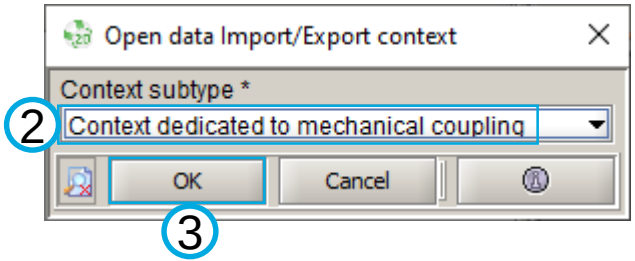
ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Open data import / export context

Step	Action
1	Click on [Data exchange] – [Open data Import / Export context]
2	Select the [Context dedicated to mechanical coupling]
3	Click on [OK]

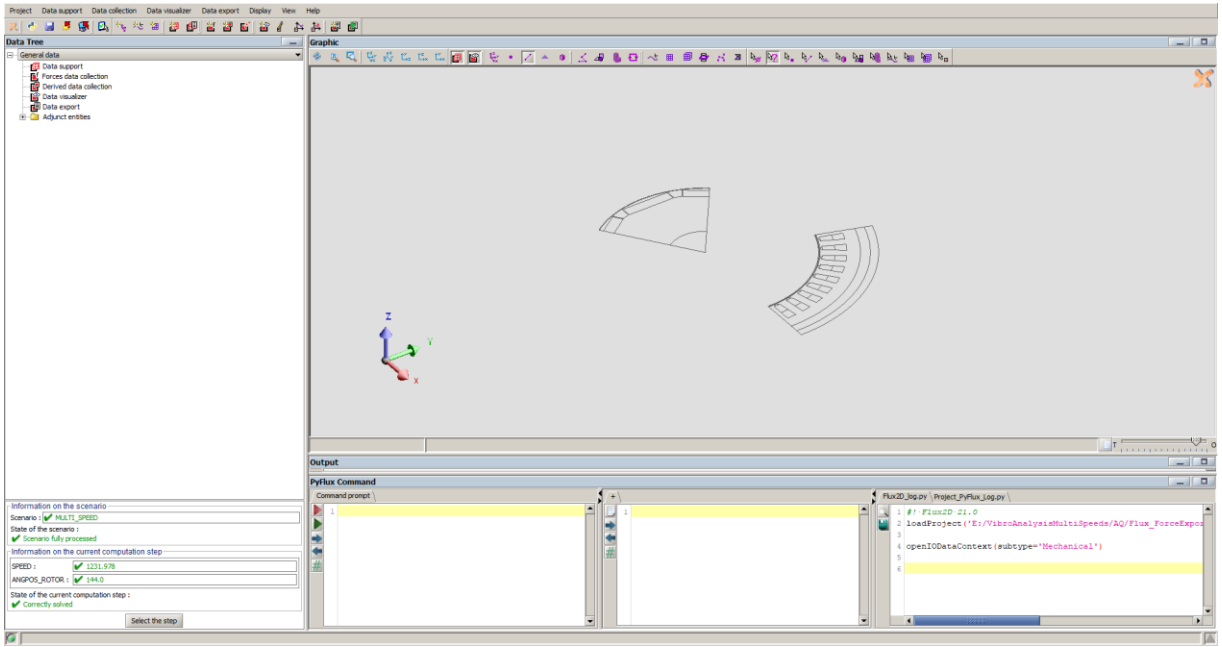




ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Open data import / export context

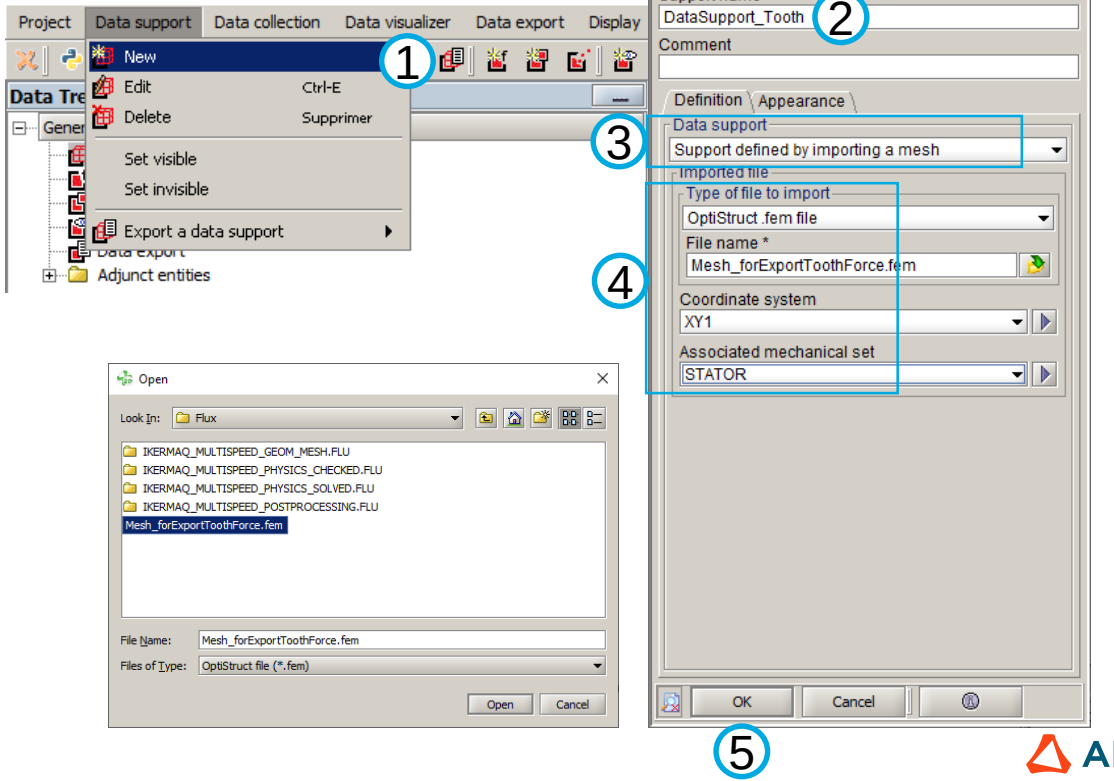


ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

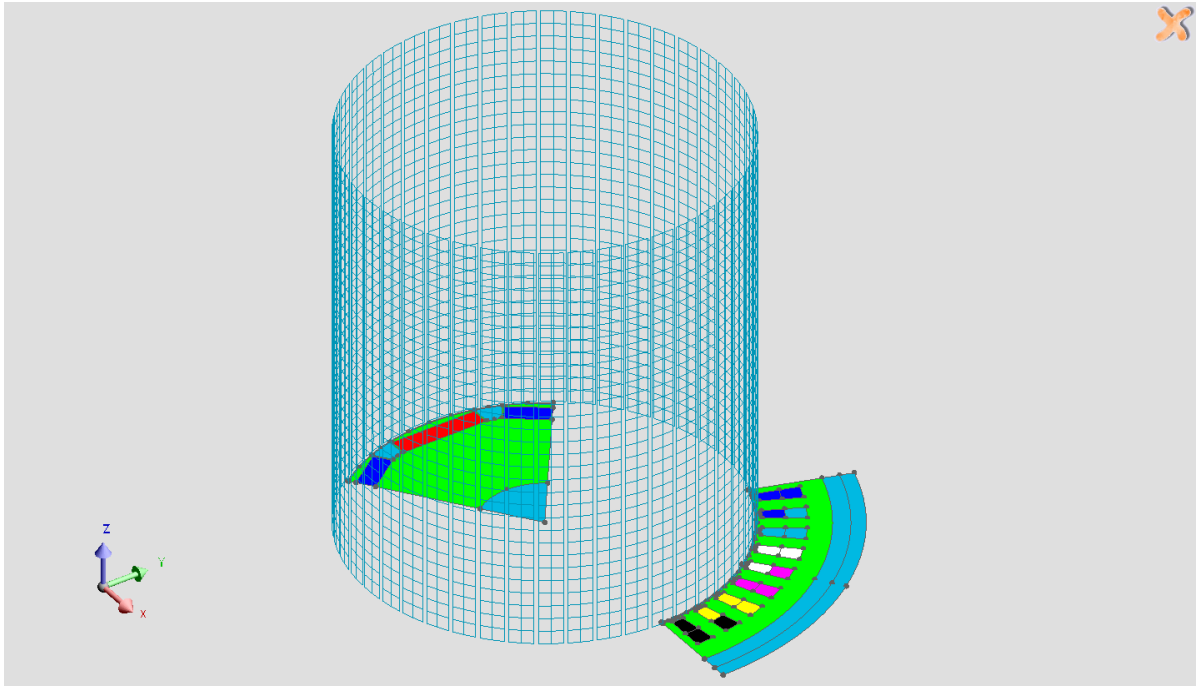
- Flux 2D project: post-processing
  - Create data support

Step	Action
1	Click on [Data support] – [New]
2	Define the support name “DataSupport_Tooth”
3	Select the support type as “Support defined by importing a mesh”
4	Select the nodal mesh file of motor teeth “Mesh_forExportTooth Force.fem”
5	Click on [OK]



MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Create data support

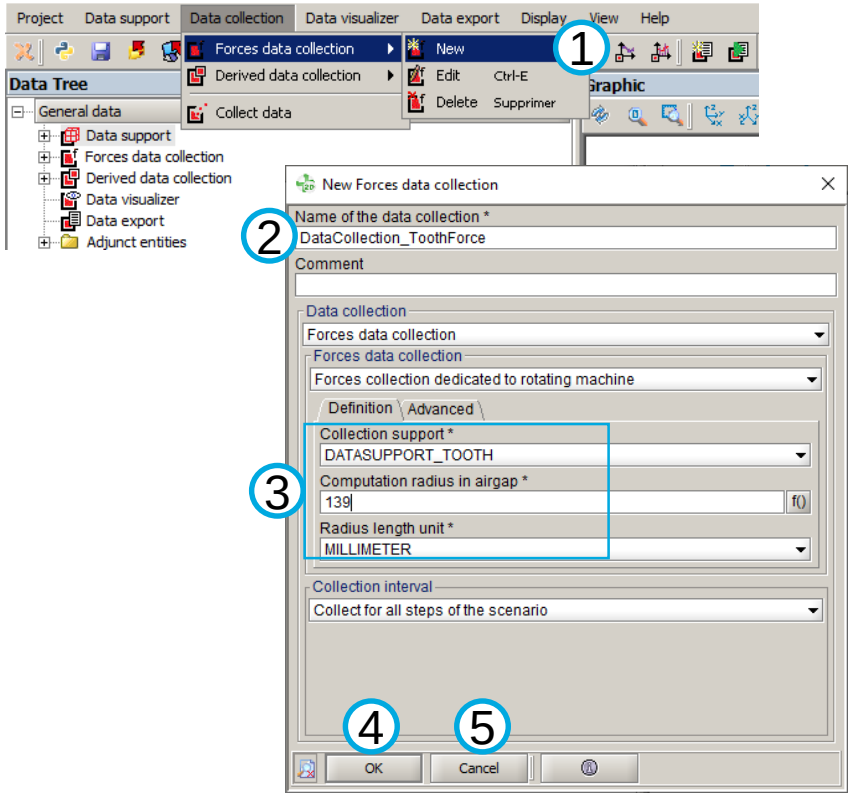


ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Create force data collection

Step	Action
1	Click on [Data collection] – [Forces data collection] – [New]
2	Define the force data collection name “DataCollection_Tooth Force”
3	Define the data support and the computation radius as 139 mm
4	Click on [OK]
5	Click on [Cancel]

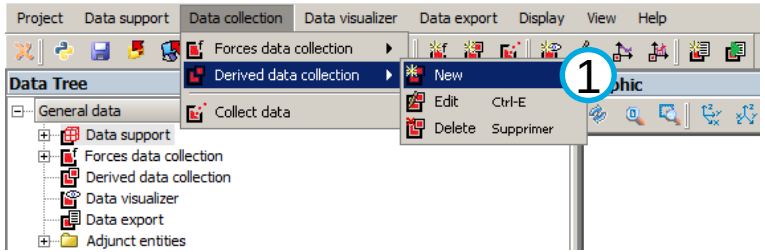


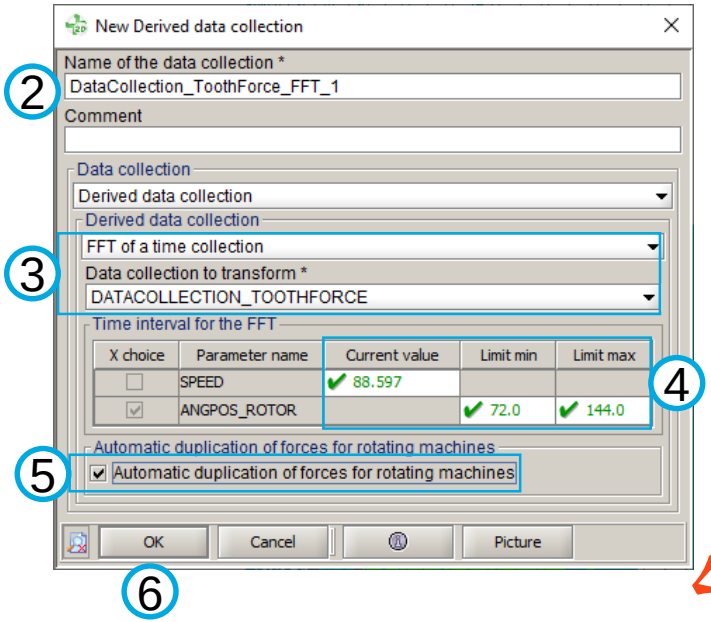
ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Create derived data collection for each speed

Step	Action
1	Click on [Data collection] – [Derived data collection] – [New]
2	Define the support name “DataCollection_ToothForce_FFT_1”
3	Define the data collection
4	Select the speed value and the ANGLE interval [72, 144]
5	Click on the “Automatic duplication of forces for rotating machines”
6	Click on [OK]

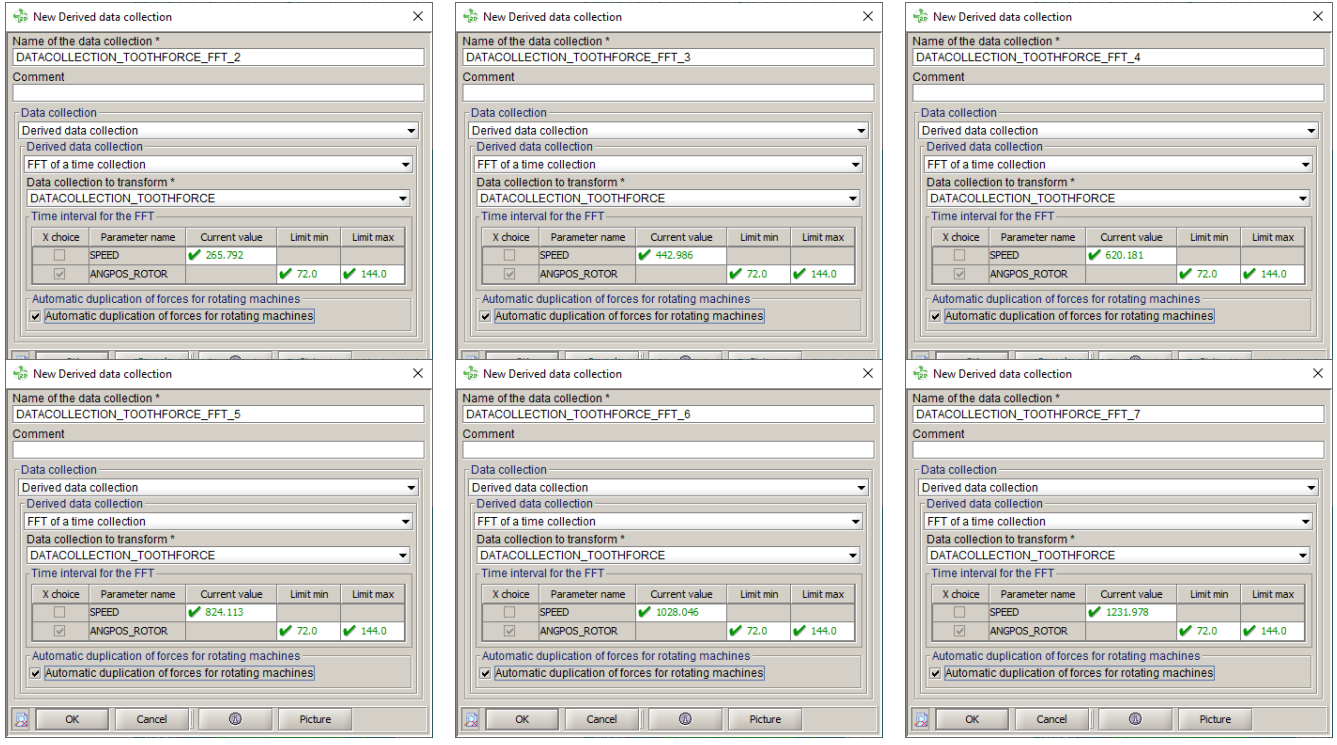




ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Create derived data collection for each speed

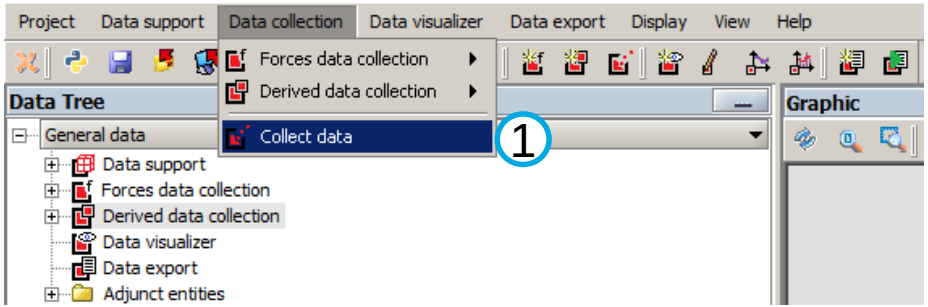


ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Collect data

Step	Action
1	Click on [Data collection] – [Collect Data]



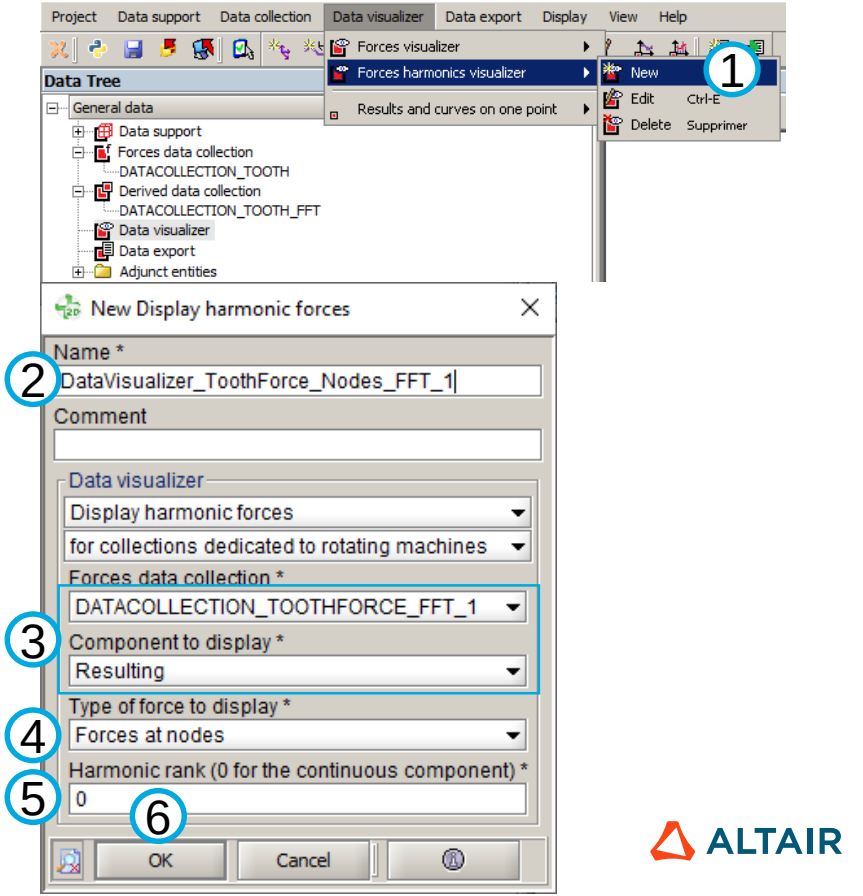
Execution time: 10 min

ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

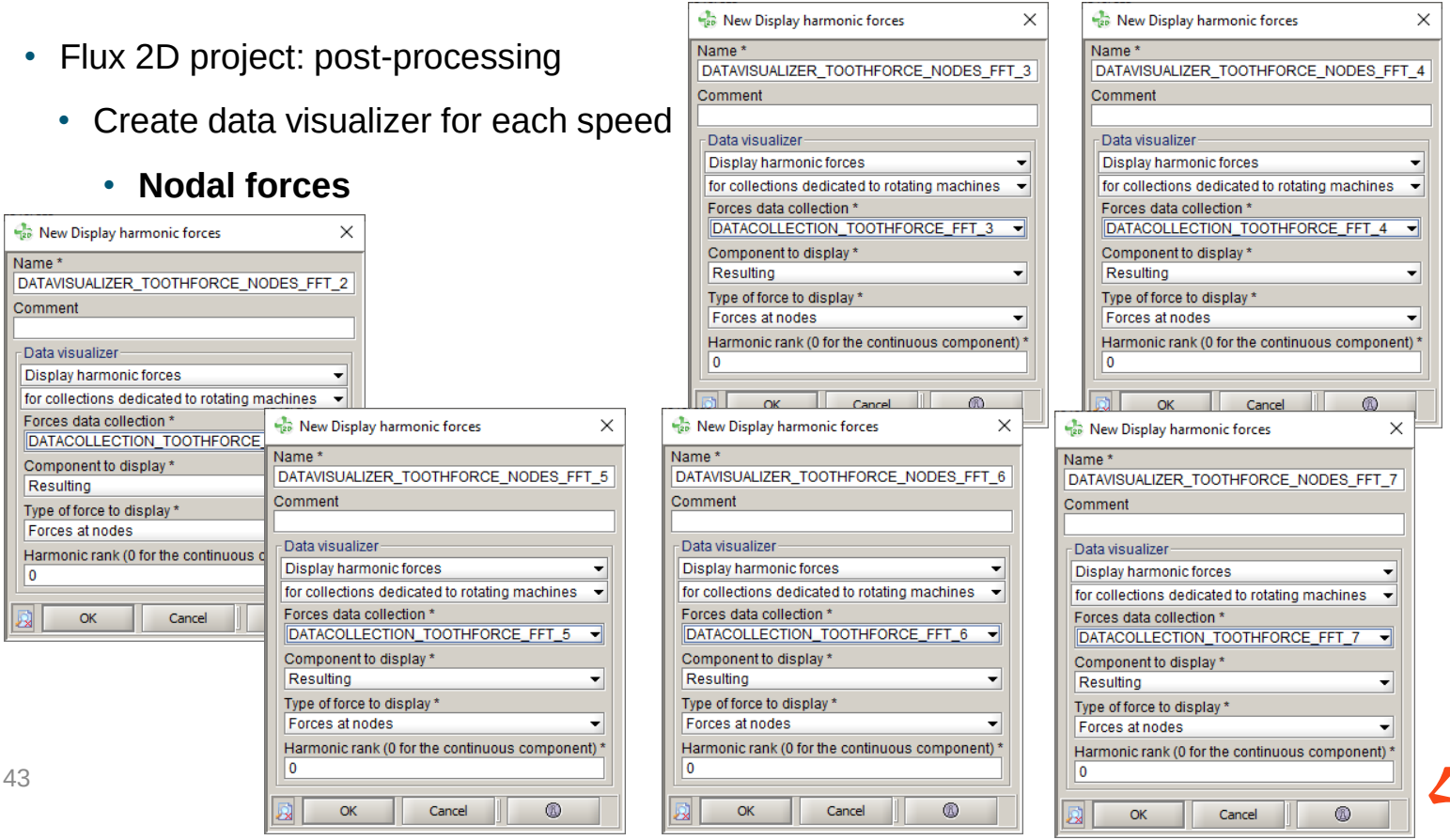
- Flux 2D project: post-processing
  - Create data visualizer for each speed
    - **Nodal forces**

Step	Action
1	Click on [Data visualizer] – [Forces harmonics visualizer] – [New]
2	Define the visualizer name “DataVisualizer_ToothForce_Nodes_FFT_X”
3	Select the objective speed and the component as “Resulting”
4	Select the type as “Forces at nodes”
5	Select the rank as 0
6	Click on [OK]



MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

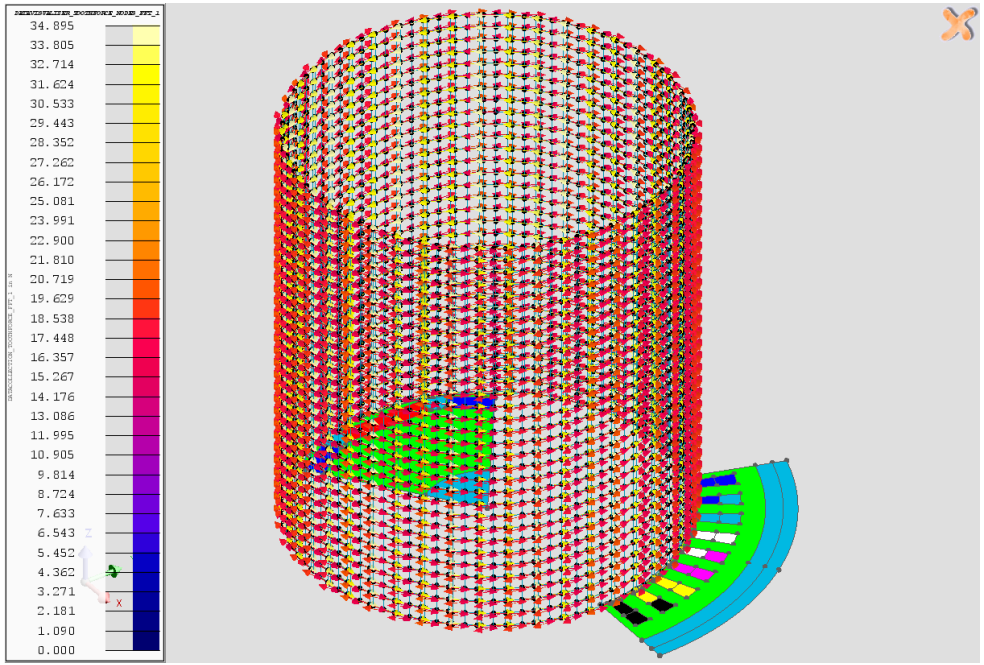
- Flux 2D project: post-processing
  - Create data visualizer for each speed
    - **Nodal forces**



ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Create data visualizer for each speed
    - **Nodal forces**

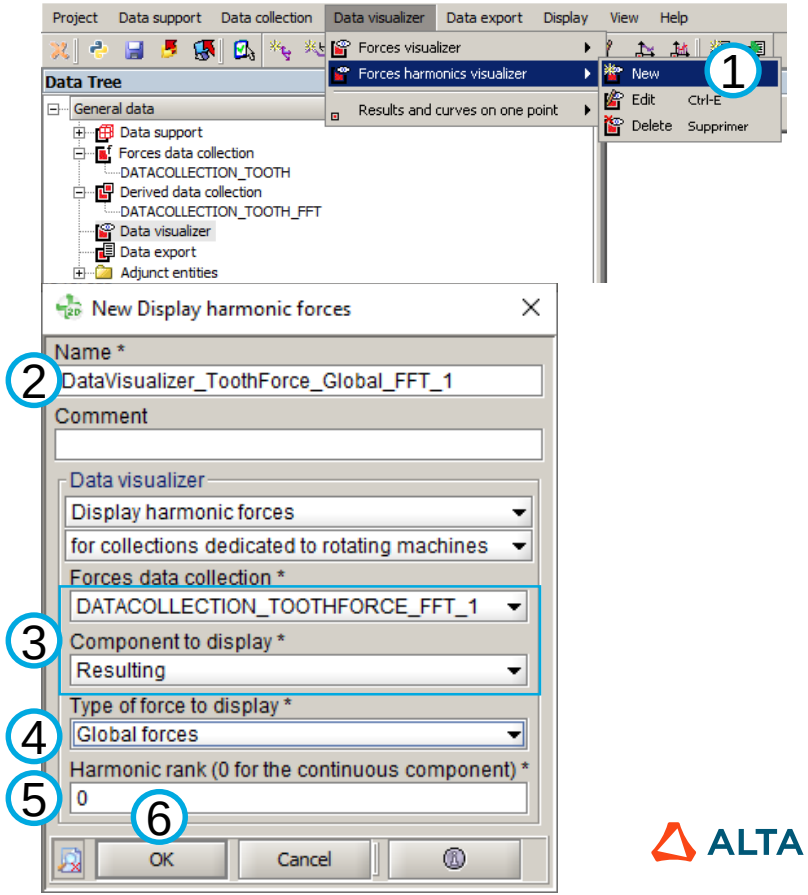


ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

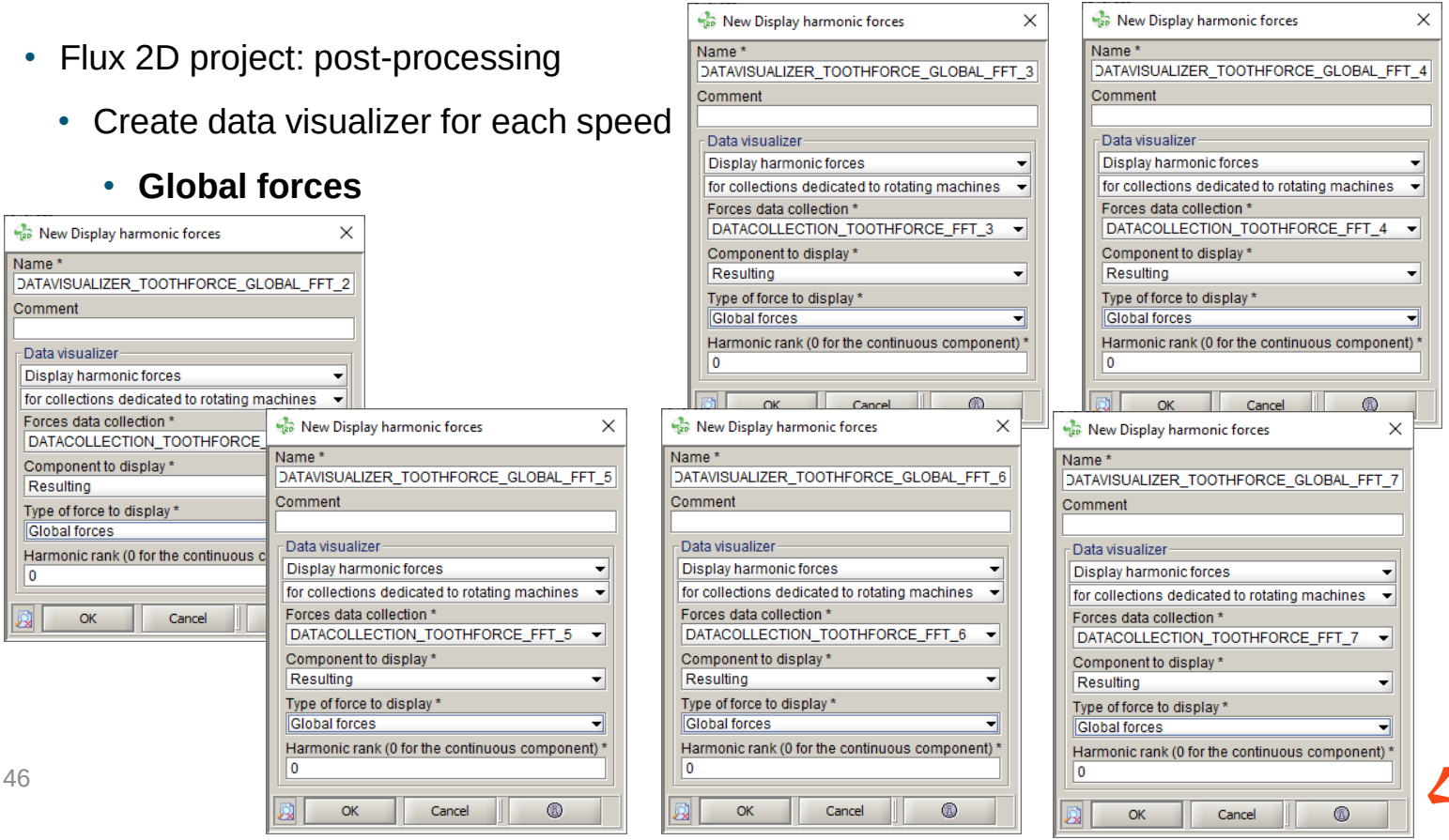
- Flux 2D project: post-processing
  - Create data visualizer for each speed
    - **Global forces**

Step	Action
1	Click on [Data visualizer] – [Forces harmonics visualizer] – [New]
2	Define the visualizer name “DataVisualizer_ToothForce_ Global_FFT_X”
3	Select the objective speed and the component as “Resulting”
4	Select the type as “Global forces”
5	Select the rank as 0
6	Click on [OK]



MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

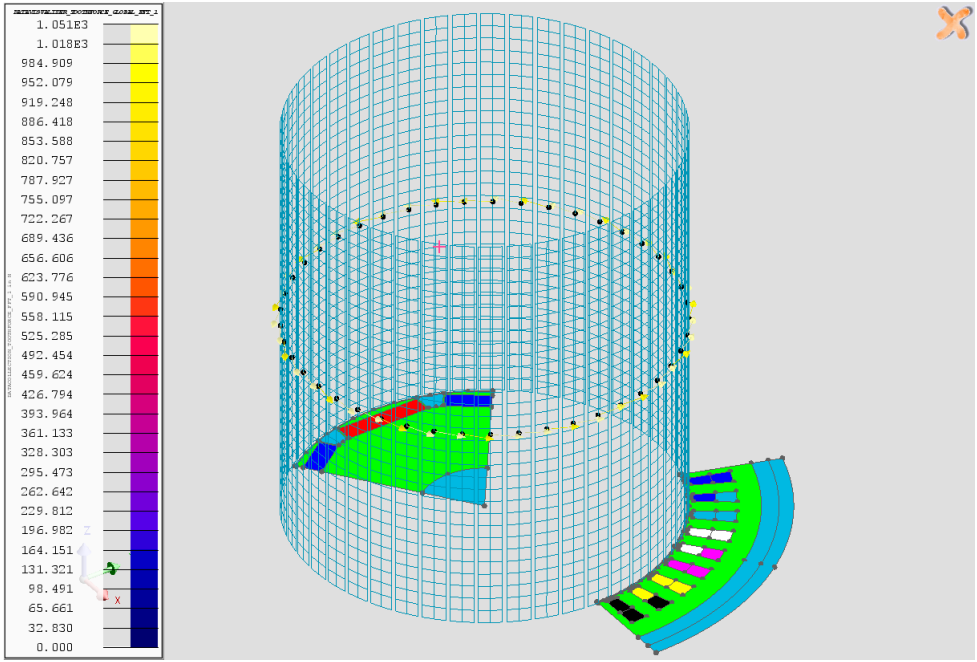
- Flux 2D project: post-processing
  - Create data visualizer for each speed
    - **Global forces**



ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Create data visualizer for each speed
    - **Global forces**

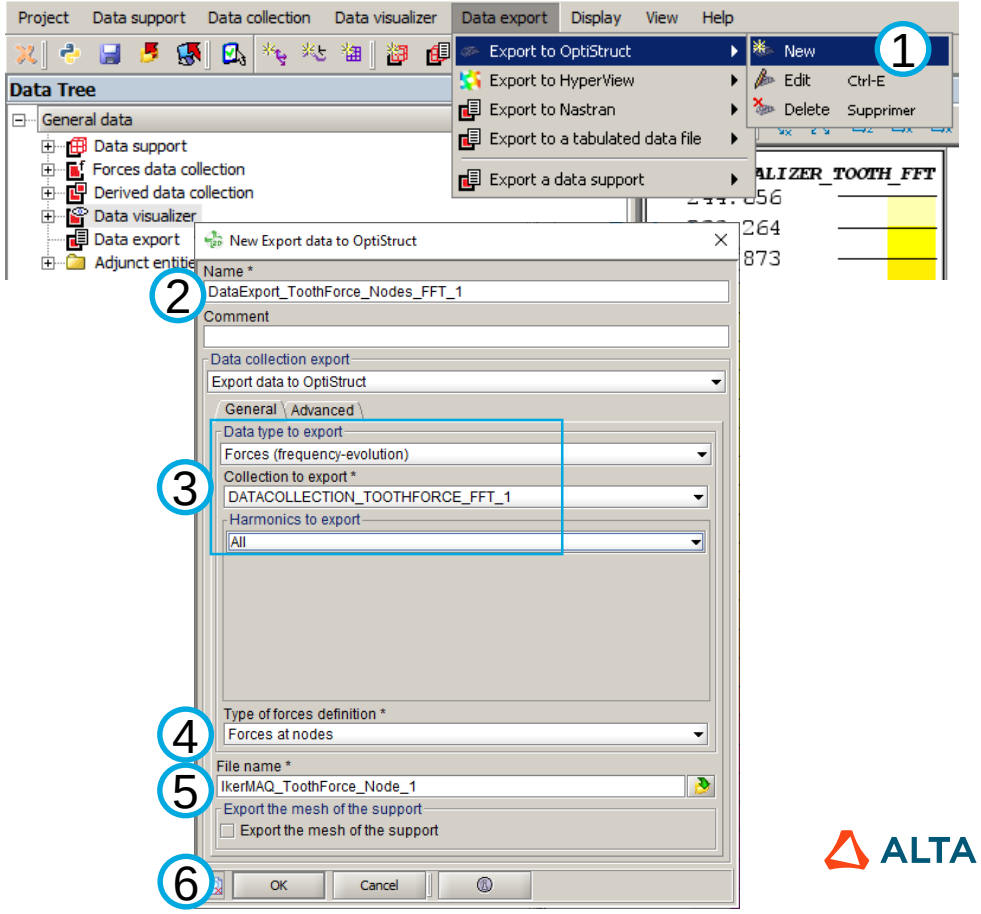


ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

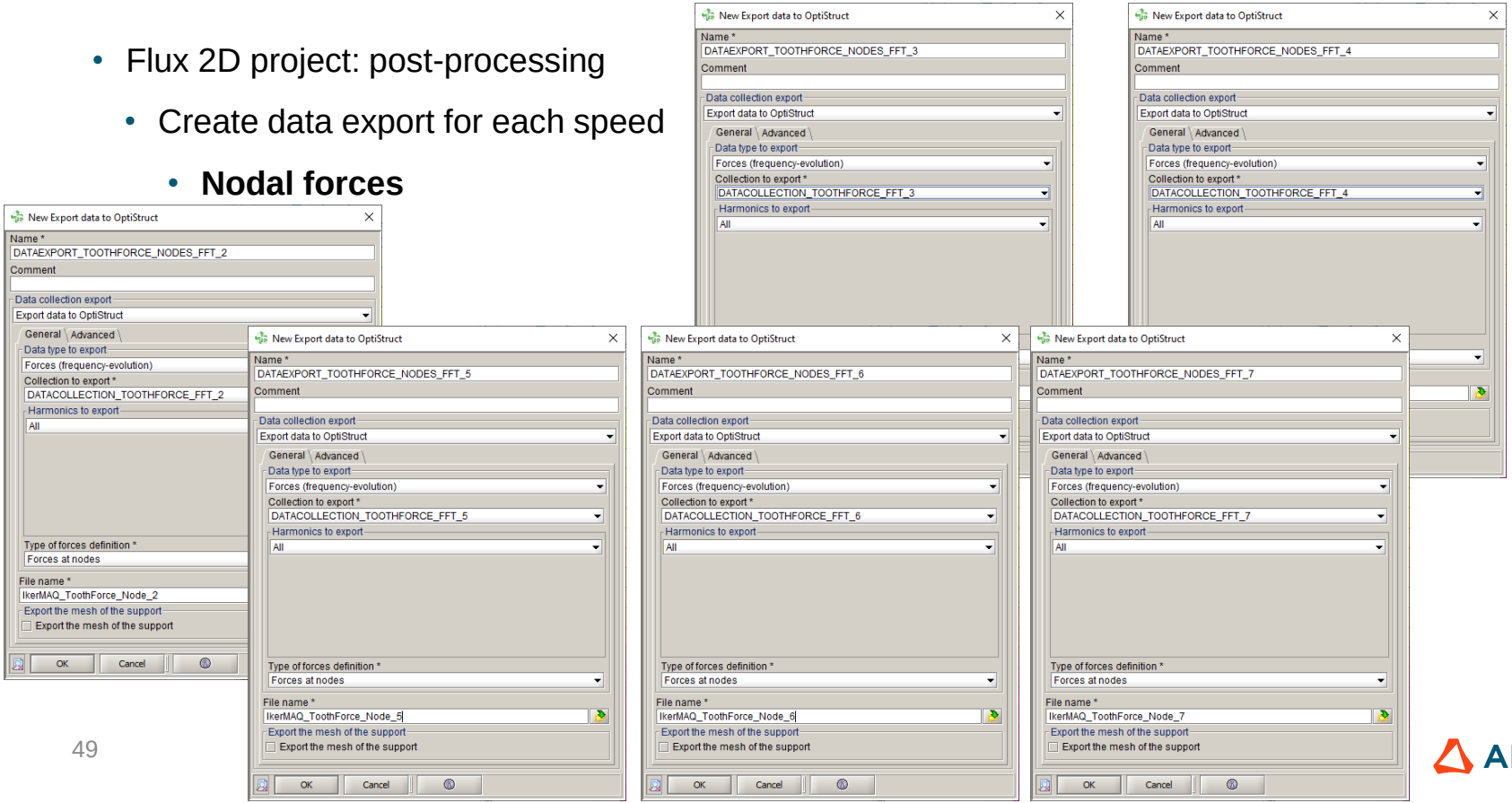
- Flux 2D project: post-processing
  - Create data export for each speed
    - **Nodal forces**

Step	Action
1	Click on [Data export] – [Export to OptiStruct] – [New]
2	Define the data export name “DataExport_ToothForce_Nodes_FFT_X”
3	Define the data type and the collection
4	Select the type as “Forces at nodes”
5	Define the export file name “IkerMAQ_ToothForce_Node_X”
6	Click on [OK]



MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Create data export for each speed
    - **Nodal forces**

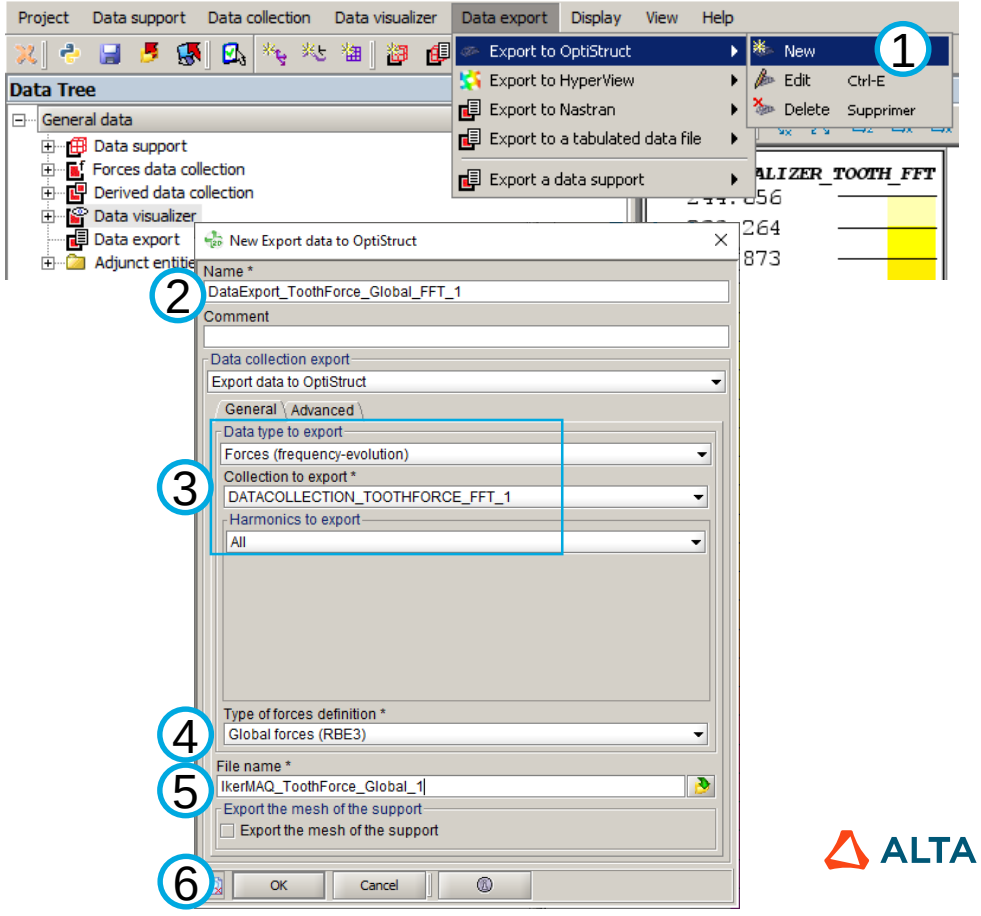


ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

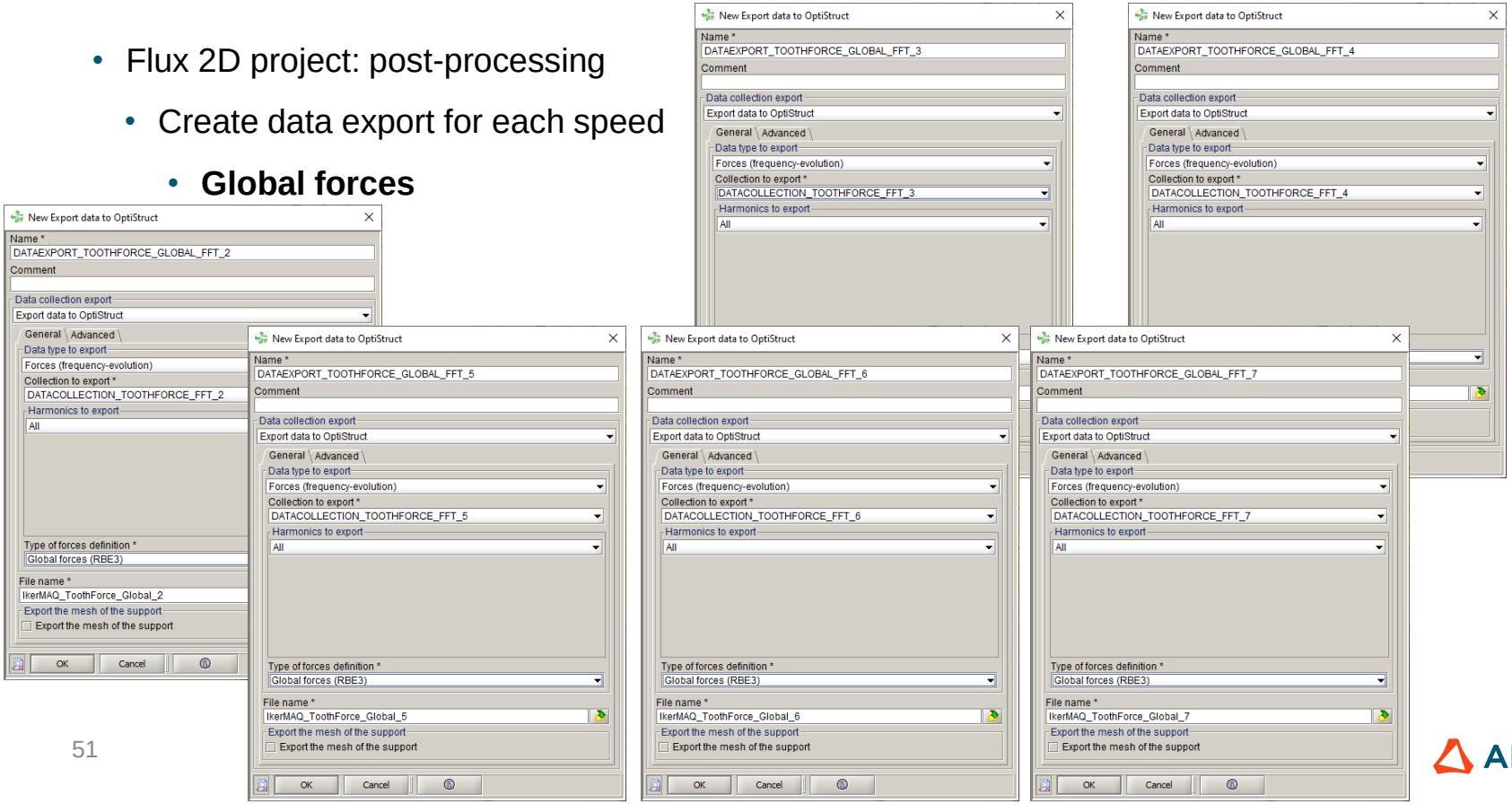
- Flux 2D project: post-processing
  - Create data export for each speed
    - **Global forces**

Step	Action
1	Click on [Data export] – [Export to OptiStruct] – [New]
2	Define the data export name “DataExport_ToothForce_Global _FFT_X”
3	Define the data type and the collection
4	Select the type as “Global forces (RBE3)”
5	Define the export file name “IkerMAQ_ToothForce_Global_X”
6	Click on [OK]



MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Create data export for each speed
    - **Global forces**

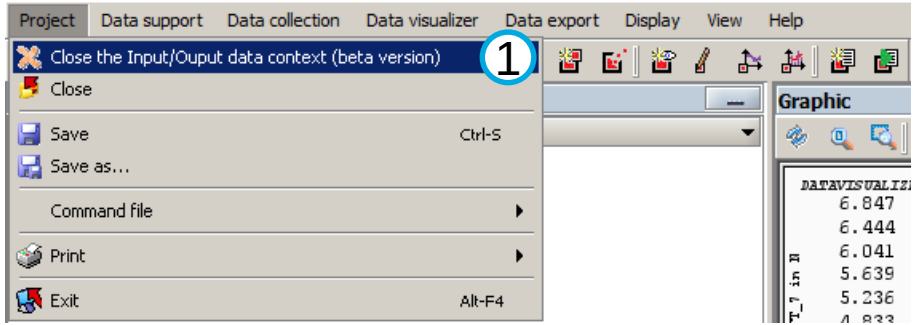


ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Close the I/O context

Step	Action
1	Click on [Project] – [Close the Input/Output data context]

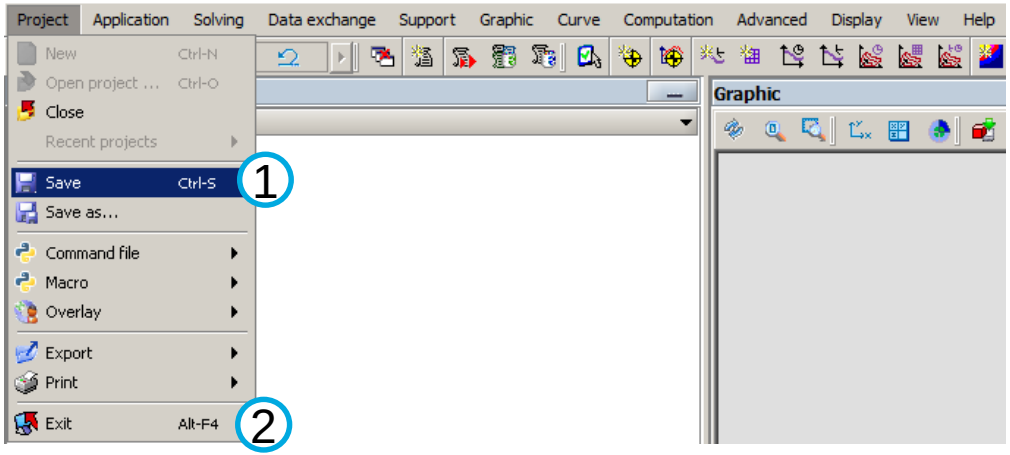


ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Close the project

Step	Action
1	Click on [Project] – [Save]
2	Click on [Project] – [Exit]

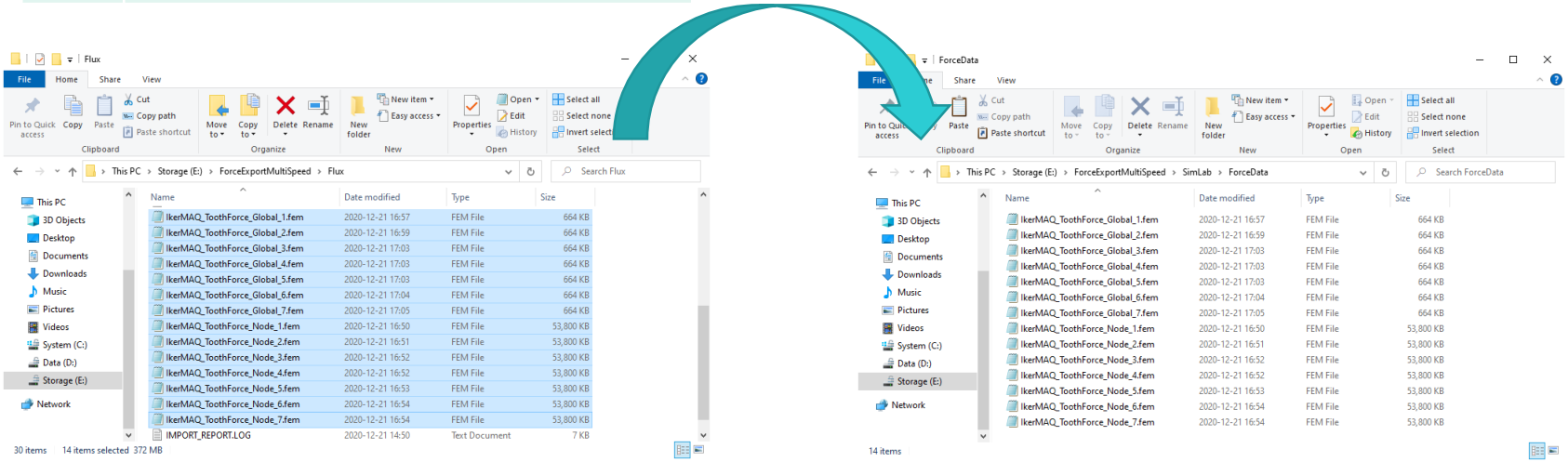


ALTAIR

MAGNETIC FORCE EXPORT IN FLUX (MULTI-SPEEDS)

- Flux 2D project: post-processing
  - Export the force per tooth

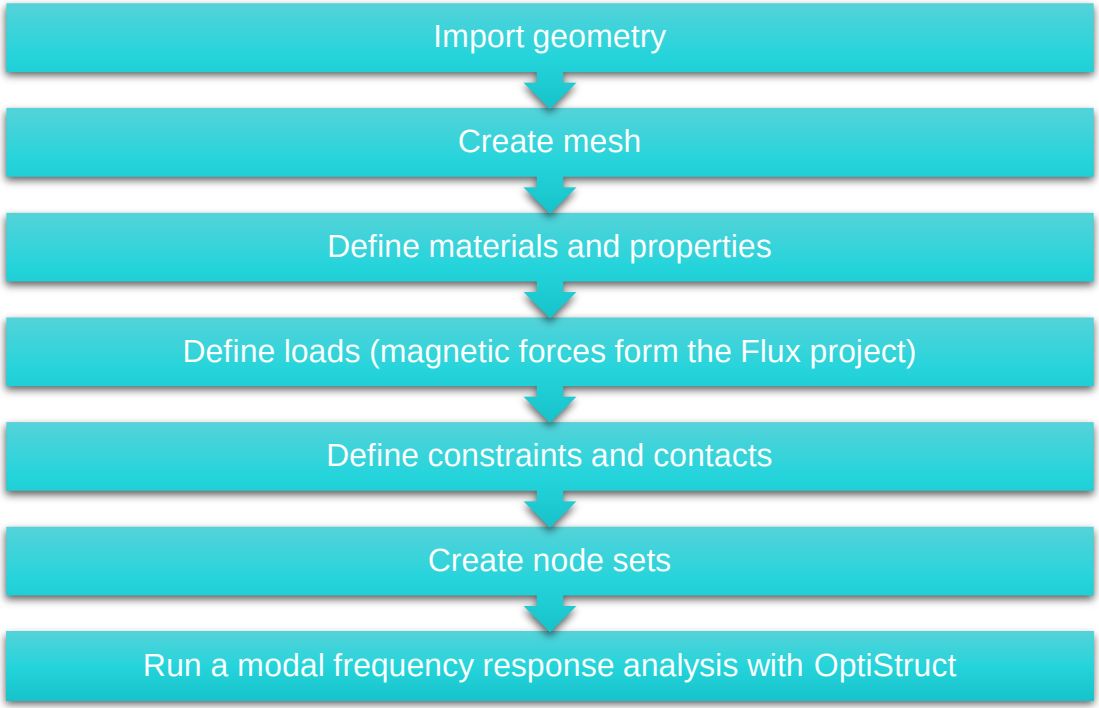
Step	Action
1	Copy the force data files to the SimLab folder



IV. VIBRATION ANALYSIS IN SIMLAB

VIBROACOUSTIC ANALYSIS IN SIMLAB

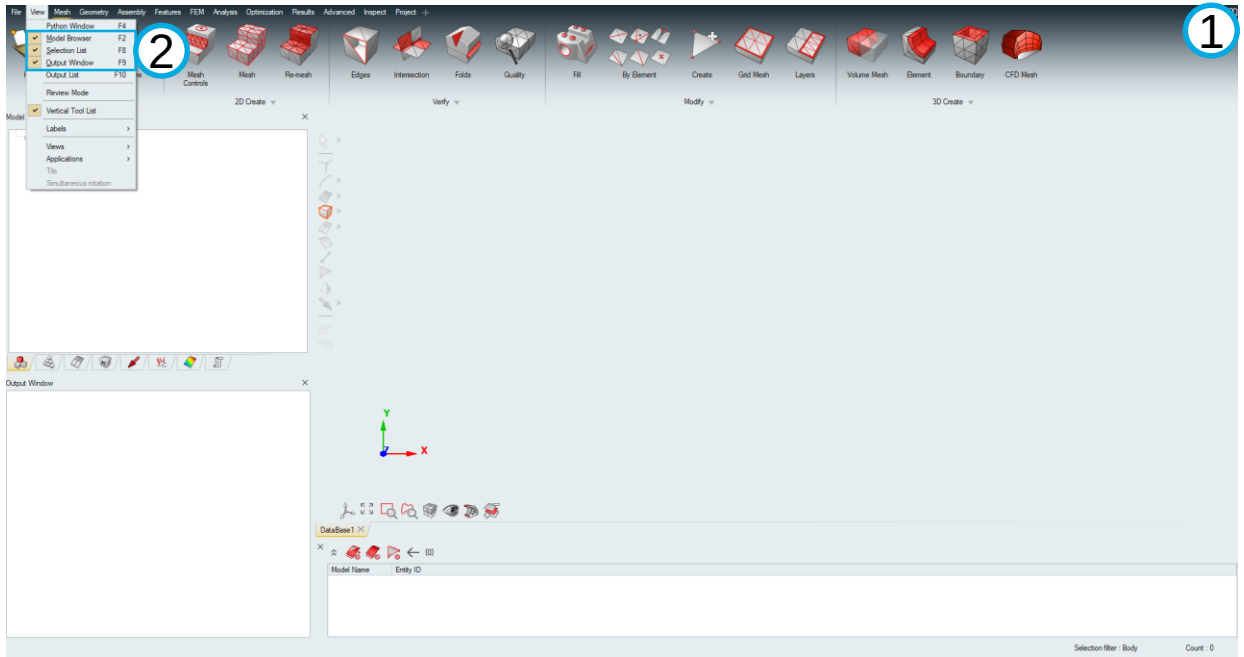
- SimLab modeling process



VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: initiation
  - Windows view configuration

Step	Action
1	Open the software SimLab
2	Click on [View], select “Model Browser”, “Selection List” and “Output Window”

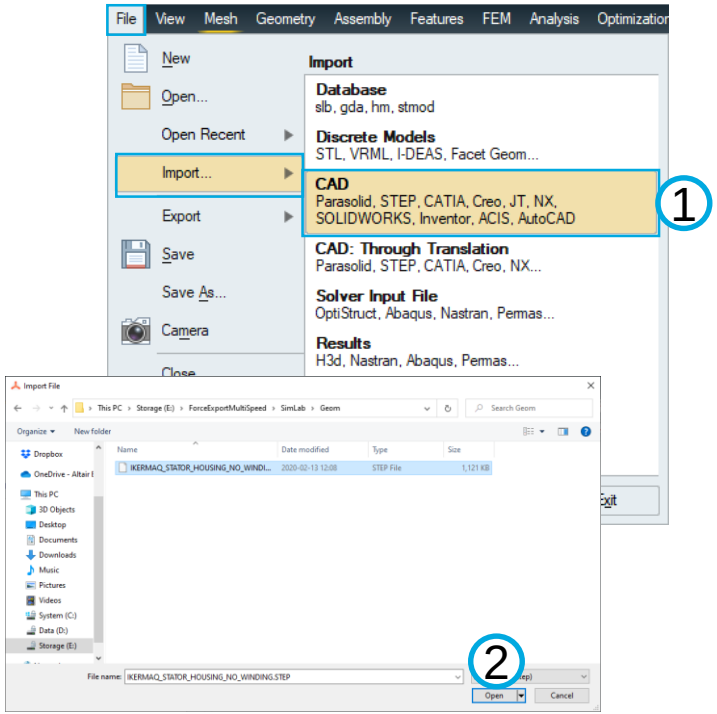


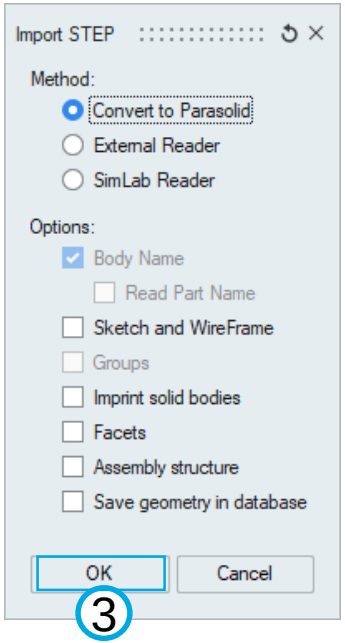
ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: initiation
  - Import motor geometry (.STEP file)

Step	Action
1	Click on [File] – [Import] – [CAD]
2	Select the file “IKERMAQ_STATOR_HOUSING_NO_WINDING.STEP” in the folder “~/ForceExportMultiSpeed/SimLab/Geom”, click on [Open]
3	Verify the import setting, and click on [OK]

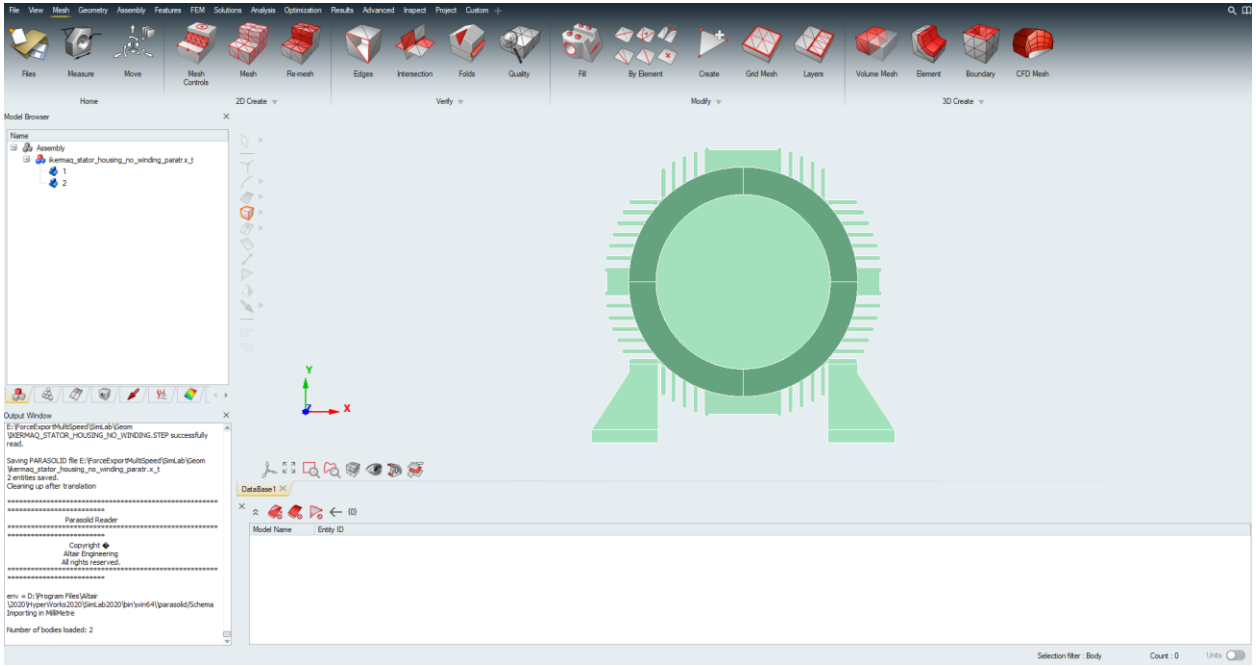




ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: initiation
  - Import motor geometry (.STEP file)

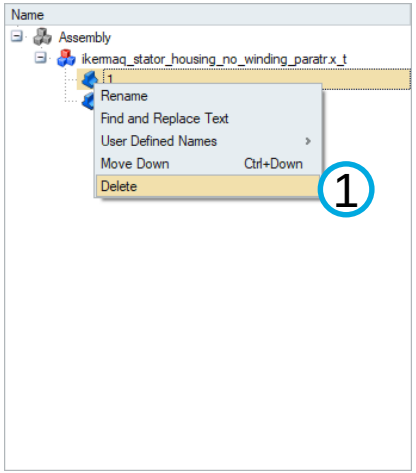


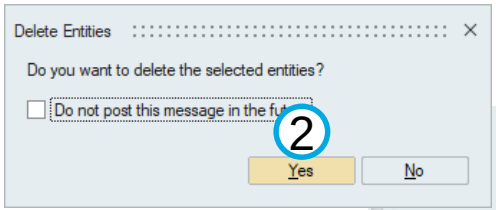
ALTAIR

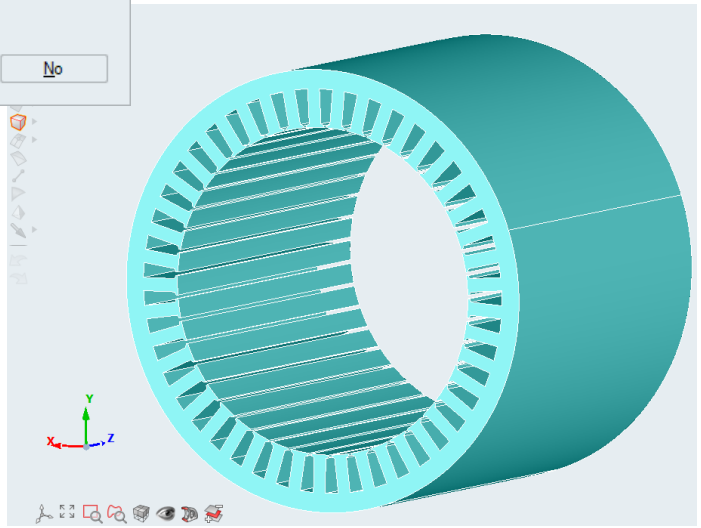
VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: initiation
  - Motor geometry modification

Step	Action
1	Right click on the component “1”, and click on [Delete]
2	Click on [Yes]





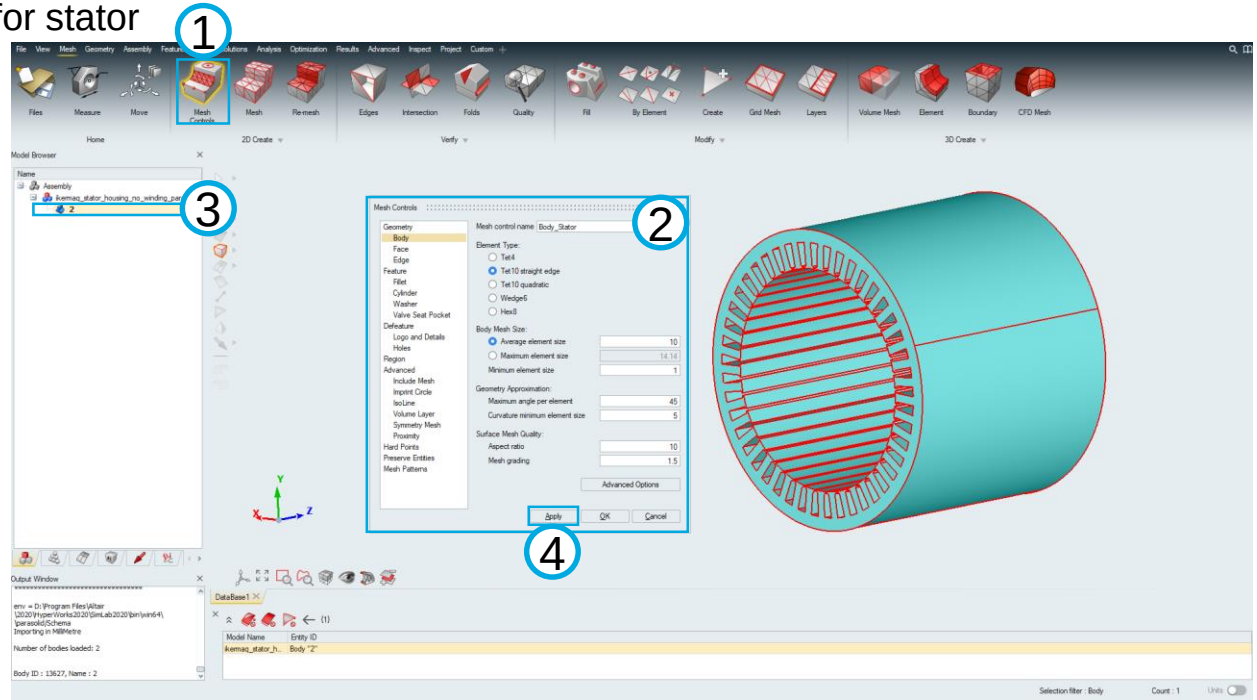


ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: meshing
  - Create *Mesh Control* for stator

Step	Action
1	In the [Mesh] ribbon, click on [Mesh Control] icon
2	Define the mesh control parameters
3	Select the body stator
4	Click on [OK]



Name	Body_Stator
Type	Body
Element type	Tet 10 straight edge
Body mesh size	Average element size: **10**

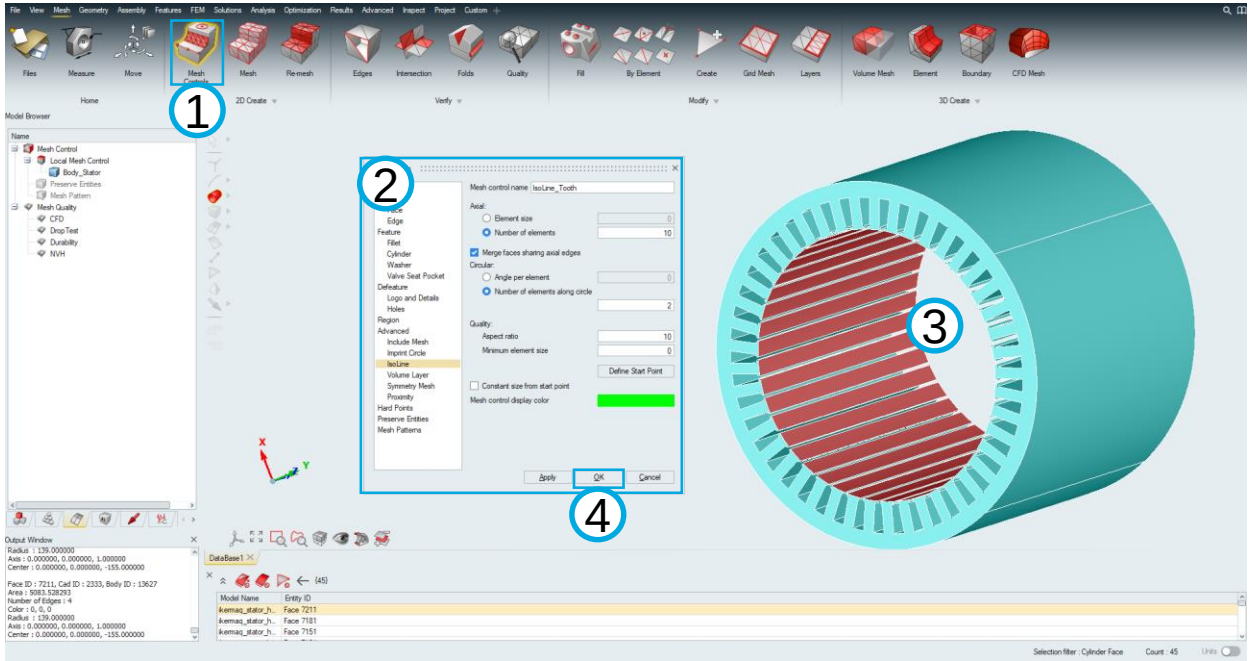
ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: meshing
  - Create *Mesh Control* for tooth face

Step	Action
1	In the [Mesh] ribbon, click on [Mesh Control] icon
2	Define the mesh control parameters
3	Select all the 45 tooth faces
4	Click on [OK]

Name	IsoLine_Tooth
Type	IsoLine
Element size	10
Number of elements along circle	2

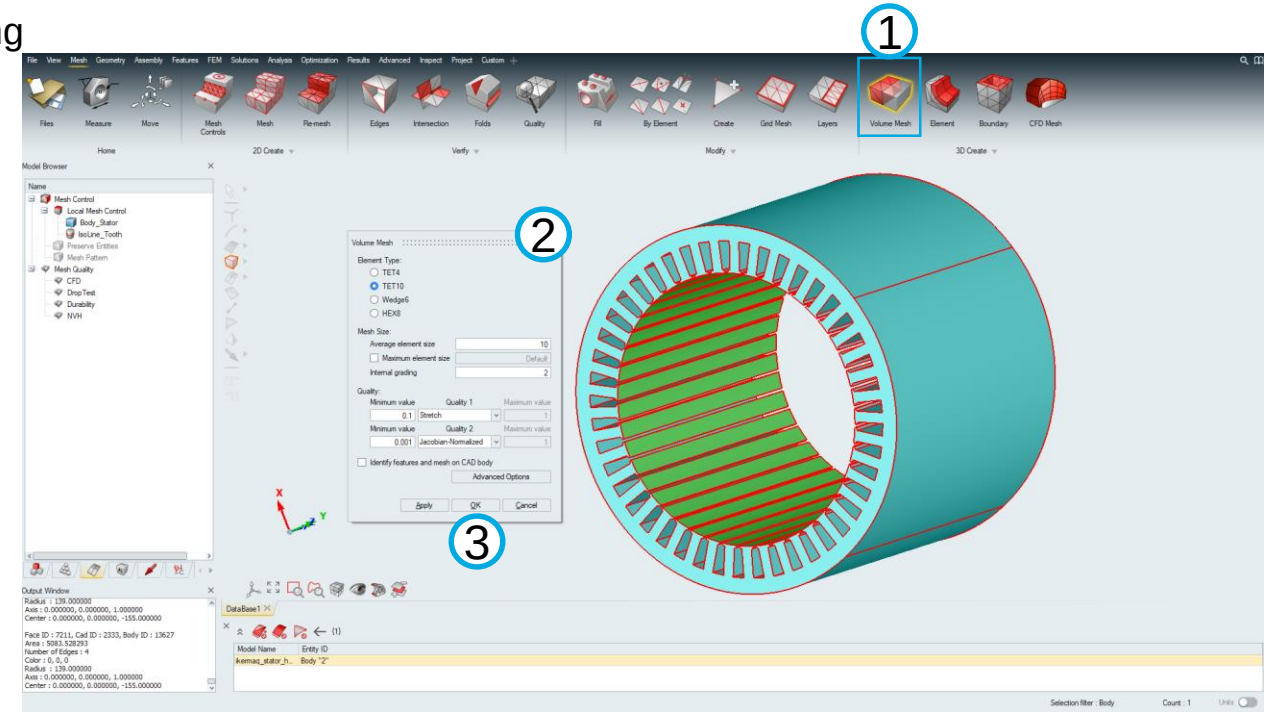


ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: meshing
  - Create volume mesh

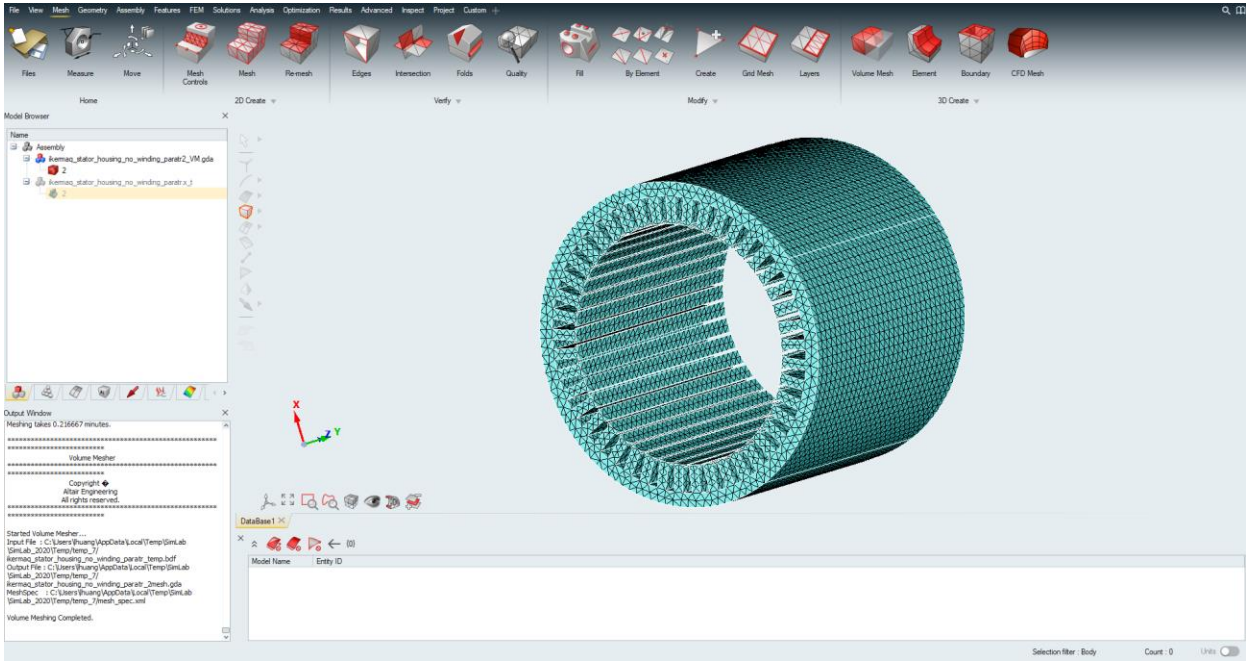
Step	Action
1	In the [Mesh] ribbon, click on [Volume Mesh] icon
2	Verify the volume mesh parameters
3	Click on [OK]



ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: meshing
  - Create volume mesh

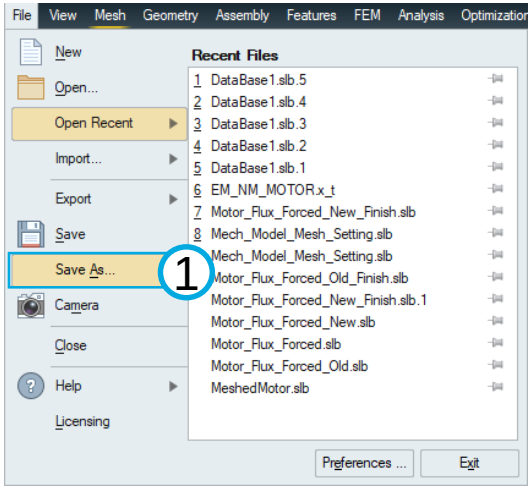


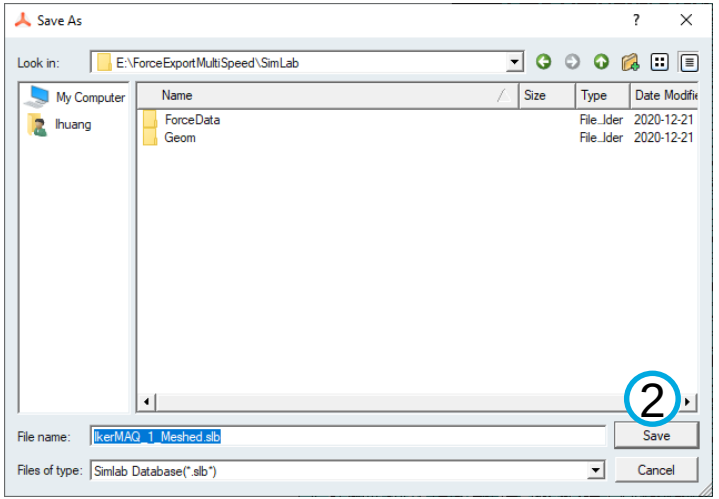
ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: meshing
  - Save the Data Base

Step	Action
1	Click on [File] – [Save as]
2	Enter a name of the Data Base, for example, “IkerMAQ_1_Meshed”, click on [Save]



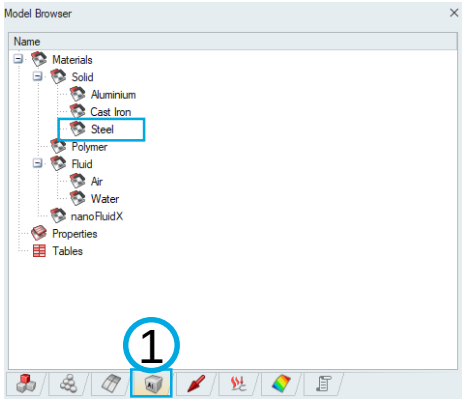


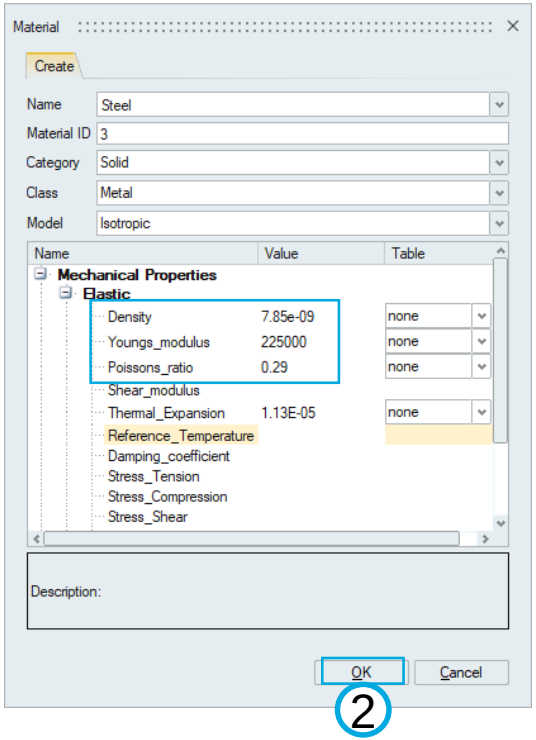
ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: material update
  - Update predefined material properties

Step	Action
1	Click on the [Property] tab of the Model Browser
2	Double click on the material “Steel”, update the material properties.



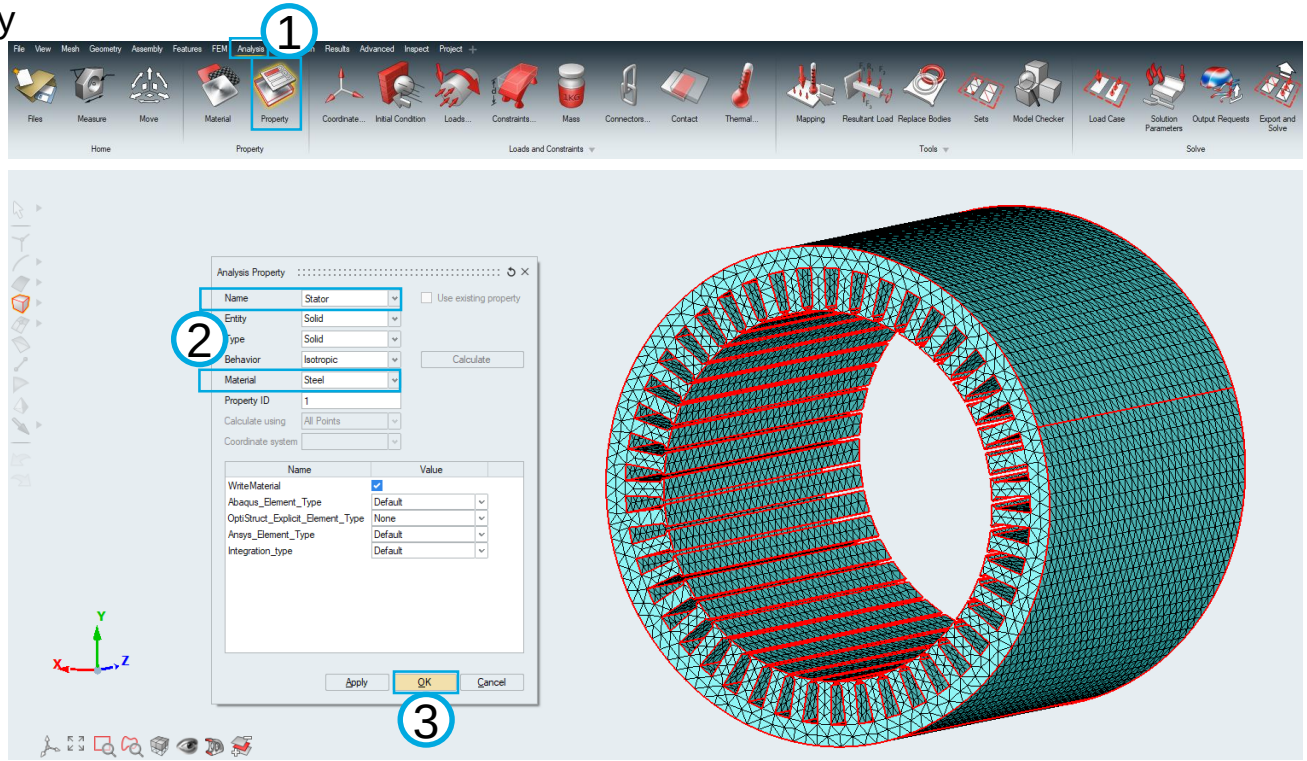


ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: property
  - STATOR

Step	Action
1	In the [Analysis] ribbon, click on [Property] icon
2	Create the Property “Stator”, select the material as “Steel”, and assign to the “STATOR” region.
3	Click on [OK]

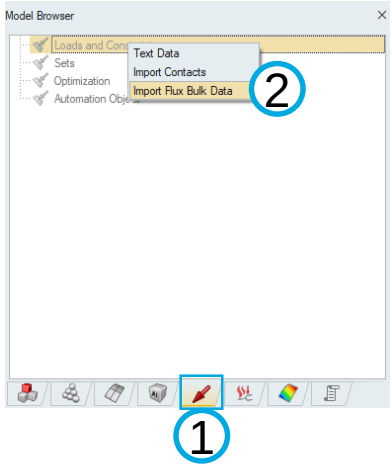


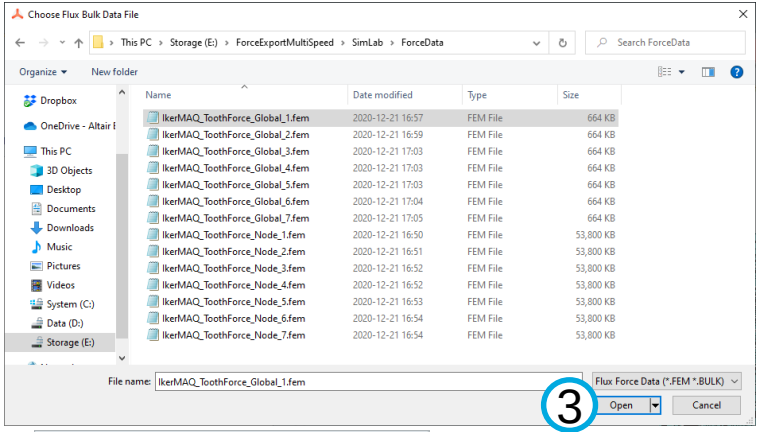
ALTAIR

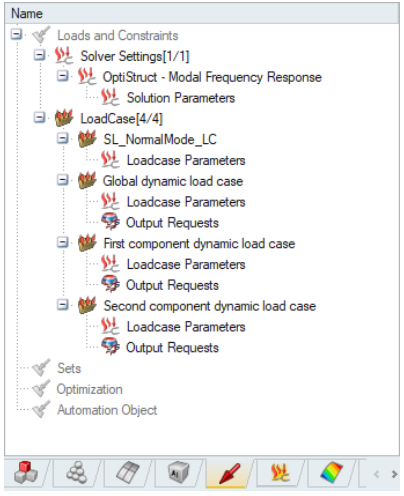
VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: magnetic force definition
  - Import magnetic force data

Step	Action
1	In [Model Browser], click on the [Loads and Constraints] tab
2	Right click on the [Loads and Constraints] and click on [Import Flux Bulk Data]
3	Select ***one of the force data*** files in the folder ~\ForceExportMultiSpeed\SimLab\ForceData, and click on [Open]





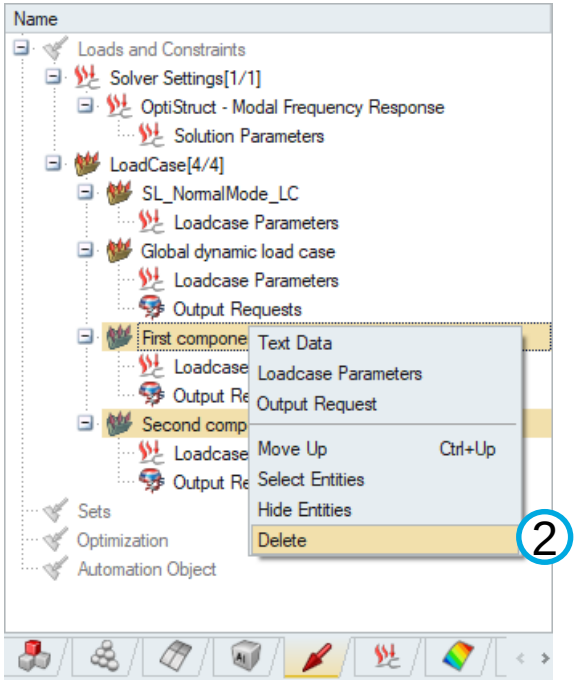


ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: magnetic force definition
  - Delete other load cases

Step	Action
1	Select the following two load cases[1]: - First component dynamic load case - Second component dynamic load case
2	Right click and click on [Delete]

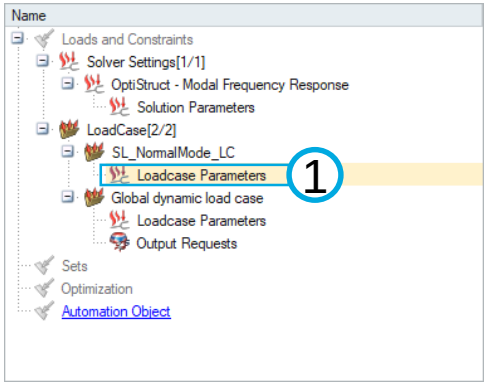


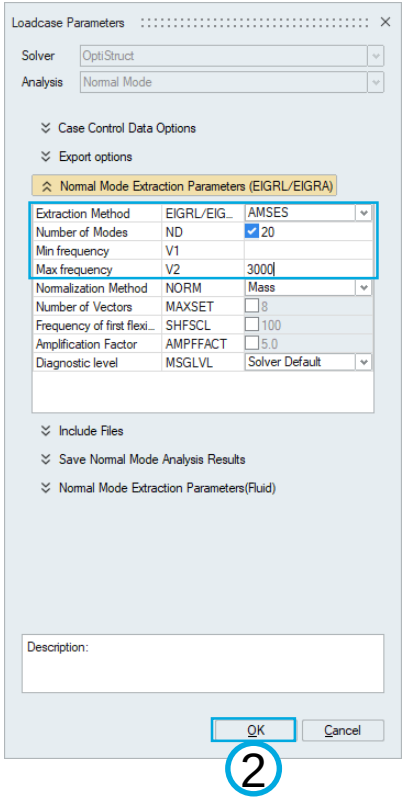
[1] if the force file is generated by Flux 2020, there will be a third component, which is also to be deleted.

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: magnetic force definition
  - Modify load setting: SL_NormalMode_LC

Step	Action
1	Double click on the load case “Loadcase Parameters” for the first LoadCase
2	Modify the setting in the “Normal Mode Extraction Parameters (EIGRL/EIGRA)”, and click on OK



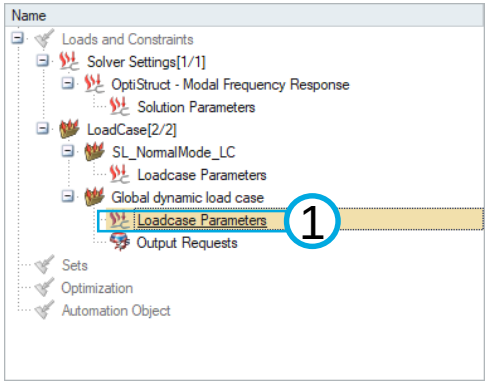


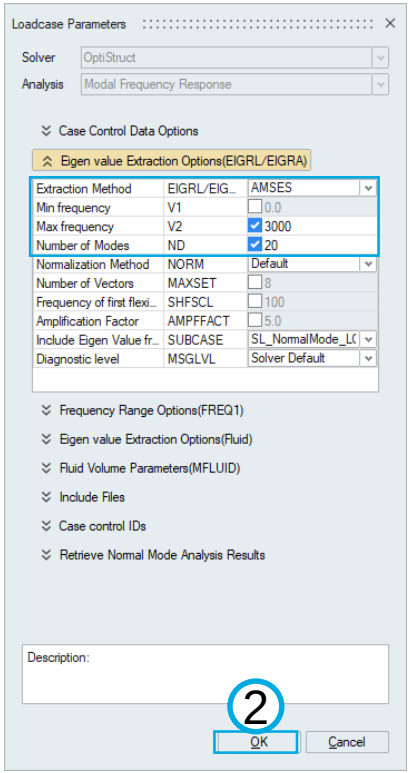
ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: magnetic force definition
  - Modify load setting: Global dynamic load case

Step	Action
1	Double click on the load case “Loadcase Parameters” for the second LoadCase
2	Modify the setting in the “Normal Mode Extraction Parameters (EIGRL/EIGRA)”, and click on OK



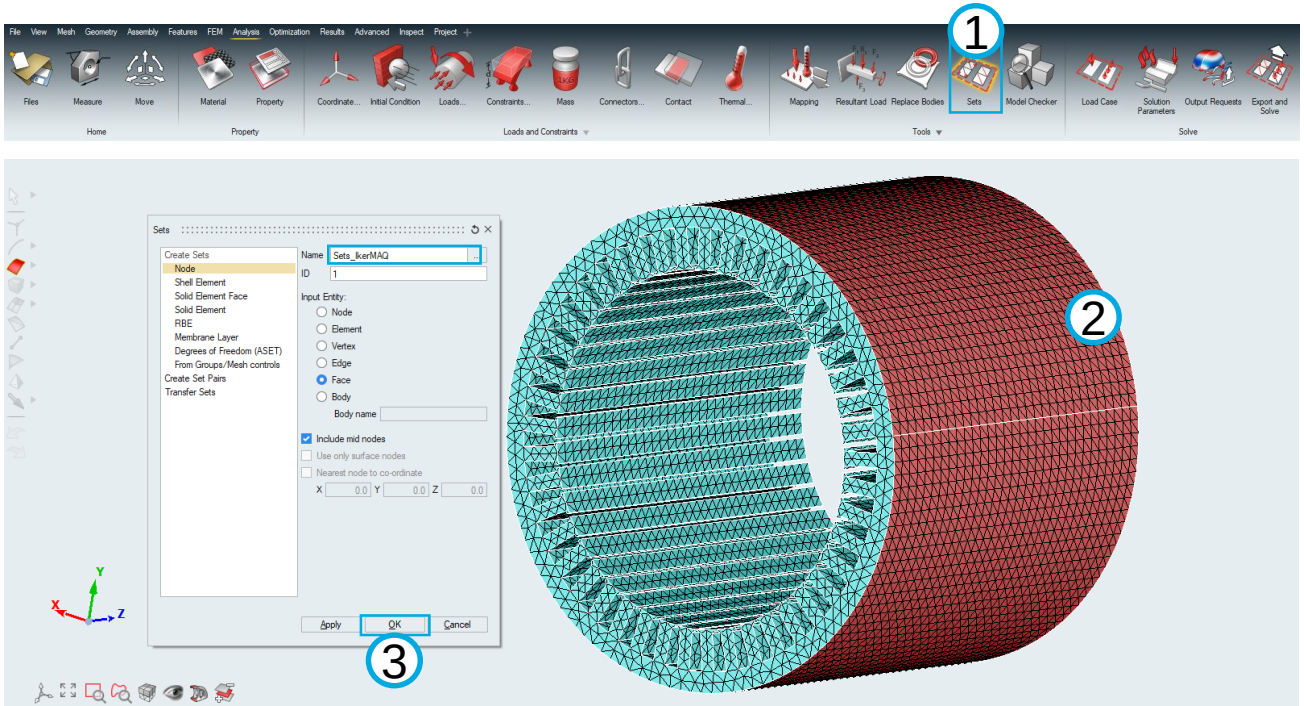


ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: node set
  - Define node set

Step	Action
1	In the [Analysis] ribbon, click on icon [Sets]
2	Select all the outer faces of the Stator body.
3	Define the name of the sets and click on [OK]

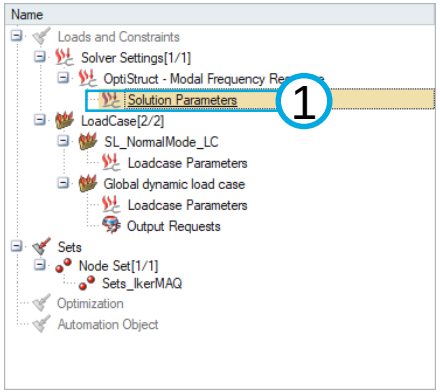


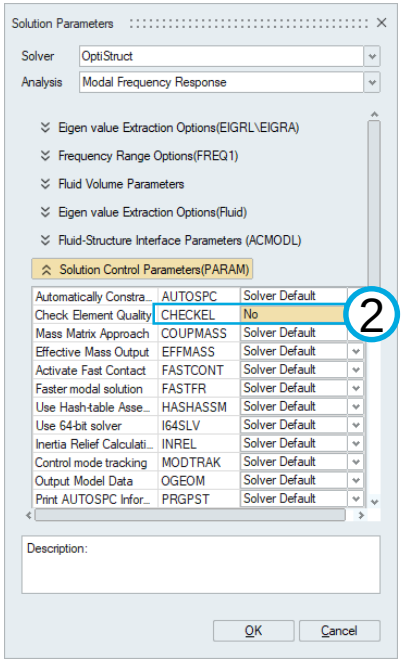
ALTAIR

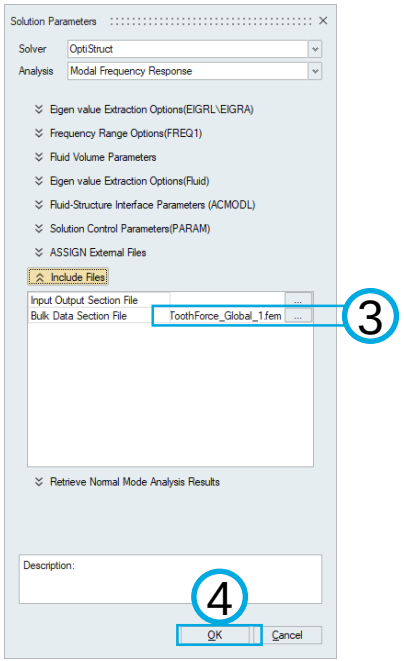
VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: solver setting
  - Define solver settings

Step	Action
1	In the [Loads and Constraints] tab, double click on the [Solution Parameters]
2	Select “No” for the option “Check Element Quality” in the “Solution Control Parameters” tab
3	Verify the force information (Bulk Data) in the “Include Files” tab
4	Click on [OK]





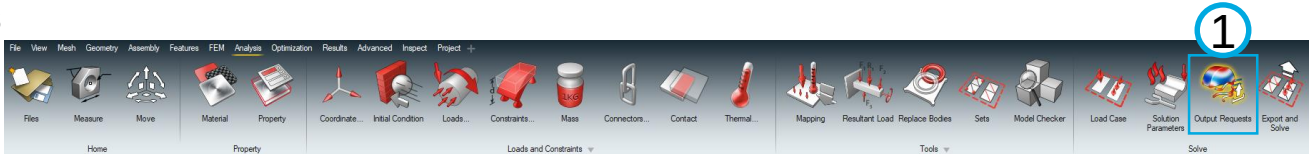


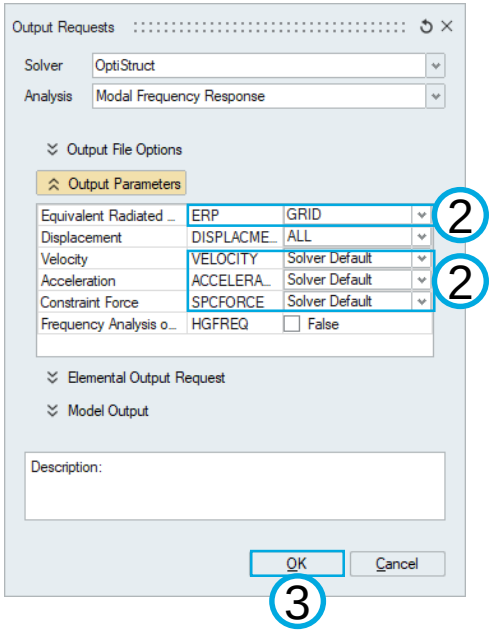
ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: solver setting
  - Define output requests

Step	Action
1	In the [Analysis] ribbon, click on the icon [Output Requests]
2	In the [Output Parameters] tab, modify the setting for different parameters
3	Click on [OK]



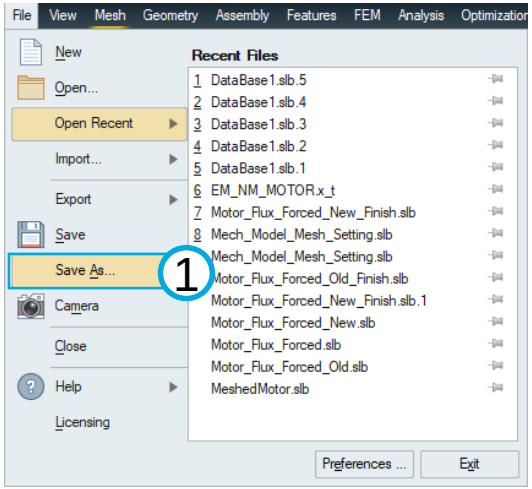


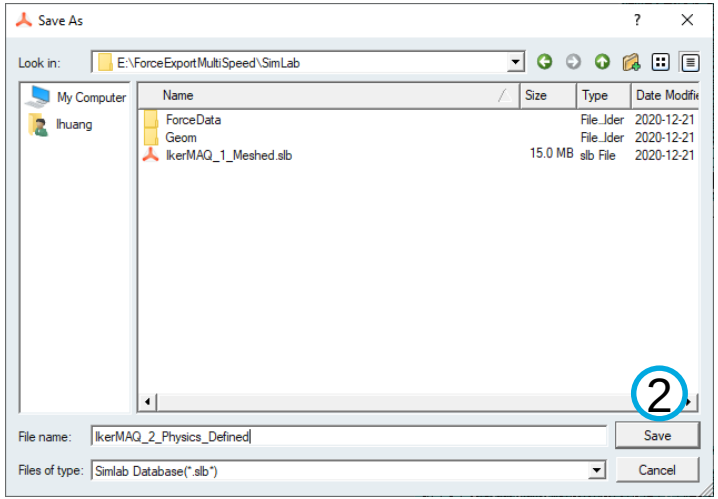
ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: export and solve
  - Save the Data Base

Step	Action
1	Click on [File] – [Save as]
2	Enter a name of the Data Base, for example, “IkerMAQ_2_Physics_Defined”, click on [Save]



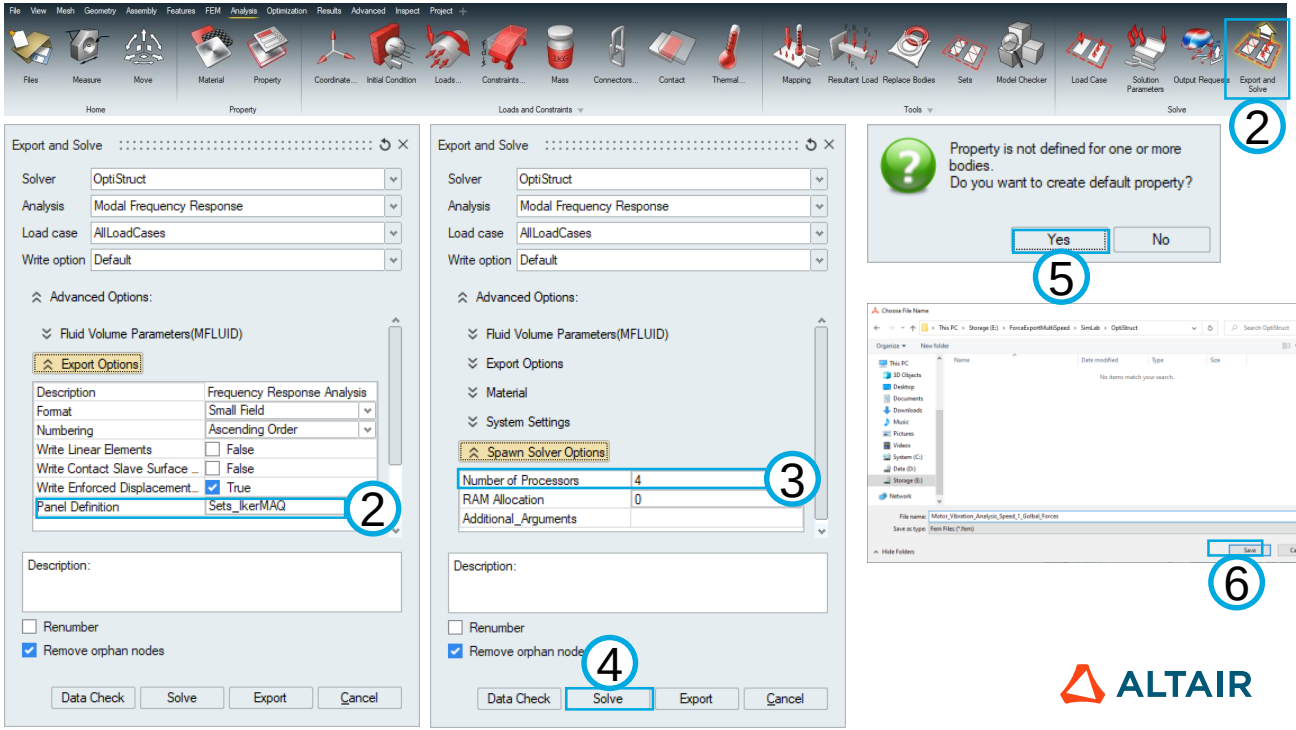


ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- SimLab project: export and solve
  - Export the OptiStruct project

Step	Action
1	In the [Analysis] ribbon, click on the icon [Export and Solve]
2	In the tab [Advanced Options] – [Export Options], select “Set_IkerMAQ” for the [Panel Definition]
3	In the tab [Advanced Options] – [Spawn Solver Options], verify the setting for solving the problem
4	Click on [Solve] for solving the project or click on [Solve] for solving the OptiStruct project
5	Define the name of the OptiStruct project as “Motor_Vibration_Analysis_Speed_X_Golbal_Forces”, click on [Save]

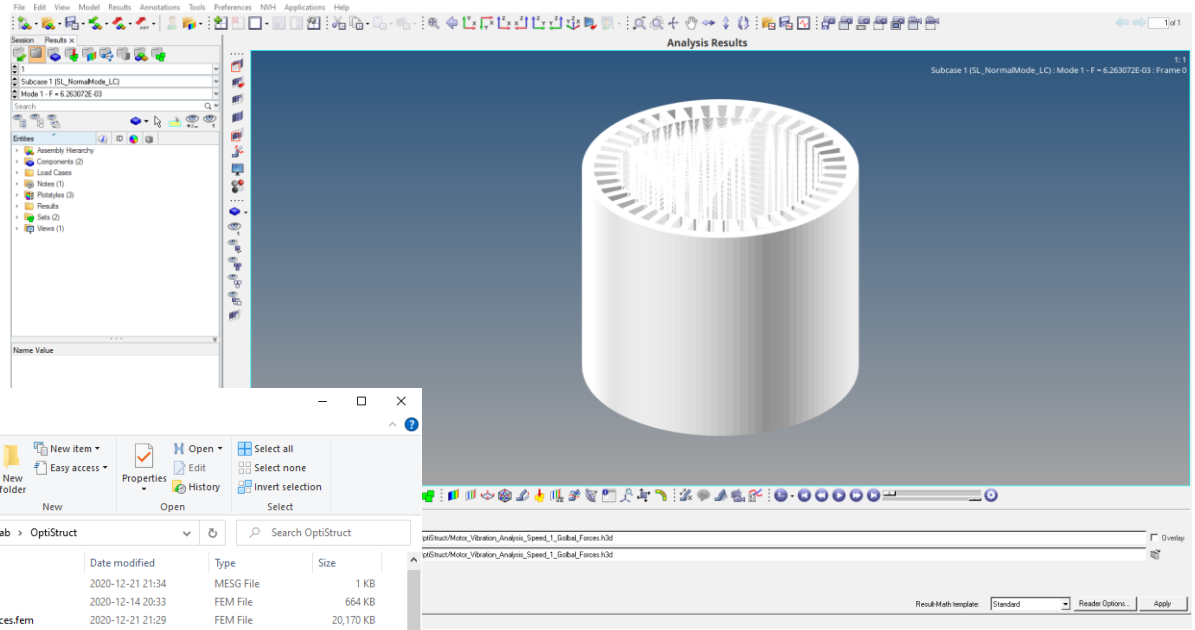


ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- Result visualization

Step	Action
1	Open the generated .mvw file

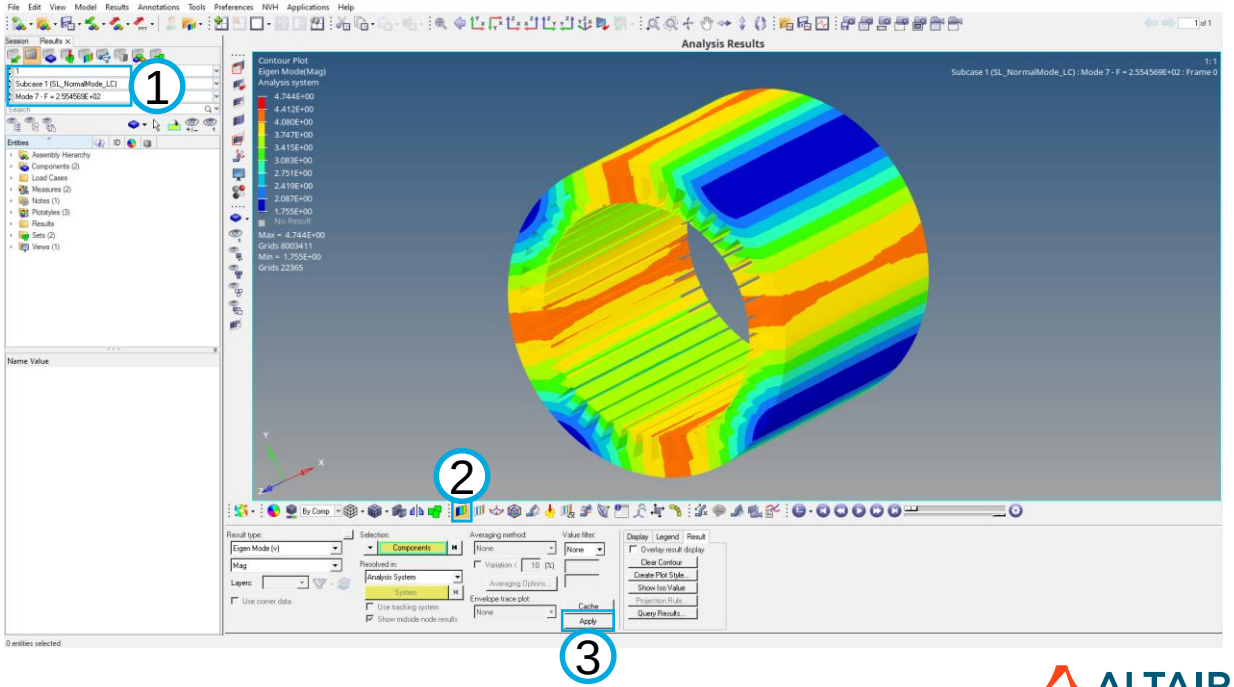


ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- Result visualization
  - Eigen modes

Step	Action
1	Select the [Subcase 1], and select the mode to be visualized
2	Click on the icon
3	Click on [Apply]

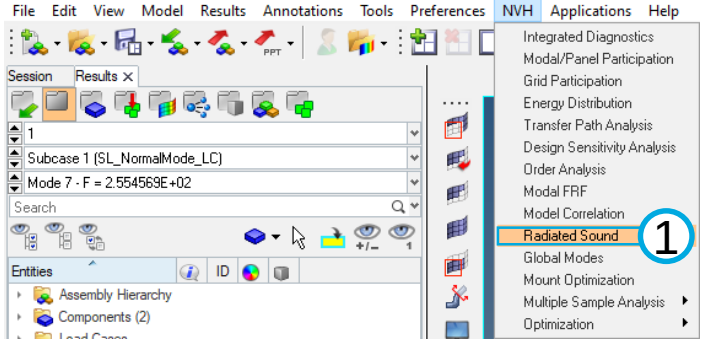


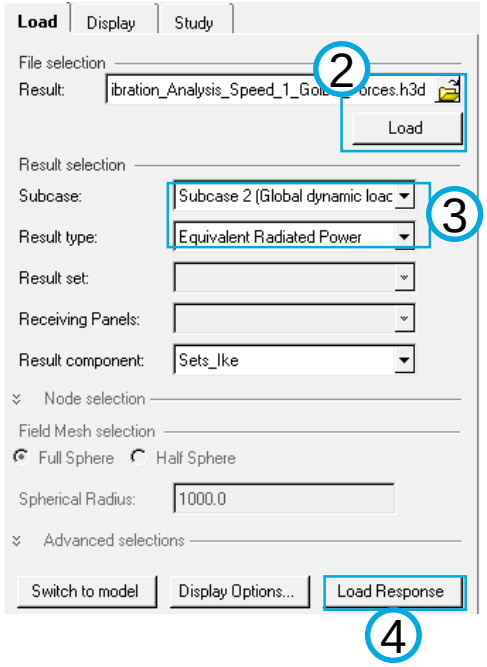
ALTAIR

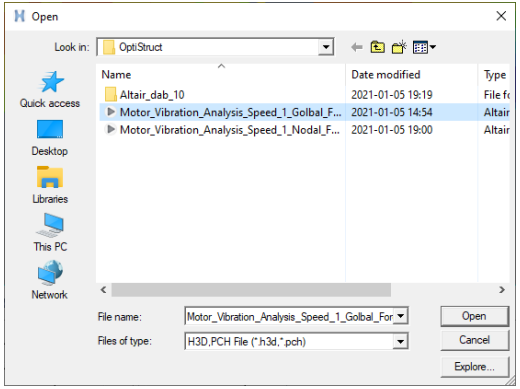
VIBROACOUSTIC ANALYSIS IN SIMLAB

- Result visualization
  - ERP (equivalent radiated power) curve

Step	Action
1	Click on [NVH] – [Radiated Sound]
2	Click on the icon to open the result file, and click on [Load]
3	Select [Subcase 2]
4	Click on [Load Response]



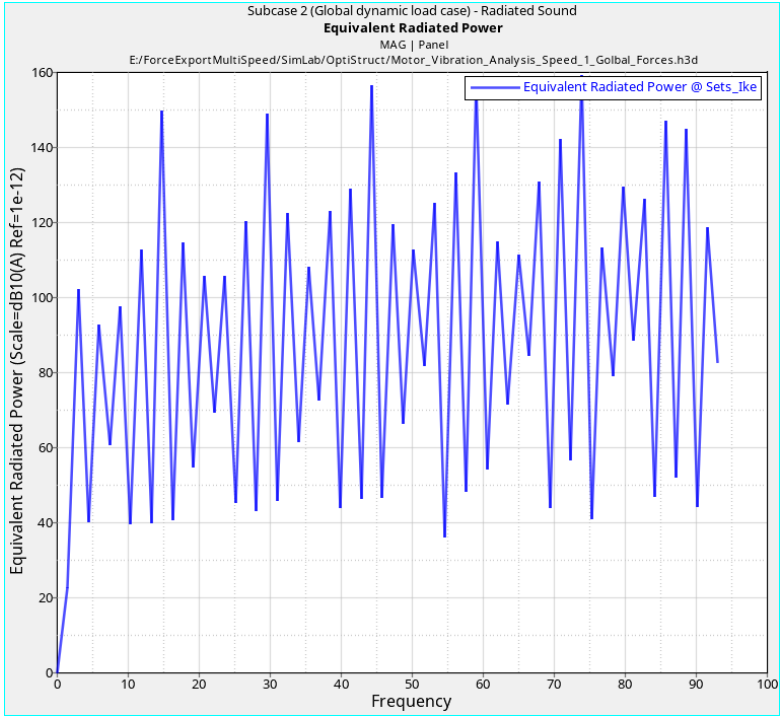




ALTAIR

VIBROACOUSTIC ANALYSIS IN SIMLAB

- Result visualization
  - ERP (equivalent radiated power) curve



V. CONCLUSION

CONCLUSION

- Altair **HyperWorks** provide a convenient analysis process for the vibration problem of electric motors.
- **Flux** is used as the solver for electromagnetic problems.
- Magnetic forces at multispeed point are computed in the mechanical I/O context in **Flux**.
- **OptiStruct** is used as the solver for mechanical problems.
- **SimLab** is used for modeling the device.

THANK YOU

altair.com

#ONLYFORWARD

ALTAIR