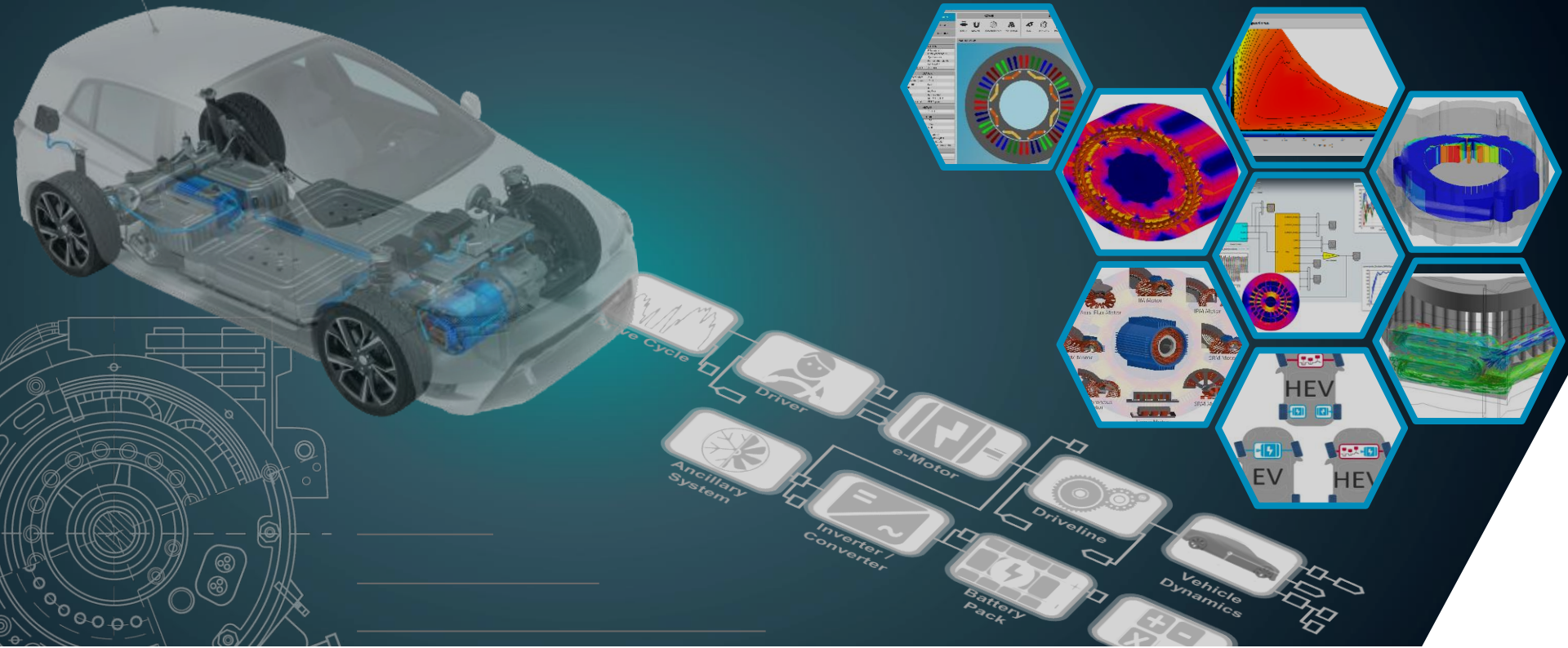
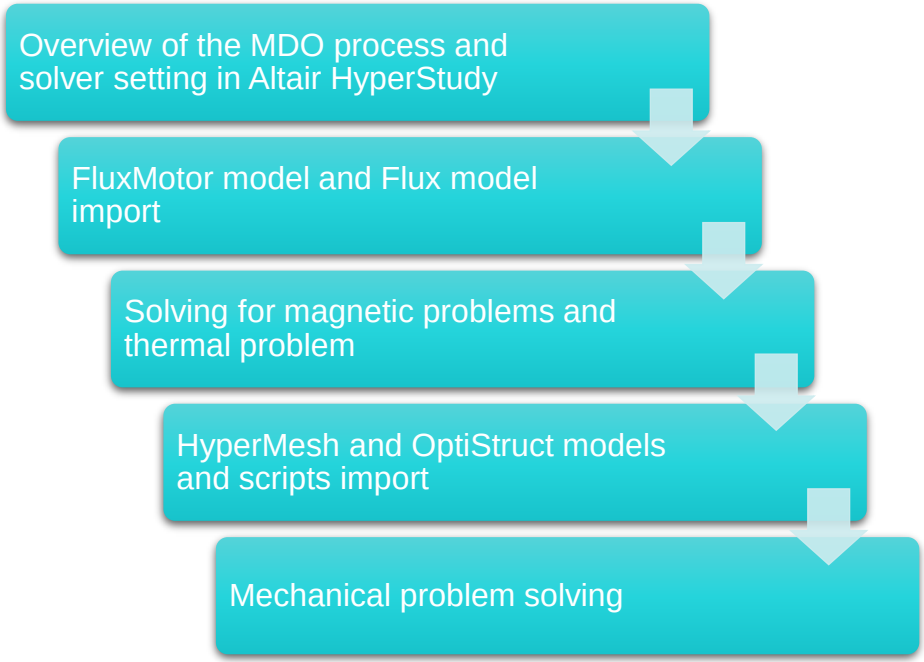


HYPERSTUDY: MULTIDISCIPLINARY DESIGN OPTIMIZATION INITIATION

ALTAIR MULTIDISCIPLINARY DESIGN OPTIMIZATION PLATFORM FOR ELECTRIC MOTORS

October 2021, Altair Flux / FluxMotor Valorization and Support Team

OUTLINE



Input files

- **FluxMotor / HyperStudy connector**: Connector_FM_Hst.fm2hst
- **Flux / HyperStudy connector** Connector_Fx_Hst_BasePoint.F2HST
- **Flux / HyperStudy connector**: Connector_Fx_Hst_OperatingPoint.F2HST
- **Flux / HyperStudy connector** Connector_Fx_Hst_Thermal.F2HST
- **HyperMesh / OptiStruct project and script**
  - reference1.hm
  - Update_motor.tcl

Software

- Altair HyperStudy 2019.1 (or later versions)

Output file

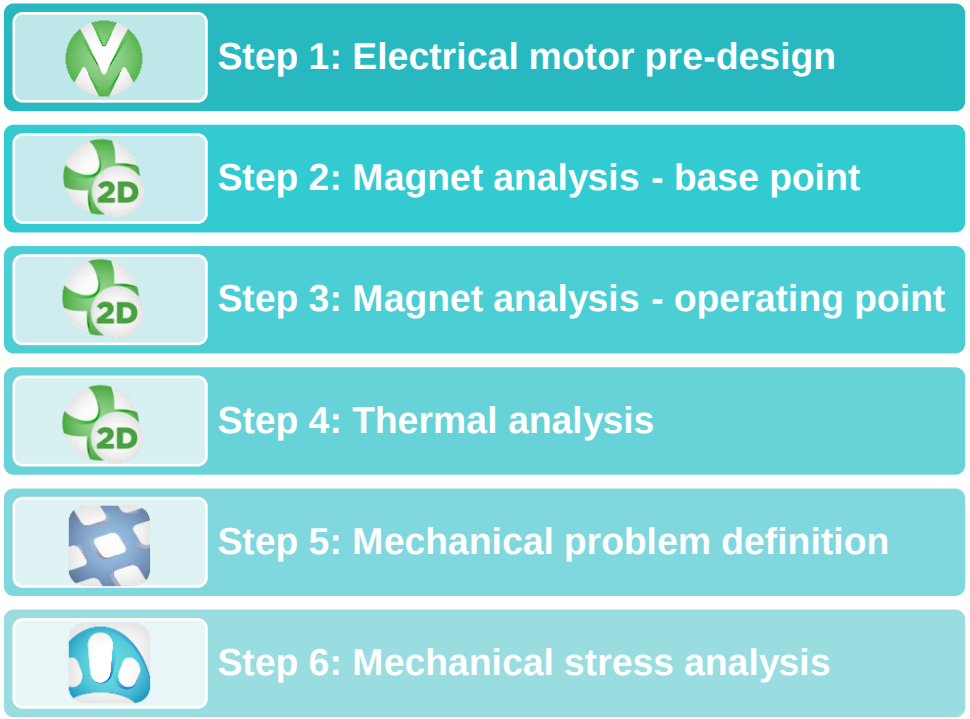
- **HyperStudy project** MDO_EMOTOR_STUDY.hstudy

ALTAIR

OVERVIEW OF THE MDO PROCESS AND THE SOLVER SETTING IN HYPERSTUDY

OVERVIEW OF THE E-MOTOR MDO PROCESS IN HYPERSTUDY

- Multidisciplinary design optimization (MDO) process with Altair HyperWorks for electrical machines



1) The MDO process is built and run with Altair HyperStudy.

2) All the five phases are represented by "Connector" between software.

3) Solvers required for HyperStudy:

- Flux
- FluxMotor
- HM Batch (HyperMesh)
- OptiStruct

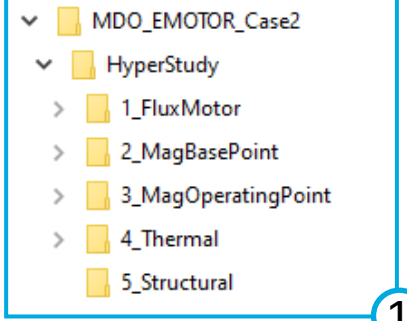
ALTAIR

SOLVER SETTING IN HYPERSTUDY

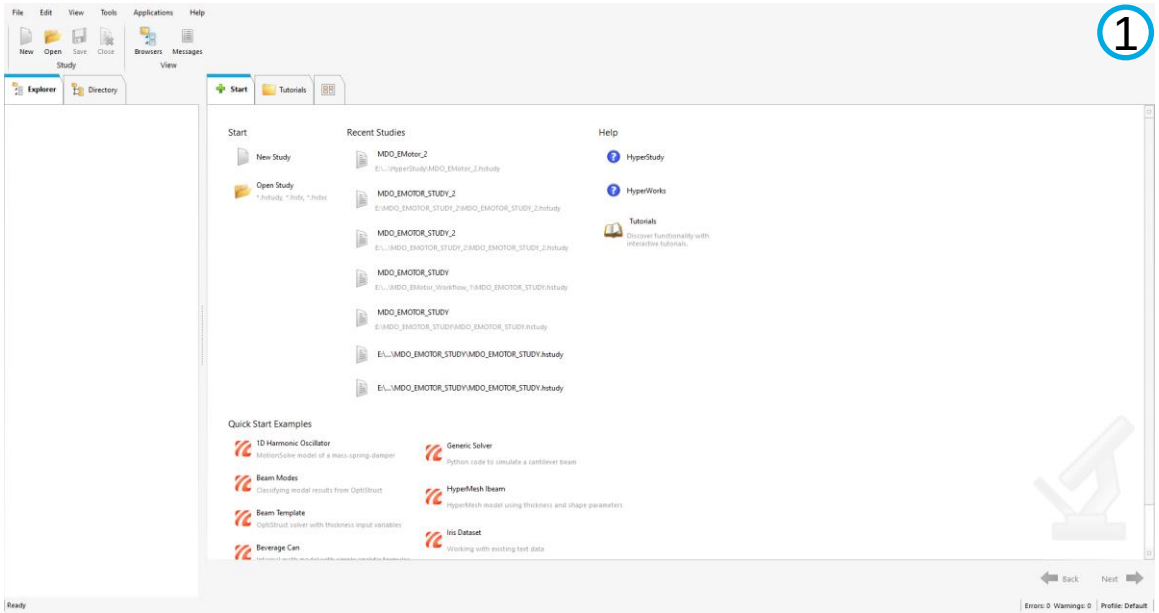
- Prepare the MDO E-Motor project with HyperStudy

Step	Action
1	Open Altair HyperStudy

Project folder structure



1



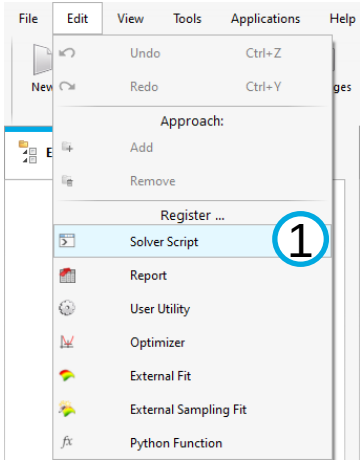
1

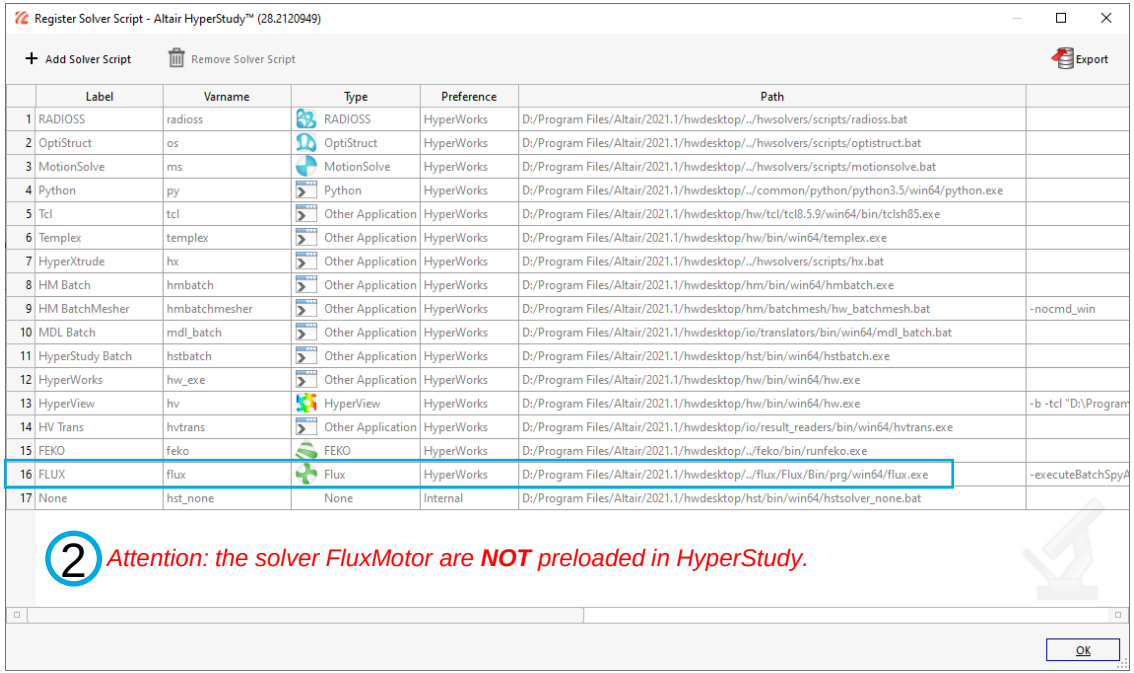
ALTAIR

SOLVER SETTING IN HYPERSTUDY

- Verify the solver script registration in HyperStudy

Step	Action
1	Click on [Edit] – [Register Solver Script]
2	Verify if the “Flux”, “FluxMotor” and “OptiStruct” solvers are already correctly loaded

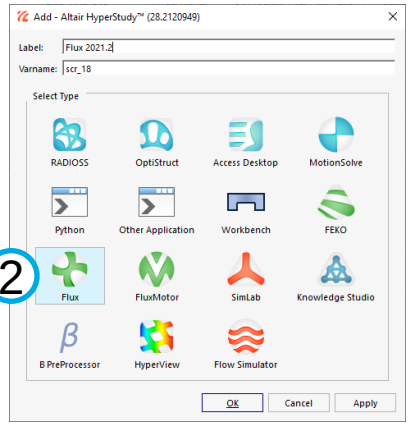


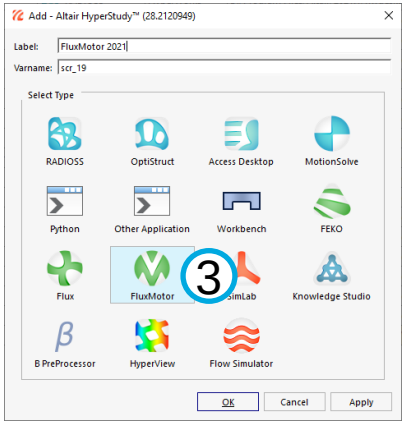


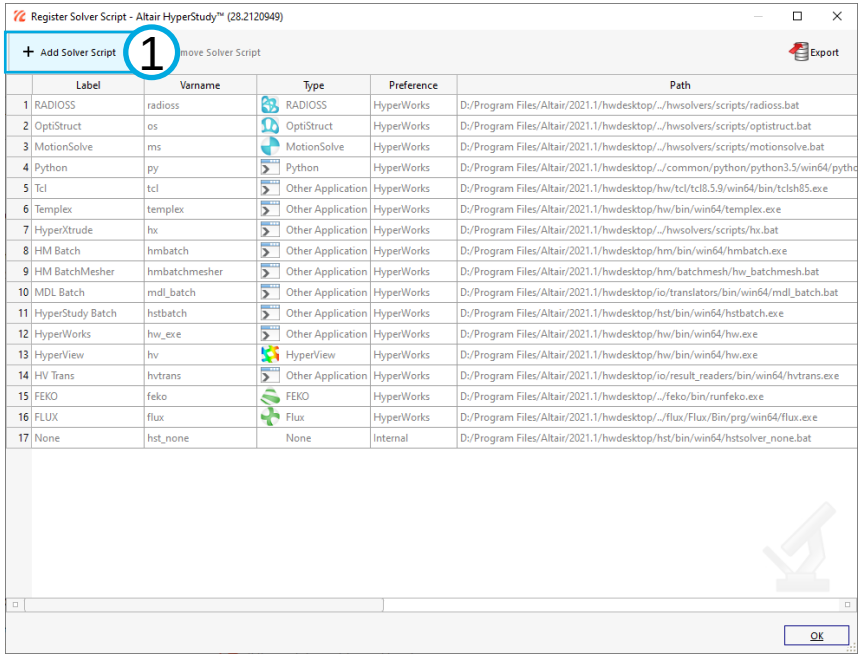
SOLVER SETTING IN HYPERSTUDY

- Register solver scripts for the solvers “Flux” and “FluxMotor”

Step	Action
1	Click on [Add Solver Script] to add new solvers
2	Add the first solver “Flux”
3	Add the second solver “FluxMotor”







Solver recommended: Flux 2021.2 and FluxMotor 2021

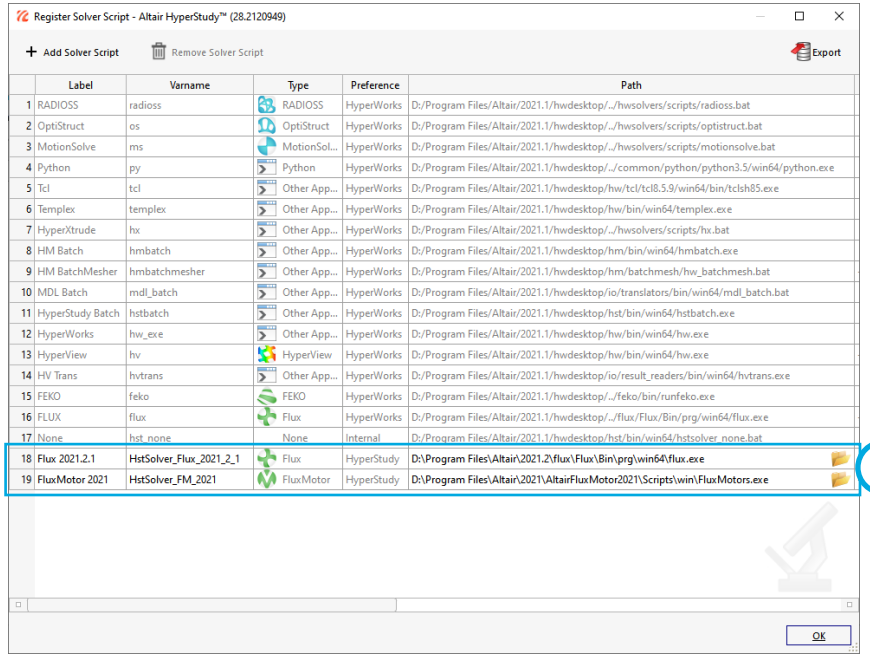
ALTAIR

SOLVER SETTING IN HYPERSTUDY

- Modify the solver settings

Step	Action
1	Modify the solver settings (Label, Varname, and Path) for the two solvers “Flux” and “FluxMotor”

Label	Varname	Path
Flux 2021.2.1	HstSolver_Flux _2021_2_1	<Flux installation directory> /flux/Flux/Bin/prg/win64/Flux.exe
FluxMotor 2021	HstSolver_FM_ 2021	<FluxMotor installation directory> /Scripts/win/FluxMotor.exe

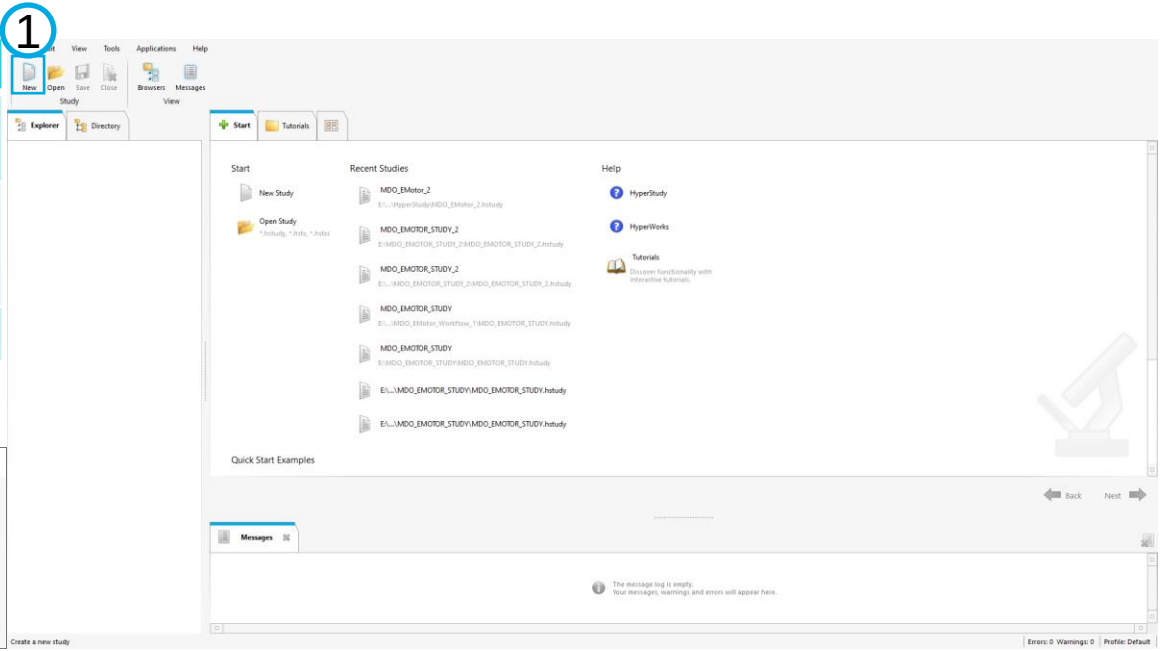


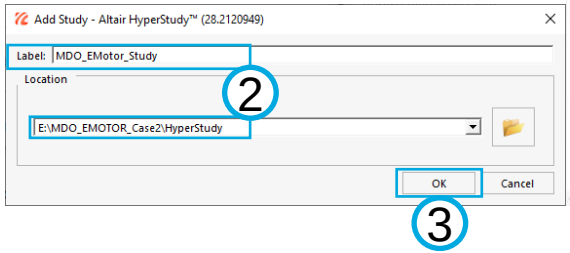
ALTAIR

OPTIMIZATION INITIALIZATION

- Create a new **HyperStudy** project

Step	Action
1	Click on the icon [New]  to start a new HyperStudy project
2	Define the project name “MDO_EMotor_Study”, and select the saving location
3	Click on [OK]





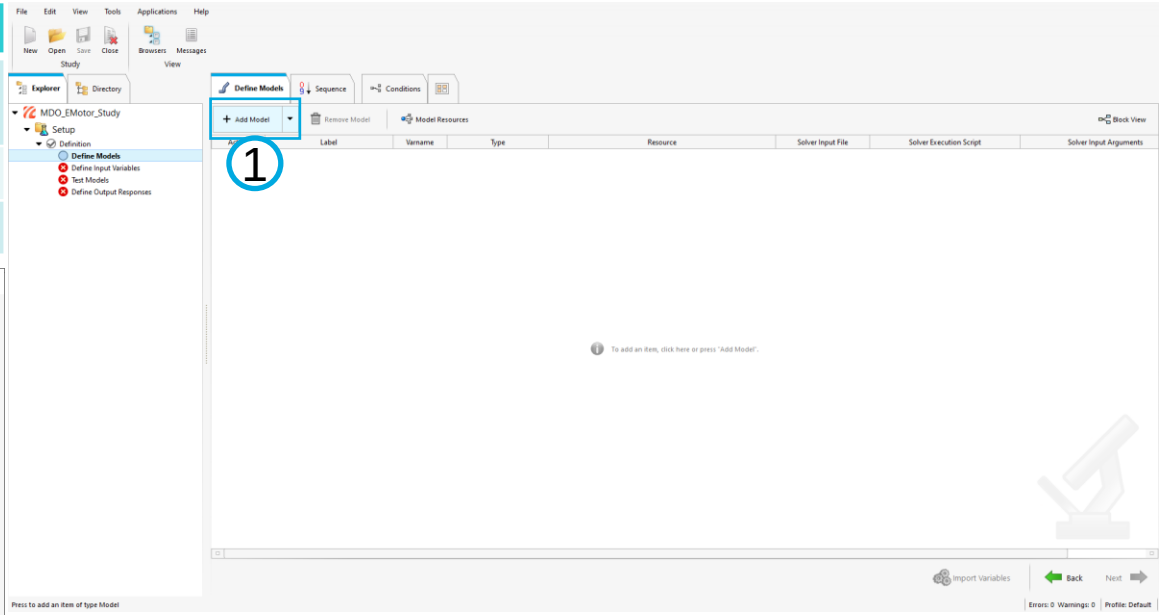
FLUXMOTOR MODEL AND FLUX MODEL IMPORT

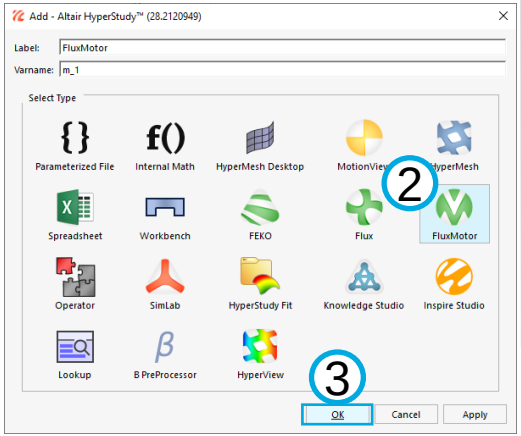
ADD FLUXMOTOR AND FLUX MODELS

FluxMotor model import

- Create a new **FluxMotor** model

Step	Action
1	Click on [Add Model] to create the model “FluxMotor”
2	Select the type “FluxMotor”
3	Click on [OK]





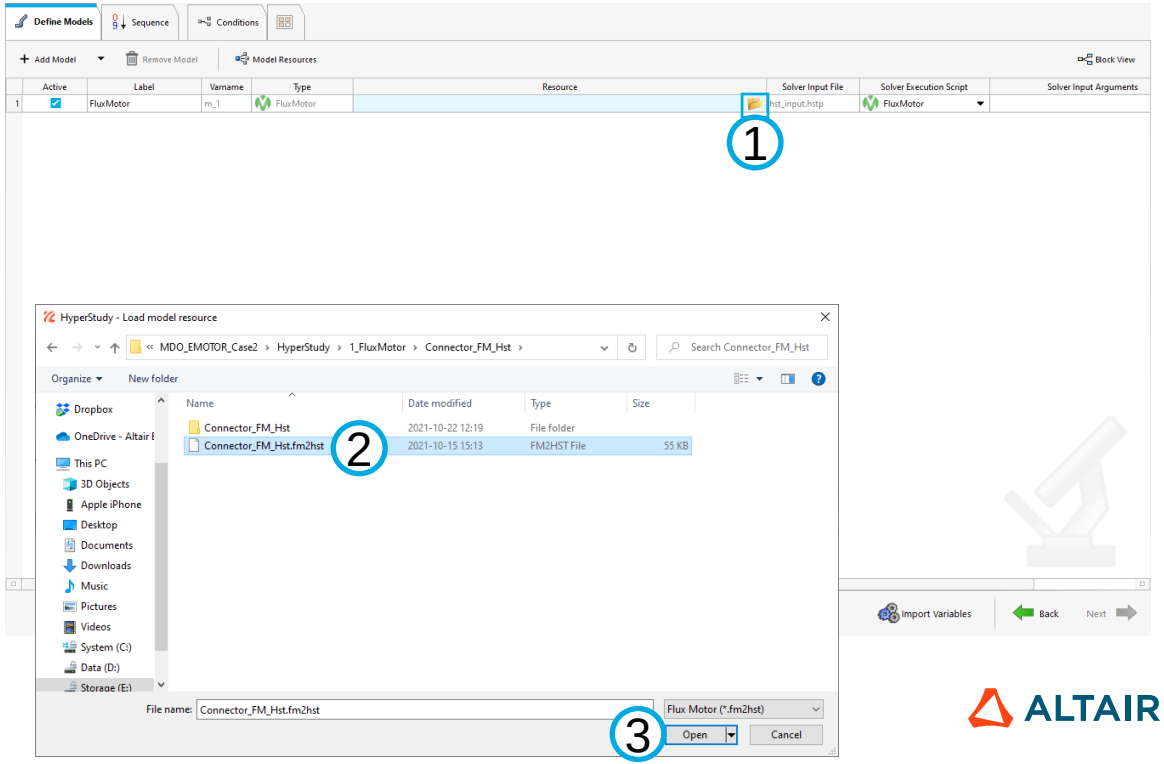
ALTAIR

ADD FLUXMOTOR AND FLUX MODELS

FluxMotor model import

- Link the model to the **FluxMotor / Hyperstudy** connector

Step	Action
1	Click on the icon to define the resource connector path
2	Select the connector “Connector_FM_Hst.fm2hst” in 1_FluxMotor
3	Click on [Open]



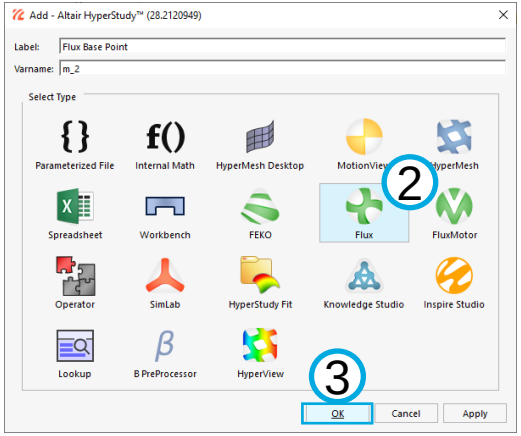
ALTAIR

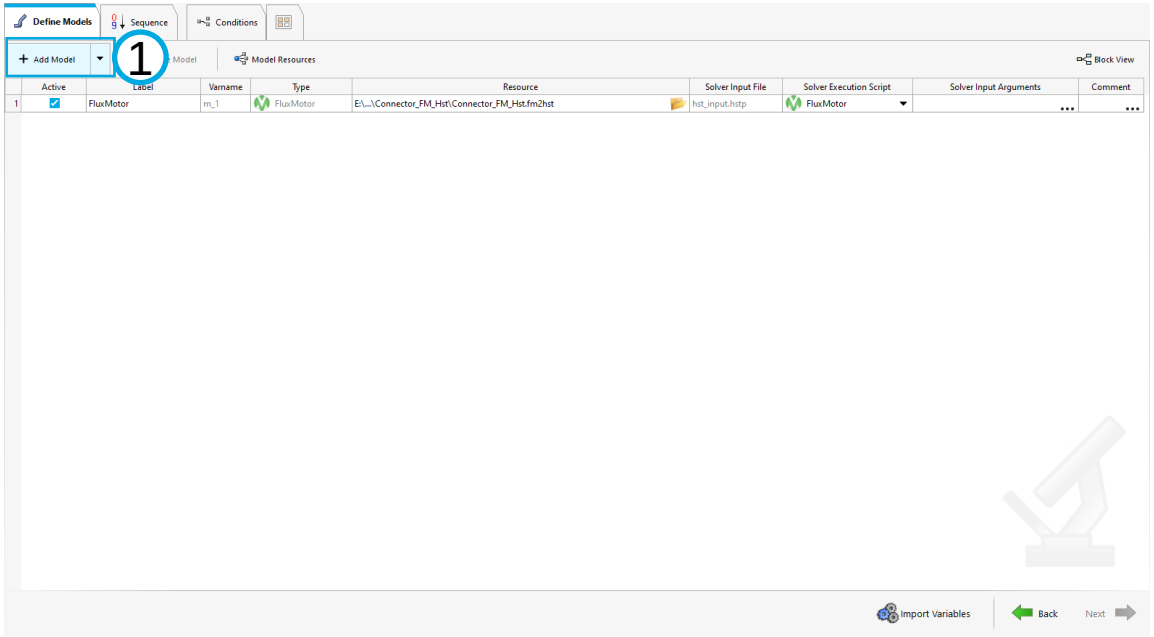
ADD FLUXMOTOR AND FLUX MODELS

Flux model import: magnetic analysis at base speed point

- Create a new **Flux** model

Step	Action
1	Click on [Add Model] to create the model “Flux Base Point”
2	Select the type “Flux”
3	Click on [OK]





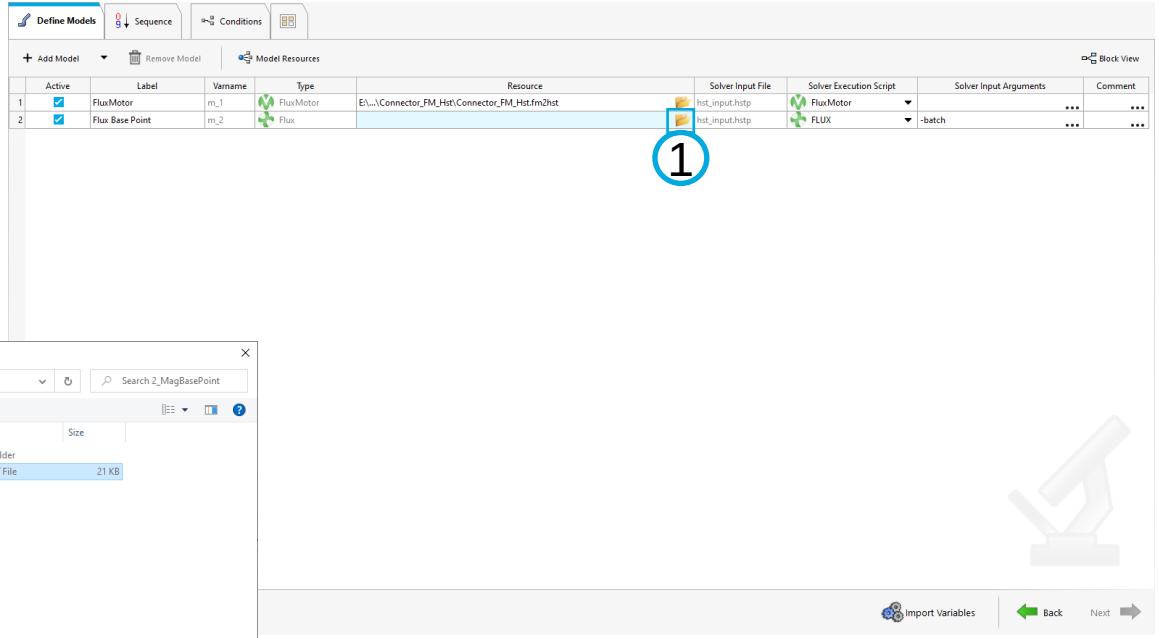
ALTAIR

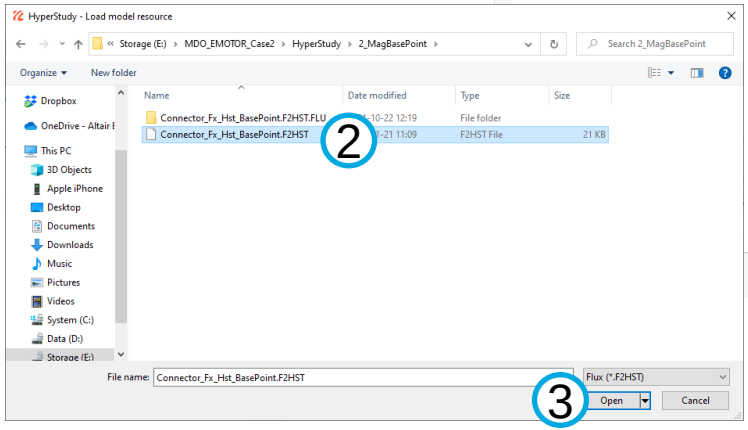
ADD FLUXMOTOR AND FLUX MODELS

Flux model import: magnetic analysis at base speed point

- Link the model to the **Flux / Hyperstudy** connector

Step	Action
1	Click on the icon to define the resource connector path
2	Select the Flux connector “Connector_Fx_Hst_BasePoint .F2HST” in 2_MagBasePoint
3	Click on [Open]





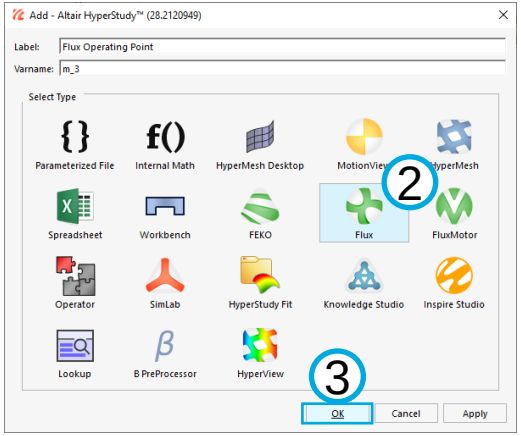
ALTAIR

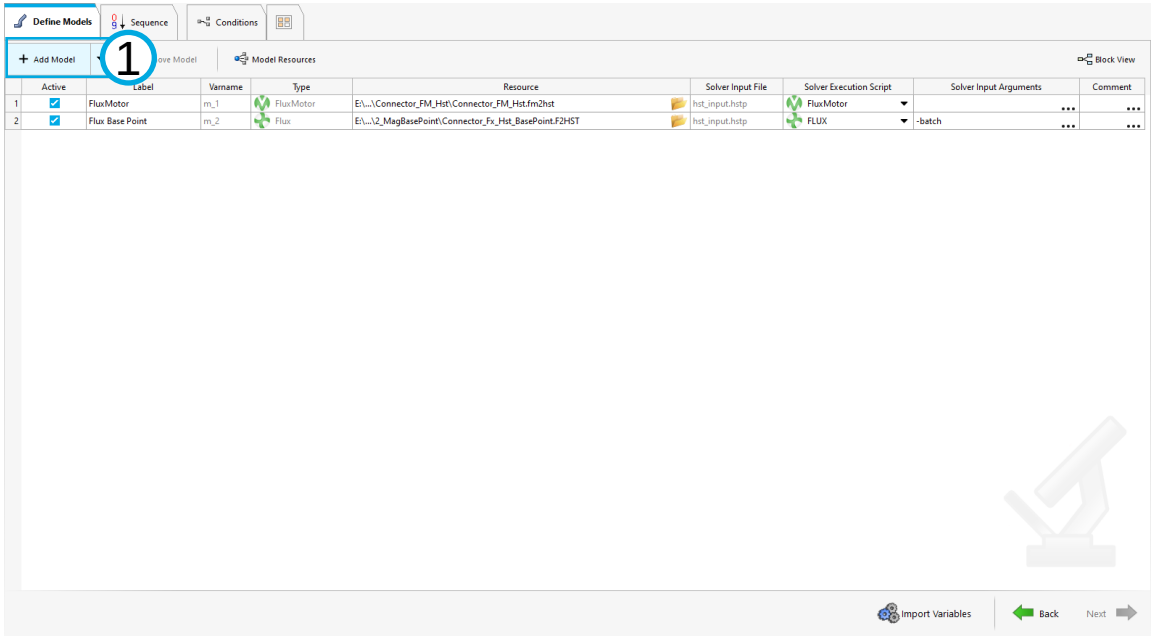
ADD FLUXMOTOR AND FLUX MODELS

Flux model import: magnetic analysis at specific operating point

- Create a new **Flux** model

Step	Action
1	Click on [Add Model] to create the model “Flux Operating Point”
2	Select the type “Flux”
3	Click on [OK]





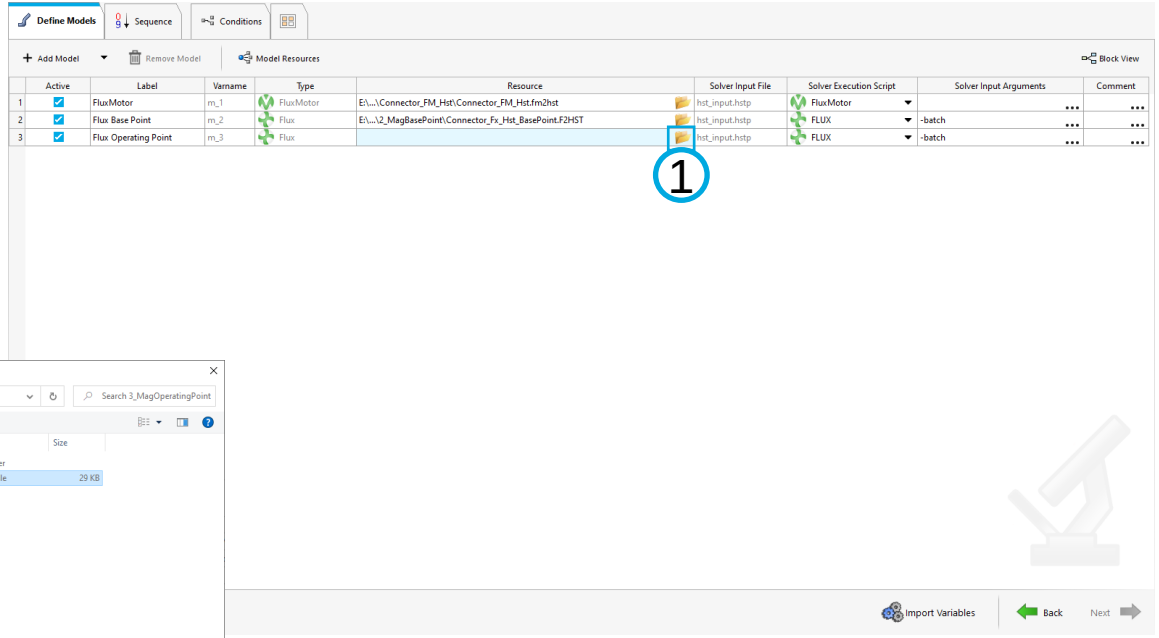
ALTAIR

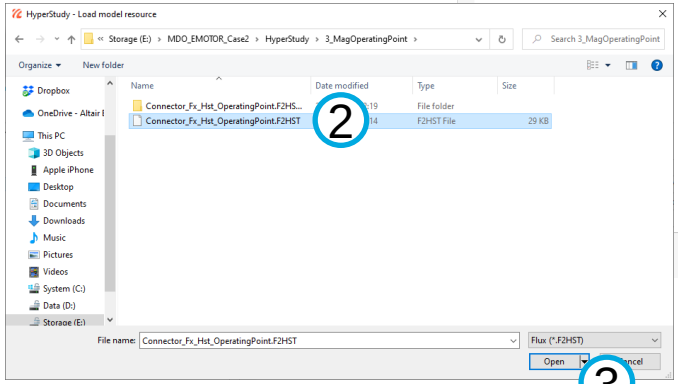
ADD FLUXMOTOR AND FLUX MODELS

Flux model import: magnetic analysis at specific operating point

- Link the model to the **Flux / Hyperstudy** connector

Step	Action
1	Click on the icon to define the resource connector path
2	Select the Flux connector “Connector_Fx_Hst_Operating Point.F2HST” in 3_MagOperatingPoint
3	Click on [Open]





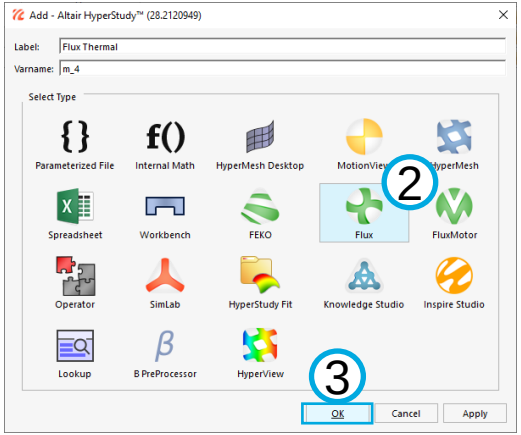
ALTAIR

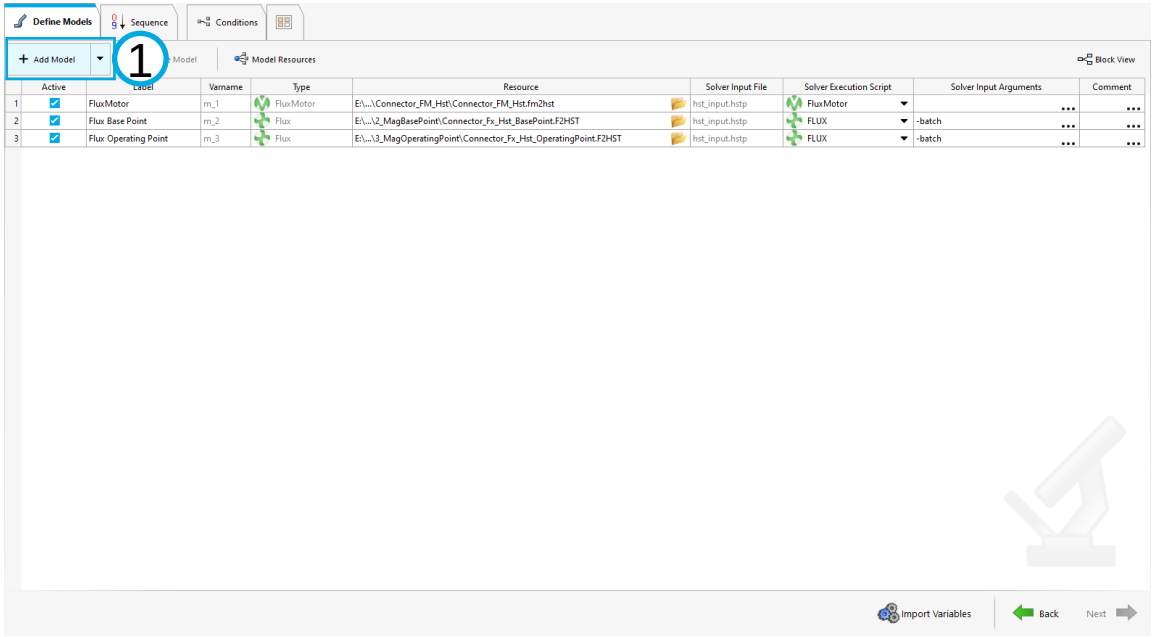
ADD FLUXMOTOR AND FLUX MODELS

Flux model import: thermal analysis

- Create a new **Flux** model

Step	Action
1	Click on [Add Model] to create the model “Flux Thermal”
2	Select the type “Flux”
3	Click on [OK]





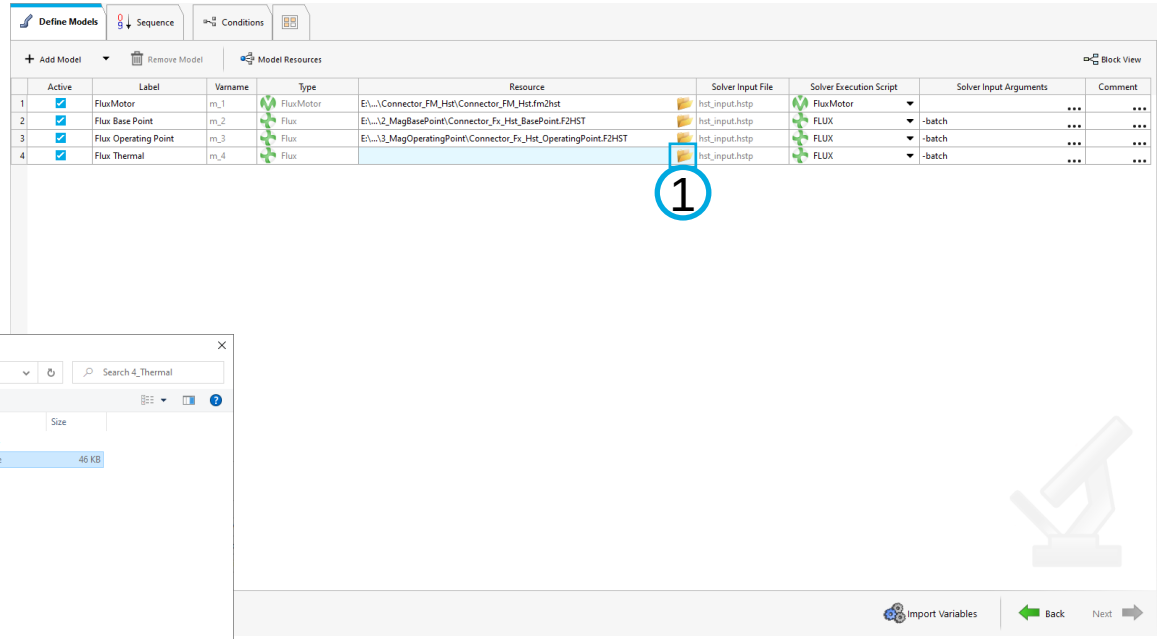
ALTAIR

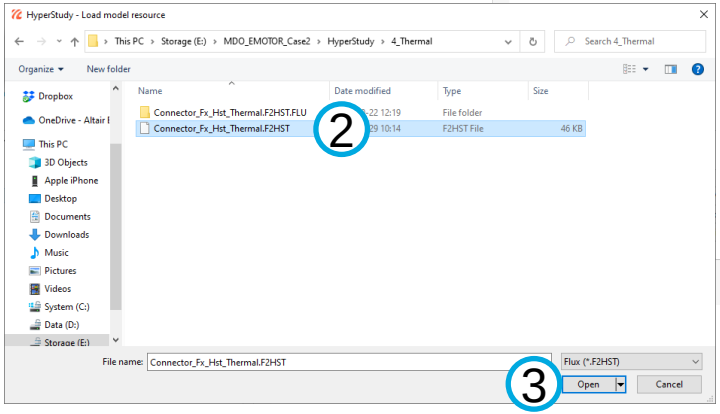
ADD FLUXMOTOR AND FLUX MODELS

Flux model import: thermal analysis

- Link the model to the **Flux / Hyperstudy** connector

Step	Action
1	Click on the icon to define the Flux connector path
2	Select the Flux connector “Connector_Fx_Hst_Thermal.F2HST” in 4_Thermal
3	Click on [Open]





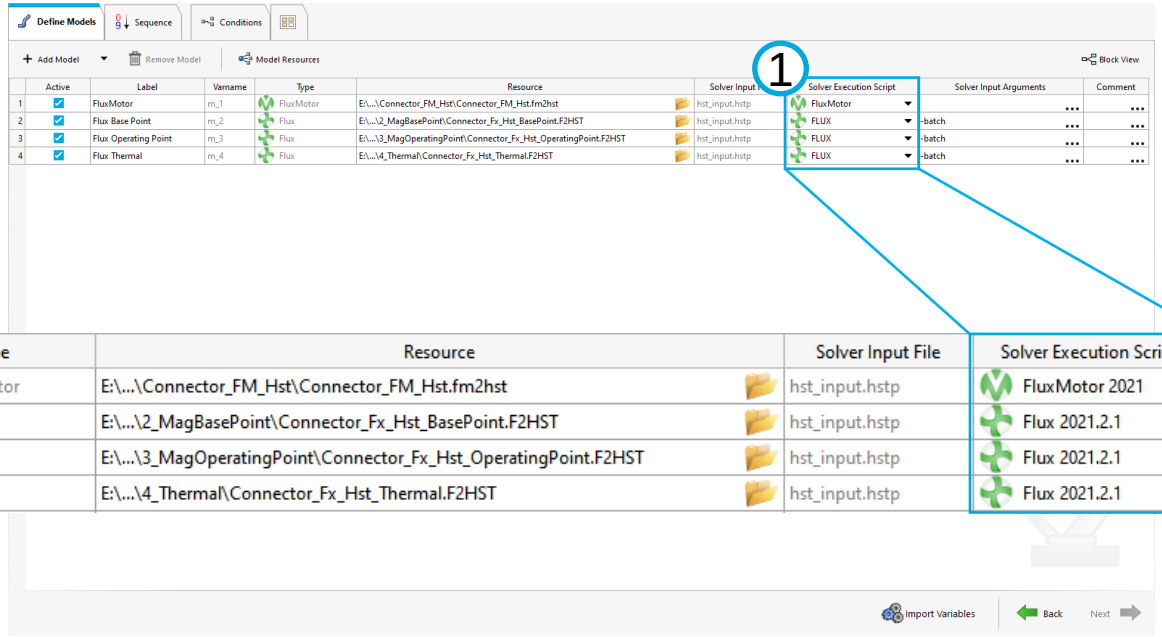
ALTAIR

ADD FLUXMOTOR AND FLUX MODELS

Update solvers (Flux and FluxMotor)

- Update the solver as defined in page 8

Step	Action
1	Click on the flash ▾ to select the solver
2	Select the created solvers



3

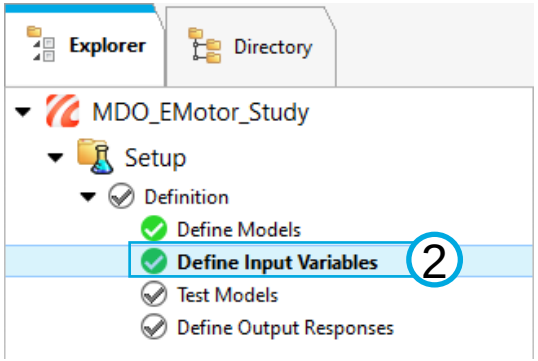
SOLVING FOR MAGNETIC PROBLEMS AND THERMAL PROBLEM

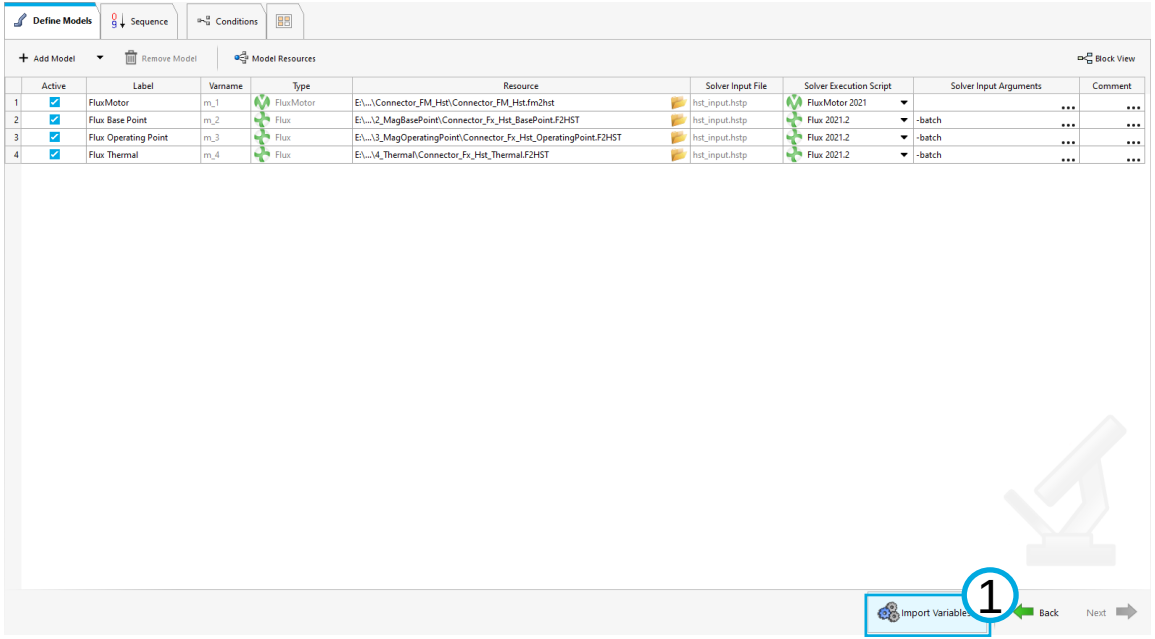
ALTAIR

SOLVING FOR MAGNETIC PROBLEMS AND THERMAL PROBLEM

Model variable import

Step	Action
1	Click on the icon [Import Variables]
2	Click on [Define Input Variables] to verify all the imported variables



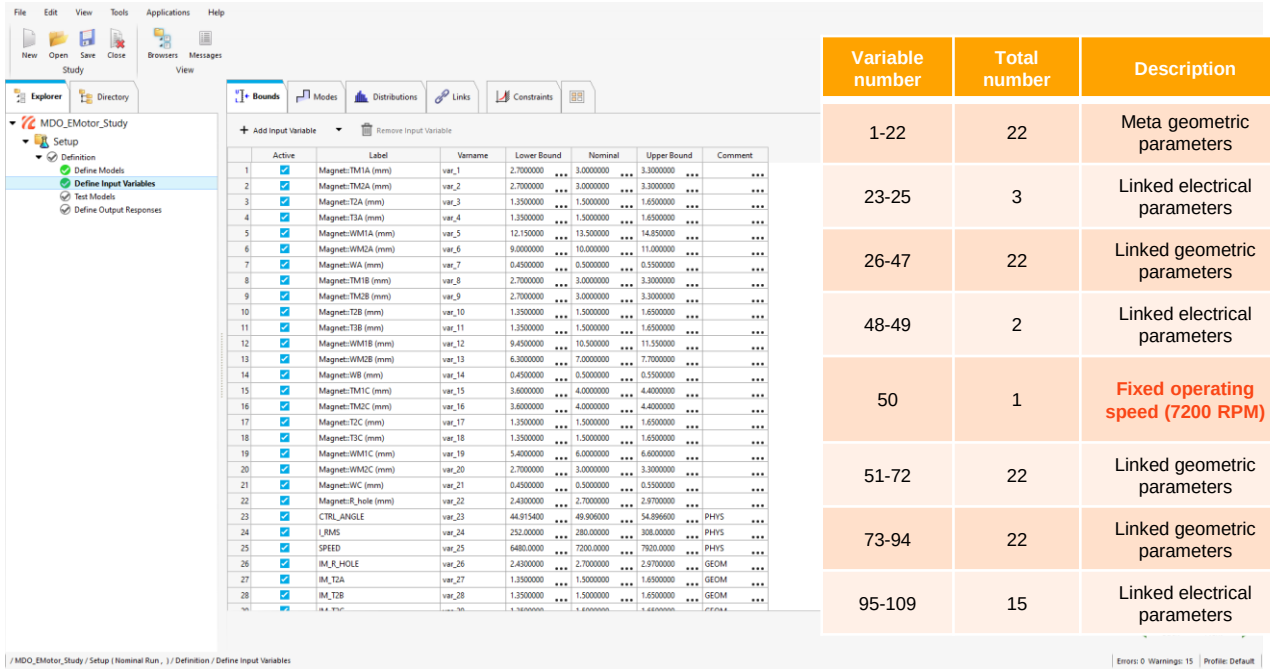


ALTAIR

SOLVING FOR MAGNETIC PROBLEMS AND THERMAL PROBLEM

Model variable import

- Define parameter links



Variable number	Total number	Description	Original model	Action in HyperStudy
1-22	22	Meta geometric parameters	FluxMotor	Modify upper and lower bounds
23-25	3	Linked electrical parameters	Flux Mag Base Point	Link to model output responses
26-47	22	Linked geometric parameters	Flux Mag Base Point	Link to variable 1-22
48-49	2	Linked electrical parameters	Flux Mag Operating Point	Link to model output responses
50	1	**Fixed operating speed (7200 RPM)**	Flux Mag Operating Point	To be **deleted** !! Because the operating speed is a constant
51-72	22	Linked geometric parameters	Flux Mag Operating Point	Link to variable 1-22
73-94	22	Linked geometric parameters	Flux Thermal	Link to variable 1-22
95-109	15	Linked electrical parameters	Flux Thermal	Link to model output responses

Note: all the modification for variable information in HyperStudy can be copied and pasted from the file "ParameterTableInHS.xlsx"

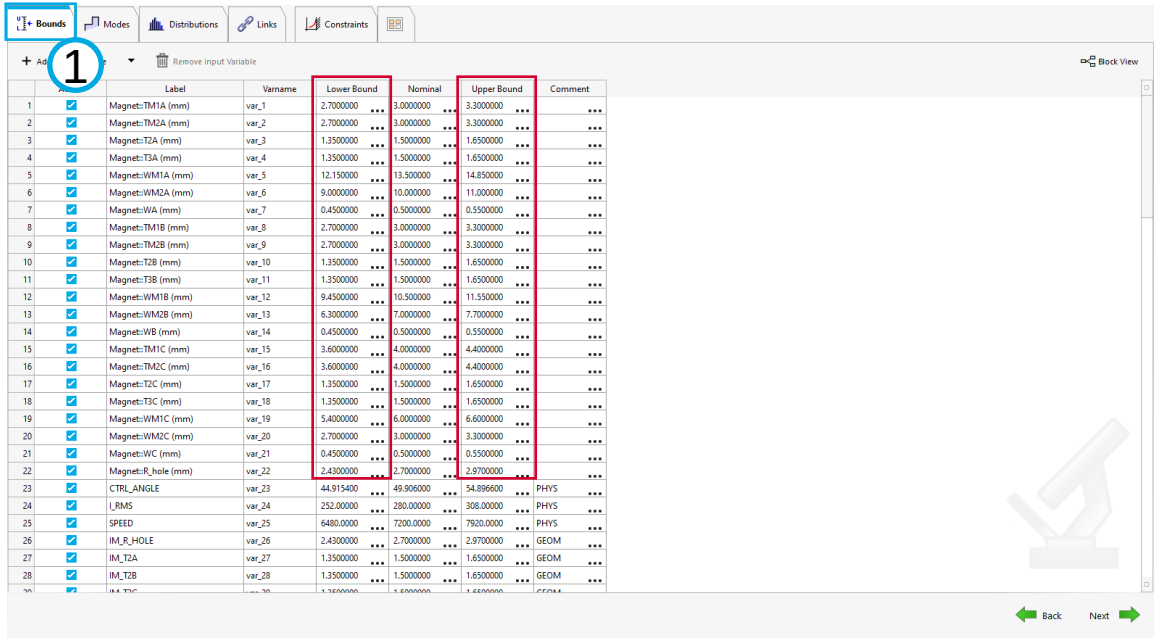
ALTAIR

SOLVING FOR MAGNETIC PROBLEMS AND THERMAL PROBLEM

Model variable import

- Define parameter variation intervals for geometric parameters from the FluxMotor model

Step	Action
1	Click on [Bounds]
2	Define lower bounds and upper bounds for the meta motor geometric parameters (variable 1-22)



		Label	Varname	Lower Bound	Nominal	Upper Bound	Comment
1	☑	Magnet::TM1A (mm)	var_1	2.7000000	3.0000000	3.3000000	
2	☑	Magnet::TM2A (mm)	var_2	2.7000000	3.0000000	3.3000000	
3	☑	Magnet::T2A (mm)	var_3	1.3500000	1.5000000	1.6500000	
4	☑	Magnet::T3A (mm)	var_4	1.3500000	1.5000000	1.6500000	
5	☑	Magnet::WM1A (mm)	var_5	12.150000	13.500000	14.850000	
6	☑	Magnet::WM2A (mm)	var_6	9.0000000	10.000000	11.000000	
7	☑	Magnet::WA (mm)	var_7	0.4500000	0.5000000	0.5500000	
8	☑	Magnet::TM1B (mm)	var_8	2.7000000	3.0000000	3.3000000	
9	☑	Magnet::TM2B (mm)	var_9	2.7000000	3.0000000	3.3000000	
10	☑	Magnet::T2B (mm)	var_10	1.3500000	1.5000000	1.6500000	
11	☑	Magnet::T3B (mm)	var_11	1.3500000	1.5000000	1.6500000	
12	☑	Magnet::WM1B (mm)	var_12	9.4500000	10.500000	11.550000	
13	☑	Magnet::WM2B (mm)	var_13	6.3000000	7.0000000	7.7000000	
14	☑	Magnet::WB (mm)	var_14	0.4500000	0.5000000	0.5500000	
15	☑	Magnet::TM1C (mm)	var_15	3.6000000	4.0000000	4.4000000	
16	☑	Magnet::TM2C (mm)	var_16	3.6000000	4.0000000	4.4000000	
17	☑	Magnet::T2C (mm)	var_17	1.3500000	1.5000000	1.6500000	
18	☑	Magnet::T3C (mm)	var_18	1.3500000	1.5000000	1.6500000	
19	☑	Magnet::WM1C (mm)	var_19	5.4000000	6.0000000	6.6000000	
20	☑	Magnet::WM2C (mm)	var_20	2.7000000	3.0000000	3.3000000	
21	☑	Magnet::WC (mm)	var_21	0.4500000	0.5000000	0.5500000	
22	☑	Magnet::R_hole (mm)	var_22	2.4300000	2.7000000	2.9700000	
23	☑	CTRL_ANGLE	var_23	44.915400	49.906000	54.896600	PHYS
24	☑	I_RMS	var_24	252.00000	280.00000	308.00000	PHYS
25	☑	SPEED	var_25	6480.0000	7200.0000	7920.0000	PHYS
26	☑	IM_R_HOLE	var_26	2.4300000	2.7000000	2.9700000	GEOM
27	☑	IM_T2A	var_27	1.3500000	1.5000000	1.6500000	GEOM
28	☑	IM_T2B	var_28	1.3500000	1.5000000	1.6500000	GEOM

Note: the bound values can be copied and pasted from the file "ParameterTableInHS.xlsx"

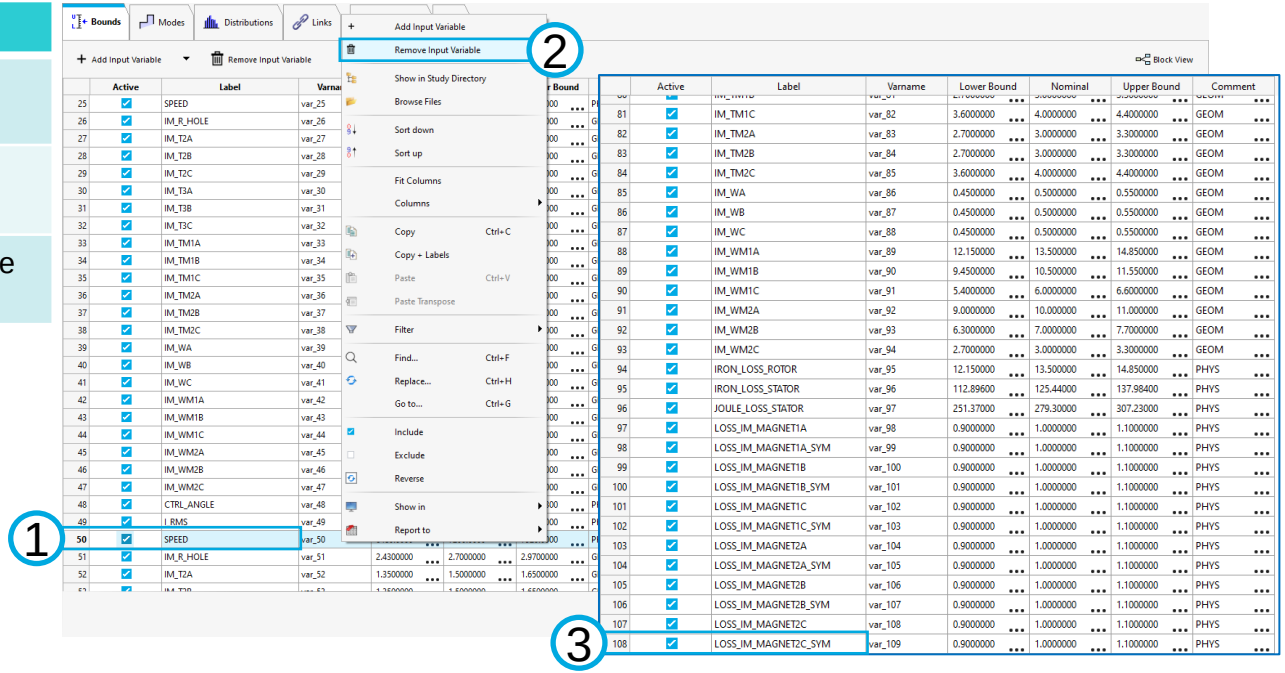
ALTAIR

SOLVING FOR MAGNETIC PROBLEMS AND THERMAL PROBLEM

Model variable import

- Delete the input variable “SPEED” from operating point model

Step	Action
1	Select the No.50 variable “SPEED”
2	Right click and click on [Remove Input Variables]
3	Verify the total input variable number is 108



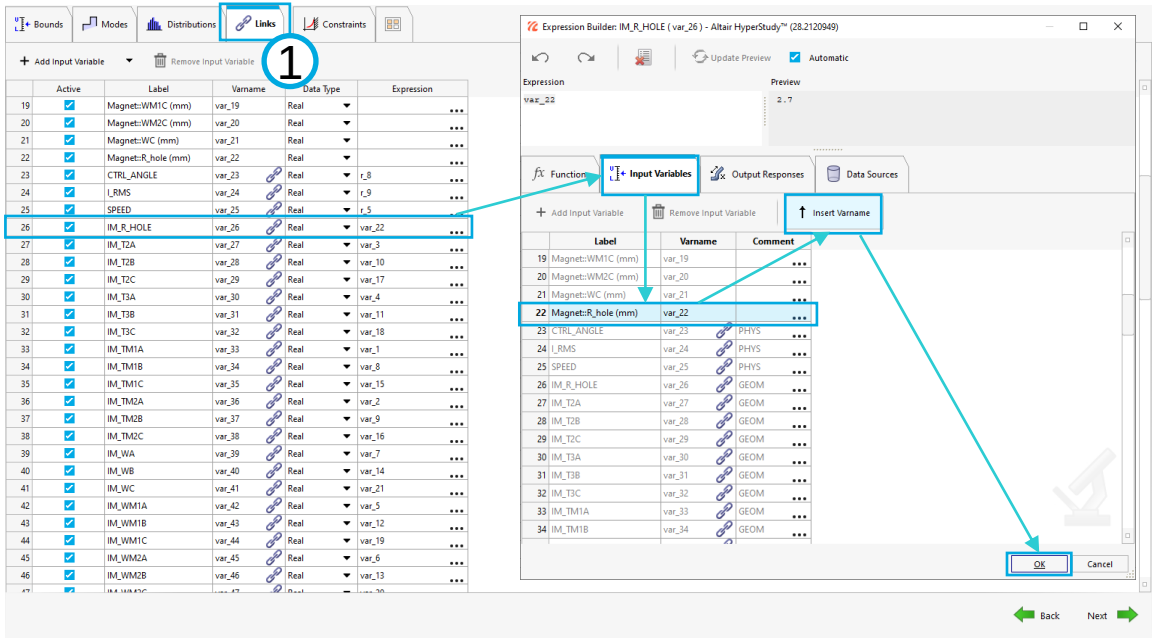
ALTAIR

SOLVING FOR MAGNETIC PROBLEMS AND THERMAL PROBLEM

Model variable import

- Define parameter links for geometric parameters between models

Step	Action
1	Click on [Links]
2	Link all the geometric parameters from Flux models (26-47, 51-72, 73-94) to the meta geometric parameters form FluxMotor model (1-22).
2.a	Example: for the variable 26: IM_R_HOLE - Click on […] - Select [Input Variables], - Select the target variable 22 - Click on [Insert Varname] - Click on [OK]



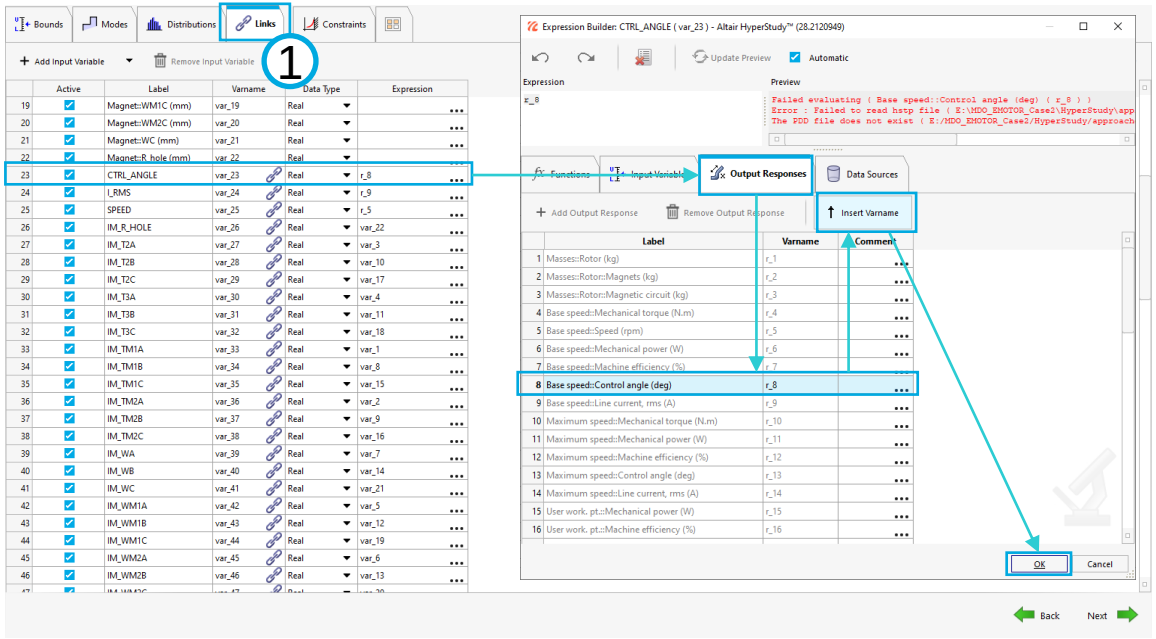
ALTAIR

SOLVING FOR MAGNETIC PROBLEMS AND THERMAL PROBLEM

Model variable import

- Define parameter links between electrical parameters and output responses

Step	Action
1	Click on [Links]
2	Link all the electrical parameters (23-25, 48-50, 95-109) to output responses
2.a	Example: for the variable 23: CTRL_ANGLE - Click on […] - Select [Output Responses], - Select the target response 8 - Click on [Insert Varname] - Click on [OK]

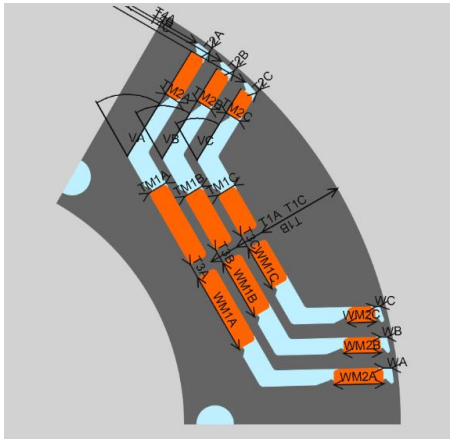


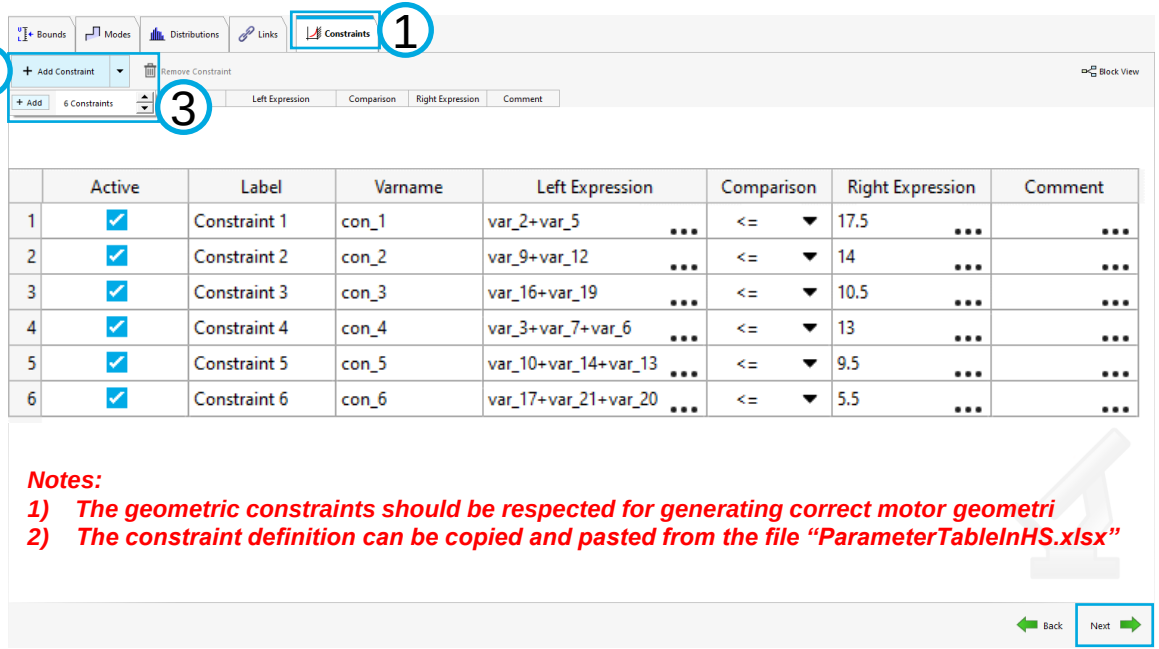
SOLVING FOR MAGNETIC PROBLEMS AND THERMAL PROBLEM

Model variable import

- Define parameter constraints

Step	Action
1	Click on [Constraints]
2	Click on [Add Constraints]
3	Add the following 6 constraints
4	Click on the icon Next





	Active	Label	Varname	Left Expression	Comparison	Right Expression	Comment
1	☑	Constraint 1	con_1	var_2+var_5	<=	17.5	
2	☑	Constraint 2	con_2	var_9+var_12	<=	14	
3	☑	Constraint 3	con_3	var_16+var_19	<=	10.5	
4	☑	Constraint 4	con_4	var_3+var_7+var_6	<=	13	
5	☑	Constraint 5	con_5	var_10+var_14+var_13	<=	9.5	
6	☑	Constraint 6	con_6	var_17+var_21+var_20	<=	5.5	

Notes:

1) The geometric constraints should be respected for generating correct motor geometri

2) The constraint definition can be copied and pasted from the file "ParameterTableInHS.xlsx"

Back Next

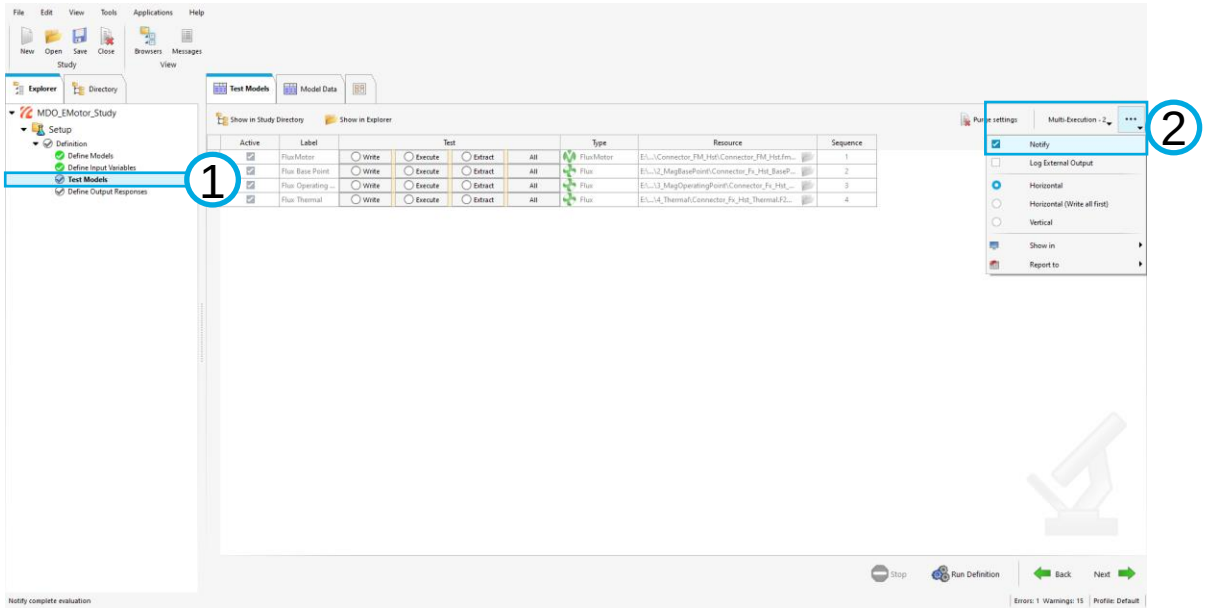
ALTAIR

SOLVING FOR MAGNETIC PROBLEMS AND THERMAL PROBLEM

Model solving

- Test model definition

Step	Action
1	Click on [Test Models]
2	Click on the icon ···, Click on [Notify]



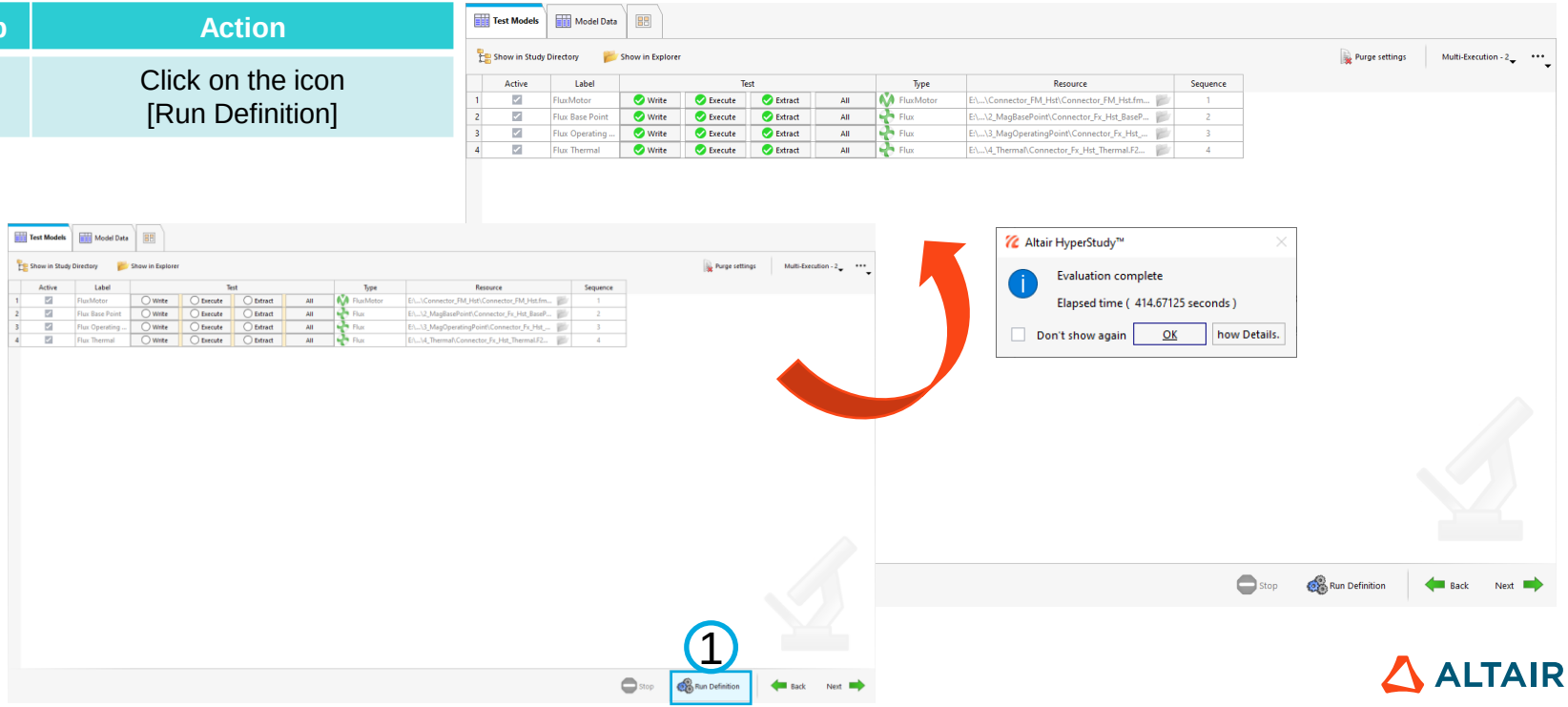
ALTAIR

SOLVING FOR MAGNETIC PROBLEMS AND THERMAL PROBLEM

Model solving

- Test model definition

Step	Action
1	Click on the icon [Run Definition]



ALTAIR

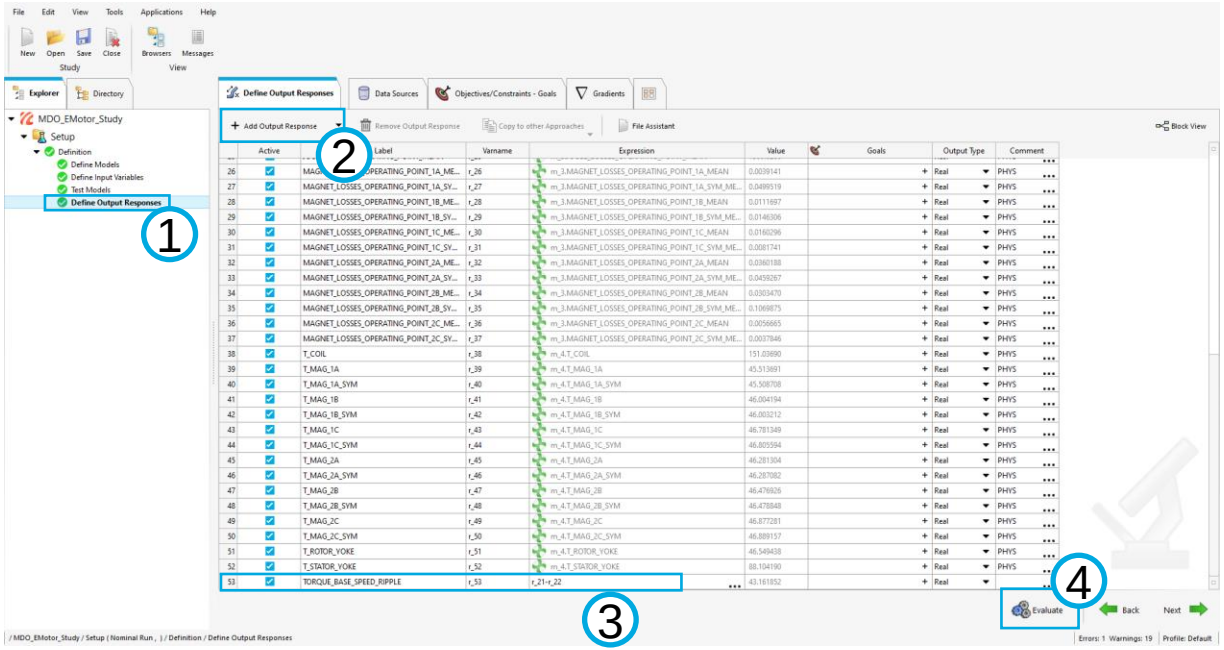
SOLVING FOR MAGNETIC PROBLEMS AND THERMAL PROBLEM

Model output responses

- Create a new output response for the magnetic torque ripple value

Step	Action
1	Click on [Define Output Response]
2	Click on [Add output Response]
3	Define the “Label” and “Expression”
4	Click on [Evaluate]

Label	TORQUE_BASE_SPEED_RIPPLE
Expression	Expression in HS: r_21-r_22 TORQUE_BASE_SPEED_MAX – TORQUE_BASE_SPEED_MIN



ALTAIR

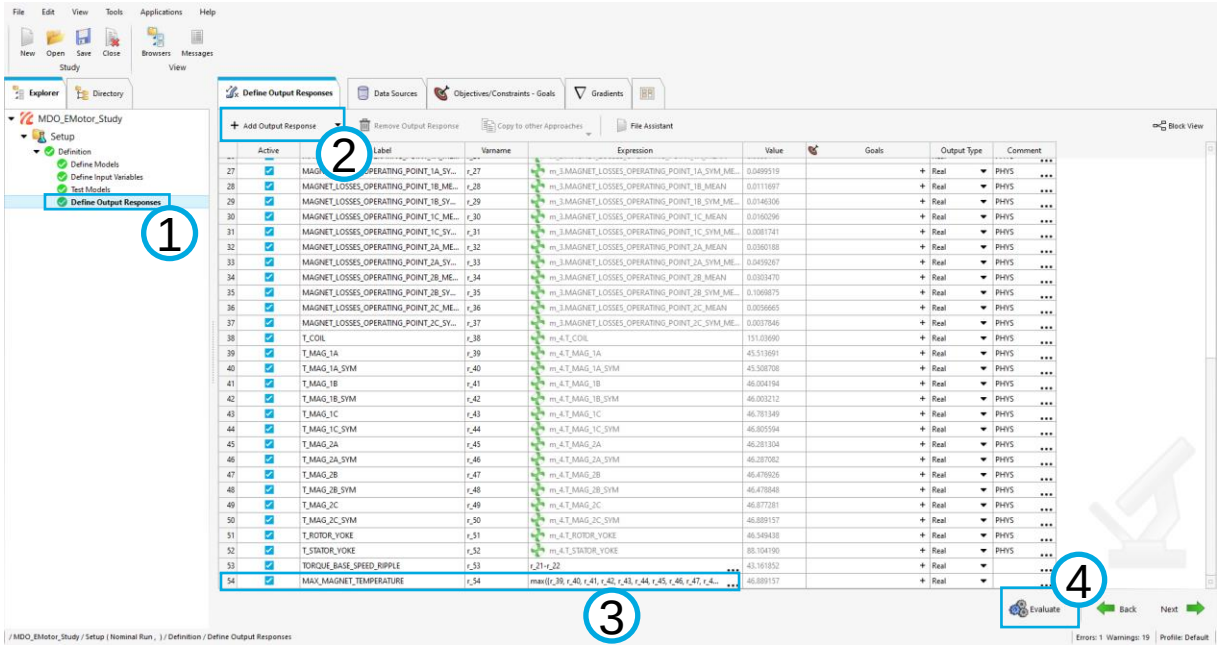
SOLVING FOR MAGNETIC PROBLEMS AND THERMAL PROBLEM

Model output responses

- Create a new output response for the maximum magnet temperature value

Step	Action
1	Click on [Define Output Response]
2	Click on [Add output Response]
3	Define the “Label” and “Expression”
4	Click on [Evaluate]

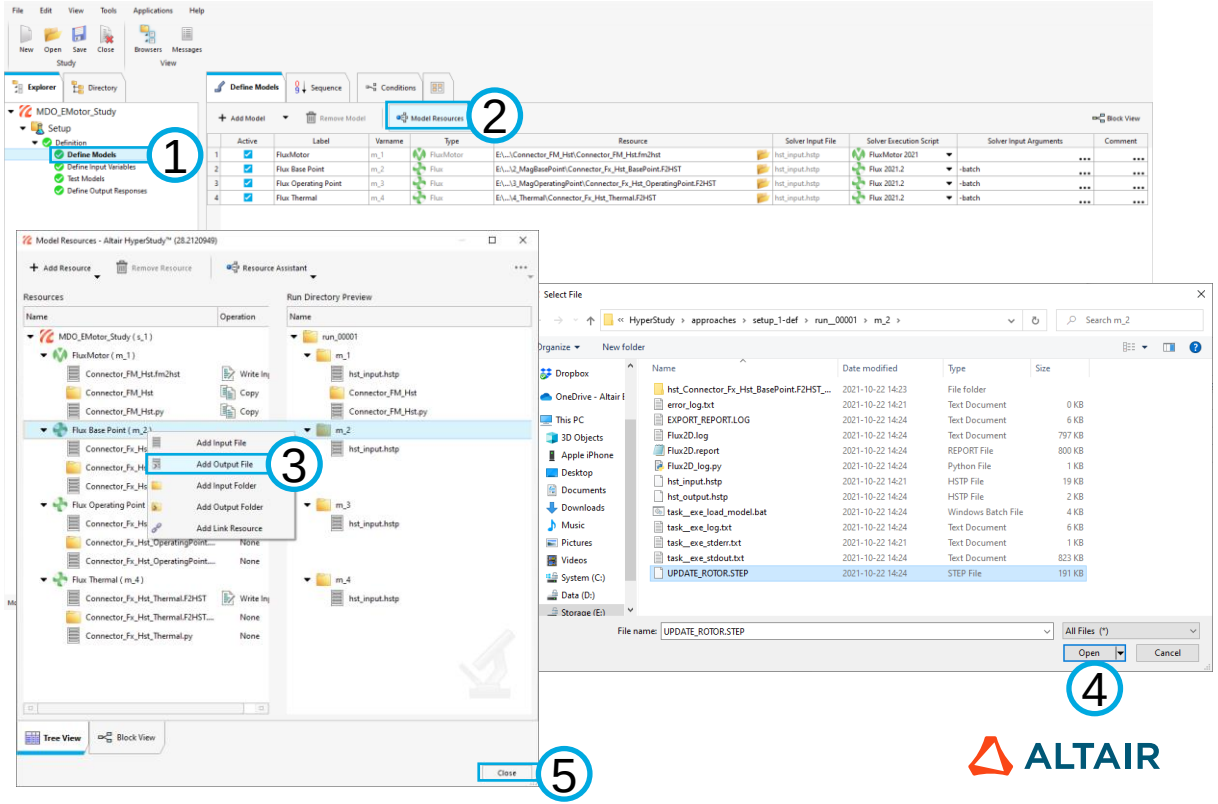
Label	MAX_MAGNET_TEMPERATURE
Expression	max({r_39, r_40, r_41, r_42, r_43, r_44, r_45, r_46, r_47, r_48, r_49, r_50})



SOLVING FOR MAGNETIC PROBLEMS AND THERMAL PROBLEM

Define the output motor geometry file (.STEP) for structural analysis

Step	Action
1	Click on [Define Models]
2	Click on [Model Resources]
3	Right click on the model [Flux Base Speed], click on [Add Output File]
4	Select on the output file [UPDATE_ROTOR.STEP], click on [Open]
5	Click on [Close]



ALTAIR

HYPERMESH AND OPTISTRUCT MODELS AND SCRIPTS IMPORT

ALTAIR

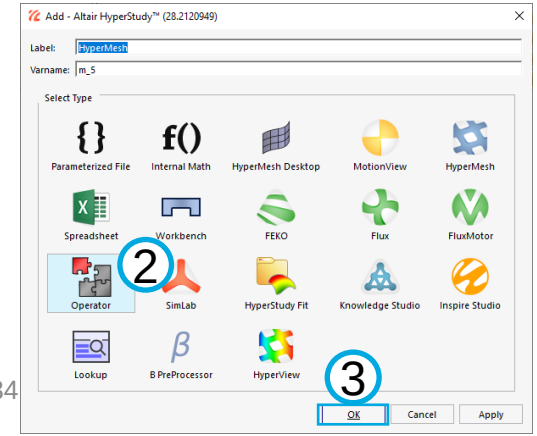
ADD HYPERMESH AND OPTISTRUCT MODELS

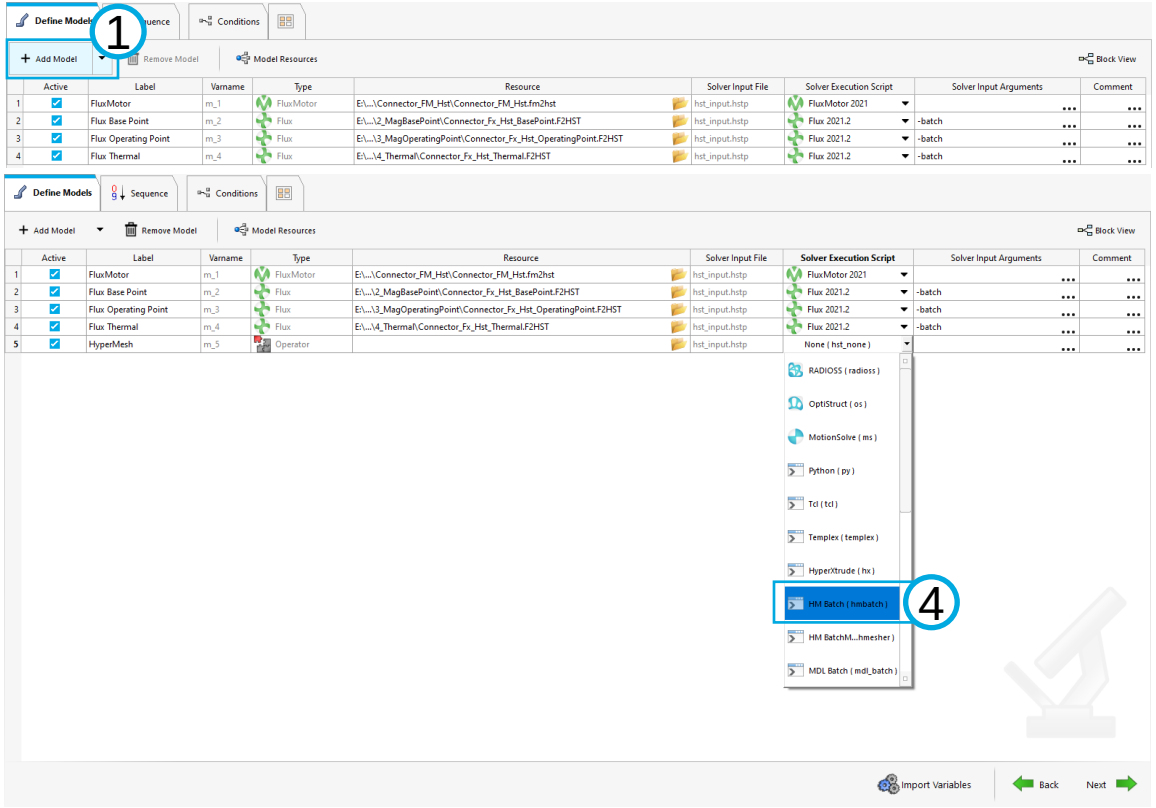
HyperMesh | OptiStruct

HyperMesh model import

- Create a new **Operator**

Step	Action
1	Click on [Add Model] to create the model “HyperMesh”
2	Select the type “Operator”
3	Click on [OK]
4	Select the solver “HM Batch”





ALTAIR

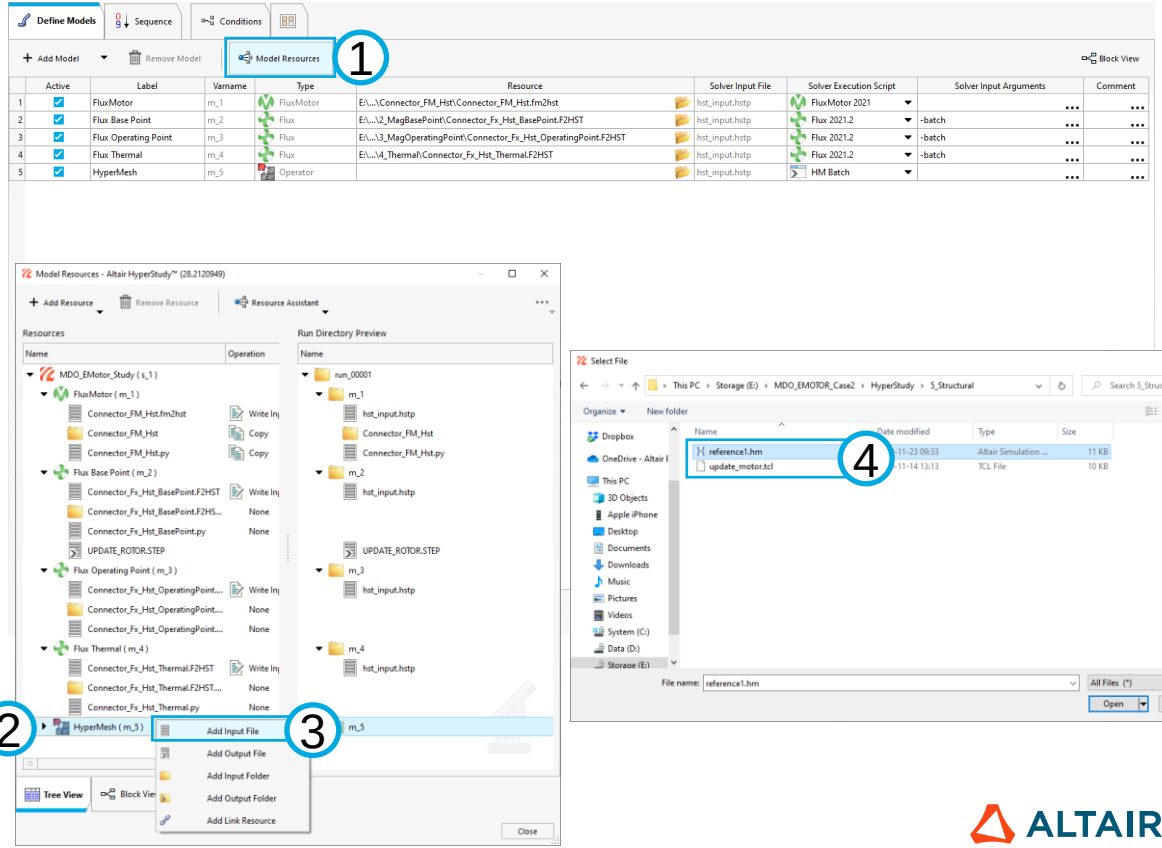
ADD HYPERMESH AND OPTISTRUCT MODELS

HyperMesh | OptiStruct

HyperMesh model import

- Add input files

Step	Action
1	Click on [Model Resource]
2	Right click on the model “HyperMesh”
3	Click on [Add Input File]
4	Add the two files: - reference1.hm - update_motor.tcl in the folder “5_Structural”



ALTAIR

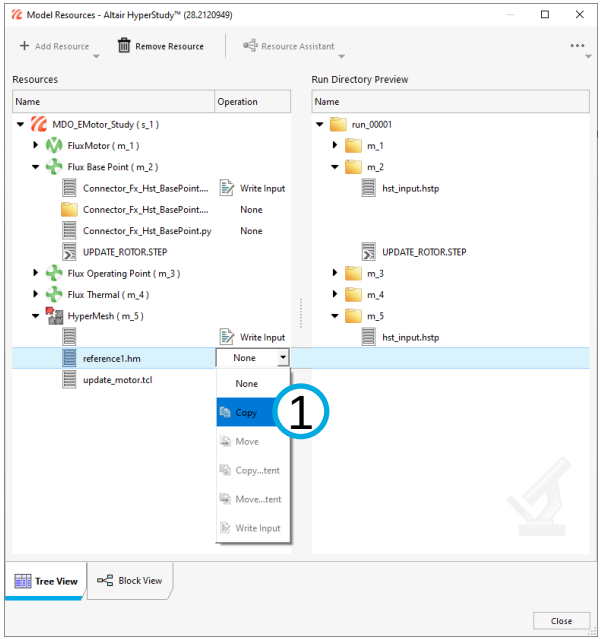
ADD HYPERMESH AND OPTISTRUCT MODELS

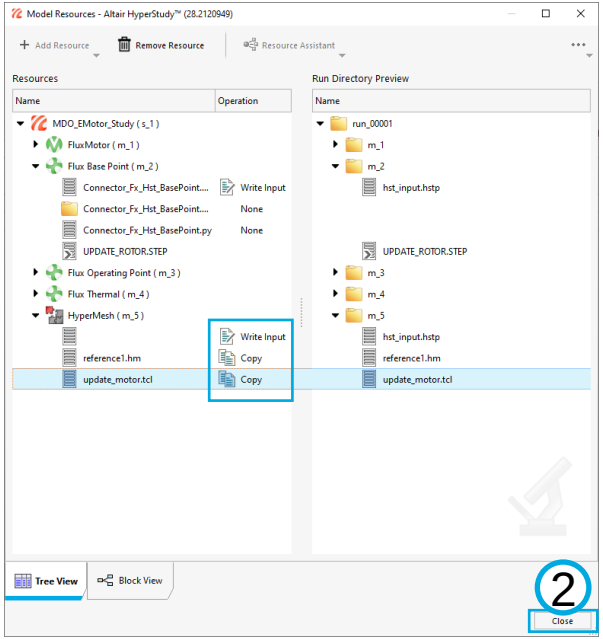
HyperMesh OptiStruct

HyperMesh model import

- Add input files

Step	Action
1	Modify the Operation of the two files: - reference1.hm - update_motor.tcl as type “Copy”
2	Click on [Close]





ALTAIR

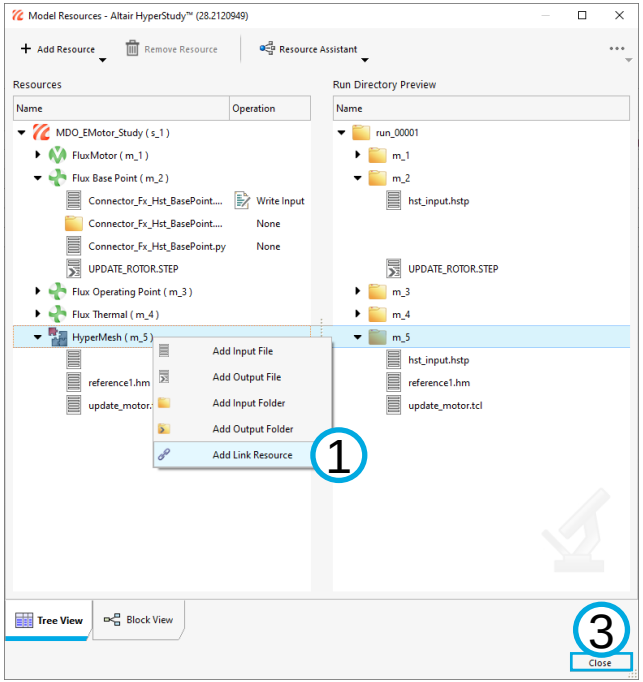
ADD HYPERMESH AND OPTISTRUCT MODELS

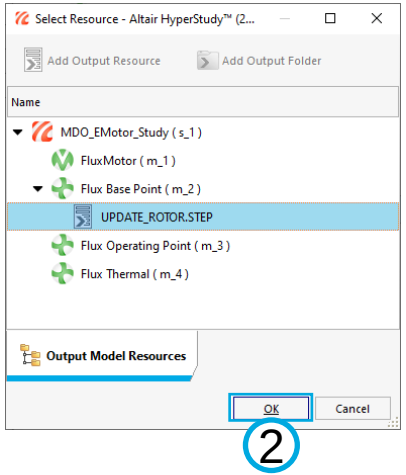
HyperMesh OptiStruct

HyperMesh model import

- Add linked file

Step	Action
1	Right click on the model “HyperMesh (m_5)”, click on [Add Link Resource]
2	Select the file “UPDATE_ROTOR.STEP” from the “Flux Base Speed” model, click on [OK]
3	Click on [Close]





ALTAIR

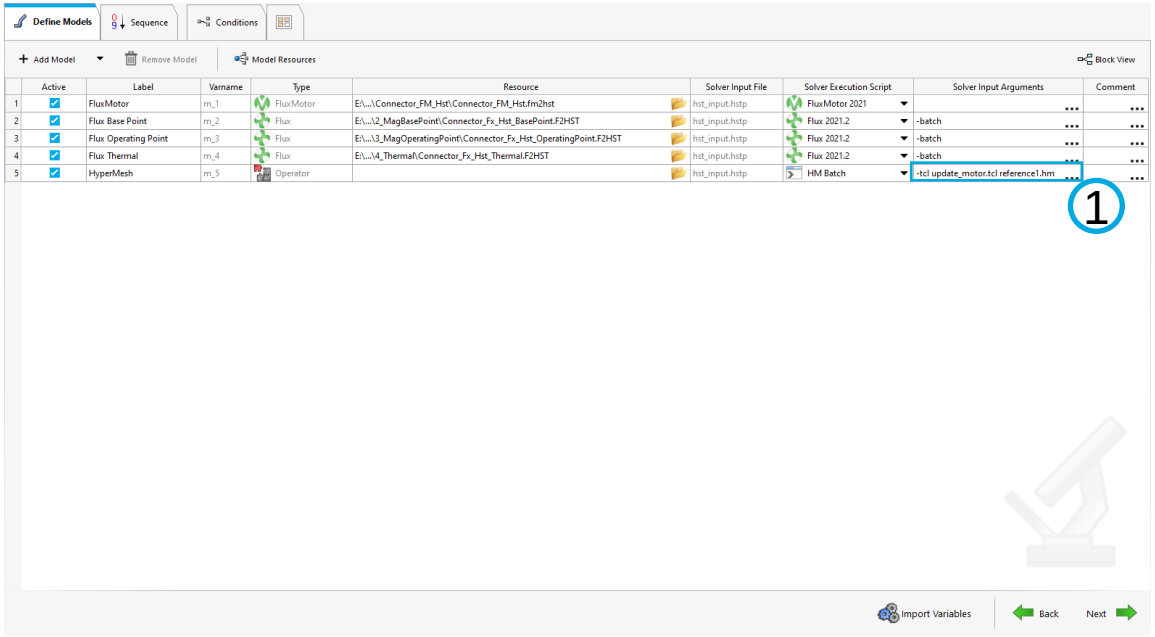
ADD HYPERMESH AND OPTISTRUCT MODELS

HyperMesh | OptiStruct

HyperMesh model import

- Define solver input arguments

Step	Action
1	In the [Solver Input Arguments], input -tcl update_motor.tcl reference1.hm



ALTAIR

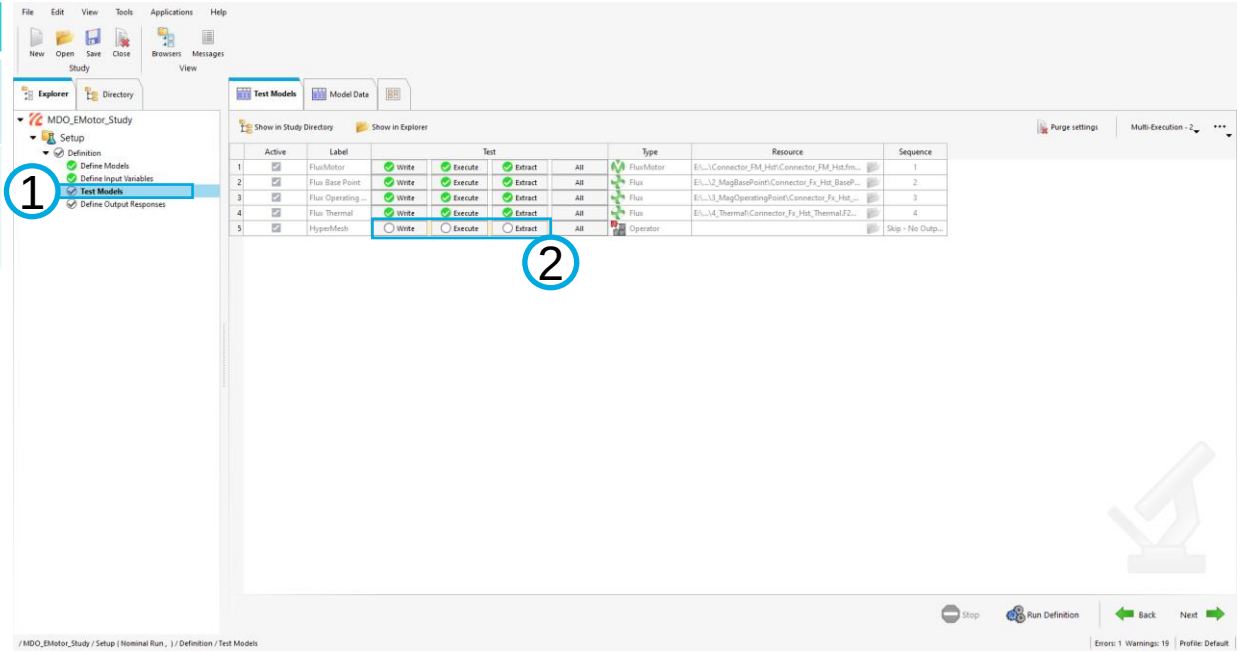
ADD HYPERMESH AND OPTISTRUCT MODELS

HyperMesh | OptiStruct

HyperMesh model import

- Pre-processing for the HyperMesh model

Step	Action
1	Click on [Test Models]
2	Click successively on [Write] – [Execute] – [Extract] for the model HyperMesh



ALTAIR

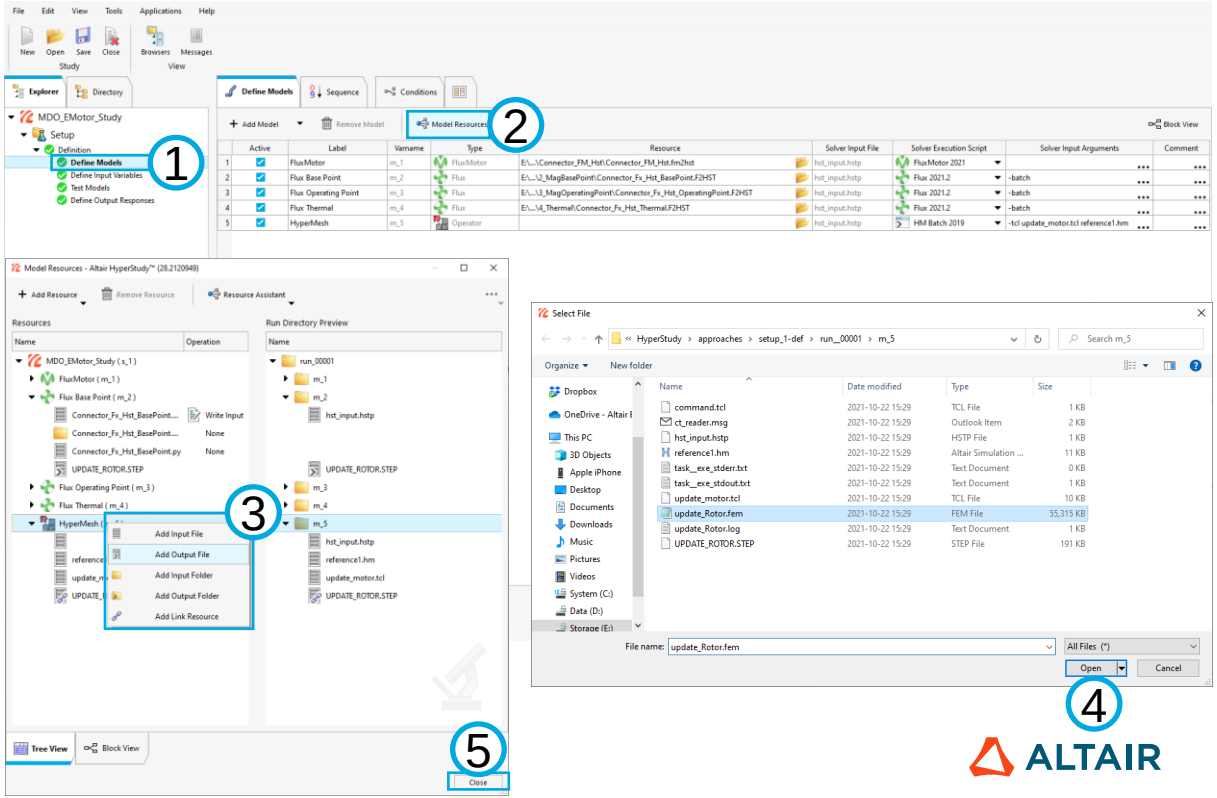
ADD HYPERMESH AND OPTISTRUCT MODELS

HyperMesh | OptiStruct

HyperMesh model import

- Define output file

Step	Action
1	Click on [Define Models]
2	Click on [Model Resources]
3	Right click on the model “HyperMesh (m_5)”, click on [Add Output File]
4	Select on the file “update_Rotor.fem” in the output folder, click on [OK]
5	Click on [Close]



ALTAIR

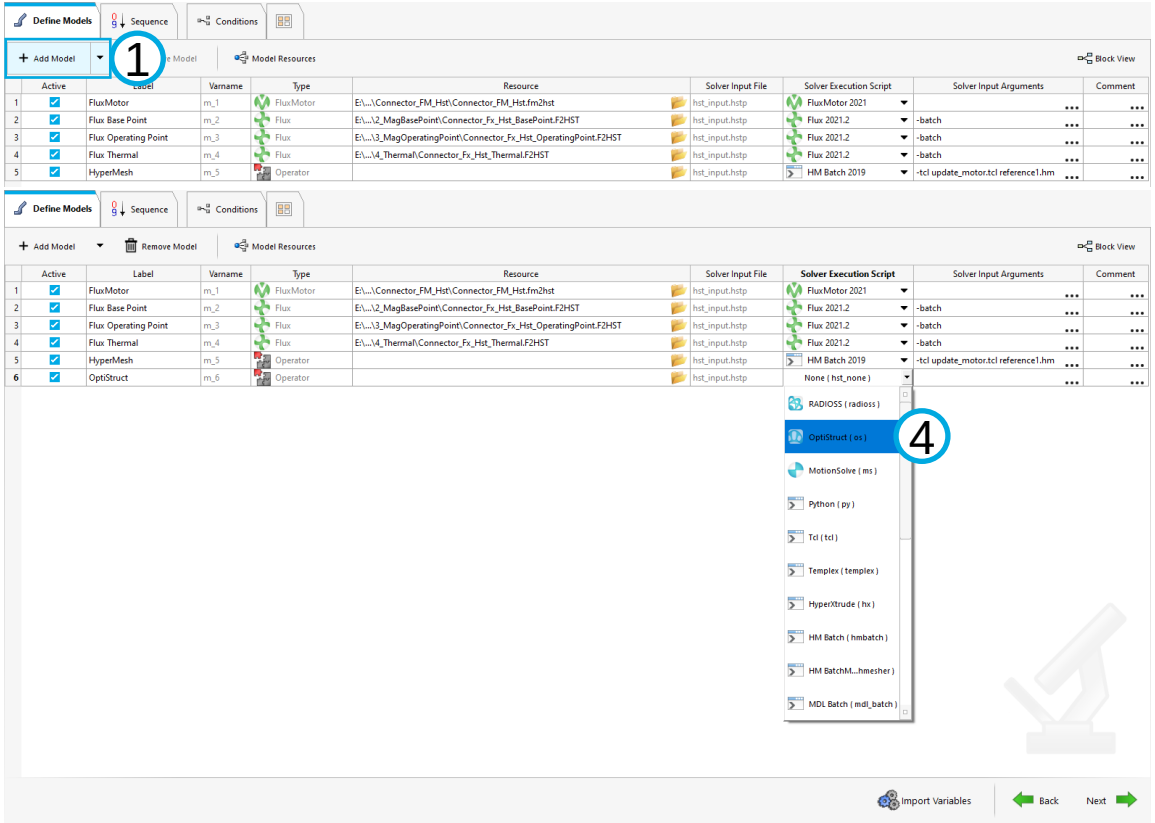
ADD HYPERMESH AND OPTISTRUCT MODELS

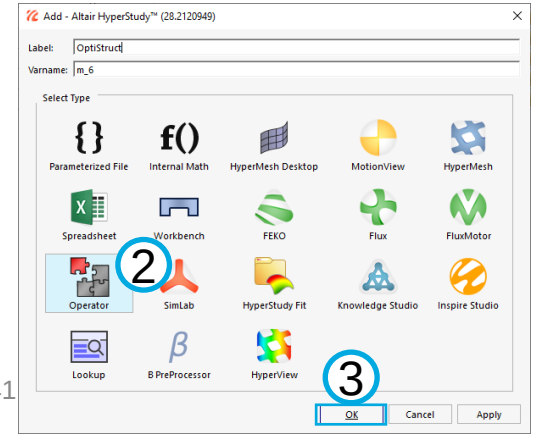
HyperMesh | OptiStruct

OptiStruct model import

- Create a new **Operator**

Step	Action
1	Click on [Add Model] to create the model “OptiStruct”
2	Select the type “Operator”
3	Click on [OK]
4	Select the solver “OptiStrut”





ALTAIR

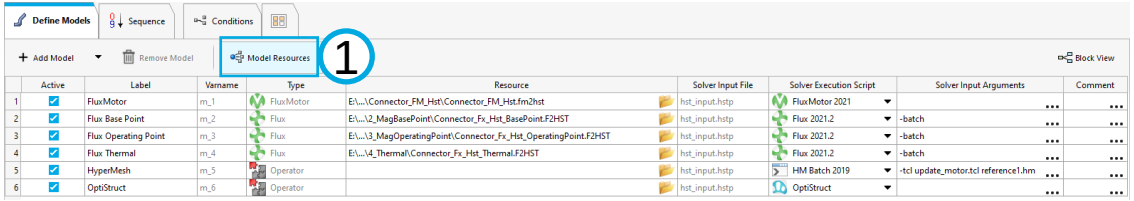
ADD HYPERMESH AND OPTISTRUCT MODELS

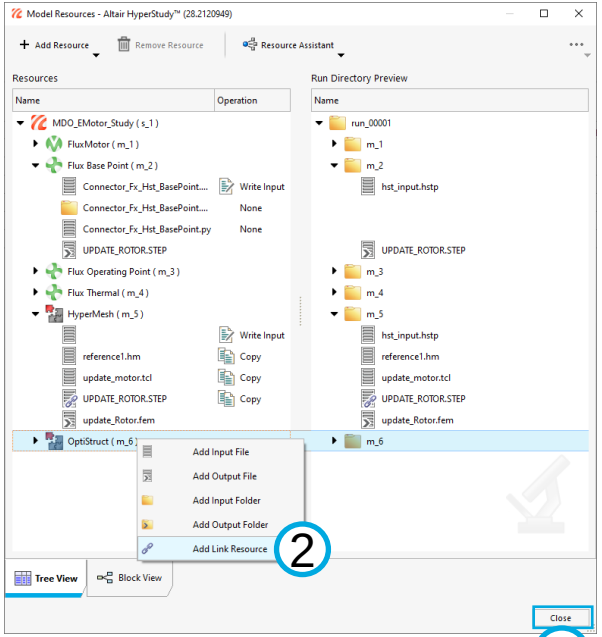
HyperMesh → OptiStruct

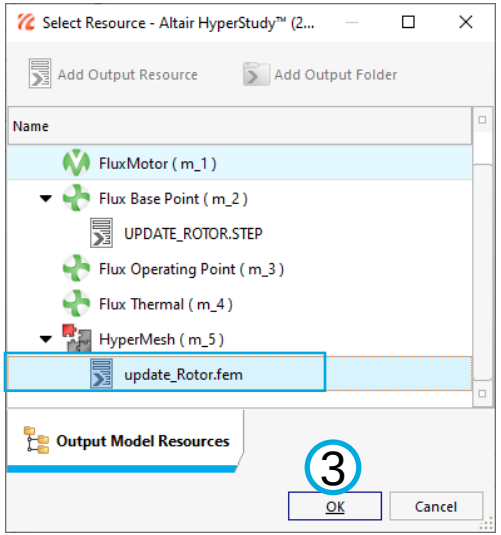
OptiStruct model import

- Add input file (link resources)

Step	Action
1	Click on [Model Resource]
2	Right click on the model “OptiStruct”, click on [Add Link Resource]
3	Select the file “update_Rotor.fem” from the HyperMesh model, click on [OK]
4	Click on [Close]







ALTAIR

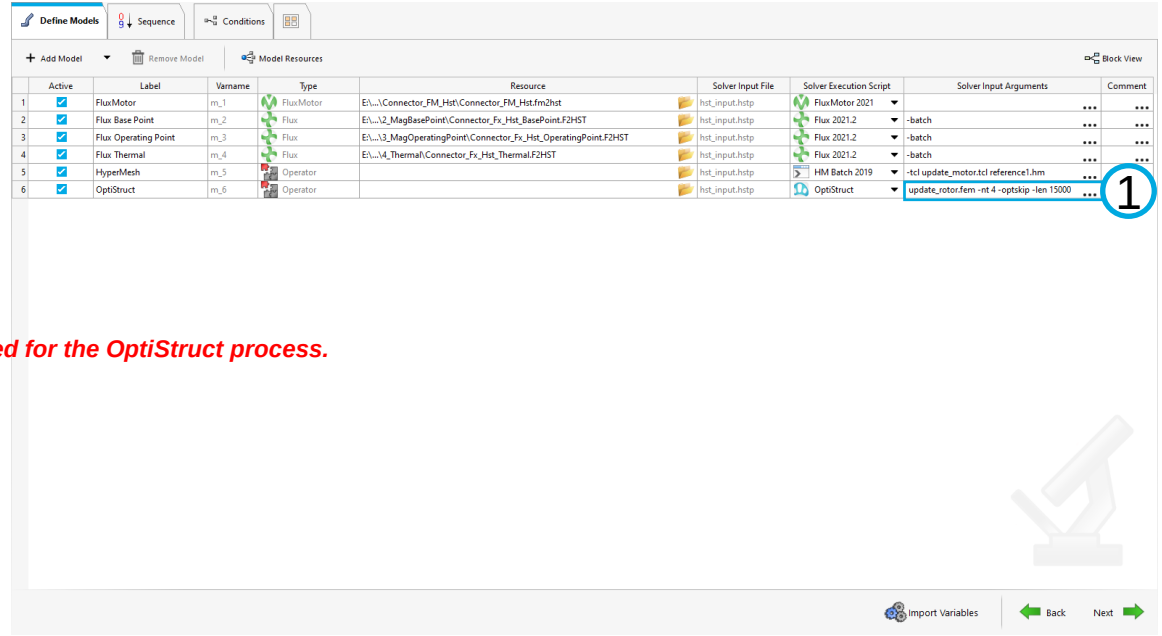
ADD HYPERMESH AND OPTISTRUCT MODELS

HyperMesh OptiStruct

OptiStruct model import

- Define input arguments

Step	Action
1	In the [Solver Input Arguments], input the command update_rotor.fem -nt 4 -optskip -len 15000



Notes:

1) -nt 4 means 4 cores in parallel are used;

2) -len 15000 means 15 Gb of RAM are reserved for the OptiStruct process.

ALTAIR

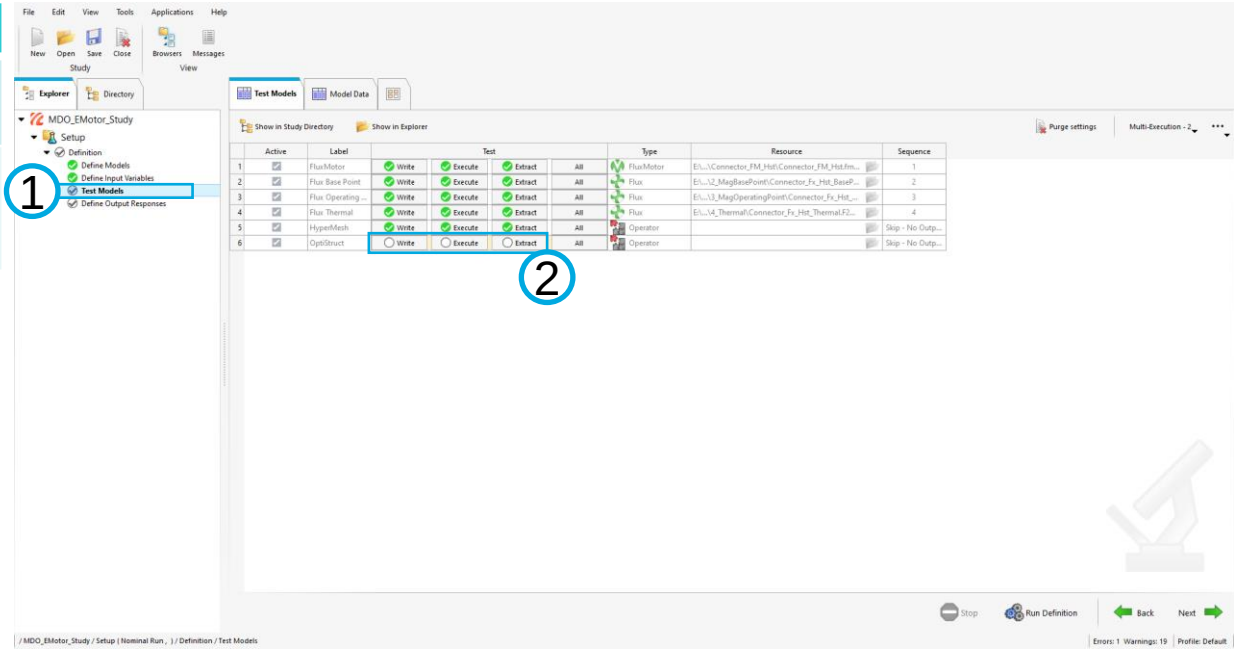
MECHANICAL PROBLEM SOLVING

ALTAIR

MECHANICAL PROBLEM SOLVING

OptiStruct model solving

Step	Action
1	Click on [Test Models]
2	Click successively on [Write] – [Execute] – [Extract] for the model HyperMesh



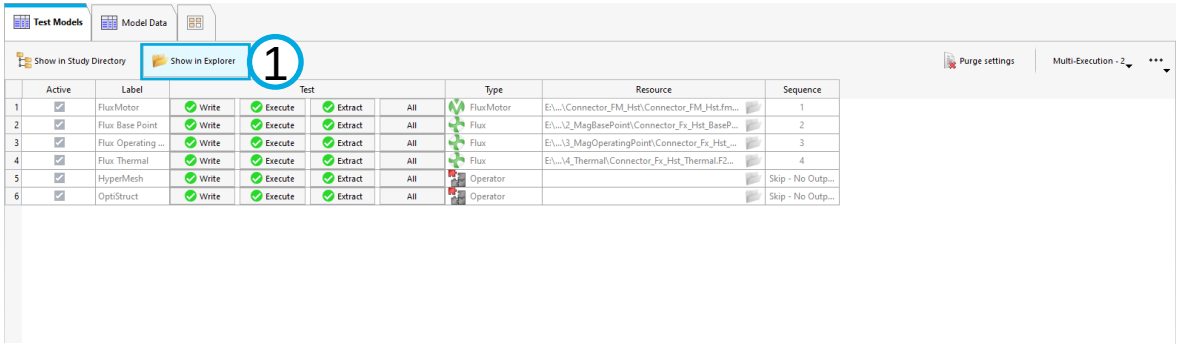
ALTAIR

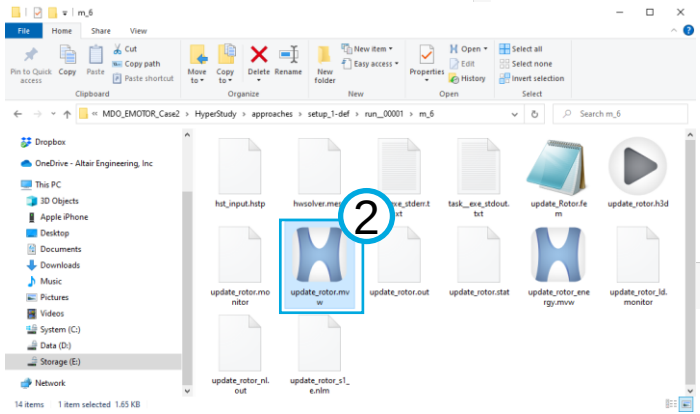
MECHANICAL PROBLEM SOLVING

OptiStruct model post-processing

- Visualize the mechanical analysis result

Step	Action
1	Click on [Show in Explorer]
2	Open the file “update_rotor.mvw” in the project output folder ~\HyperStudy\approaches\setup_1-def\run__00001\m_6



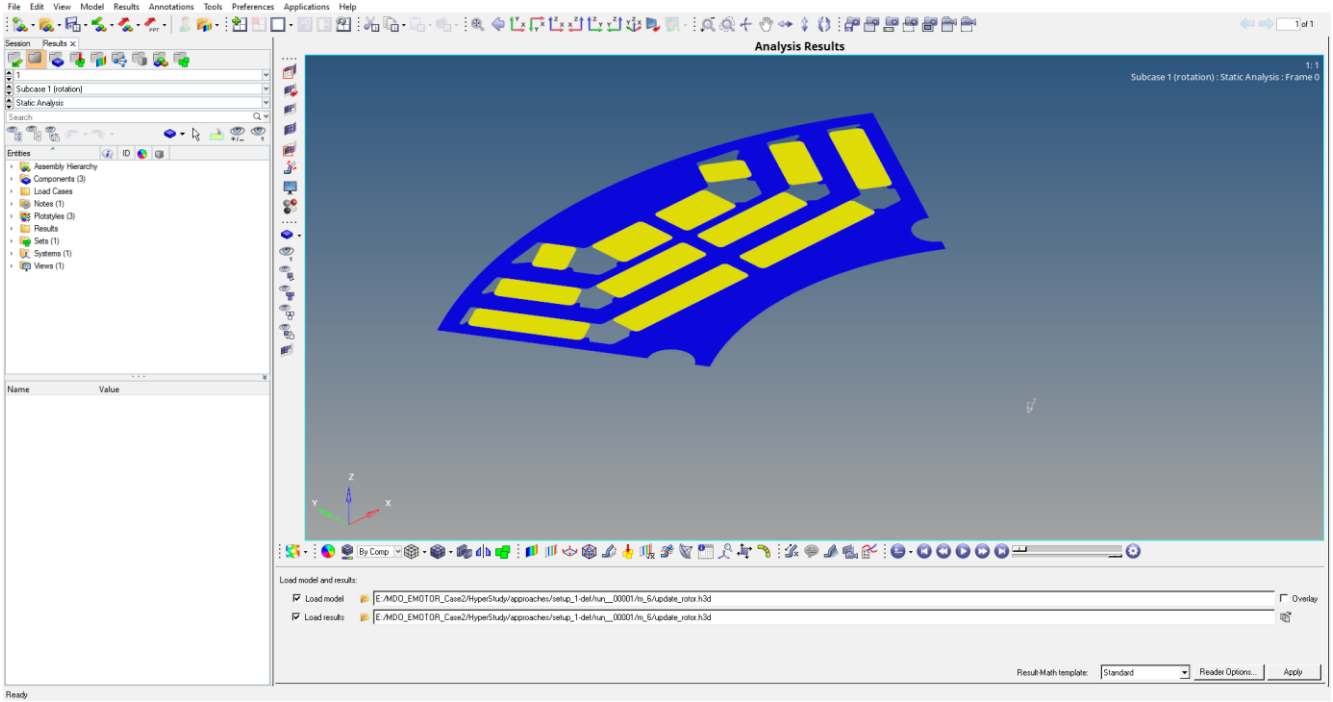


ALTAIR

MECHANICAL PROBLEM SOLVING

OptiStruct model post-processing

- Visualize the mechanical analysis result



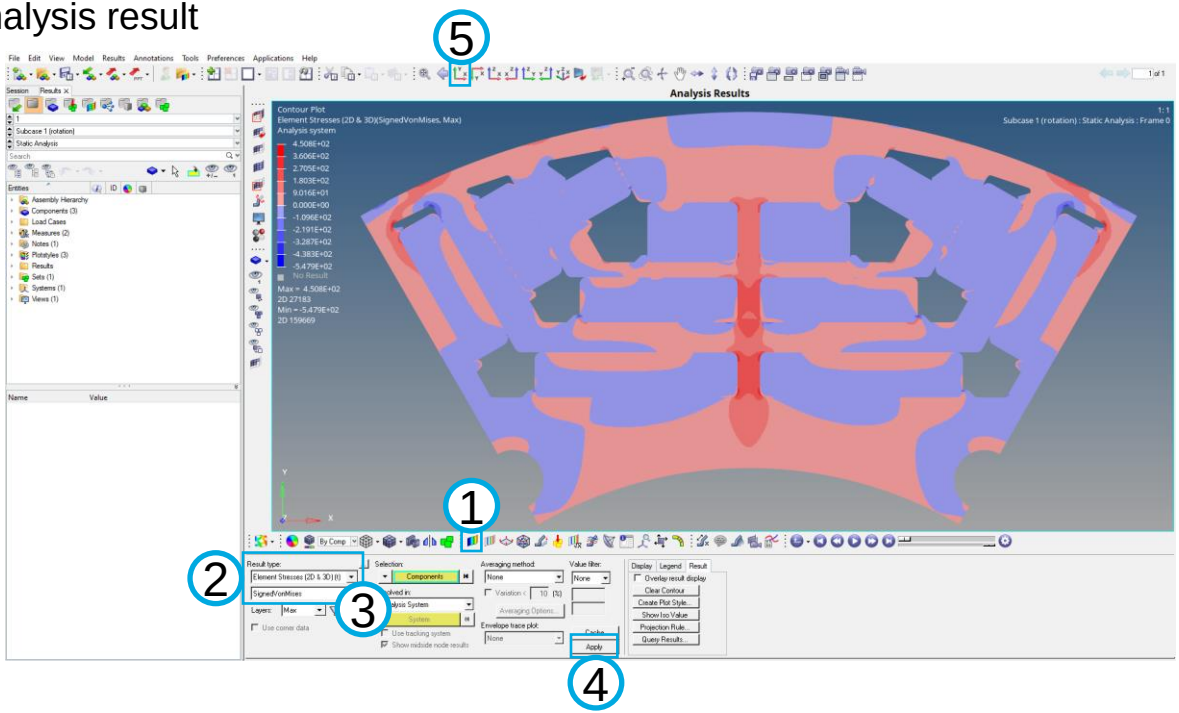
ALTAIR

MECHANICAL PROBLEM SOLVING

OptiStruct model post-processing

- Visualize the mechanical analysis result

Step	Action
1	Click on the icon
2	Select [Element Stress (2D&3D)]
3	Select [SignedVonMises]
4	Click on [Apply]
5	Click on the icon to adjust the view



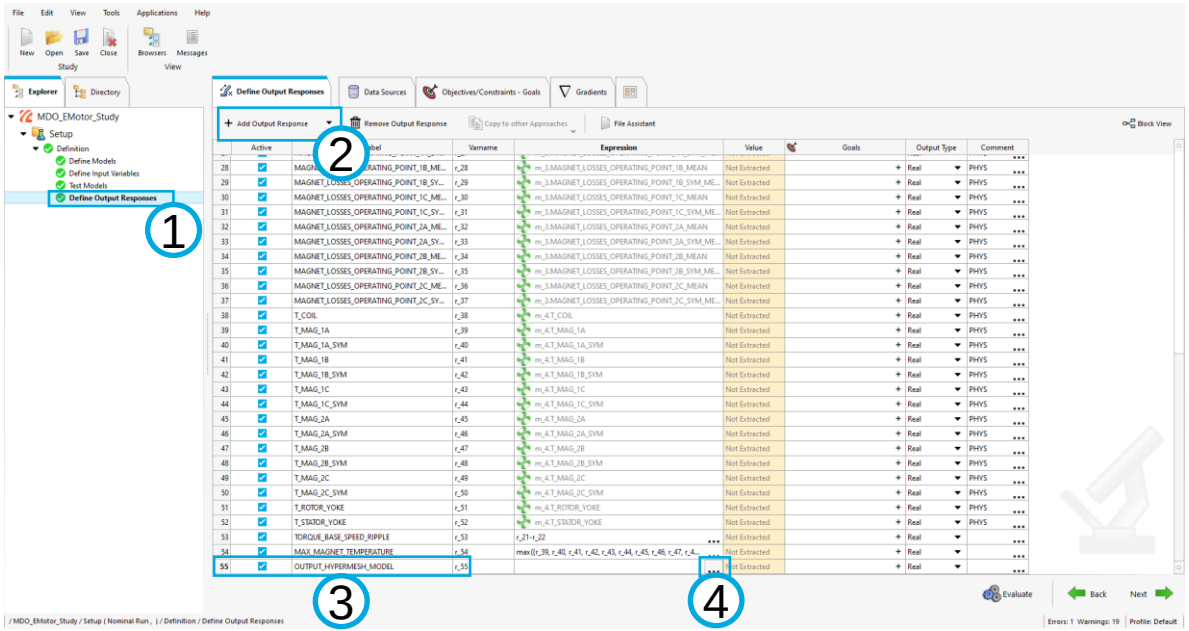
ALTAIR

MECHANICAL PROBLEM SOLVING

HyperMesh and **OptiStruct** model post-processing

- Define output response (**HyperMesh** model)

Step	Action
1	Click on [Define Output Response]
2	Click on [Add output Response]
3	Define the new output response label as “OUTPUT_HYPERMESH_MODEL”
4	Click on […]

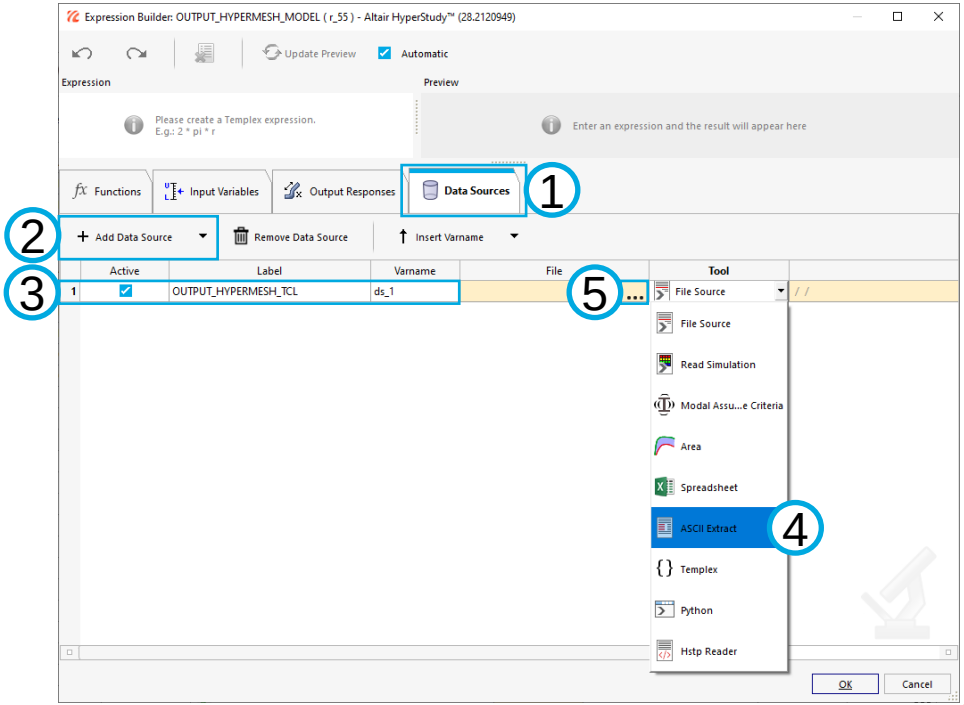


MECHANICAL PROBLEM SOLVING

HyperMesh and **OptiStruct** model post-processing

- Define output response (**HyperMesh** model)

Step	Action
1	Click on [Data Sources]
2	Click on [Add Data Source]
3	Define the new data source label as “OUTPUT_HYPERMESH_TCL”
4	Select the type as “ASCII Extract”
5	Click on […]



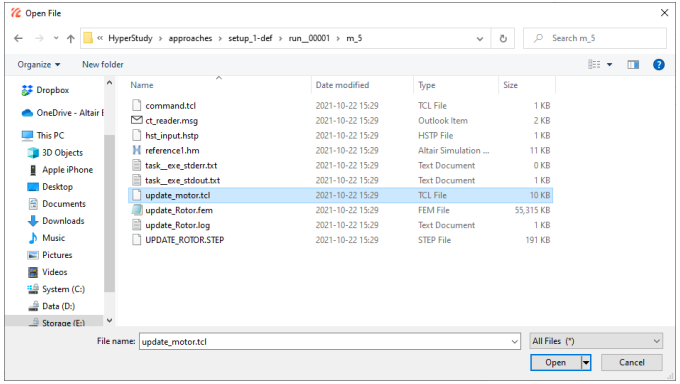
ALTAIR

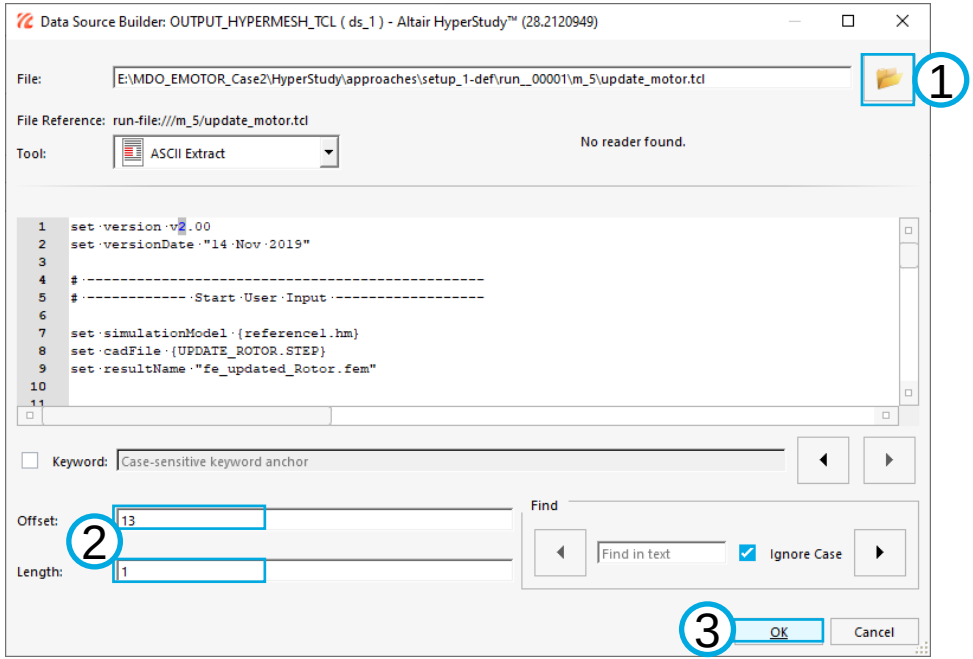
MECHANICAL PROBLEM SOLVING

HyperMesh and **OptiStruct** model post-processing

- Define output response (**HyperMesh** model)

Step	Action
1	Click on the icon to locate the file “update_motor.tcl” in the folder “run_00001/m_5”
2	Define the Offset as 13, and Length as 1
3	Click on [OK]





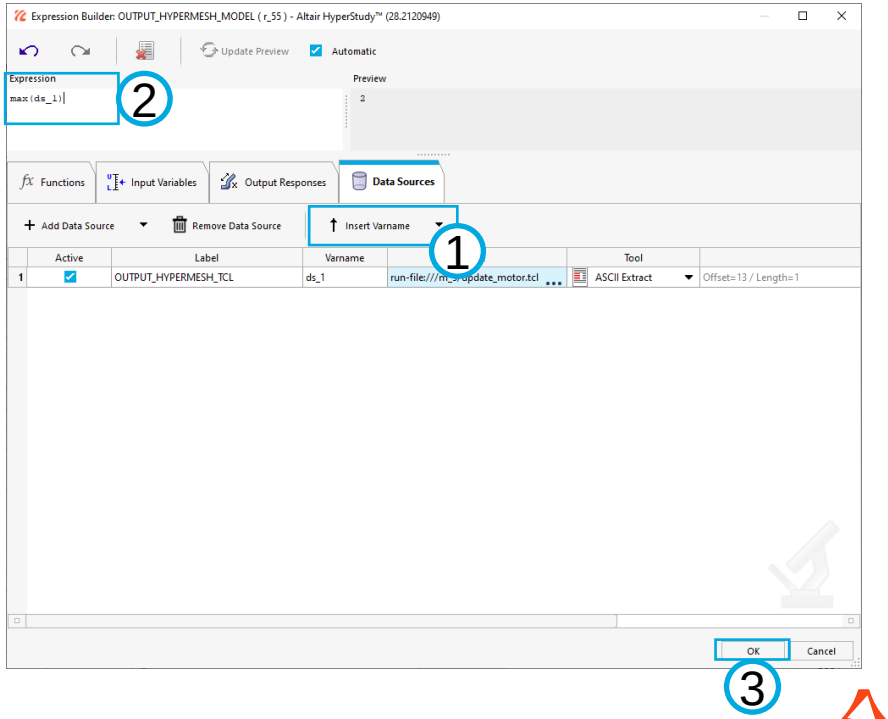
ALTAIR

MECHANICAL PROBLEM SOLVING PROBLEM SOLVING

HyperMesh and **OptiStruct** model post-processing

- Define output response (**HyperMesh** model)

Step	Action
1	Click on [Insert Varname]
2	Verify if the expression is “max(ds_1)”
3	Click on [OK]

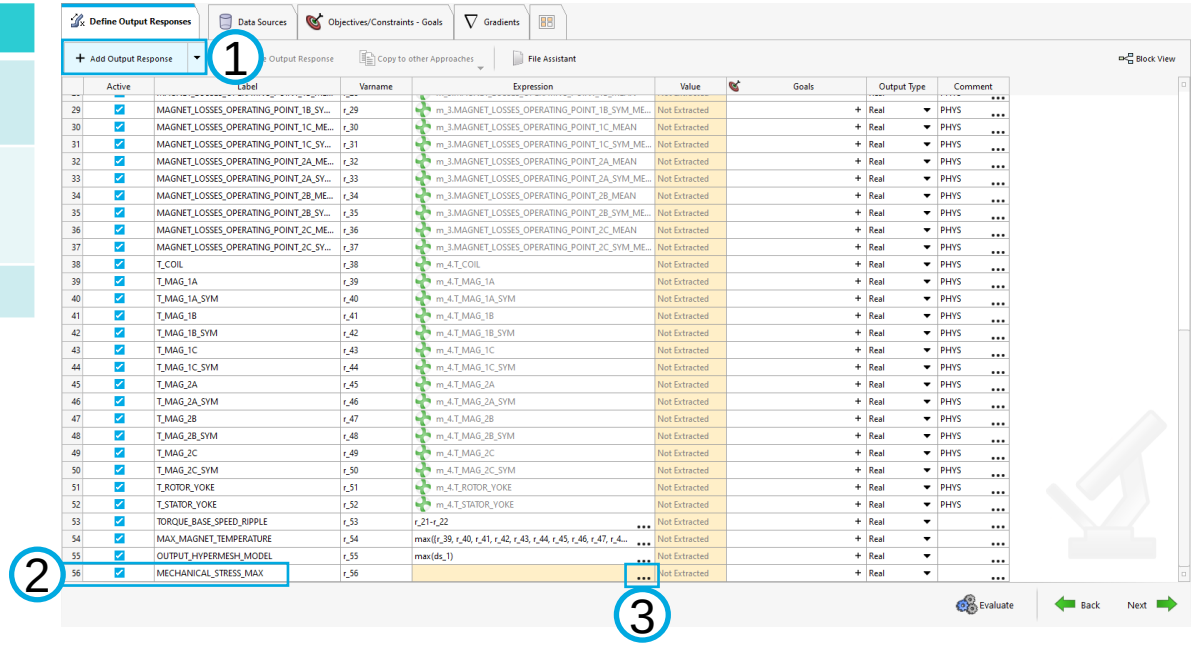


MECHANICAL PROBLEM SOLVING

HyperMesh and **OptiStruct** model post-processing

- Define output response (**OptiStruct** model)

Step	Action
1	Click on [Add output Response]
2	Define the new output response label as “MECHANICAL_STRESS_MAX”
4	Click on […]

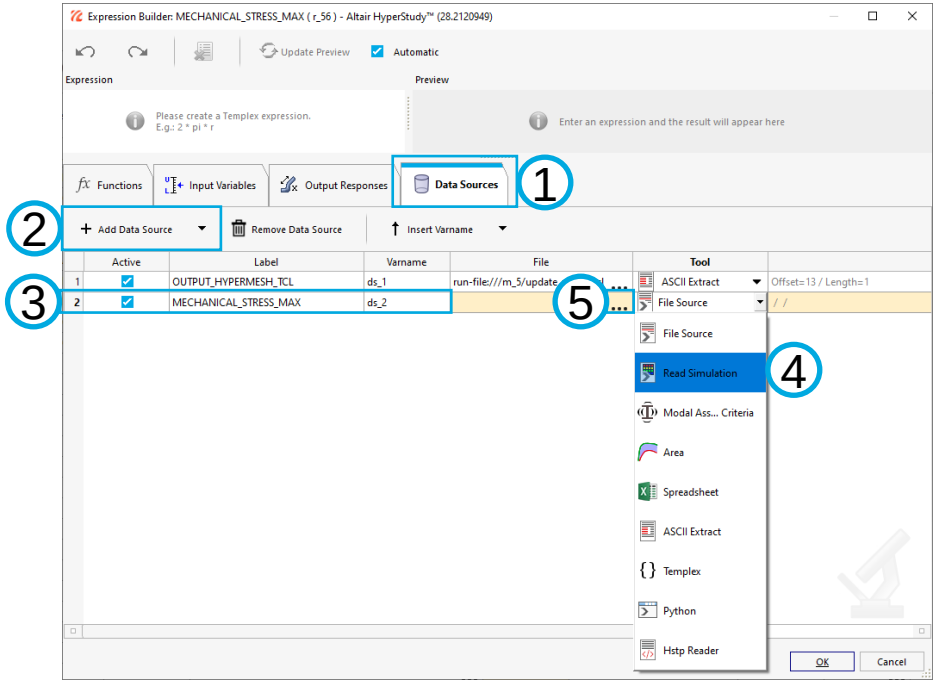


MECHANICAL PROBLEM SOLVING

HyperMesh and **OptiStruct** model post-processing

- Define output response (**OptiStruct** model)

Step	Action
1	Click on [Data Sources]
2	Click on [Add Data Source]
3	Define the new data source label as “MECHANICAL_STRESS_MAX”
4	Select the type as “Read Simulation”
5	Click on […]



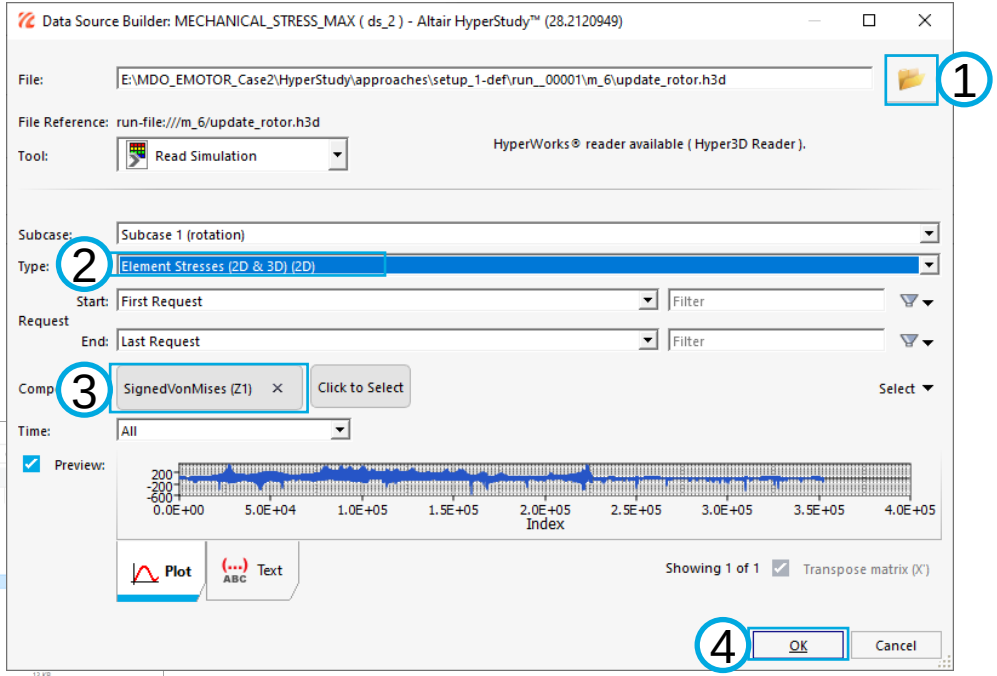
ALTAIR

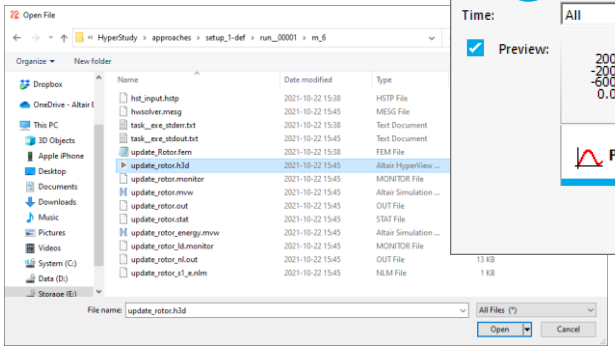
MECHANICAL PROBLEM SOLVING

HyperMesh and **OptiStruct** model post-processing

- Define output response (**OptiStruct** model)

Step	Action
1	Click on the icon to locate the file “update_rotor.h3d” in the folder “run_00001/m_6”
2	Select the type as “Element Stresses”
3	Change the component to “SignedVonMises (Mid)”
4	Click on [OK]





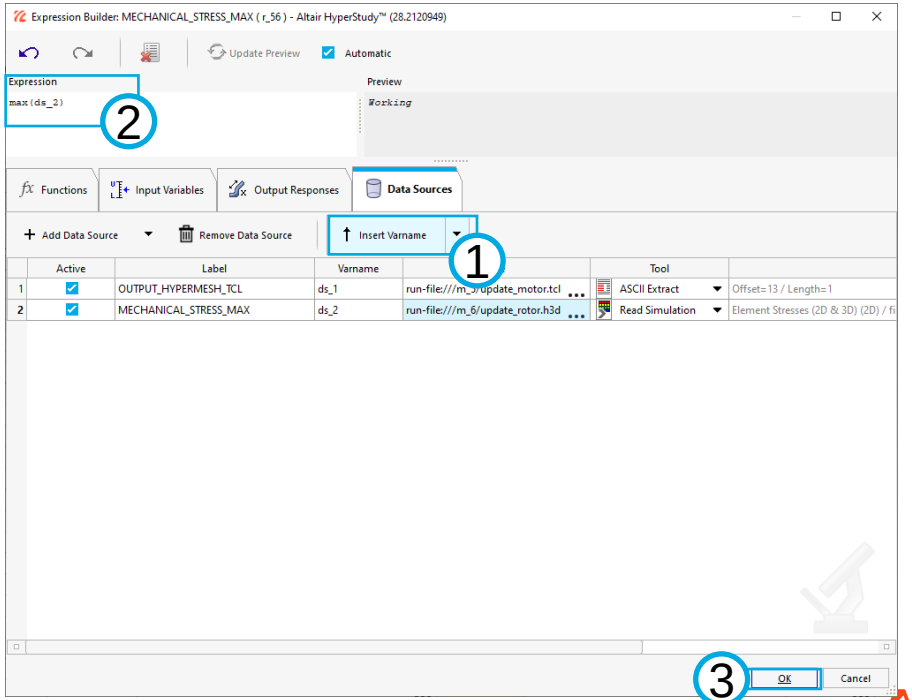
ALTAIR

MECHANICAL PROBLEM SOLVING

HyperMesh and **OptiStruct** model post-processing

- Define output response (**OptiStruct** model)

Step	Action
1	Click on [Insert Varname]
2	Verify if the expression is “max(ds_2)”
3	Click on [OK]



ALTAIR

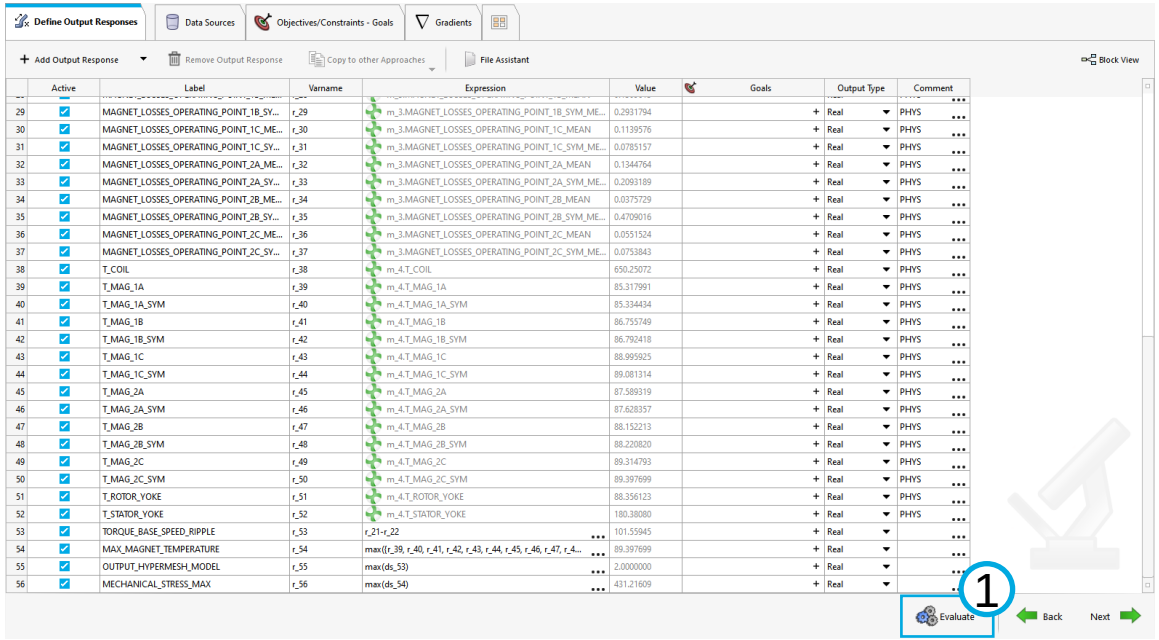
MECHANICAL PROBLEM SOLVING

HyperMesh and **OptiStruct** model post-processing

- Evaluate new responses

Step	Action
1	Click on [Evaluate]

	Original value[1]
Mechanical power at base point / W	137076.74
Rotor mass / kg	4.96
Mechanical torque at base point / N*m	183.15
Mechanical torque ripple at base point / N*m	43.16
Winding temperature / °C	151.04
Magnet temperature / °C	46.89
Mechanical stress / MPa	431.22



[1]Different software versions may cause slight changes in result values.

ALTAIR

THANK YOU

altair.com

#ONLYFORWARD

ALTAIR