Extraction of plate modes

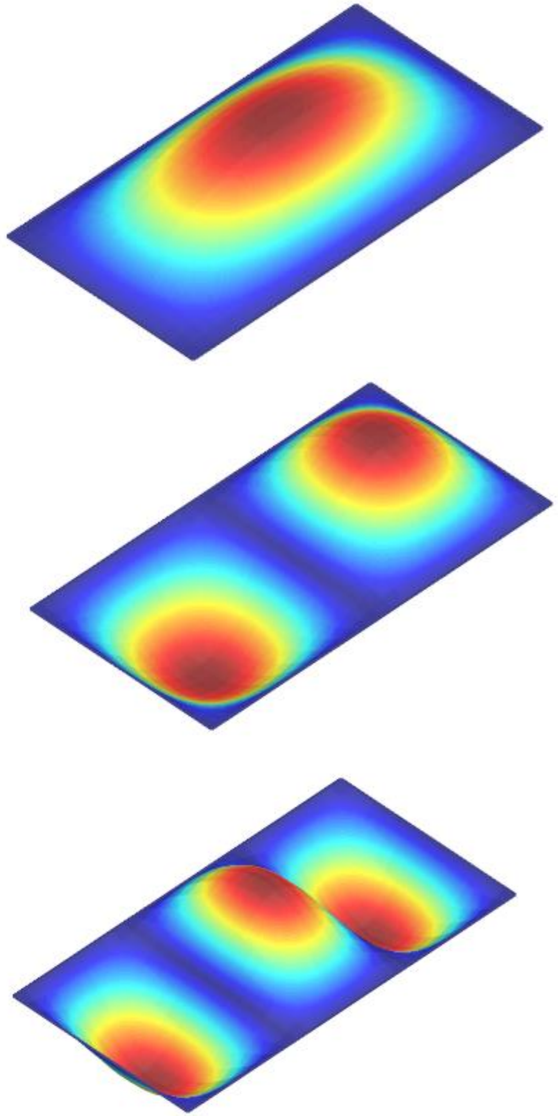
Actran Student Edition Tutorial

Introduction

- This workshop introduces the **modal extraction** analysis for structures and shows an application case on a plate

- The objectives of this workshop are the following :
  - Get introduced to structure dynamics
  - Get introduced to the modal extraction analysis
  - Calculate the vibrating modes of a structure using Actran

- Software Version:
  - Actran 2022 Student Edition



Workshop description

Model description

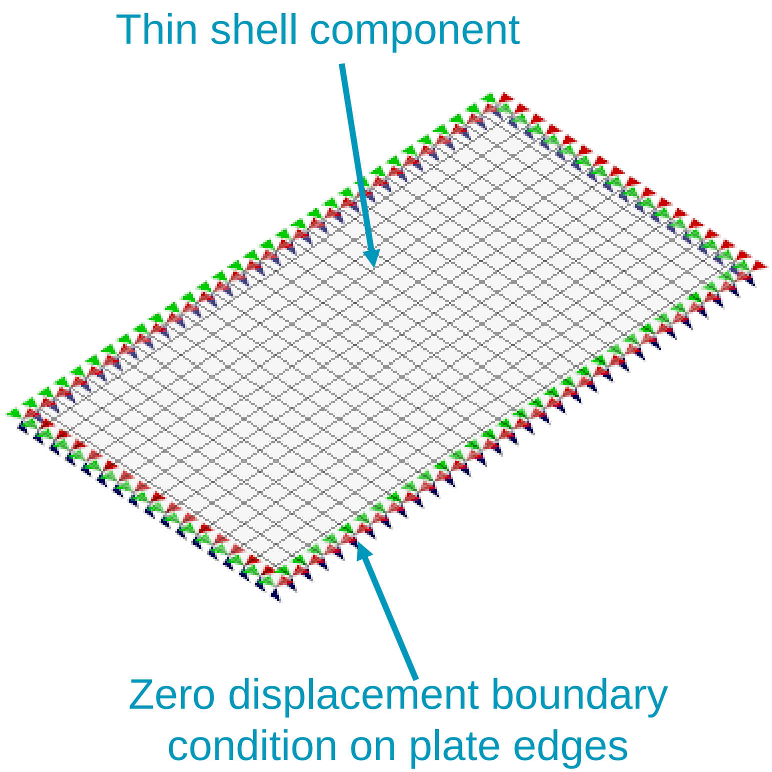
Modal extraction of a rectangular plate

Plate modelling

- A *thin shell component* is defined
- The plate is modeled by 2D elements, with its thickness defined as a property of the shell

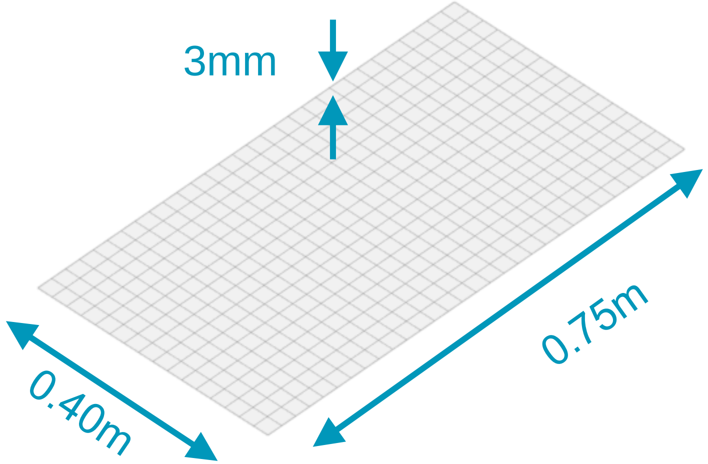
Simply supported edges

- A 0 *displacement boundary condition* is defined



Analytical solution

- Let us consider a plate with the following properties:
  - Size: Lx = 0.75 m, Ly = 0.40 m, thickness t = 0.003 m
  - Material properties: E = 7*10e10 Pa, v = 0.25, ρ = 2400 kg/m3
  - The plate is simply supported along the four edges



- Kirchhoff–Love model for Isotropic thin plate gives

$$D\nabla^4 w_{mn}(x,y) - \rho t \omega_{mn}^2 w_{mn}(x,y) = 0$$

- Which leads to eigenmodes

$$w_{mn}(x,y) = A_{mn} \sin\left(\frac{m\pi x}{L_x}\right) \sin\left(\frac{n\pi x}{L_y}\right)$$

- And eigenfrequencies $\omega_{mn} = \sqrt{\frac{D}{\rho t}}\left(\left(\frac{m\pi}{L_x}\right)^2 + \left(\frac{n\pi}{L_y}\right)^2\right)$ where $D = \frac{Et^3}{12(1-\upsilon^2)}$ is the bending stiffness

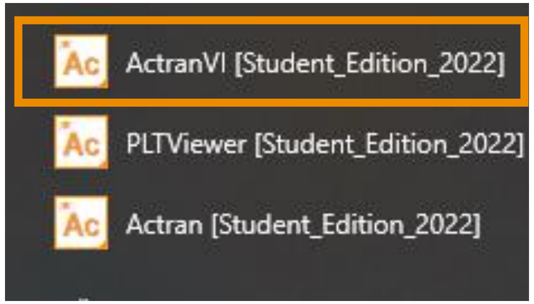
HEXAGON

Workshop pre-processing

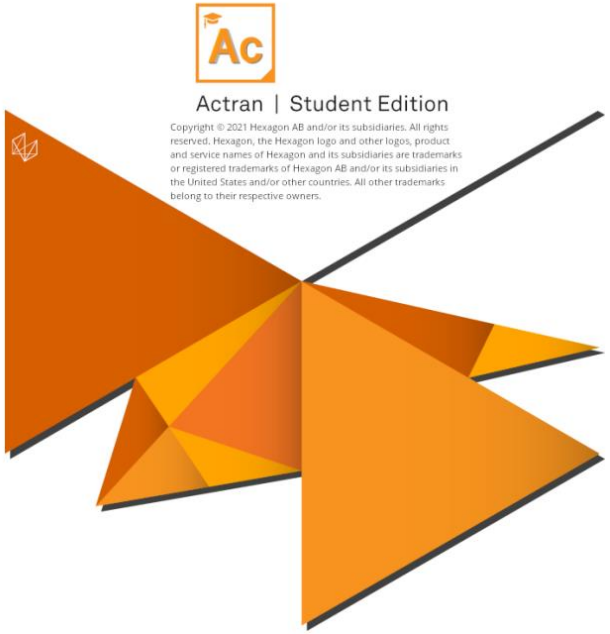
HEXAGON

Start ActranVI

- Start ActranVI:
  - Shortcut is available through the Windows Start Menu



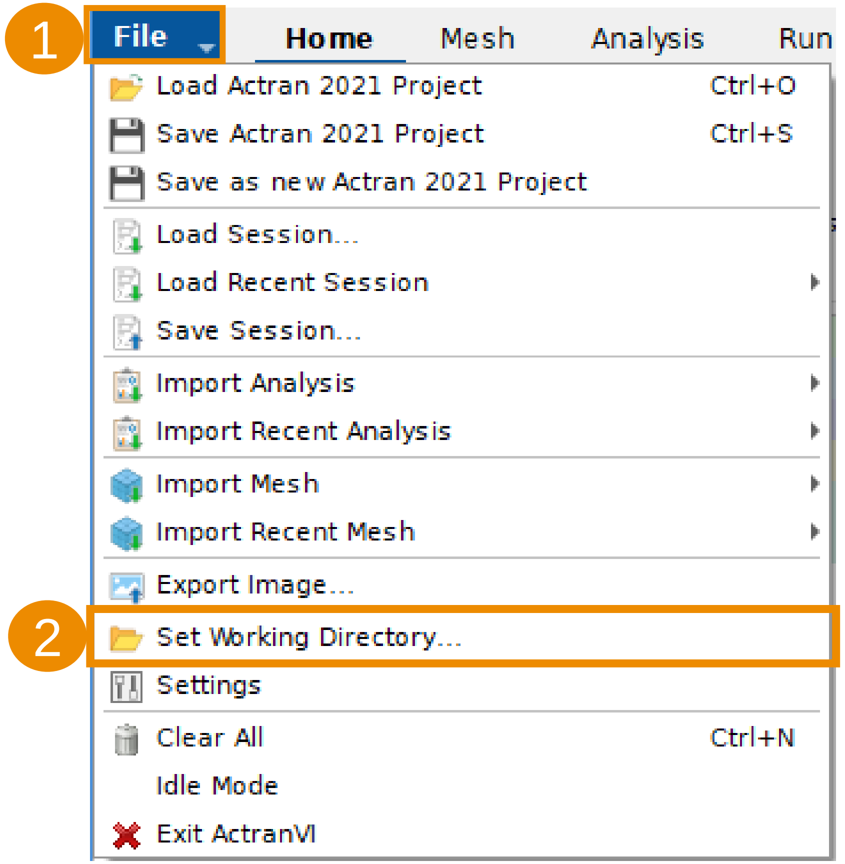
(Windows Start Menu)

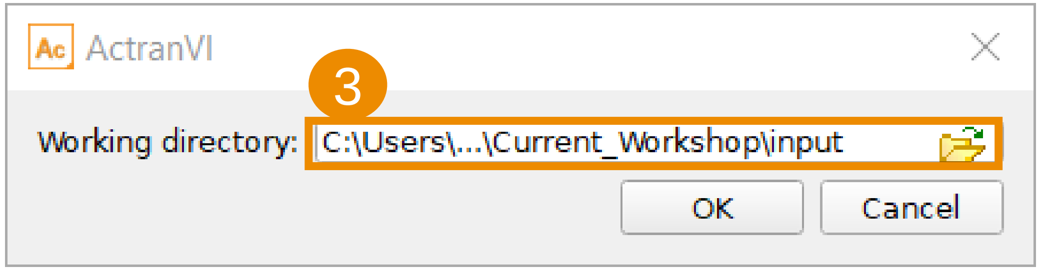


Set the working directory

Select the workshop input directory as the working directory

- The working directory is the project directory where all ActranVI related files are output





The working directory path should not contain any space or special character

Create the mesh

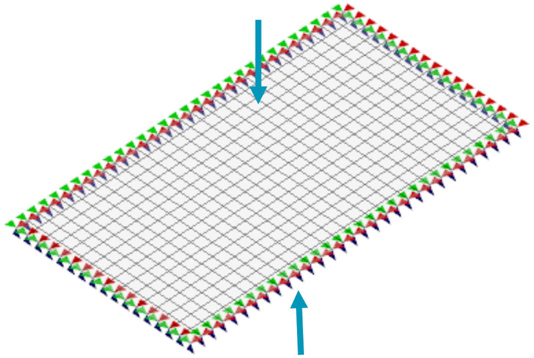
General introduction

- ActranVI includes some meshing tools allowing to design meshes in order to build an Actran analysis. These meshing tools are used to create the mesh needed for this workshop

- Two element sets must be created:
  - One 2D surface mesh element set to support the plate (thin shell component)
  - One 1D edge mesh element set to support the boundary condition

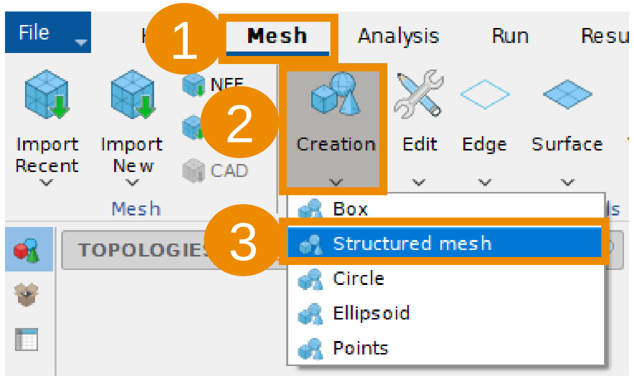
- Meshing tools can be found in ActranVI toolbox, under Mesh tab → Meshing tools

Thin shell component
2D element set



Zero displacement BC
1D element set

Select the *Structured mesh* meshing tool

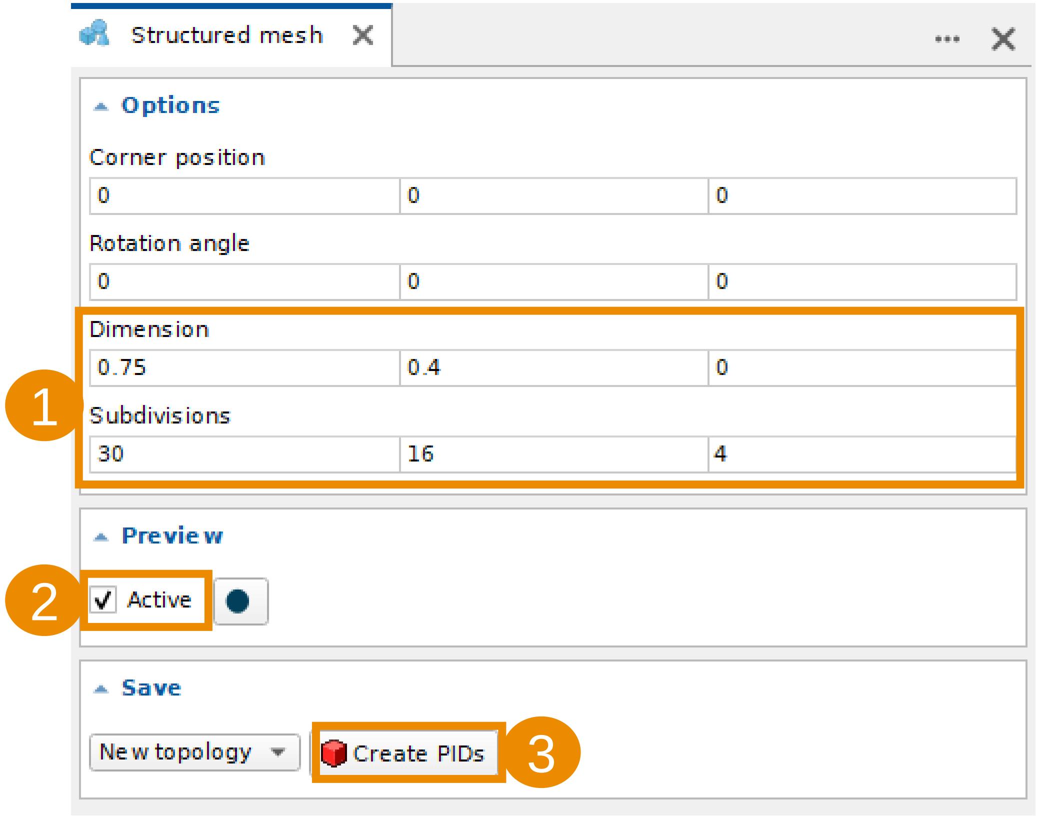


HEXAGON

Create the mesh

Create the 2D element set

- The 2D element set is a rectangle with length
  - Lx = 0.75 m and Ly = 0.40 m
- The target element size is 0.025m (mesh subdivisions are set accordingly)
- The created mesh will contain linear elements



The *Structured Mesh* function can be used to create the plate → Adjust the function parameters to the problem definition → Pre-visualize the mesh using the interactive preview → The element set can be created by clicking "Create PIDs"

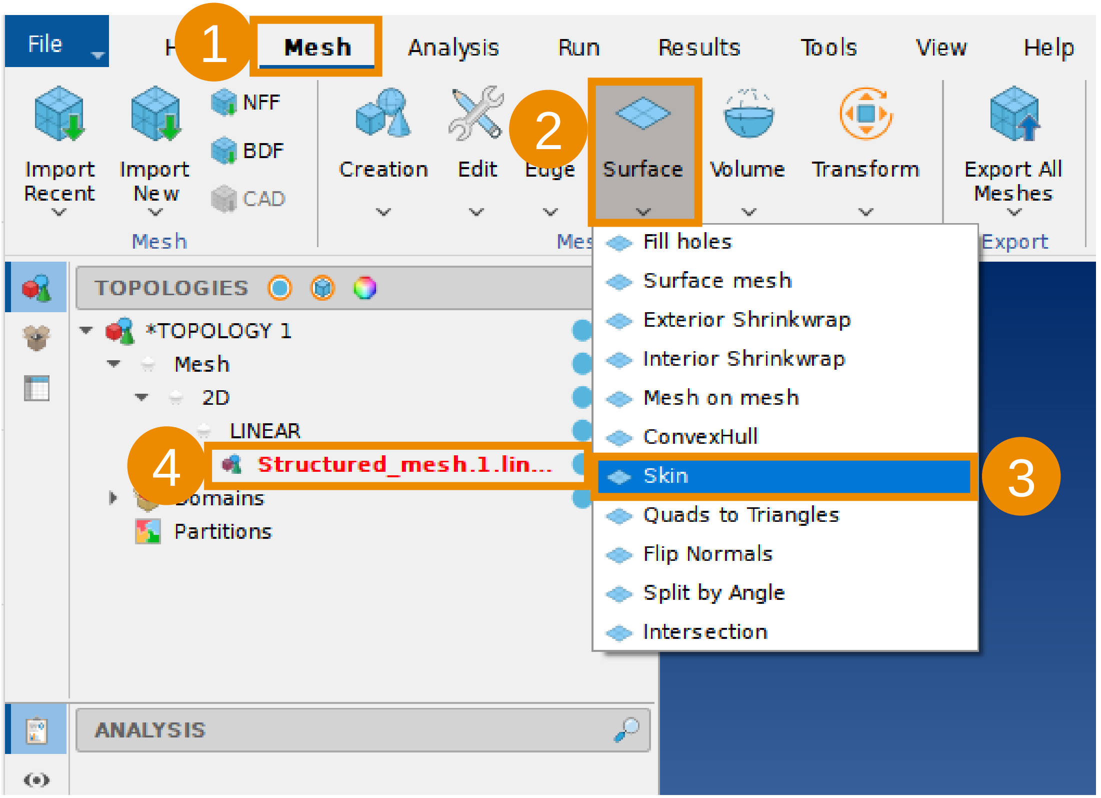
Create the mesh

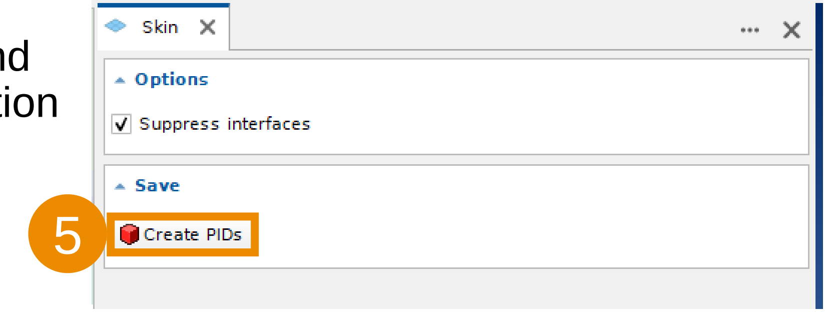
Create the 1D element set – Skin

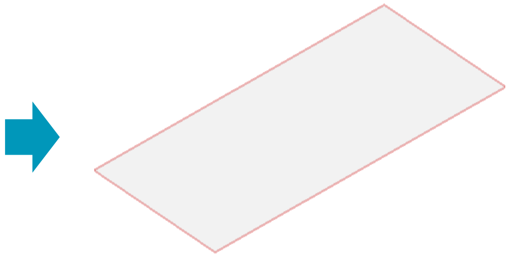
- The 1D element set must be created on the free edges of the previously created 2D element set to apply the boundary condition

- The Skin function can be used to create the element set:
  - On the meshing tools toolbox open the Skin function
  - Select the plate element set by clicking on it in the graphical tree (or CTRL+click in the render window): the selected element will color in red
  - Click on “Create PIDs” to create the 1D element set

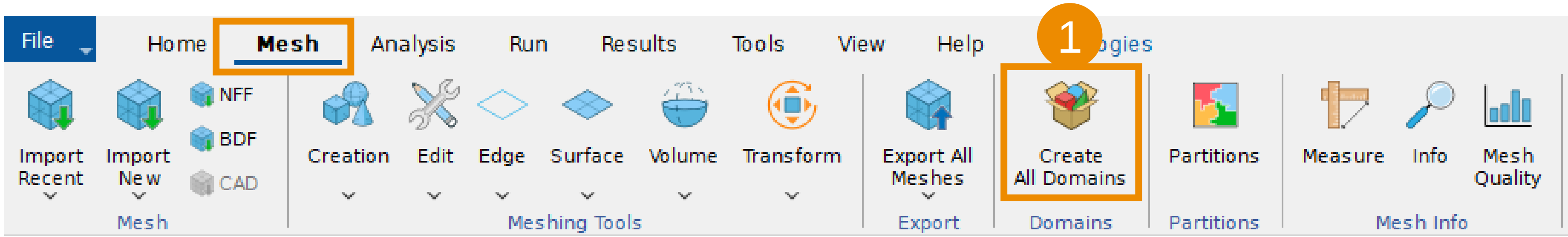
- The mesh needed to run the analysis is now ready and will be used to setup the analysis and run the calculation

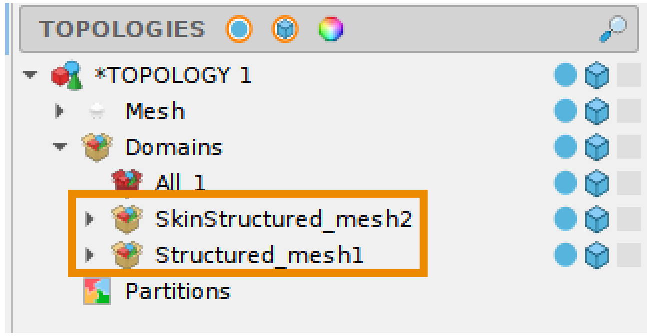






Create the domains





Auto create domain

- Automatic domain creation based on PIDs
- One domain per PID

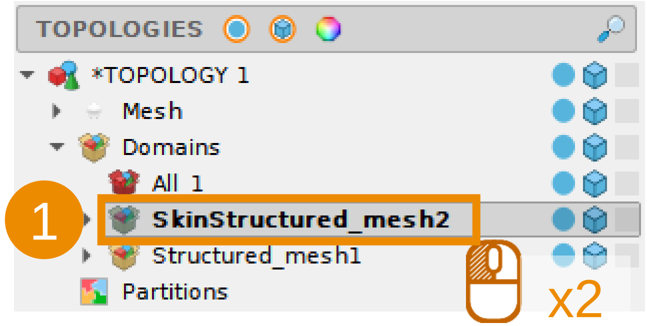
A ***Domain*** is a group of **one** or **several** PIDs

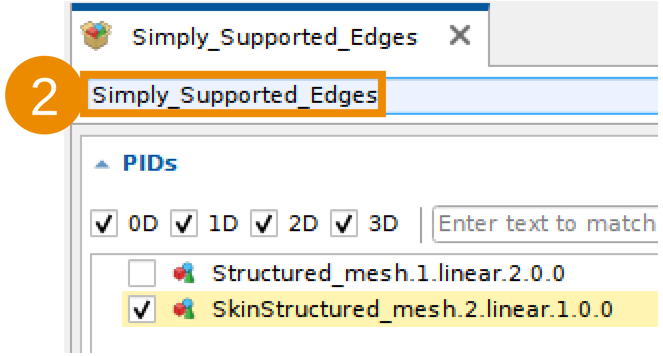
- *Domains* link PIDs to the analysis objects
- *Domains* decouple the topology from the analysis

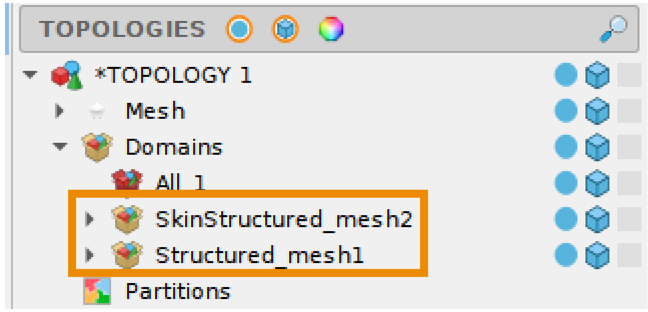
Rename the domains

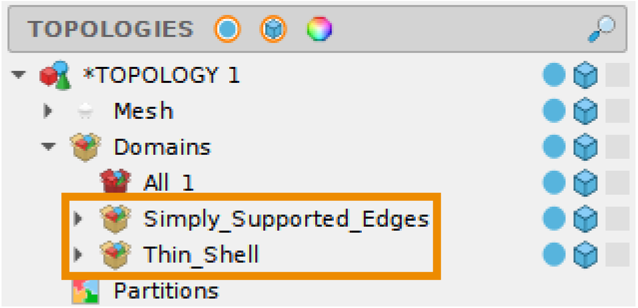
Rename both domains

- SkinStructured_mesh2 → Simply_Supported_Edges
- Structured_mesh1 →Thin_Shell







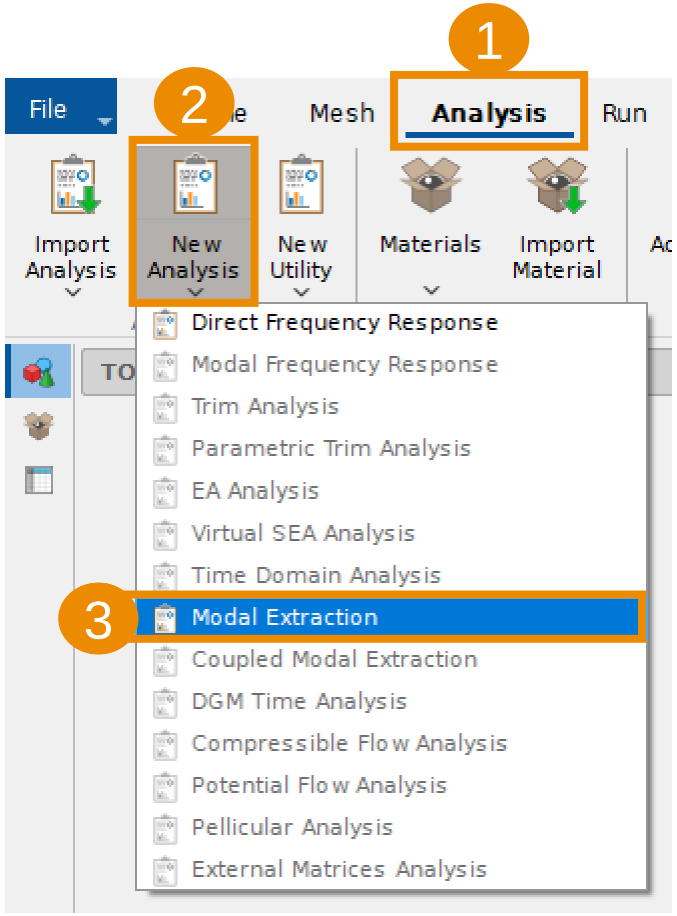


HEXAGON

Create the modal extraction analysis

Add a *Modal Extraction* analysis

- A Modal Extraction is computation procedure which provides the modes shapes and natural frequencies of a system



HEXAGON

Specify the frequency range

The plate bending wavelength must be captured

- Maximum frequency (smallest wavelength) is driven by the largest element length
- For linear elements, 10 elements per bending wavelength can be used (rule of thumb)
- Mesh largest element length is 0.25m
  → maximum frequency is 500Hz

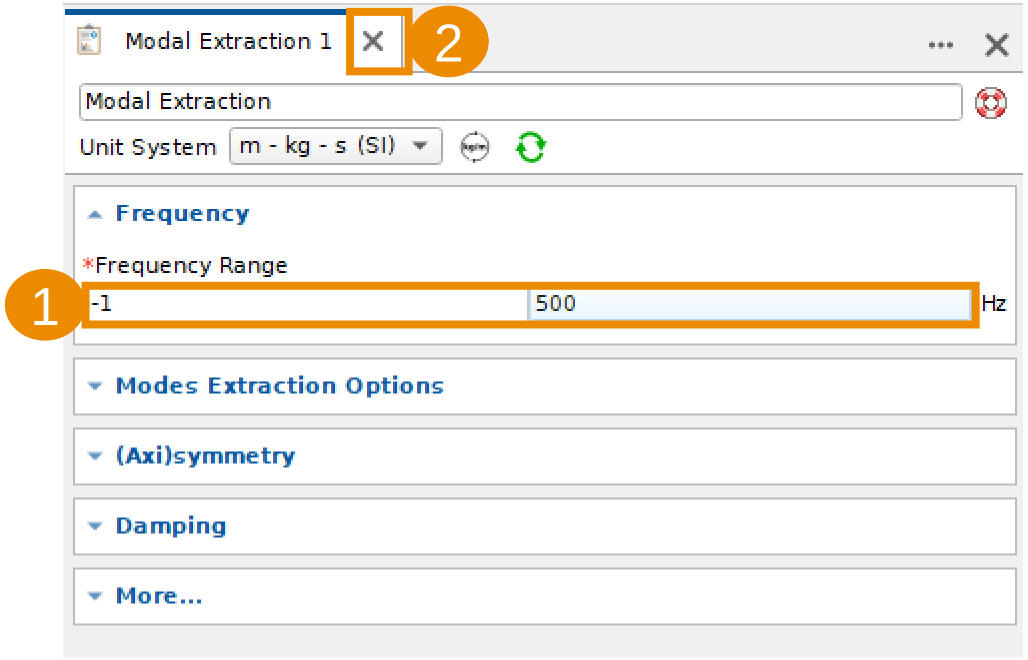
$$c_{bend} = \sqrt{\omega . t \sqrt{\frac{E}{12. \rho . (1 - \nu^2)}}}$$

$$\lambda_{min} = \frac{c_{bend}}{f_{max}} 10 * L_{max}$$

NOTE *The modal extraction starts at -1Hz to make sure that rigid body modes (numerically close to 0Hz) are calculated*

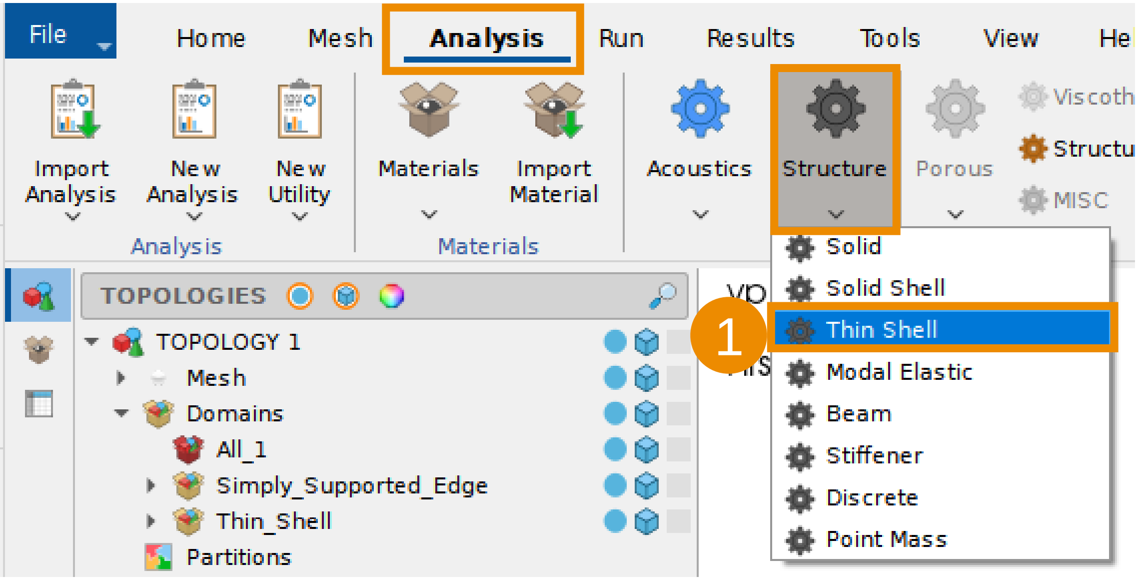
Define the range of frequency for modal extraction

- This analysis is performed from -1Hz to 500 Hz



Create the thin shell component

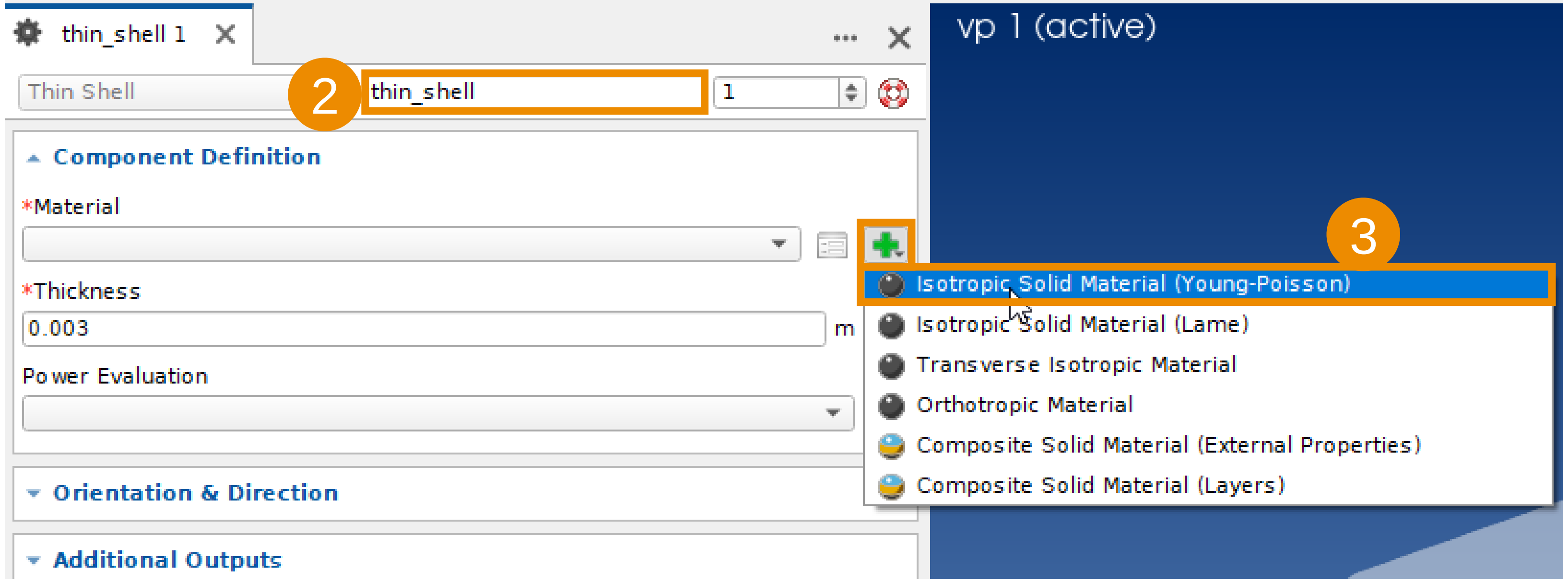
1. Create a new thin shell component



Add a Thin shell component

Specify the name of the Thin Shell component: thin_shell

Create a new Isotropic Solid Material (Defined by its Young modulus and Poisson ratio)



Create the thin shell component

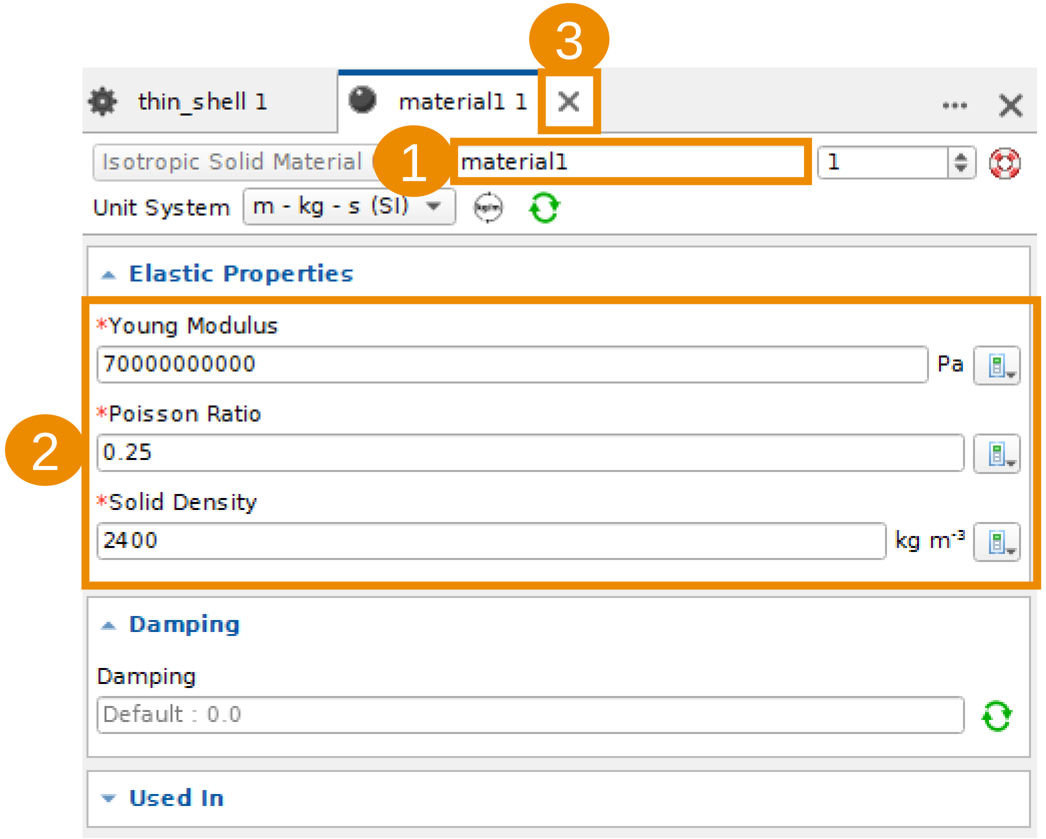
2 – Set up the Isotropic solid Material

Set the name of the *material* component: material1

Set standard properties of aluminum

- Young modulus: 7e+10 Pa
- Poisson ratio: 0.25
- Density: 2400 kg/m³

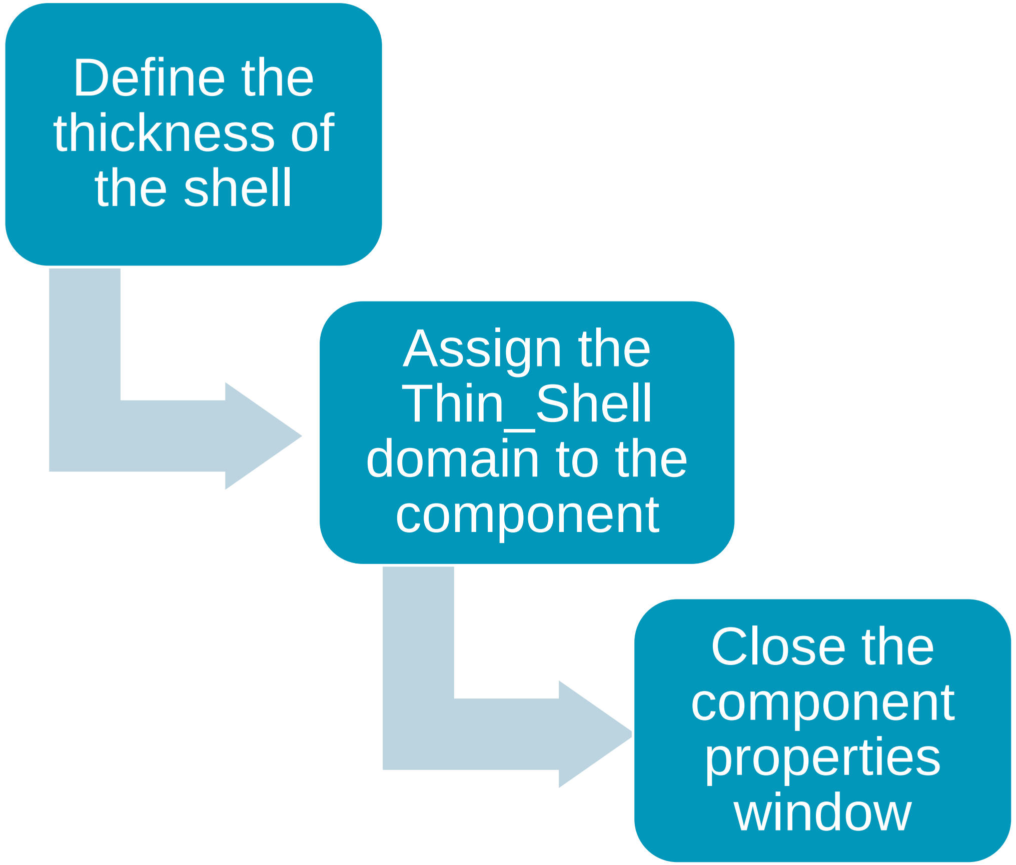
Close the material properties window

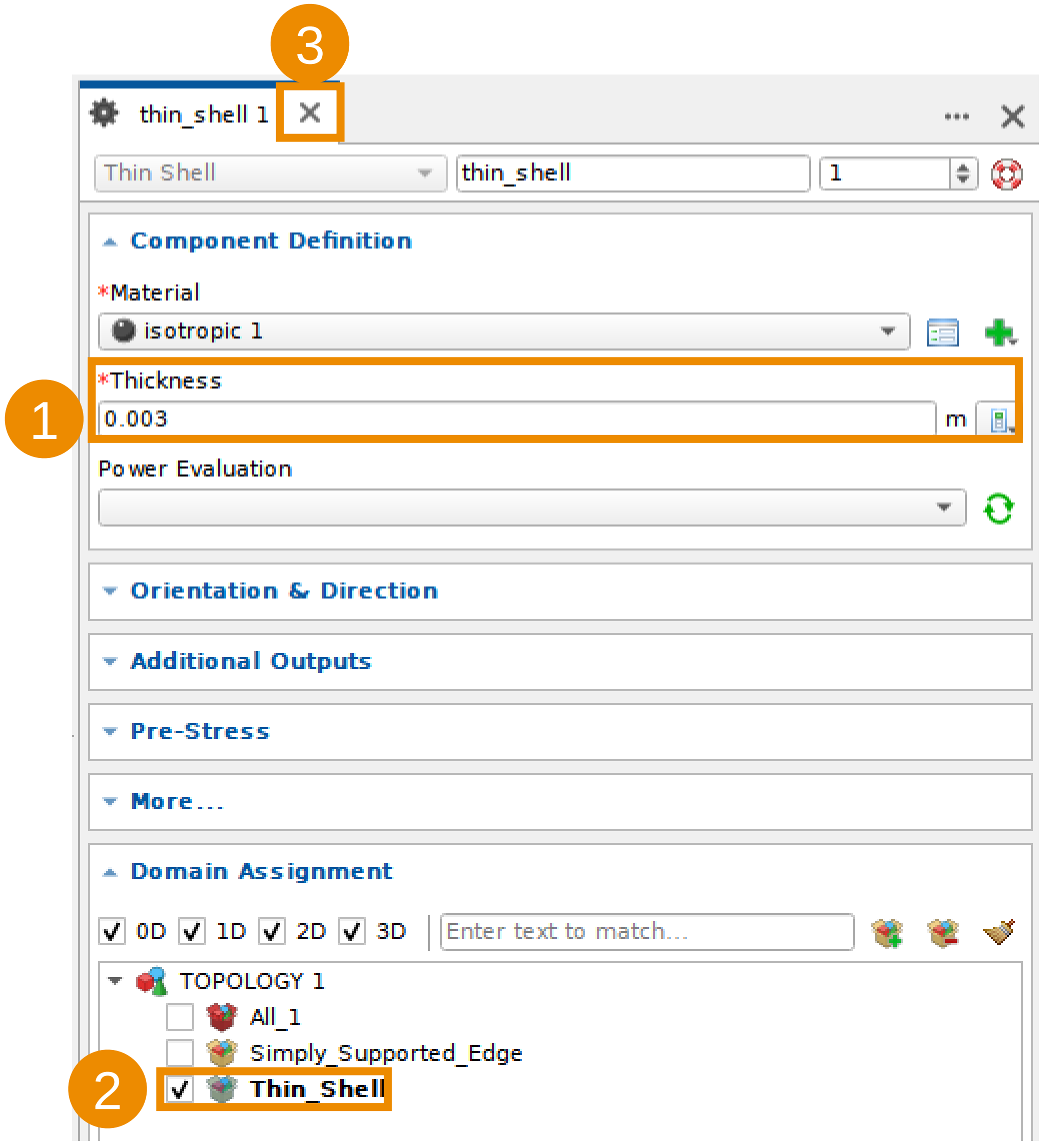


HEXAGON

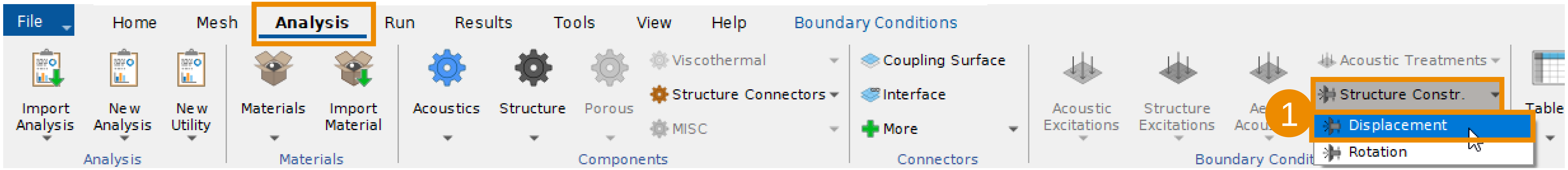
Create the thin shell component

3 – Assign the Domain





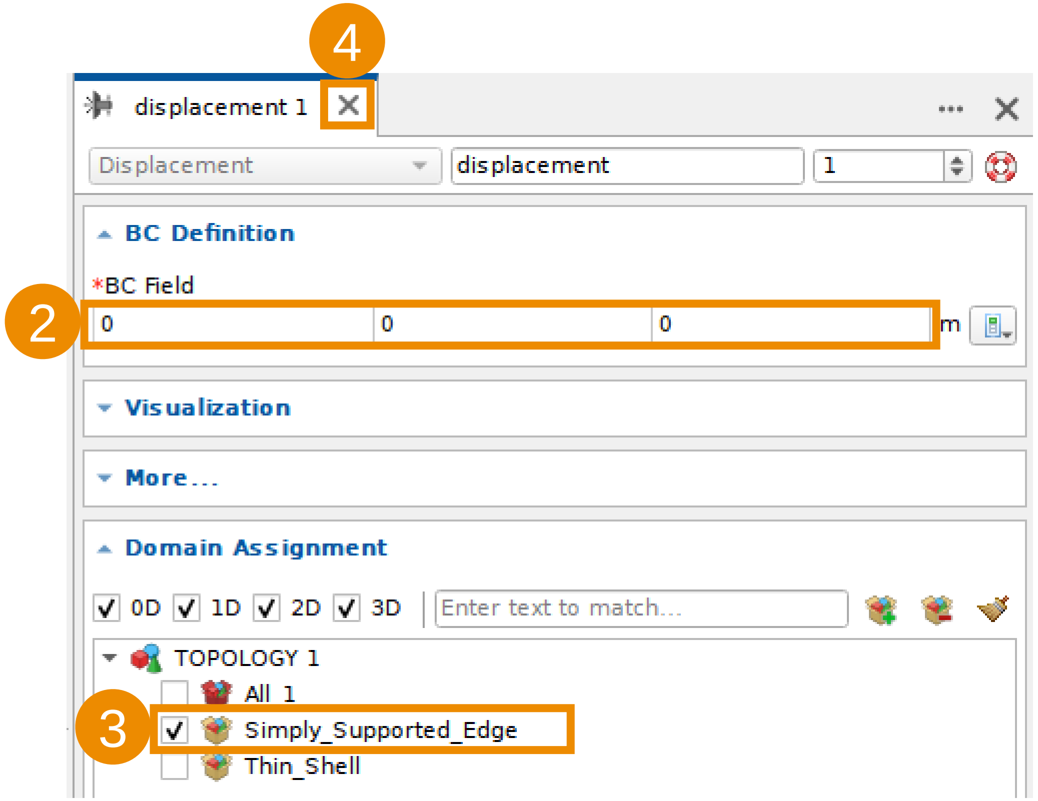
Create the simply supported boundary condition



Add a new Displacement BC

Set the boundary condition properties

Close the properties window

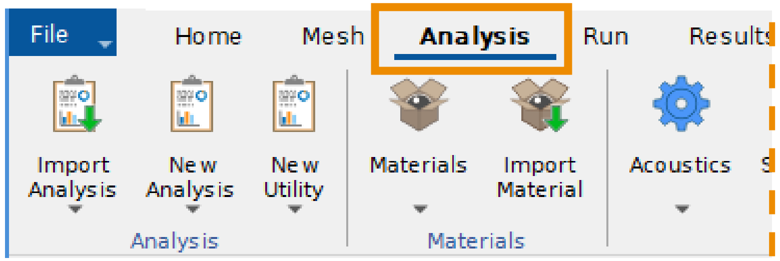


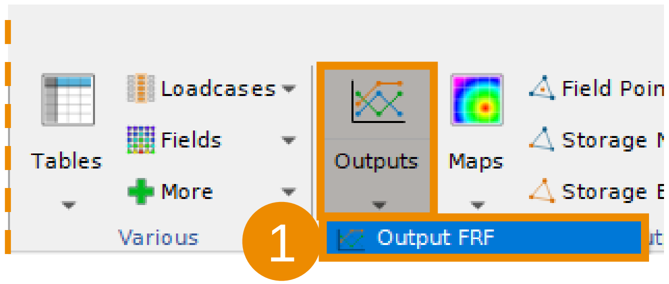
Set the post-processing parameters

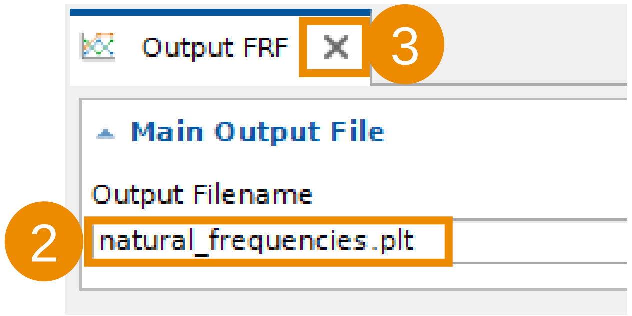
Add an Output FRF post-processing

Define the name of the output file *natural_frequencies.plt*

Close the property window

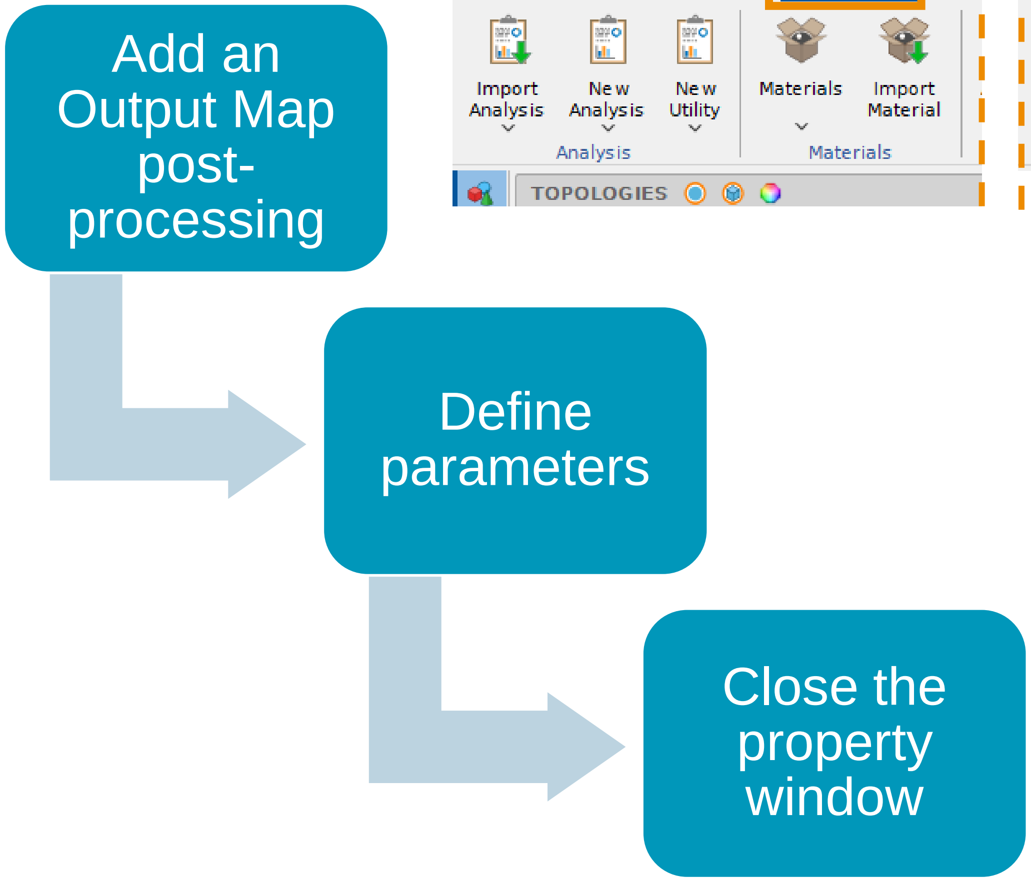


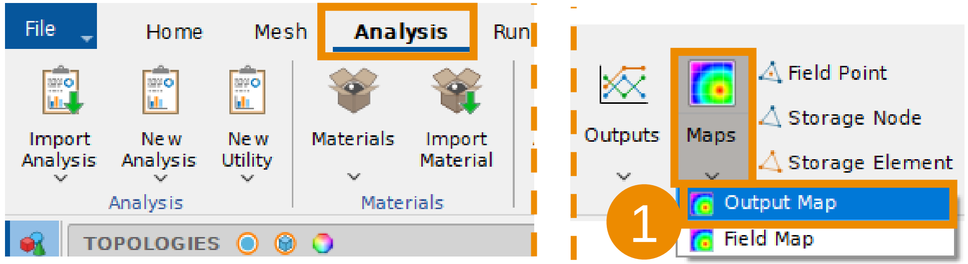


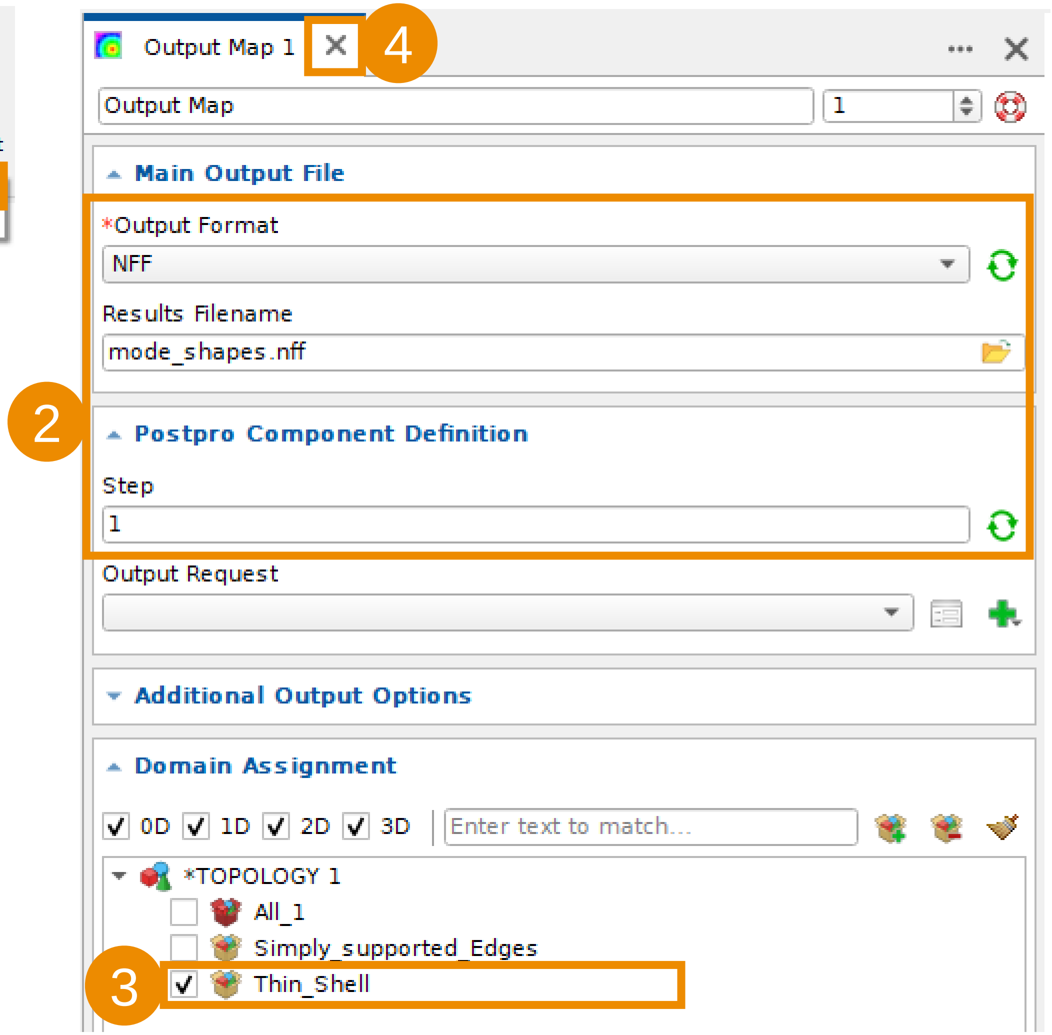


HEXAGON

Set the post-processing parameters





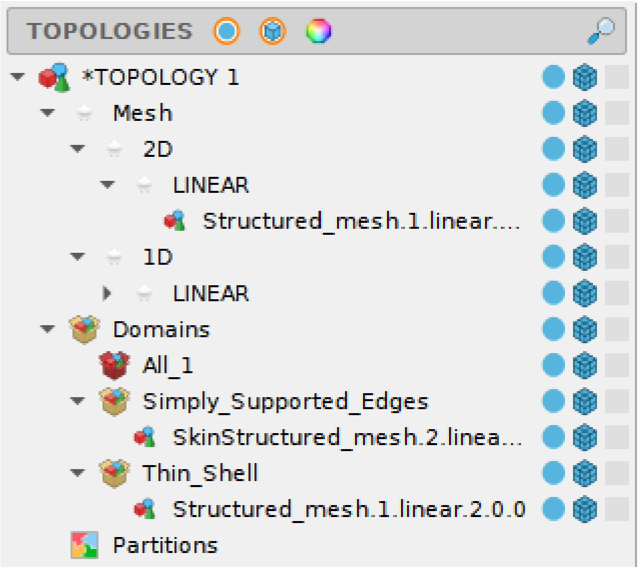


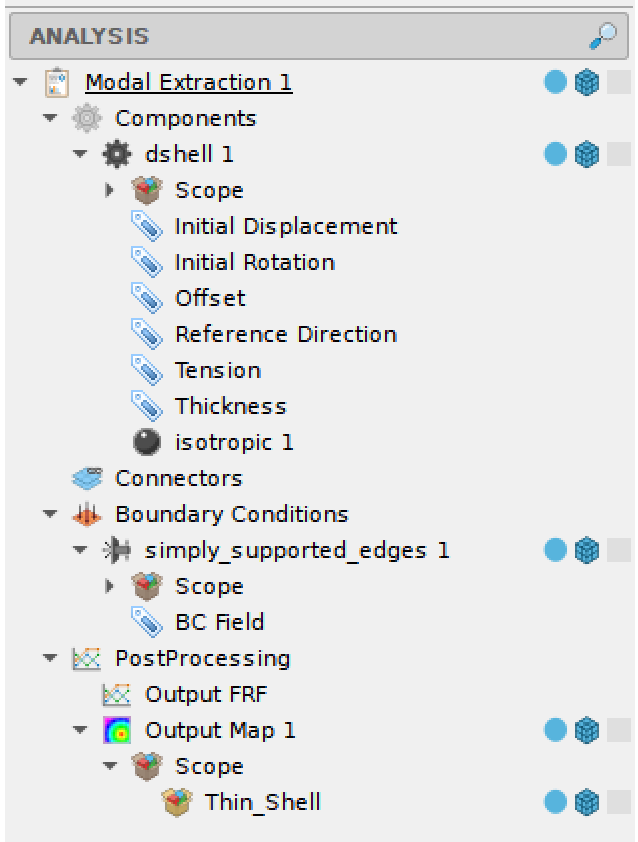
Check the analysis

The analysis set-up is now complete

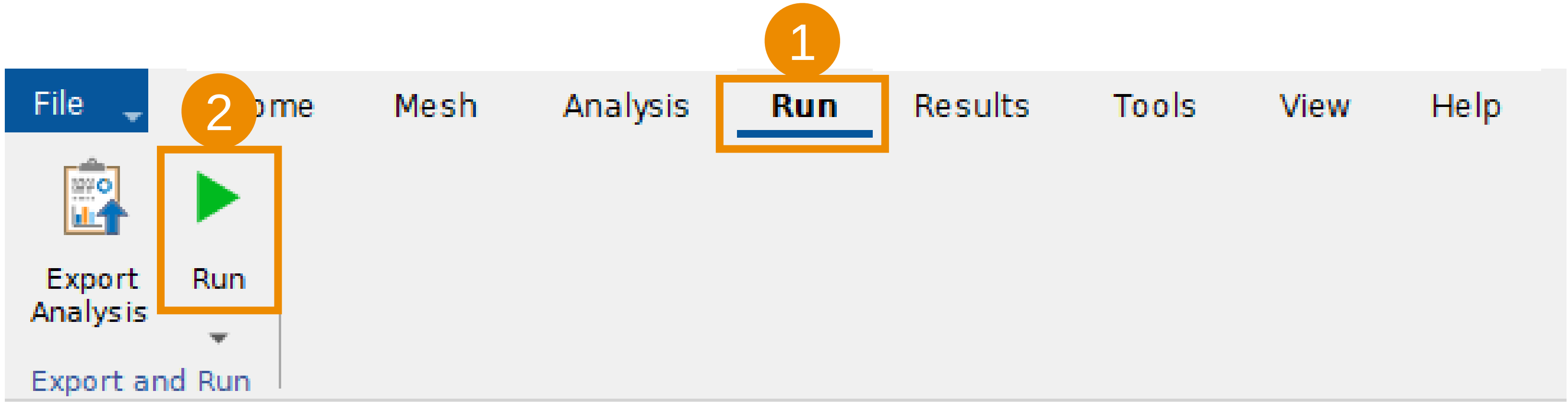
All the parts of the analysis are available and editable on the data tree panel

Check if the data tree is identical to the one shown



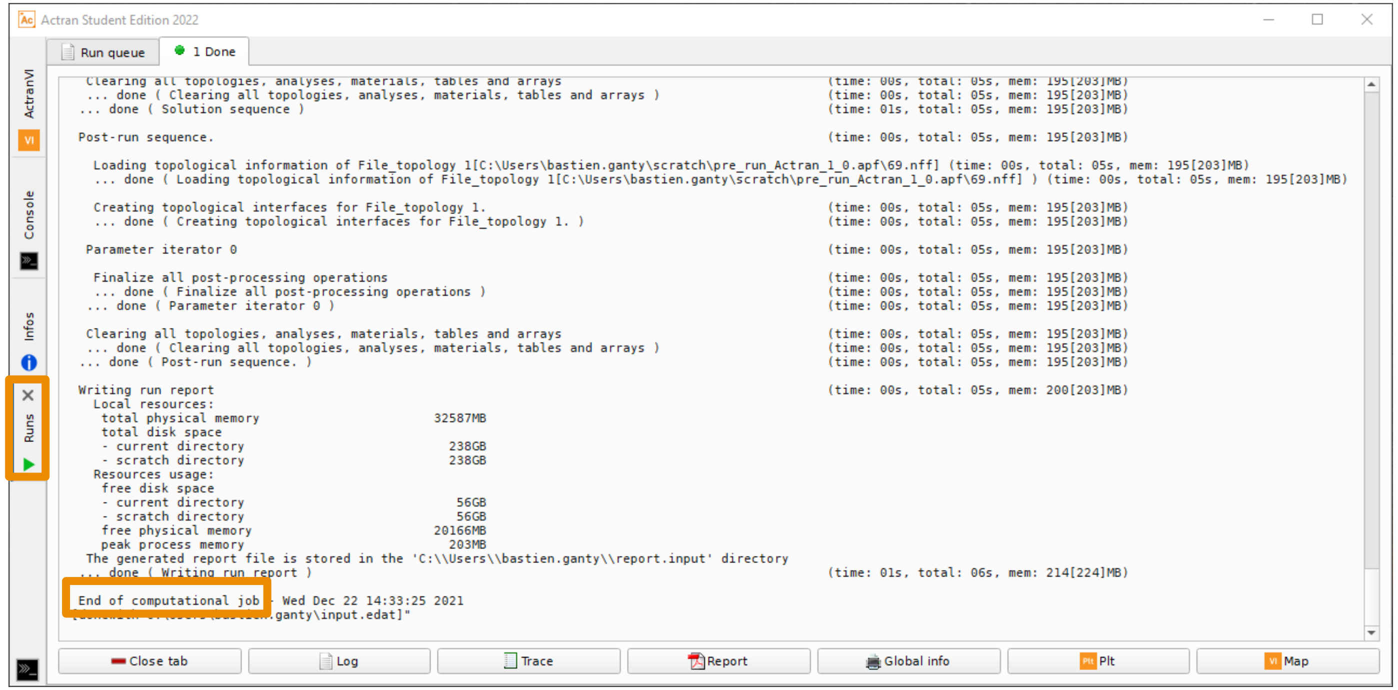


Launch the Actran analysis in ActranVI



Launch the computation

Check the log showing the computation progress



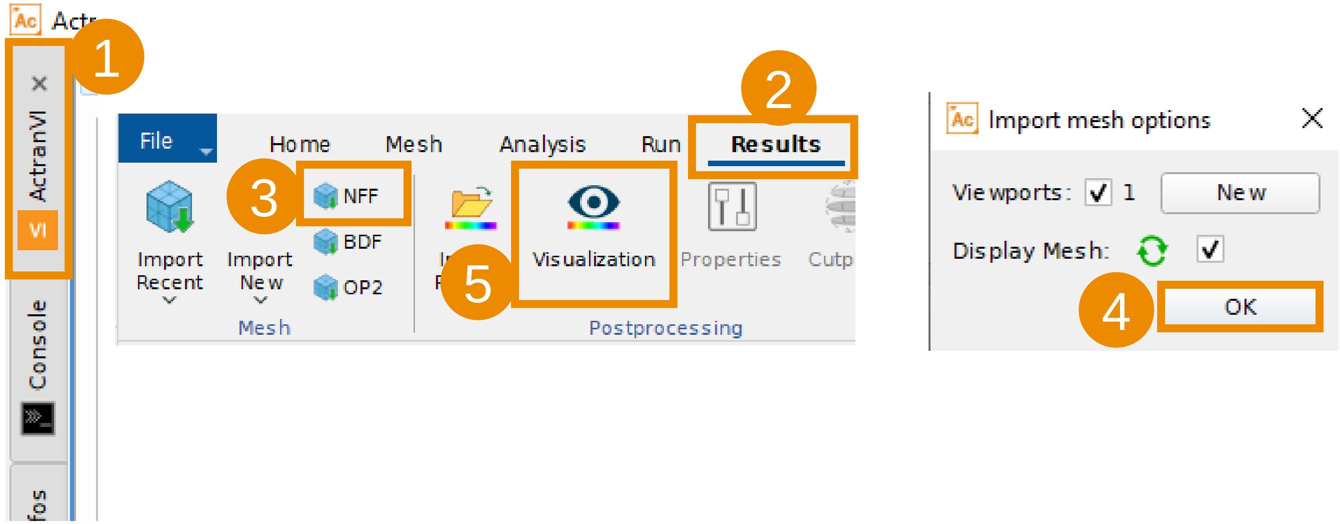
Post-processing

Examine eigenfrequencies

Visualize Mode shapes in ActranVI

HEXAGON

Visualize field maps (1)



Go back to ActranVI

Import the file *mode_shapes.nff*

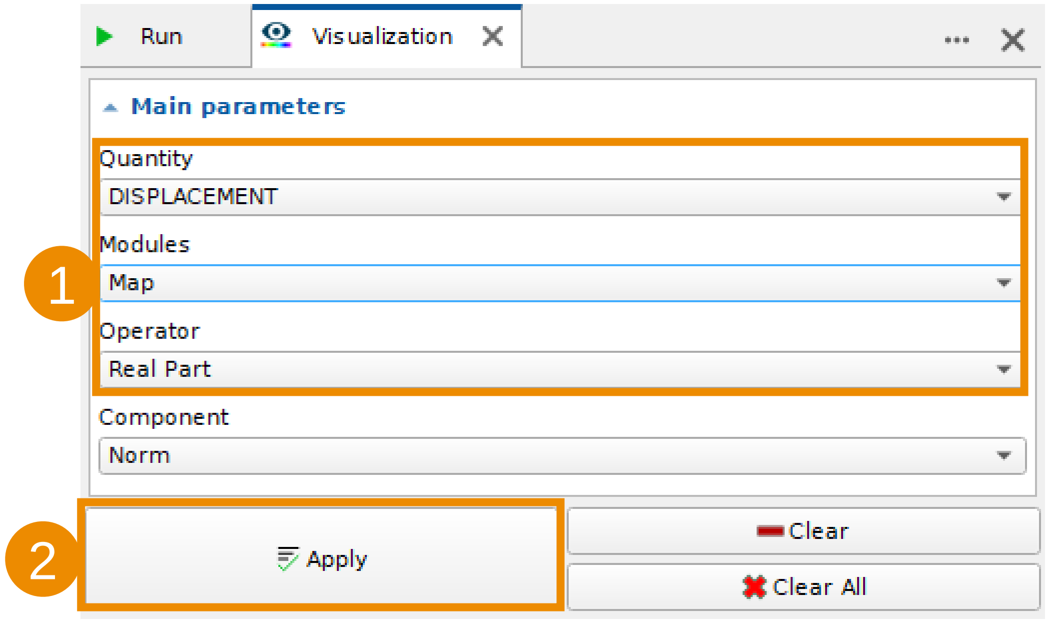
Open the Visualization window

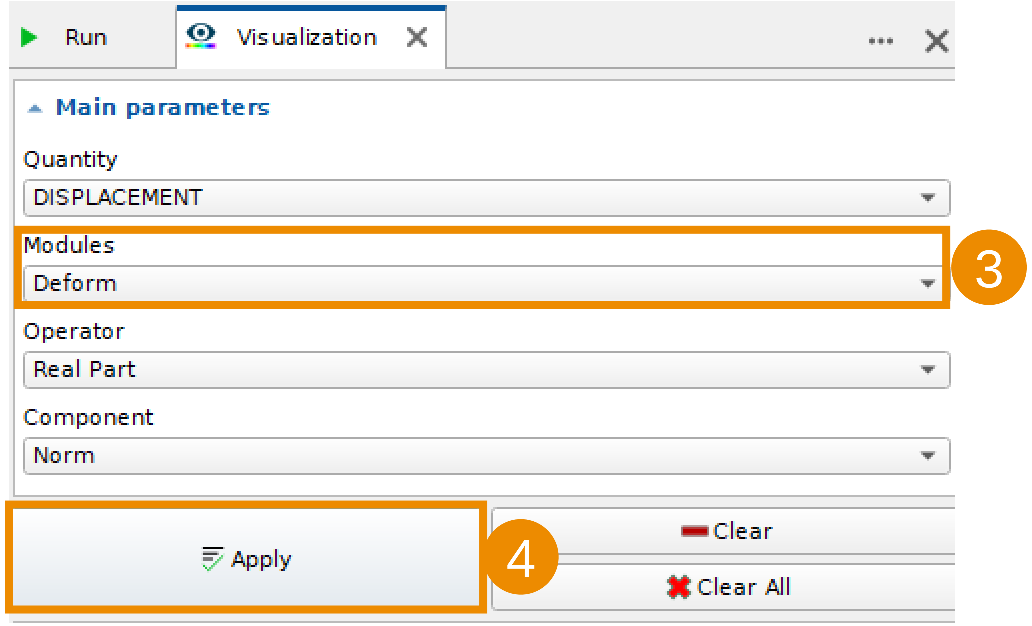
HEXAGON

Visualize field maps (2)

Plot the Real Part of Displacement field as a color map

Plot the Real Part of Displacement field as a Deform





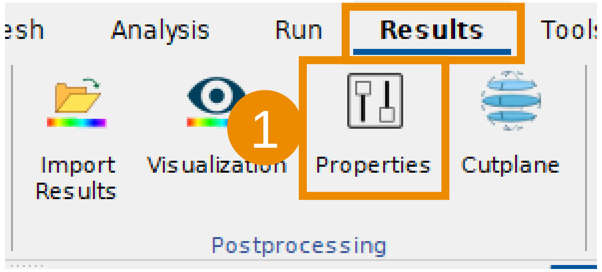
HEXAGON

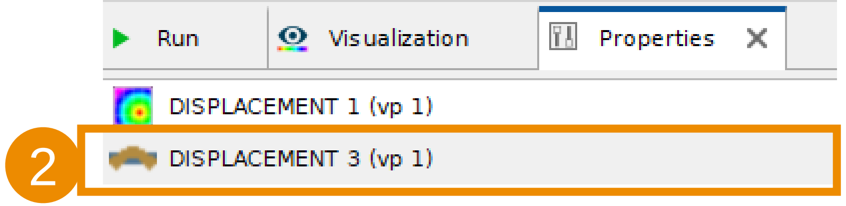
Visualize field maps (3)

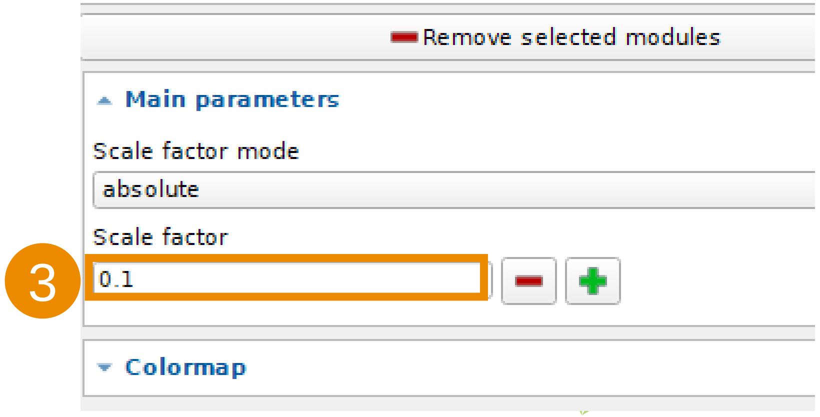
Open Properties windows of Post-processing

Select the deformed displacement plot

Adjust the scale factor

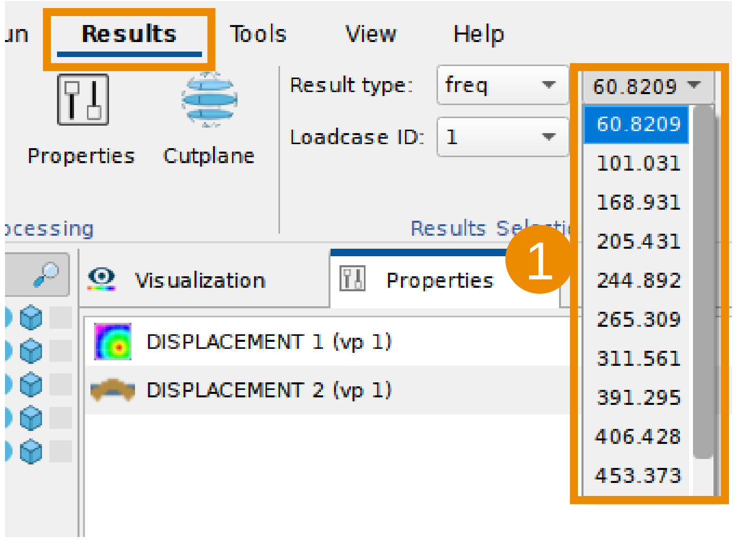


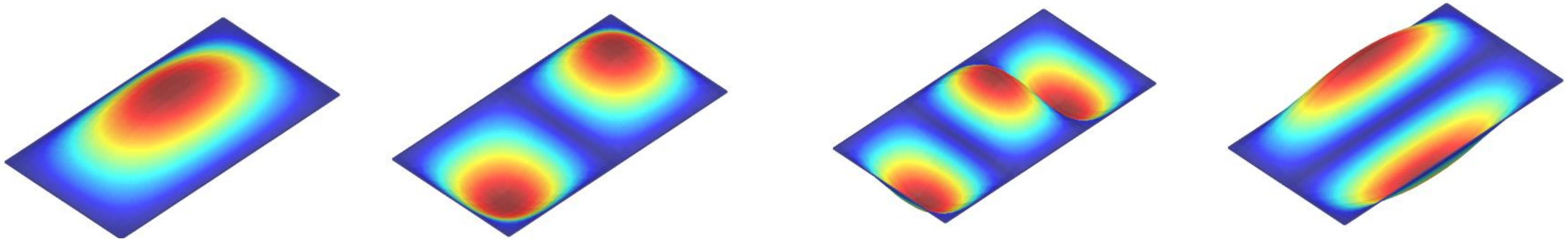




Mode shapes

- Select the natural frequencies of the modes
- The higher the frequency, the more complex the mode shape is
- The amplitude of the displacement does not have a physical meaning, the eigen vectors are normalized using the mass matrix





Mode 1 (1x1)
f = 60.820Hz

Mode 2 (2x1)
f = 101.031Hz

Mode 3 (3x1)
f = 168.931Hz

Mode 4 (1x2)
f = 205.431Hz

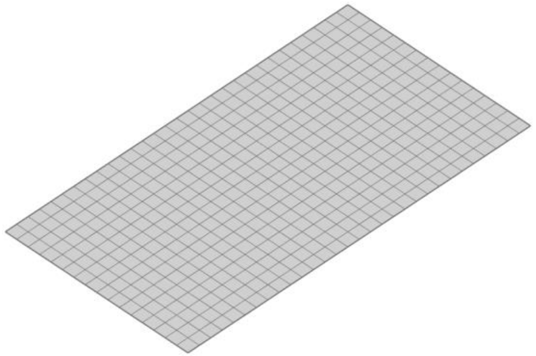
Mesh quality: impact on the accuracy

Same Analysis with Coarser Mesh

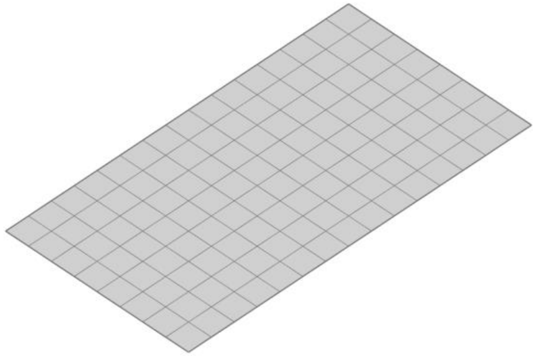
- The ability to accurately compute eigenfrequencies from input parameters in Actran is highly dependent on the mesh quality
- If the mesh is too coarse regarding the bending wavelength, it may not capture well the modes of the plate
- The provided input file input_mode_coarse.edat contains the same analysis using a coarser mesh. Eigenfrequencies calculated with this mesh will be output in natural_frequencies_coarse.plt and mode_shapes_coarse.nff
- Based on the 10 linear elements per bending wavelength criterion used to define the frequency range of the modal extraction, the maximum frequency for which the coarse mesh can be used is 120Hz

$$c_{bend} = \sqrt{\omega . t \sqrt{\frac{E}{12.\rho.(1-\nu^2)}}}$$

$$\lambda_{bend} = {c_{bend}}/{f} = 0.503 \ for \ f = 120Hz$$



Mesh created during the workshop
Elements size: 0.025



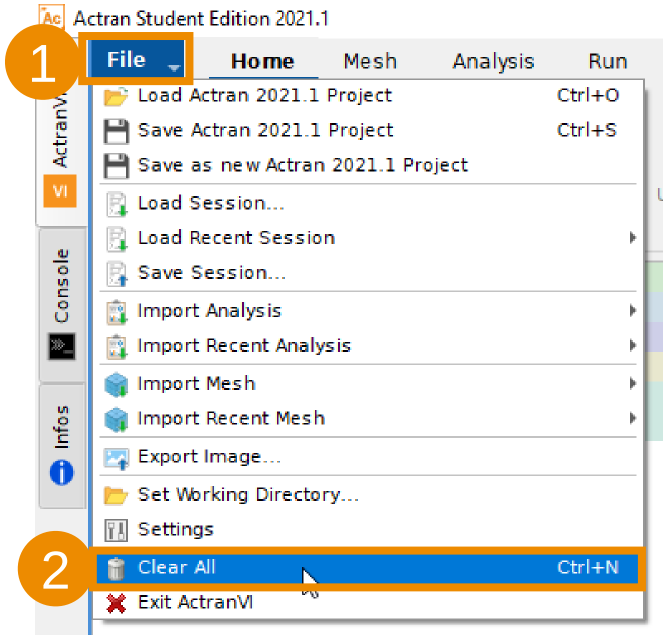
Coarse mesh used in input_mode_coarse.edat
Elements size: 0.05

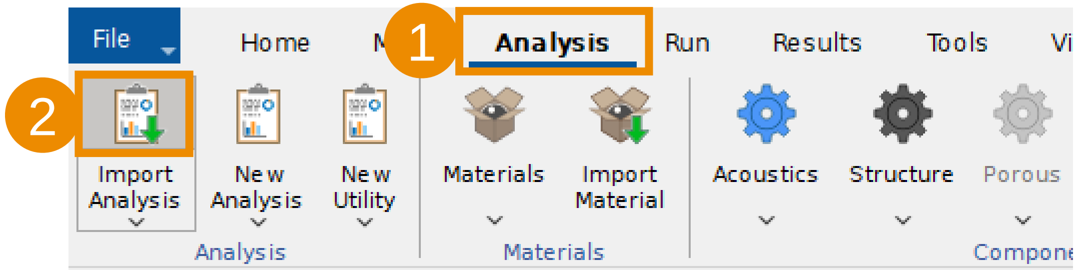
Run the provided analysis

Clear the workspace

Load the analysis *input_mode_coarse.edat* defined based on a coarse mesh

Run the analysis as in previous step





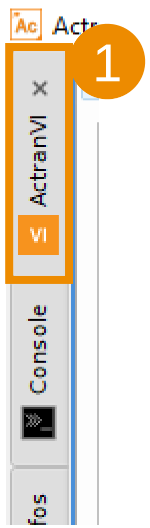
HEXAGON

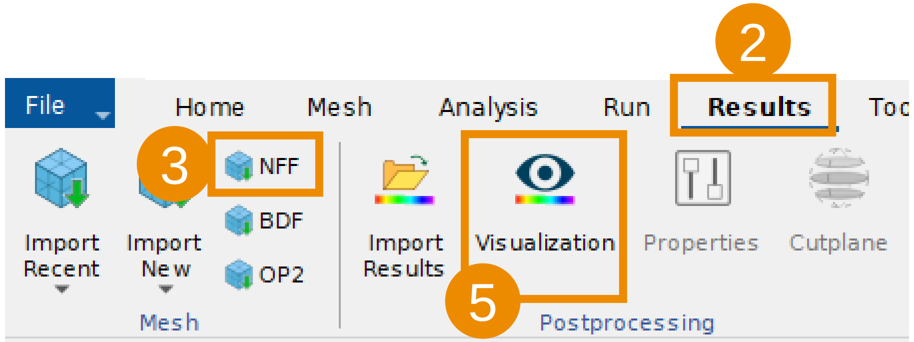
Post-processing

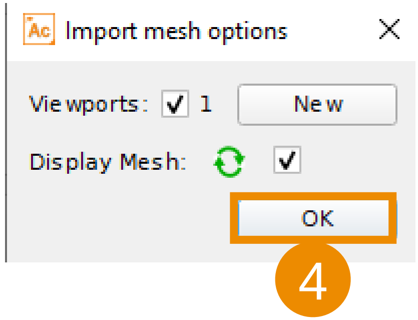
Load the mode_shapes_coarse.nff results file

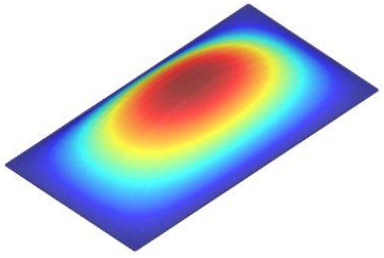
Perform similar post-processing as in previous step

Observe the difference for the natural frequencies

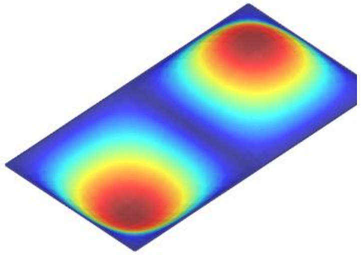




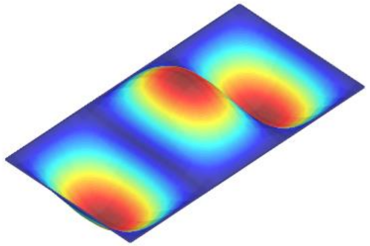




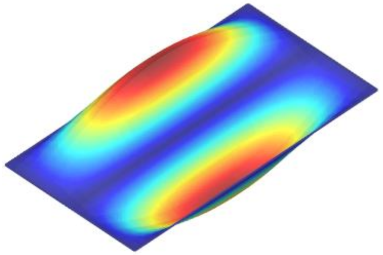
Mode 1 (1x1)
f = 61.04 Hz



Mode 2 (2x1)
f = 101.174 Hz



Mode 3 (3x1)
f = 170.979 Hz



Mode 4 (1x2)
f = 213.212 Hz

Mesh quality

Eigen Value

- Eigenfrequencies computed using the coarse mesh are accurate up to the second mode (~100Hz)
- For higher frequency modes the eigenfrequencies are less accurately calculated (confirming the criterion used for frequency range determination)
- Using the coarse mesh, the 8th and 9th modes do not appear in the same order
- Having a sufficiently refined mesh is a prerequisite for valid finite elements computation

Mode	(m x n)	Analytical	Actran Fine	Error on Fine	Actran Coarse	Error on Coarse
1	1 x 1	60,912	60.821	0.1%	61.040	0.2%
2	2 x 1	101,380	101.031	0.3%	101.174	0.2%
3	3 x 1	168,826	168.931	0.05%	170.979	1.3%
4	1 x 2	203,181	205,431	1.1%	213.212	5%
5	2 x 2	243,649	244,892	0.5%	251.028	3%
6	4 x 1	263,250	265,309	0.8%	273.916	4%
7	3 x 2	311,094	311,560	0.1%	316.904	1.9%
8	5 x 1	384,653	391,294	1.7%	415.105	8%
9	4 x 2	405,519	406,428	0.2%	414.650	2.2%
10	1 x 3	440,295	453,372	3%	496.860	13%

Relative error calculation: $\varepsilon = \frac{|f_{analytic} - f_{actran}|}{f_{analytic}} * 100$

HEXAGON

Conclusions

Conclusions

- This workshop introduced a structural modal extraction on a plate and the mode shapes have been visualized in ActranVI

- The model involved:
  - A 2D plate modeled with thin shell elements
  - A displacement constraint on the plate edges

- The results highlighted:
  - An increasing mode pattern complexity as the frequency increased
  - The importance of having a sufficiently refined mesh as a prerequisite for modal extraction

- Going further:
  - Investigate how plate modes are modified if the plate properties change
  - Young Modulus, Density, Thickness, Size of the plate can be modified
  - The plate edges can be clamped or supported on only two edges