Plate Forced Response

Actran Student Edition Tutorial

Introduction

- Pre-requisites - before going through this presentation, the reader should have read and understood the following presentation:
  - Workshop: Extraction of plate modes

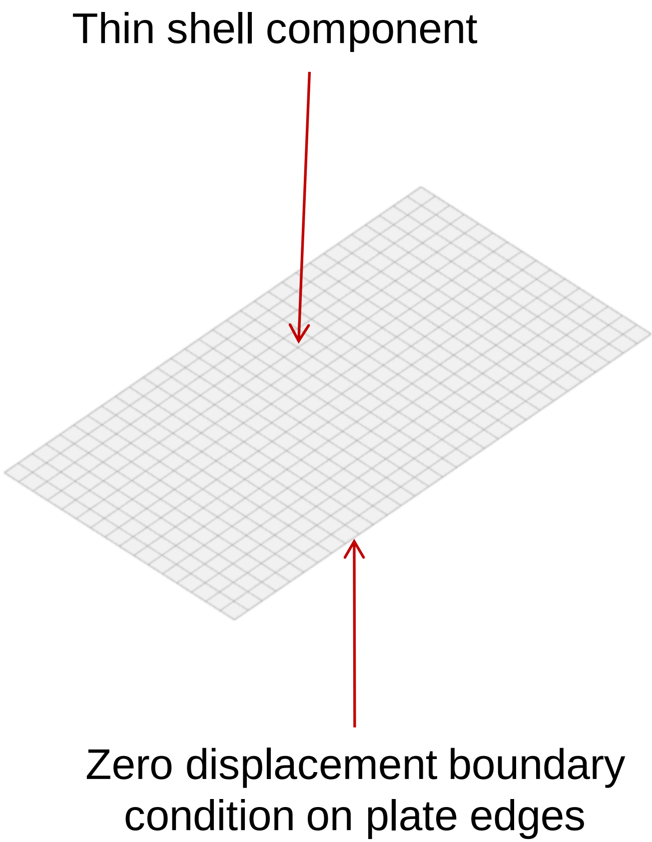
- This workshop demonstrates Actran capabilities to compute structural response to excitations

- The objectives of this workshop are the following :
  - Get introduced to structural dynamics
  - Get introduced to structural components of Actran
  - Get introduced to the direct solution sequence

- Software Version:
  - Actran 19 Student Edition

Workshop Description

- The objective of this workshop is to compute the structural response to a point load excitation
- The plate is modeled in 2D
  - A thin shell component is defined. The plate is modeled by 2D elements and its thickness is defined in the component properties
- The plate is simply supported on its edges
  - A zero displacement boundary condition is defined
- The excitation is modeled as an harmonic force applied on the plate
  - A point load boundary condition is defined



Analytical solution

- Let us consider a plate with the following properties:
  - Size: L_x = 0.75 m, L_y = 0.40 m, thickness t = 0.003 m
  - Material properties: E = $7\text{x}10^{10}$ Pa with 1% damping v = 0.25, ρ = 2400 kg/m^3
  - Plate simply supported along the four edges
  - Time-harmonic point load with unit amplitude at point x = 0.2 m, y = 0.1 m
    $F(t) = \mathcal{R}\left(Ae^{i\omega t}\right)\ where\ A = 1\ and\ \omega = 2\pi f$
- The system of differential equations characterizing the system is

$$M\ddot{U}(t) + KU(t) = F(t)$$

- For the steady solution, the time harmonic response is

$$U(t) = \mathcal{R}\left(\tilde{U}e^{i\omega t}\right)$$

- System of algebraic equation for the steady state solution

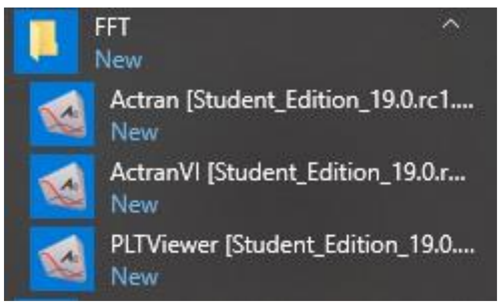
$$\left(-\omega^2 M\tilde{U} + K\tilde{U}\right)e^{i\omega t} = \tilde{R}e^{i\omega t} \quad \Rightarrow \quad (K - \omega^2 M)\tilde{U} = \tilde{R}$$

Workshop Pre-Processing

Direct Frequency response Analysis

Start ActranVI

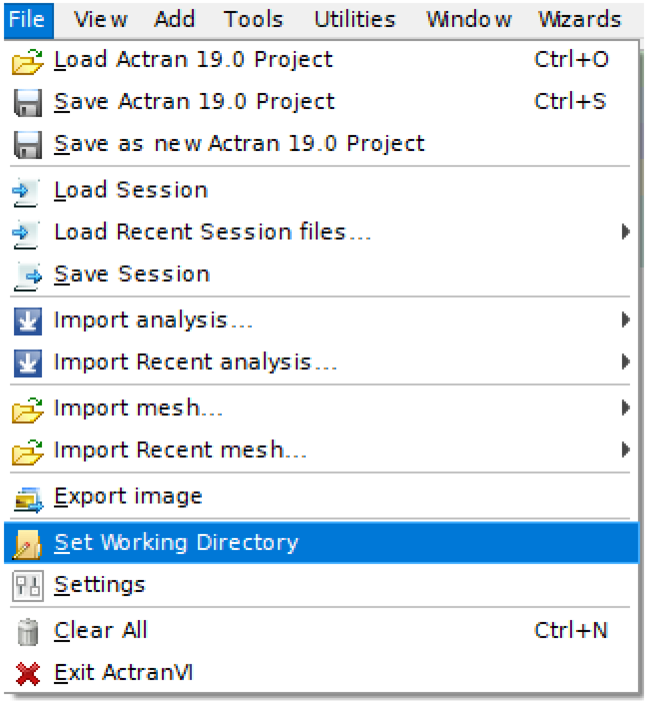
- Start ActranVI:
  - shortcut is available through the Windows Start Menu



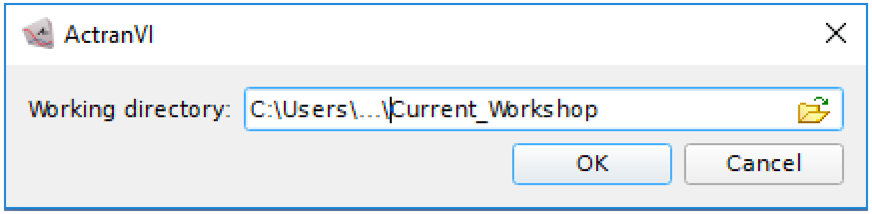
(Windows Start Menu)

Set the Working Directory

- The working directory is the default directory where all the files are output
- Click on :
  - File → Set Working Directory…
- Select the workshop directory as the working directory

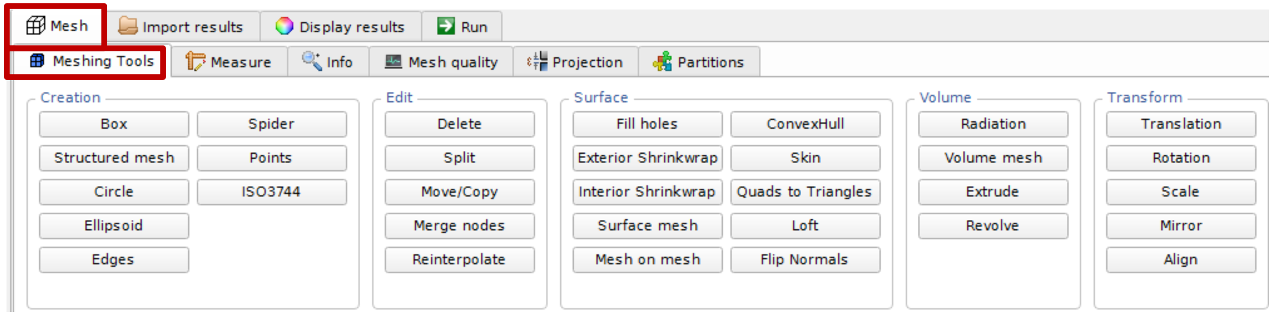


<u>Important:</u> The working directory path should not contain any space or special character

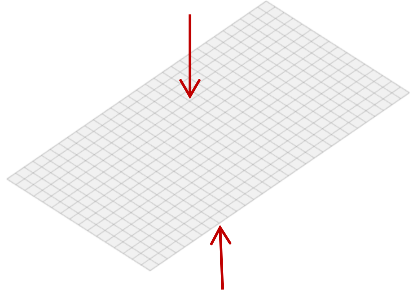


Create the mesh
1 – General introduction

- 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
- Three 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
  - One 0D point mesh element set to support the point load excitation
- Meshing tools can be found in ActranVI toolbox, under Mesh → Meshing tools



Thin shell component
2D element set

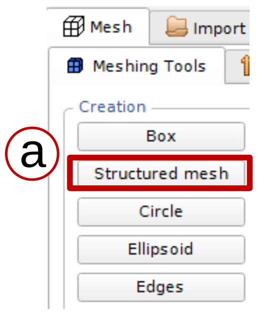


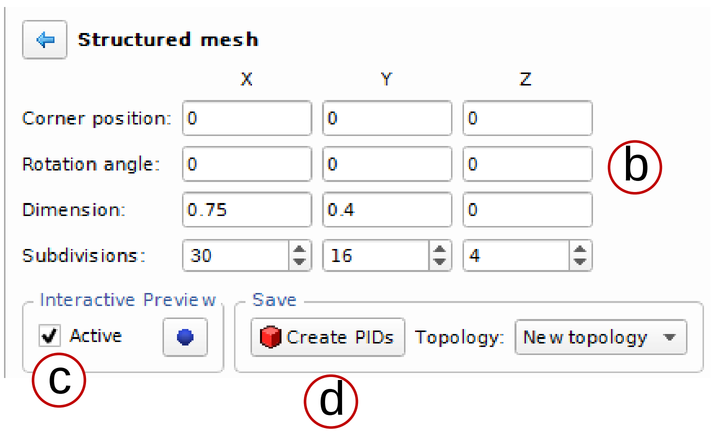
Zero displacement BC
1D element set

Create the mesh

2 – Create the 2D element set – Structured mesh

- The 2D element set is a rectangle with length $L_x = 0.75$ m and $L_y = 0.40$ m
- The target element size is 0.025m
- The Structured Mesh function is used to create the plate
  a) On the meshing tools toolbox select the Structured Mesh function
  b) Adjust the function parameters to create the mesh according to the problem definition
  c) Pre-visualize the mesh using the interactive preview
  d) If the mesh corresponds to what is expected the element set can be created by clicking “Create PIDs”



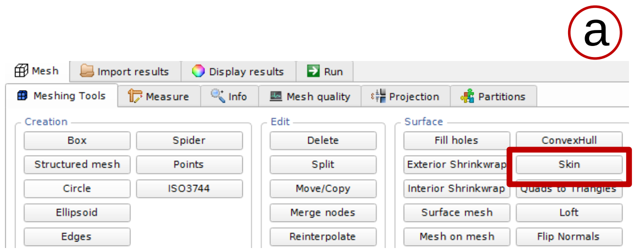


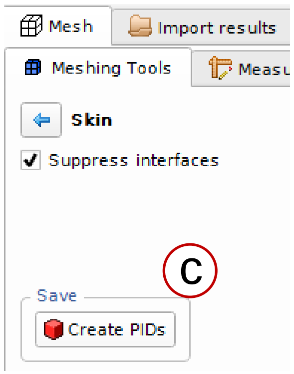
- The created mesh contains linear elements

Create the mesh

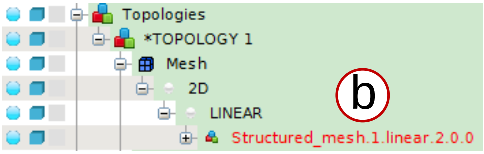
3 – Create the 1D element set – Skin

- The 1D element set that must be created corresponds to the free edges of the 2D element set that that was created in previous slide. Therefore the Skin function can be used to create this 1D element set
  a) On the meshing tools toolbox select the Skin function
  b) Make sure the plate element set is selected (An element set is selected when it is colored in red). To select an element set, click on it on the graphical tree
  c) Click on “Create PIDs” to create the 1D element set





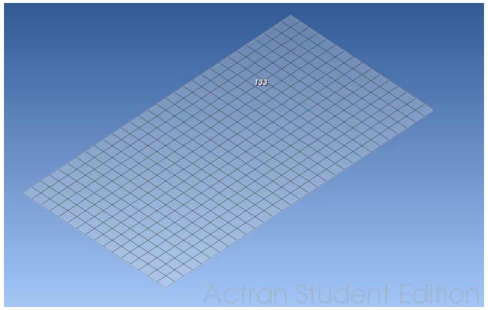
- The mesh needed to run the analysis was created
- It will be used to setup the analysis and run the calculation

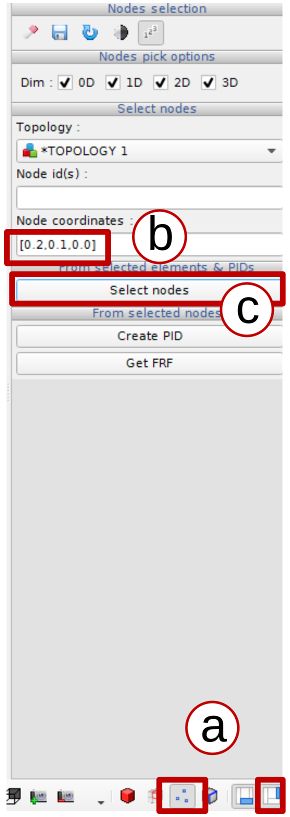


Create the mesh

4 – Create the 0D element set

- The mesh includes 1D and 2D element sets
  - The thin shell component will be defined on the 2D element set
  - The zero displacement boundary condition will be defined on the 1D element set
- The point load excitation has to be carried by a point of the mesh. The element set corresponding to this point has to be created
  a) Select the node selection mode and open the picking options
  b) Search for coordinates [0.2, 0.1, 0.0]. Write the coordinates and press the Enter key.
  c) Select the node

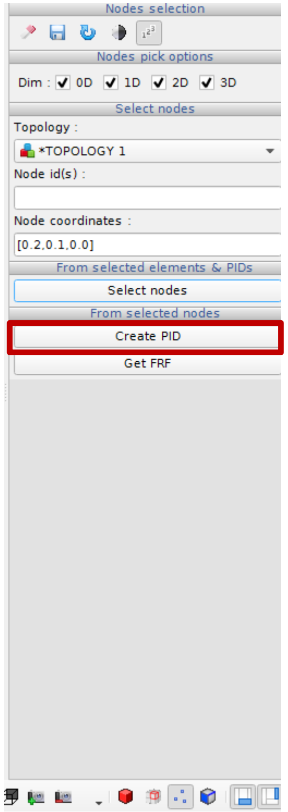




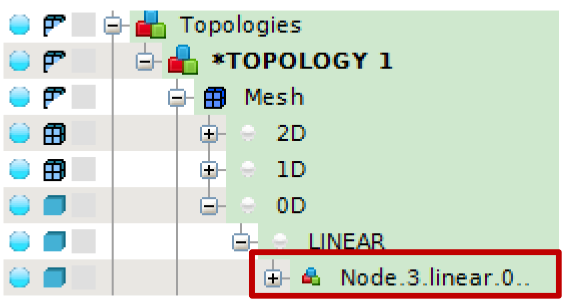
Create the mesh

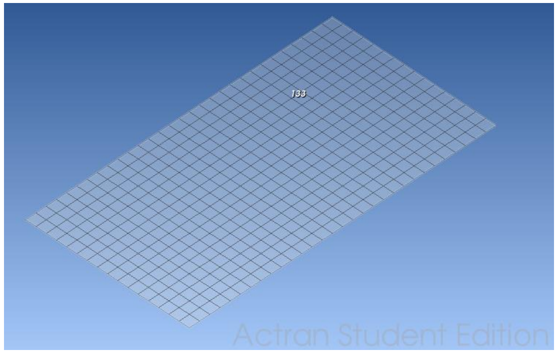
4 – Create the 0D element set

- Once the node is selected a new element set can be created to support the point load boundary condition
  - Click on Create PID from nodes selected



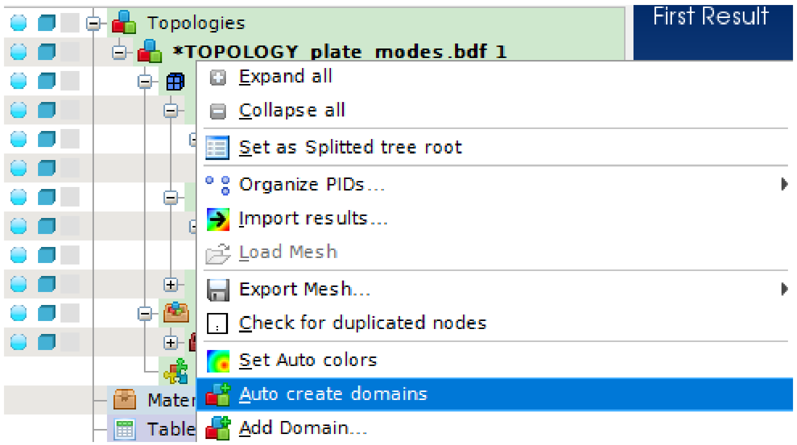
- The 0D element is created and appear in the Topologies tree and the mesh



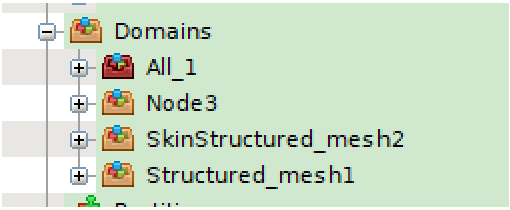


Create the Domains

- Auto create the domains of the (right click):
  → *TOPOLOGY → Auto create domains



- Rename the domains with appropriate names (right click on each domain → Properties…) :

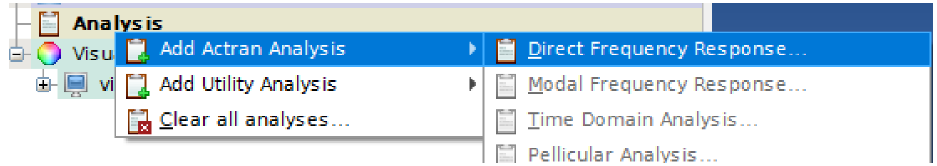


Default name	New name
SkinStructured_mesh2	Simply_supported_edges
Structured_mesh1	Thin_shell
Node3	Point_load_node

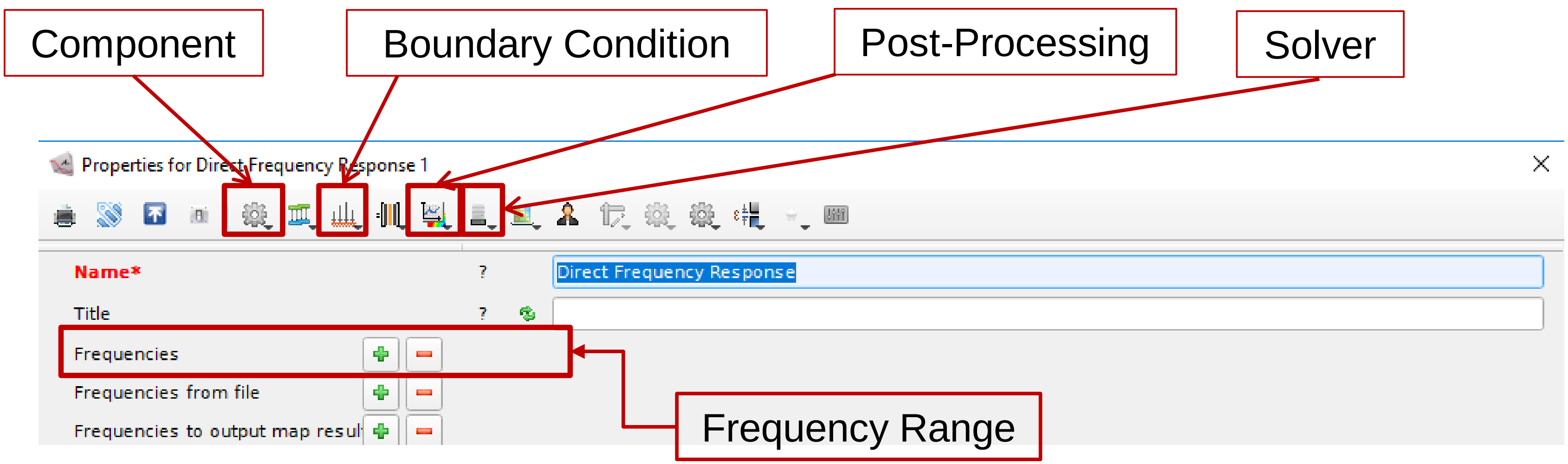
Remark: the domains are automatically sorted following their names

Create the Direct Frequency Response

- Create a Direct Frequency Response analysis by right-clicking on “Analysis”



- The analysis properties window pops up. It is the window from which the different parts of the analysis are defined

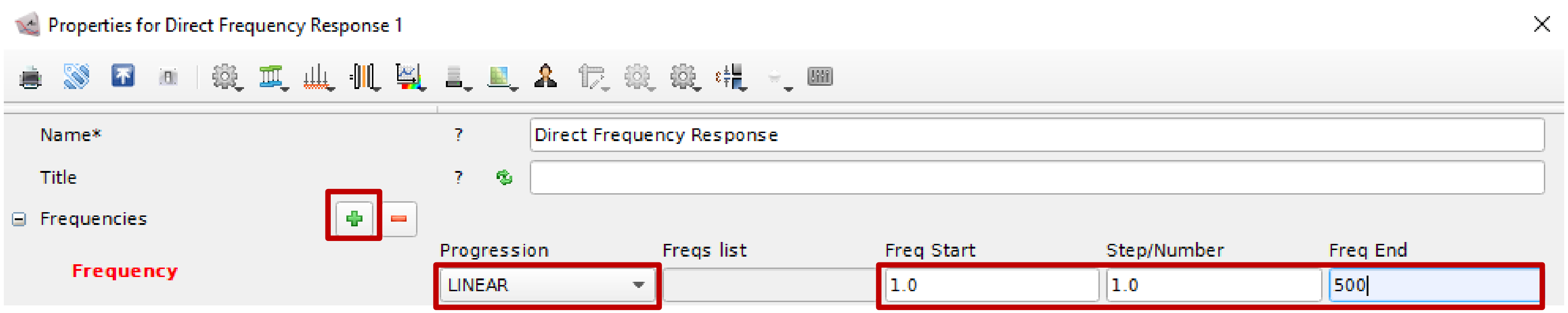


Specify the Frequency Range

- The analysis parameters are specified in the properties of the analysis
- As the largest element of this linear mesh is 2.5 cm, the smallest bending wavelength accurately modeled is : 10 * 0.025 = 0.25 m (based on 10 linear elements per wavelength criterion)
- The bending wavelength of a simply supported steel plate (3 mm thick) at 500 Hz is 0.246m

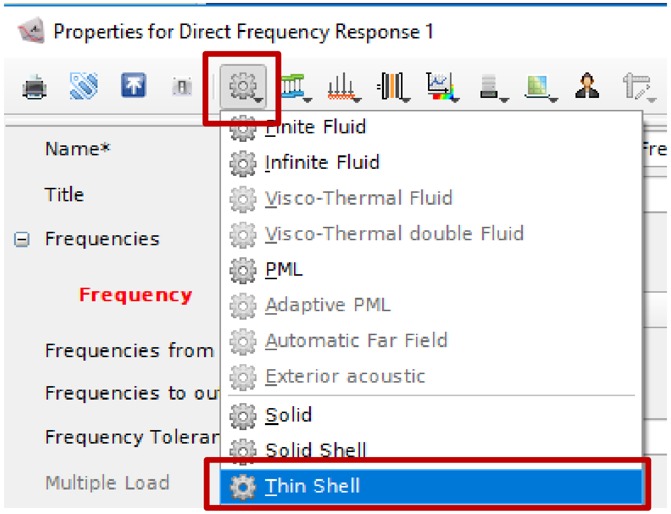
$$\lambda_{bend} = c_{bend}/f \qquad c_{bend} = \sqrt{\omega . t \sqrt{\frac{E}{12.\rho.(1-\nu^2)}}}$$

- The mesh can then be considered as valid up to 500 Hz
- This analysis is performed from 1Hz up to 500Hz with a 1Hz step

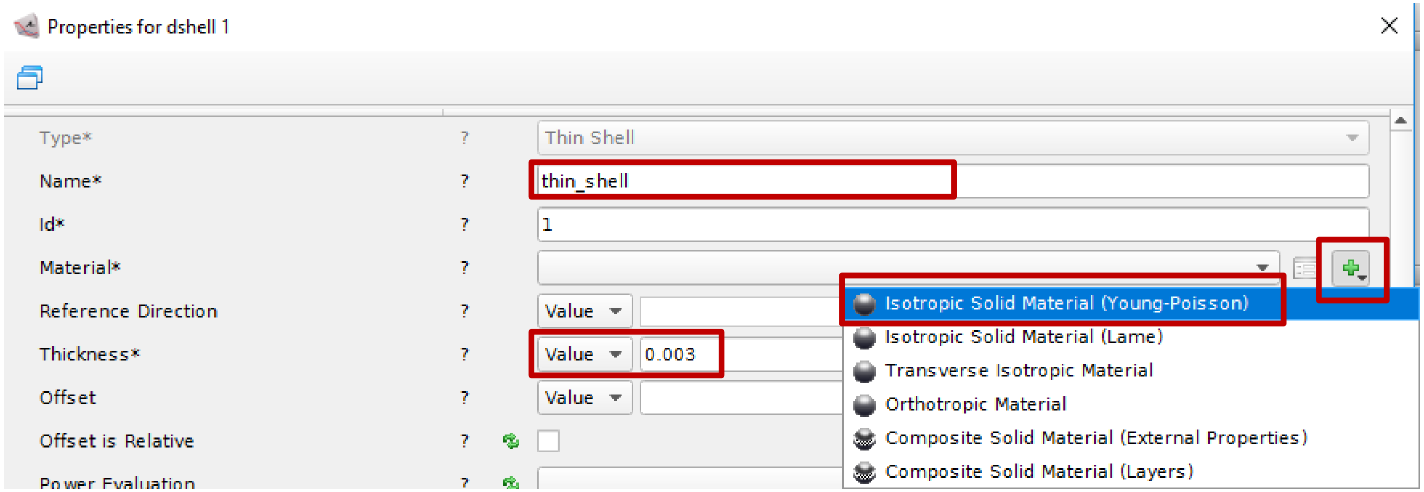


Create the Thin shell Component

1 – Add a Component



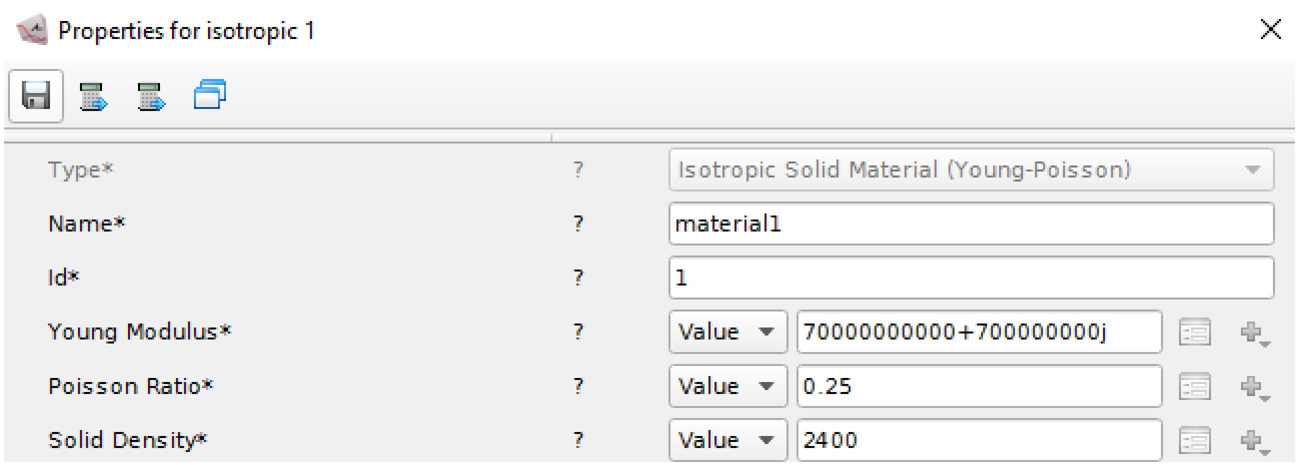
- Add a Thin shell component
- Component properties:
  - Specify the name of the Thin Shell component: thin_shell
  - Specify the thickness value: 0.003 m
  - Create a new Isotropic Solid Material (Defined by its Young modulus and Poisson ratio)



Create the Thin shell Component
2 – Set up the Isotropic solid Material

The dissipation by structural damping is taken into account through the imaginary part of the Young Modulus.

- Name: *material1*
- Set the following properties:
  - Young Modulus: 7e+10 + 7e+8j Pa
  - Poisson Ratio: 0.25
  - Density: 2400 kg/m^3

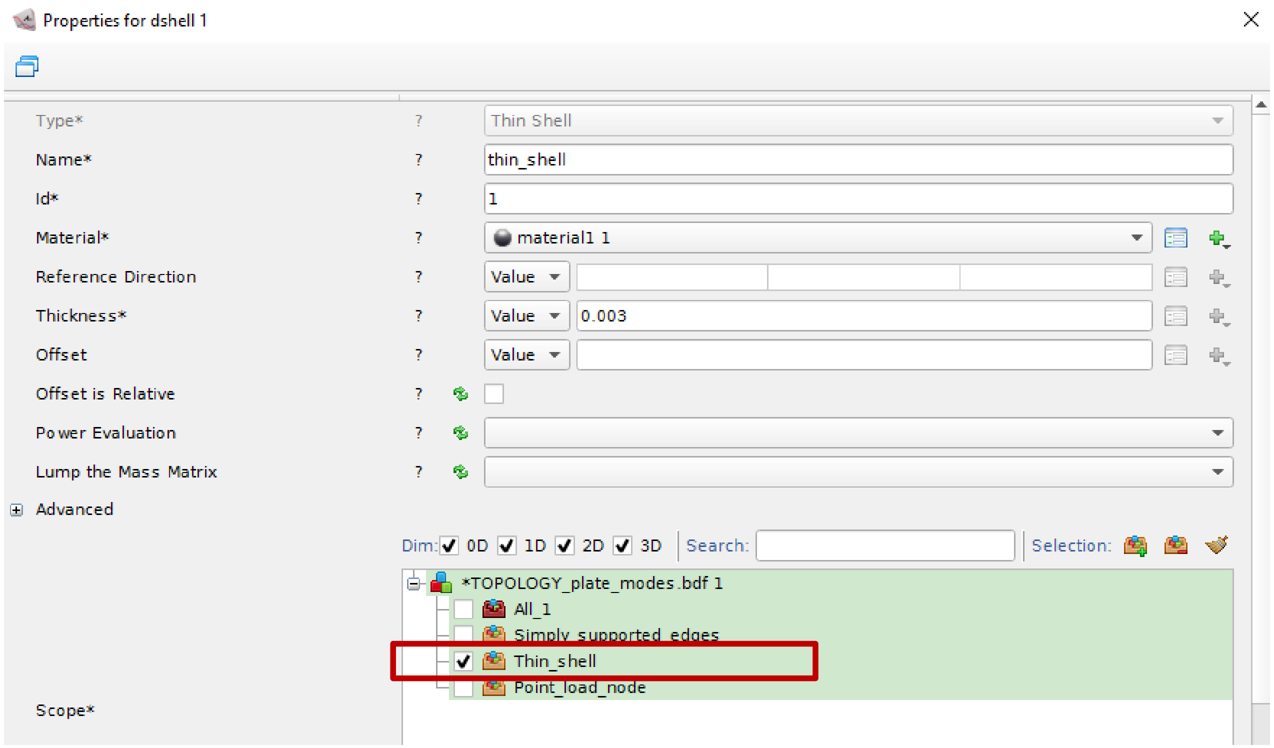


- Close the material properties window

Create the Thin shell Component

3 – Assign the Domain

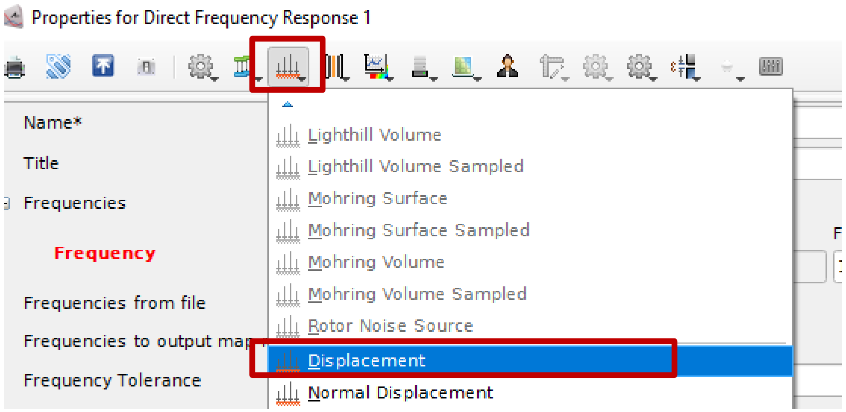
- With the Scope selector, assign the *Thin_shell* domain to the *Thin Shell* component

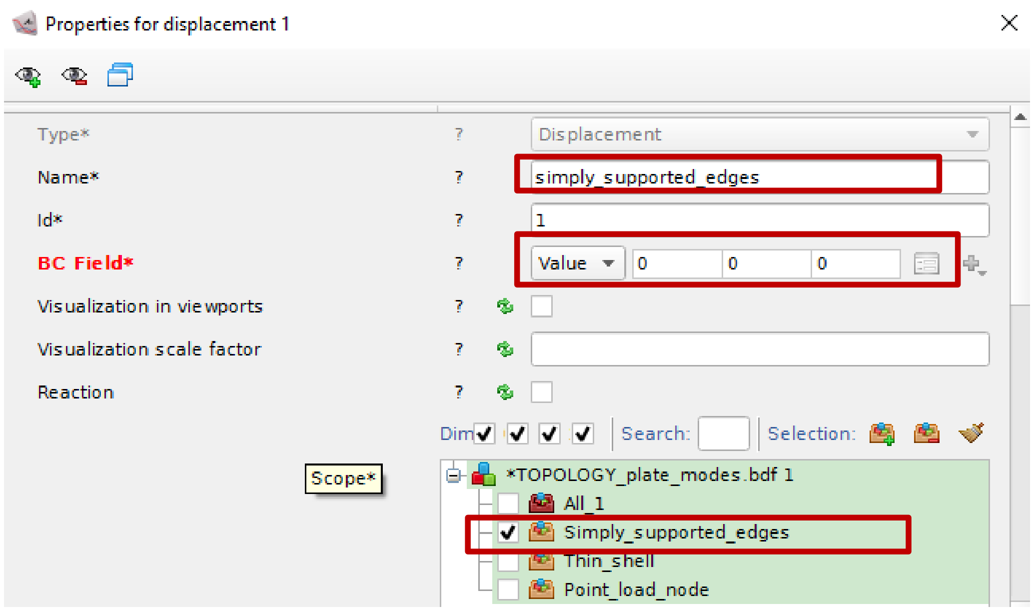


- Close the component properties window

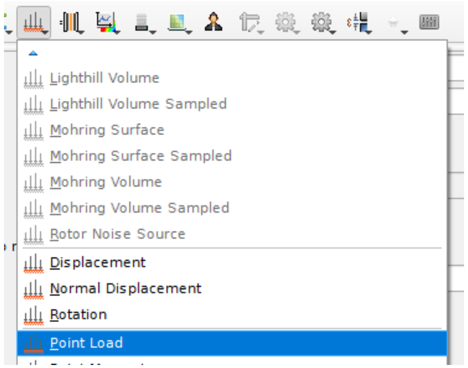
Create the Simply Supported Boundary Condition

- Add a Displacement Boundary condition
- Set the following properties
  - Name: Simply_supported_edges
  - BC Field: [0,0,0] (X and Y displacements are constrained to avoid rigid body modes)
  - Domain: Simply_supported_edges
- Close the boundary condition properties window

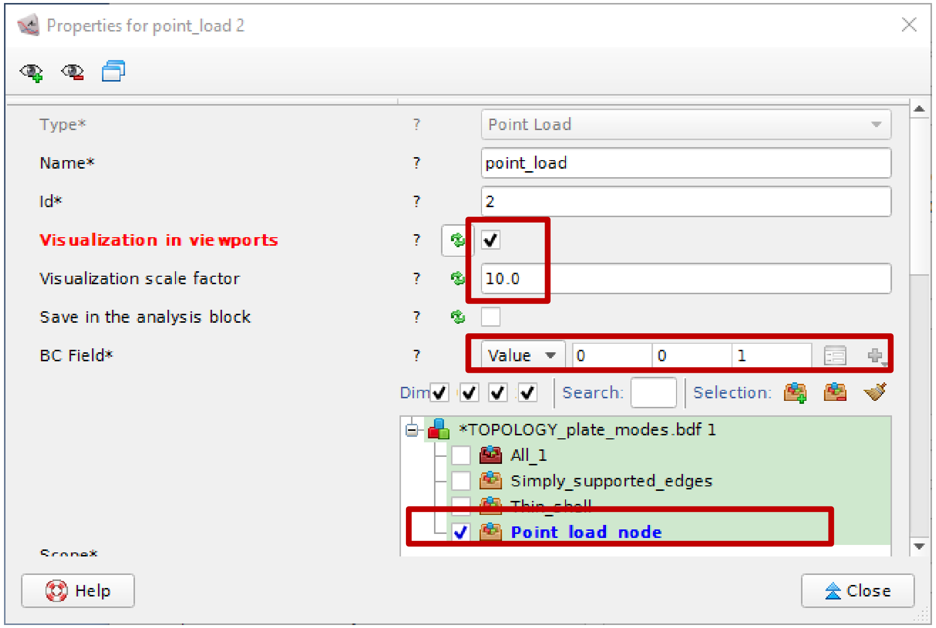


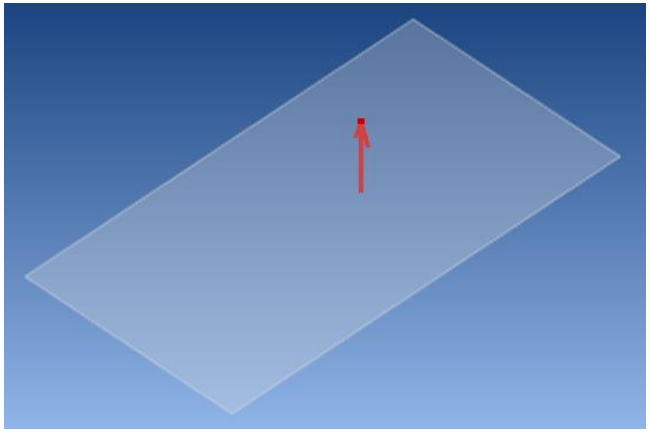


Create the Point Load Boundary condition



- Add a Point Load Boundary condition
- Set the following properties
  - Name: point_load_excitation
  - BC Field: [0,0,1]
  - Domain: Point_load_node
  - View Boundary Condition
  - Scale Factor: 10



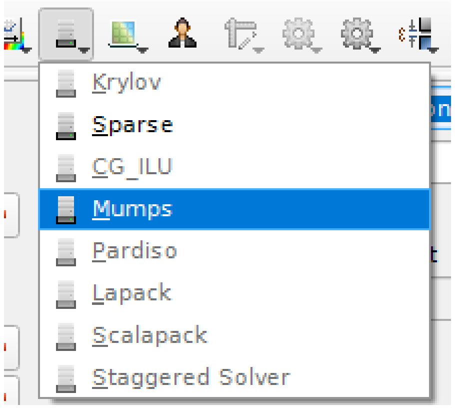


Remark: For each frequency, a time harmonic excitation of amplitude 1 in the Z direction is defined

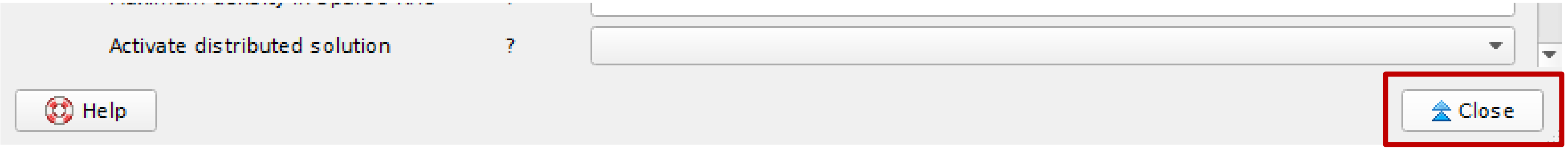
- Close the boundary condition properties window

Specify the Solver

- Define the solver of the analysis



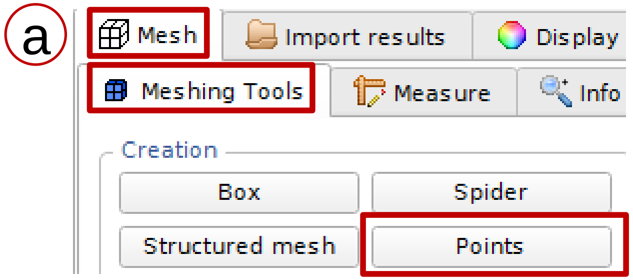
- Set the MUMPS solver
- Close the pop-up window of MUMPS



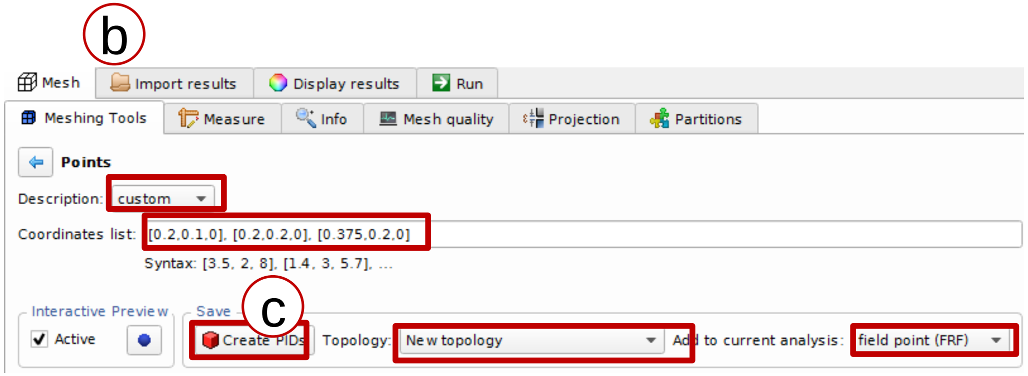
- Close the properties window of the Direct Frequency Response

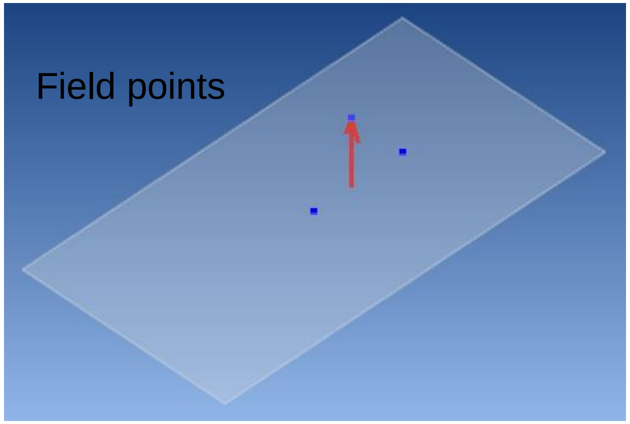
Set the Post-processing Parameters

1 – Create field points FRF output



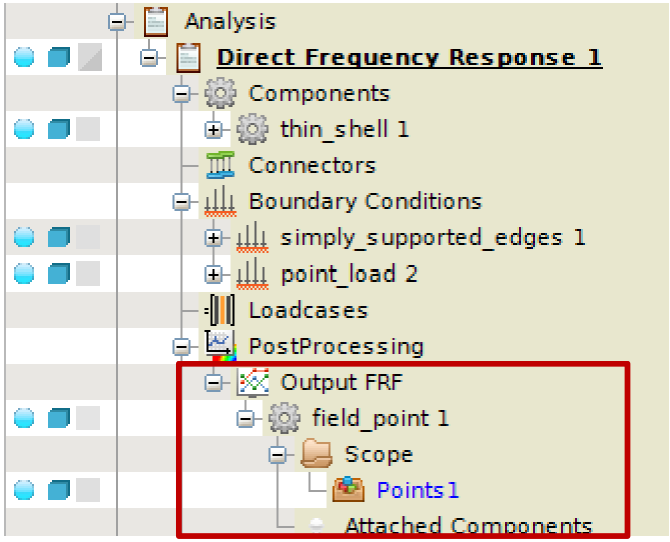
- Create Field Points. Field Points are points where the fluid pressure is output. These field points can be seen as virtual microphones:
  - a) Go to *Mesh → Meshing Tools → Points*
  - b) Select the "custom" description, and set the coordinates of the field points: [0.2,0.1,0], [0.2,0.2,0], [0.375,0.2,0]
  - c) Select "New topology" to create the points in a new topology, and add "field point (FRF)" to current analysis to save them as outputs
- Close the properties window

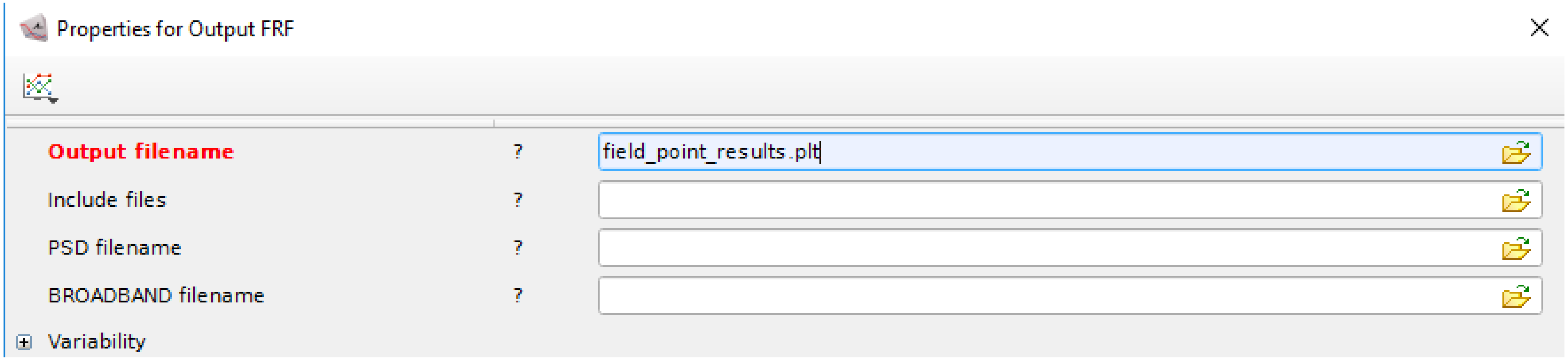




Set the Post-processing Parameters
2 – Set the output filename

- An *Output FRF* was created in the data tree panel, containing the *Field Points*
- Right click on *Output FRF* in the data tree panel and go to *Properties for Output FRF…*
- Set the Output filename to *field_point_results.plt*
  - This PLT file will be created at the end of the computation

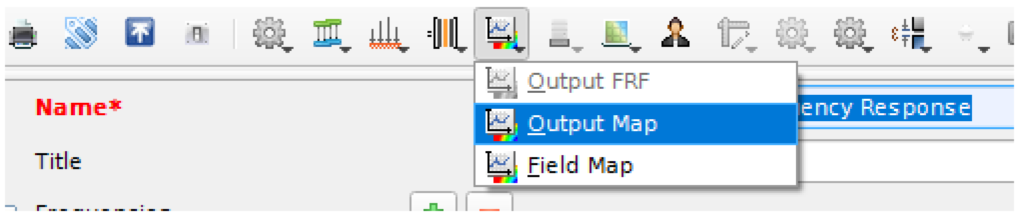


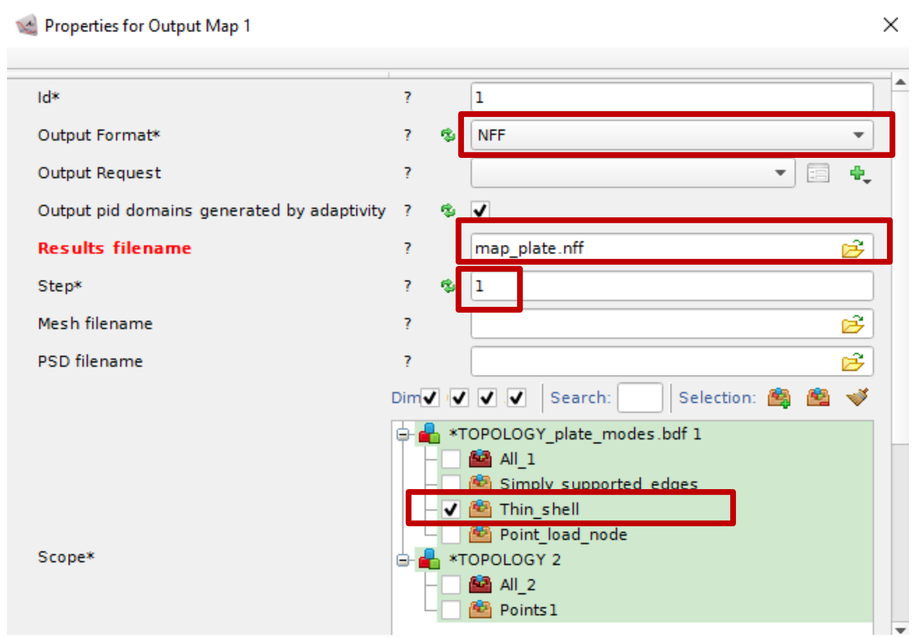


Set the Post-processing Parameters

3 – Create an Output Map

- Add an Output Map post-processing parameter

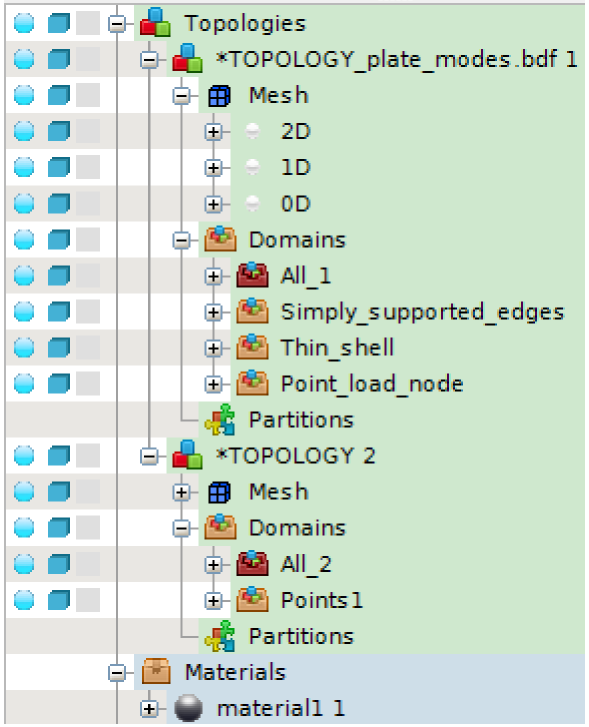


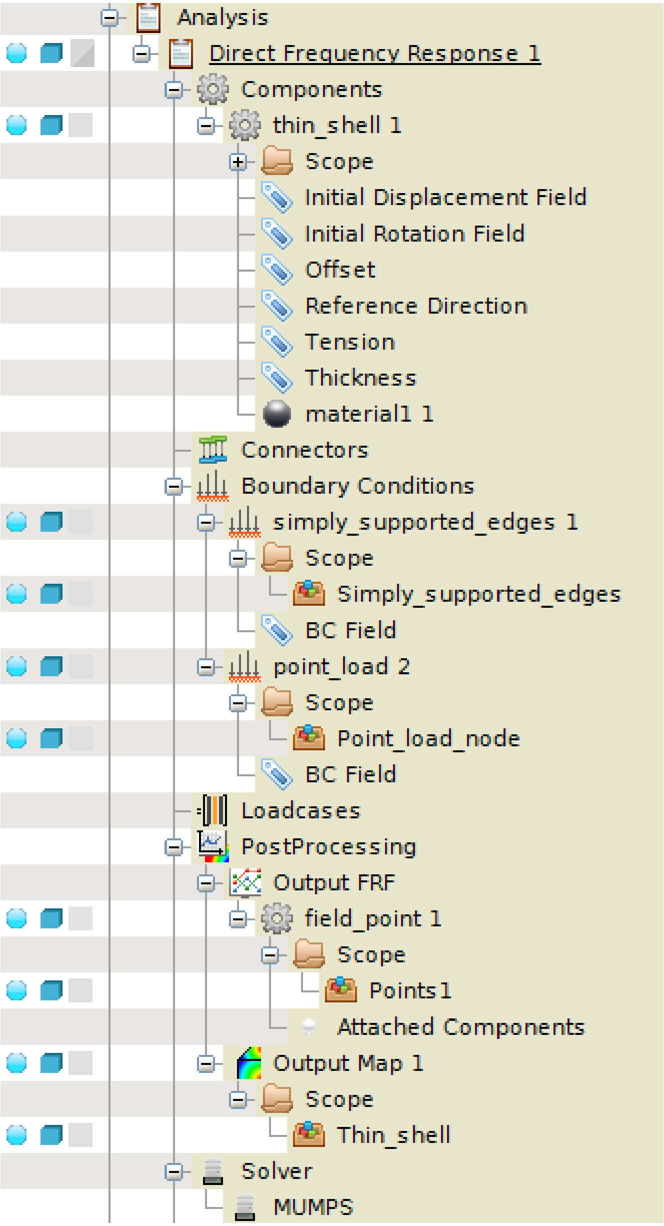


- Set output parameters:
  - Specify the output format NFF
  - Specify the filename map_plate.nff
  - Output the map for every frequency (step: 1)
  - Select the *Thin_shell* domain
- Close the Output Map properties window

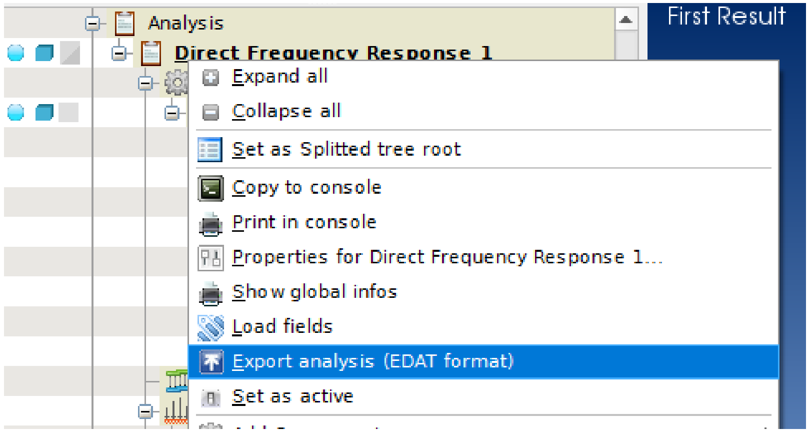
Check the Analysis

- The analysis is now completely defined
- All the parts of the analysis are available and editable on the data tree panel
- Check if the analysis tree is identical to the one shown



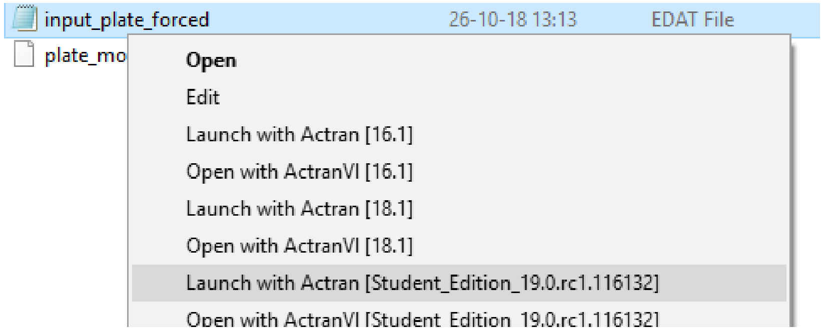


Export the Analysis File

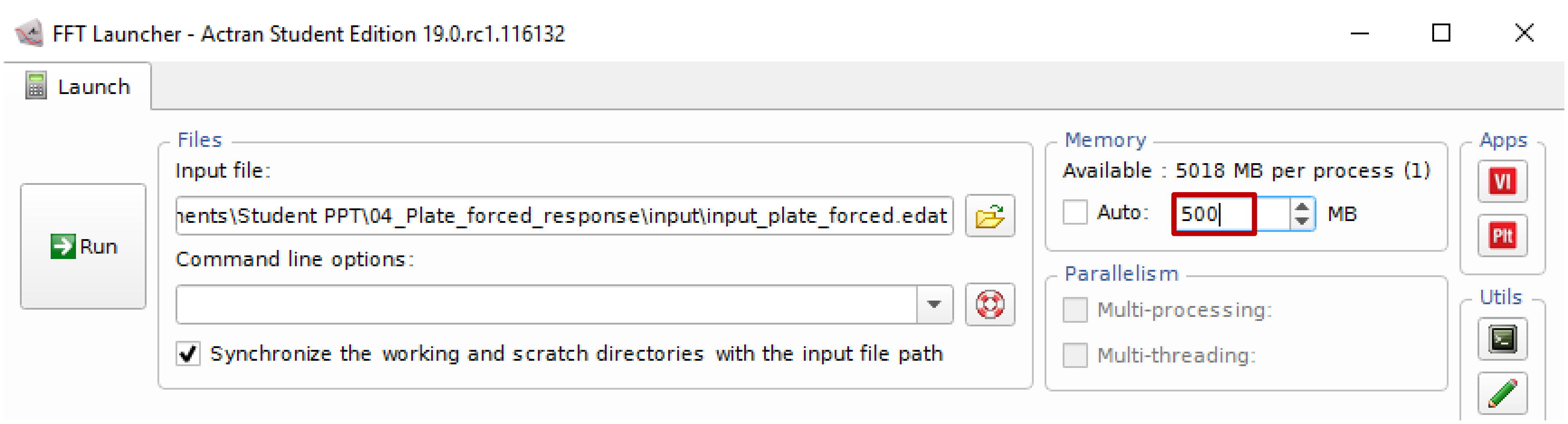


- The analysis can be exported in the EDAT Actran input file
- Right click on the *Direct Frequency Response*,
  and choose *Export analysis (EDAT format)*
- The mesh has been created in ActranVI. It will be written on the input file
- Export the analysis and name the file "input_plate_forced.edat"

Launch Actran Analysis

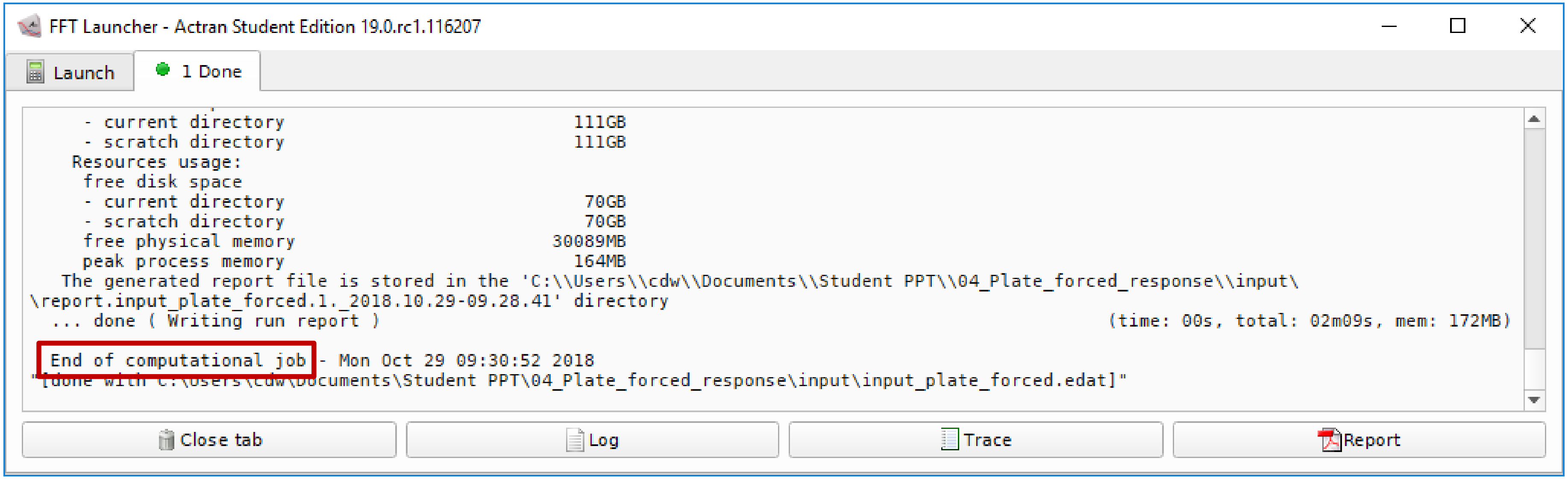


- Launch the computation:
  - Open the FFT Launcher by right clicking on the *input_plate_forced.edat* input file and selecting *Launch with ACTRAN [Student Edition]*
  - Specify the allocated memory (in MB): 500
  - Click on the green arrow to run the computation



- The computation log progresses as the model runs

- “End of computational job” indicates the computation has finished



- Close the Launcher window

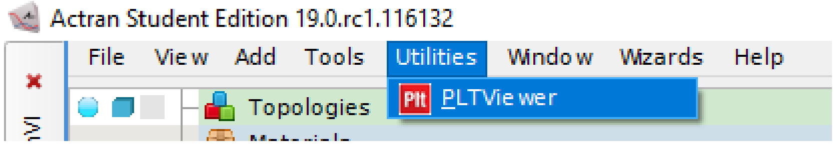
Post-processing

Plot plate displacement in PLTViewer
Visualize map of plate displacement in ActranVI

Plot the pressure with PLTViewer

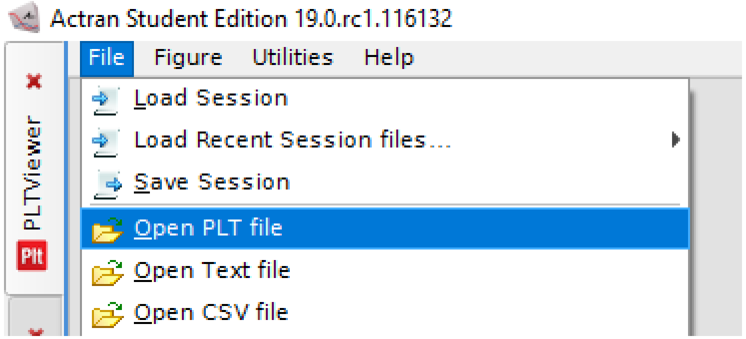
1 – Open PLTViewer and import field point results

- PLTViewer is the dedicated post-processing utility to visualize FRF's from Actran (stored in the PLT file) or from measurements



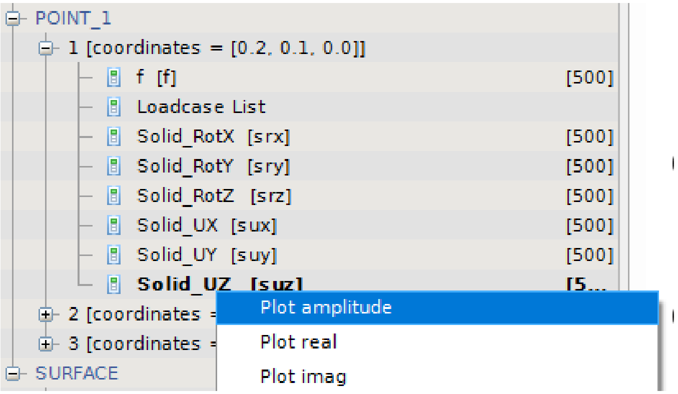
- Open the PLTViewer interface
  - PLTViewer can be launched within ActranVI from the Utilities menu

- Import the file *field_point_results.plt*
  - Under *File* select *Open PLT file*
  - Select the file *field_point_results.plt*

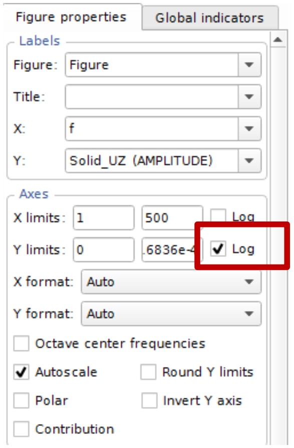


Evaluate the Structural Displacement

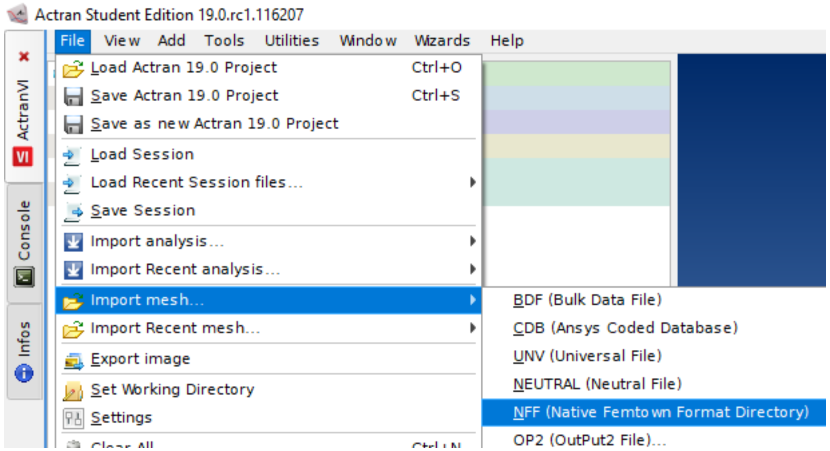
- The plate vibrates along the Z axis, the amplitude of structural displacement along the Z axis is plot <u>for each point</u> using the shortcut
  a) Unfold the point tree (“*1 [coordinates = [0.2, 0.1, 0.]]*” for point 1)
  b) Right Click on *Solid_UZ [suz]*
  c) Select *Plot Amplitude*



- Visualize the plot using a logarithm scale for displacement amplitude

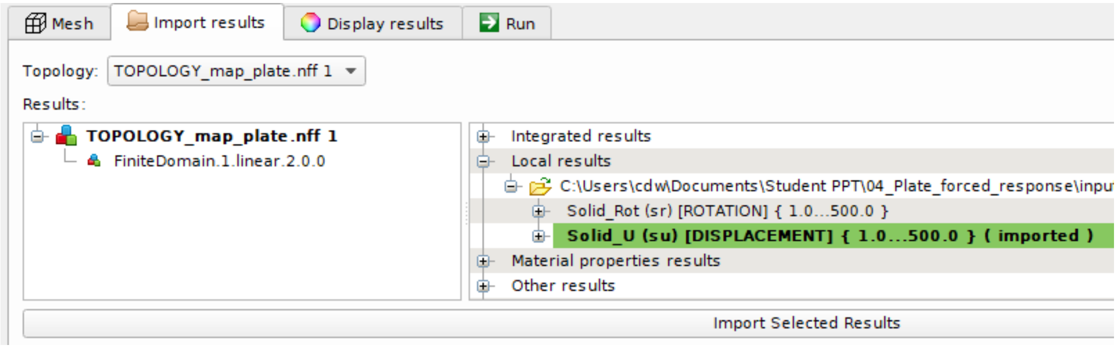


Visualize Map in ActranVI



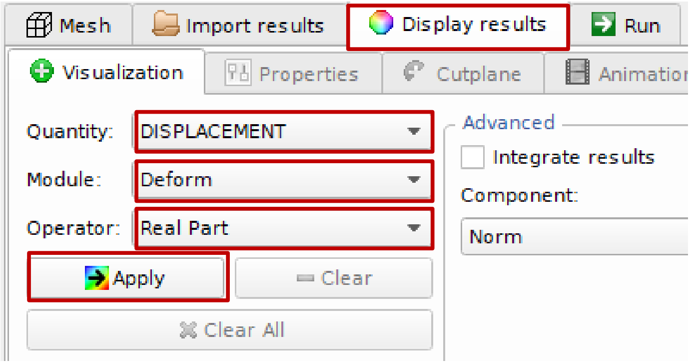
- Switch back to tab ActranVI:

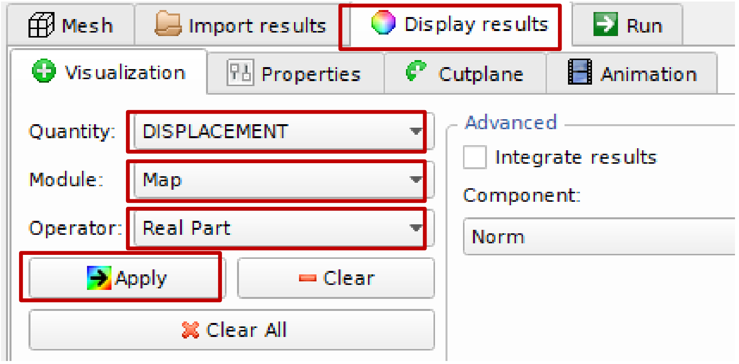
- Import the displacement results on the mesh:
  a) Import the NFF mesh created during the calculation
  b) Open tab: Import Results
  c) Select the NFF database topology
  d) Choose Solid_U (su) [DISPLACEMENT] for structure displacement
  e) Import Selected Results



Visualize Map in ActranVI

- From the *Display results* tab of the Toolbox, visualize the Displacement Deform and the Displacement Map





- Adjust the deformation scaling factor to better visualize modes shape