Extraction of Plate Modes

Actran Student Edition Tutorial

Introduction

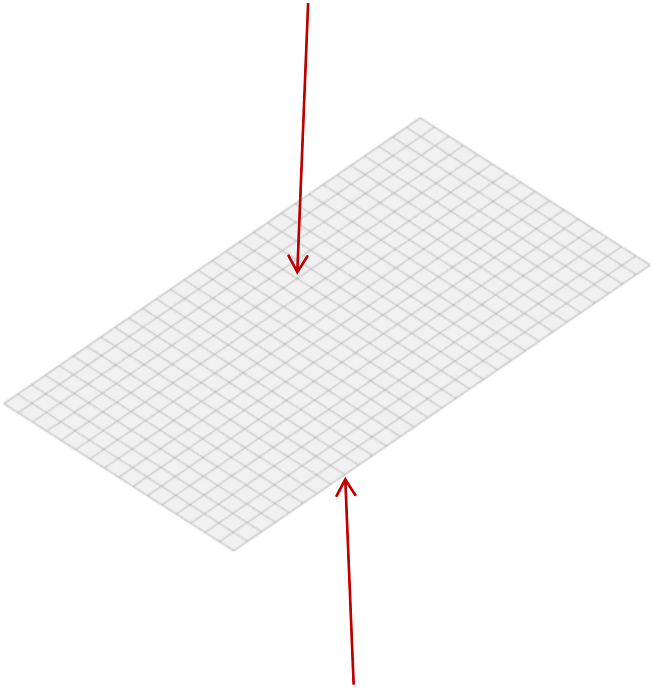
- This workshop introduces the modal extraction analysis for structures and shows an application case on a plate

- 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 19 Student Edition

Workshop Description

- A rectangular plate is studied
  - A thin shell component is defined. The plate is modeled by 2D elements and its thickness is defined as a property of the shell
- The plate is simply supported on its edges:
  - A displacement 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: $E = 7 \times 10^{10}$ Pa, v = 0.25, ρ = 2400 kg/m^3
  - The plate is simply supported along the four edges
- Kirchhoff–Love model for Isotropic thin plate gives

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

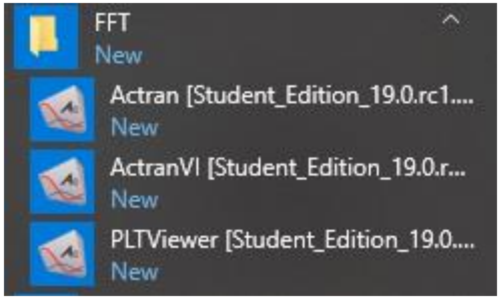
- Which leads to eigenmodes $\mathrm{w}_{\mathrm{mn}}(\mathrm{x},\mathrm{y}) = A_{mn} \sin\left(\frac{m\pi x}{L_x}\right) \sin\left(\frac{n\pi x}{L_y}\right)$
- And eigenfrequencies $\omega_{mn} = \sqrt{\frac{D}{\rho t}}\left(\left(\frac{m\pi}{L_x}\right)^2 + \left(\frac{n\pi}{L_y}\right)^2\right)$
- Where $D = \frac{Et^3}{12(1-\upsilon^2)}$ is the bending stiffness

Workshop Pre-Processing

Modal Extraction Analysis

Start ActranVI

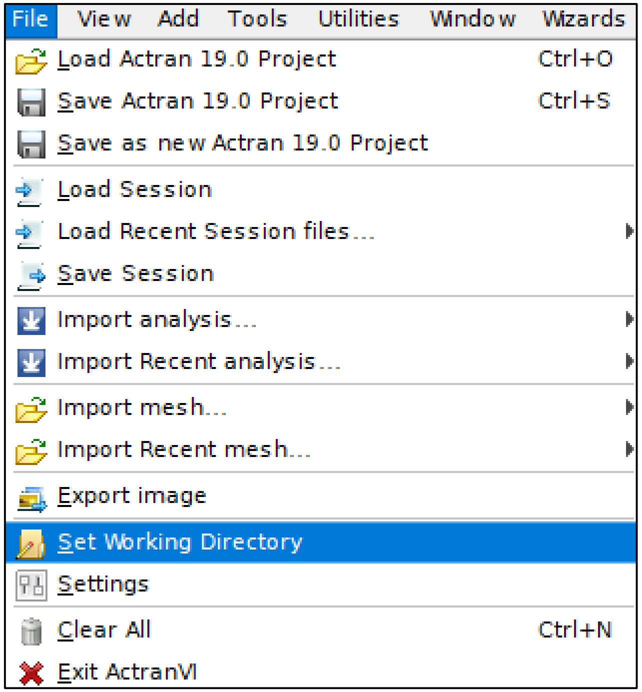
- Start ActranVI:
  - shortcut is available through the Windows Start Menu

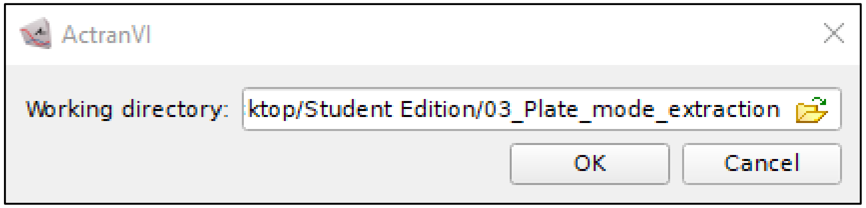


(Windows Start Menu)

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



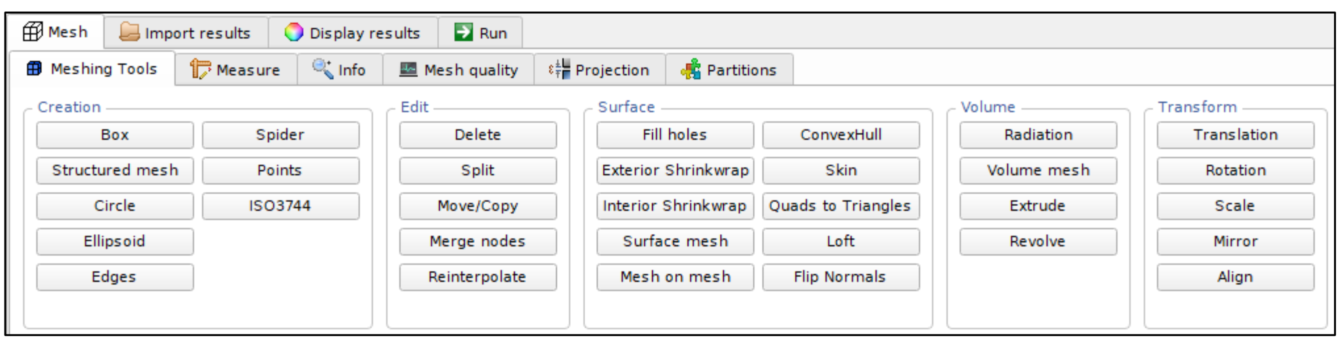


* Important: The working directory path should not contain any space or special character

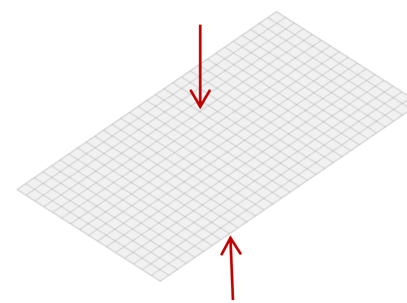
Create the mesh

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
- 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 → Meshing tools



Thin shell component
2D element set

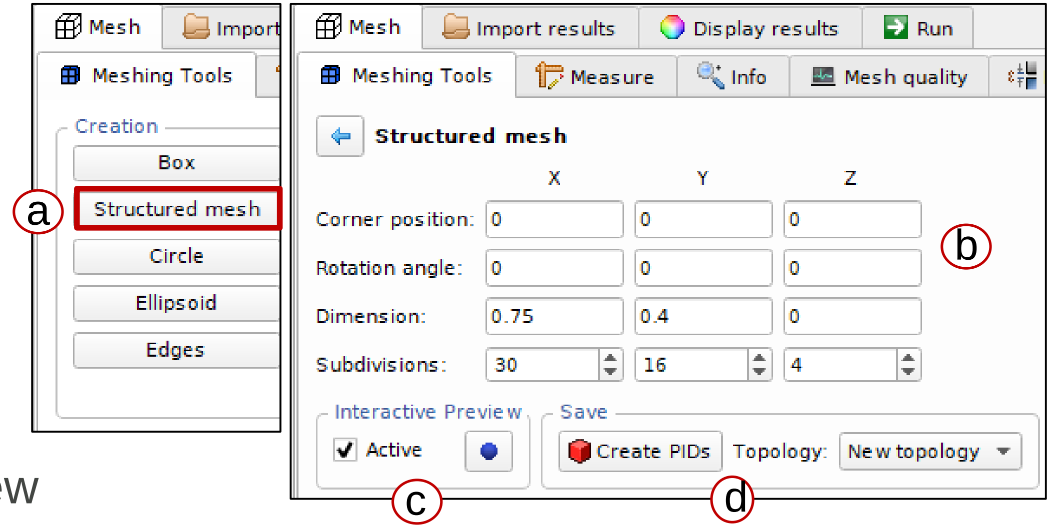


Zero displacement BC
1D element set

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”

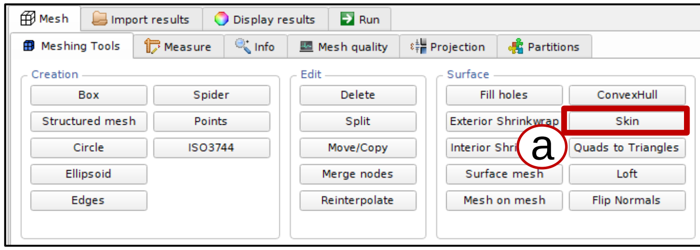


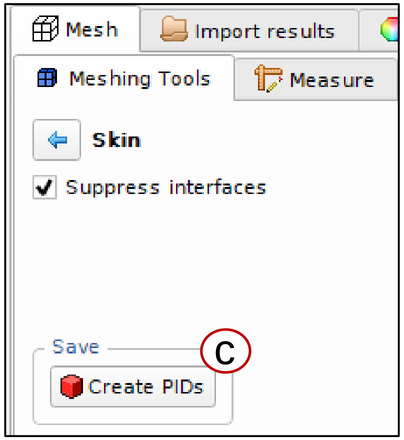
- The created mesh contains linear elements

Create the mesh

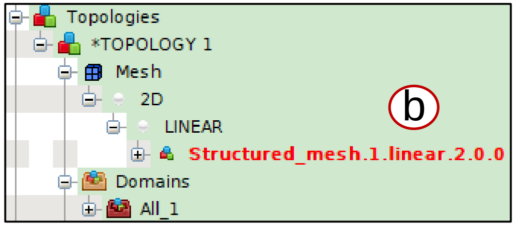
3 – Create the 1D element set – Skin

- 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



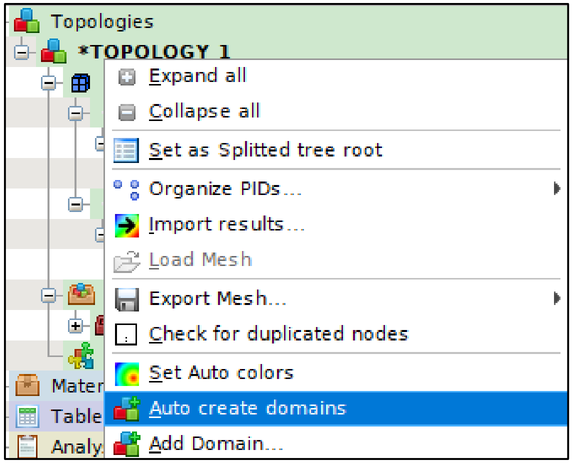


- The mesh needed to run the analysis was created
- It will be used to setup the analysis and run the calculation

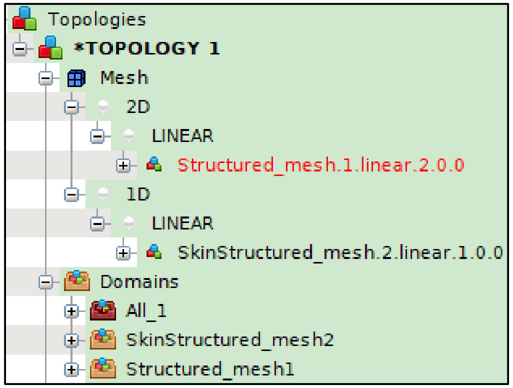


Create the Domains

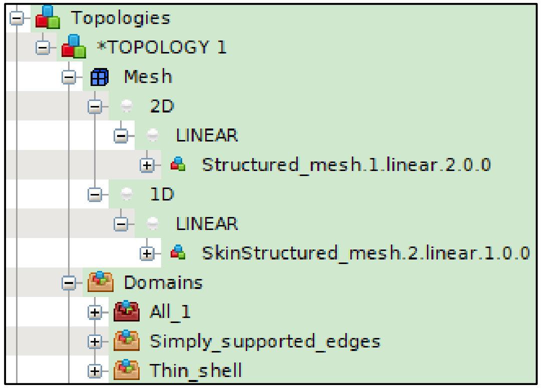
- Auto create the domains of the created mesh (right click):
  → *TOPOLOGY* → *Auto create domains*



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

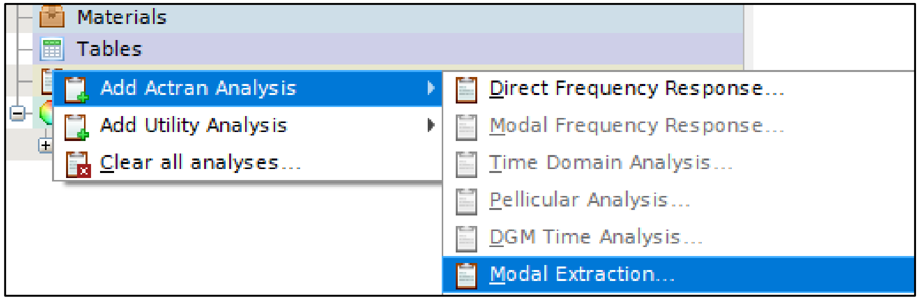


DEFAULT NAME	NEW NAME
SkinStructured_mesh2	Simply_supported_edges
Structured_msh1	Thin_shell

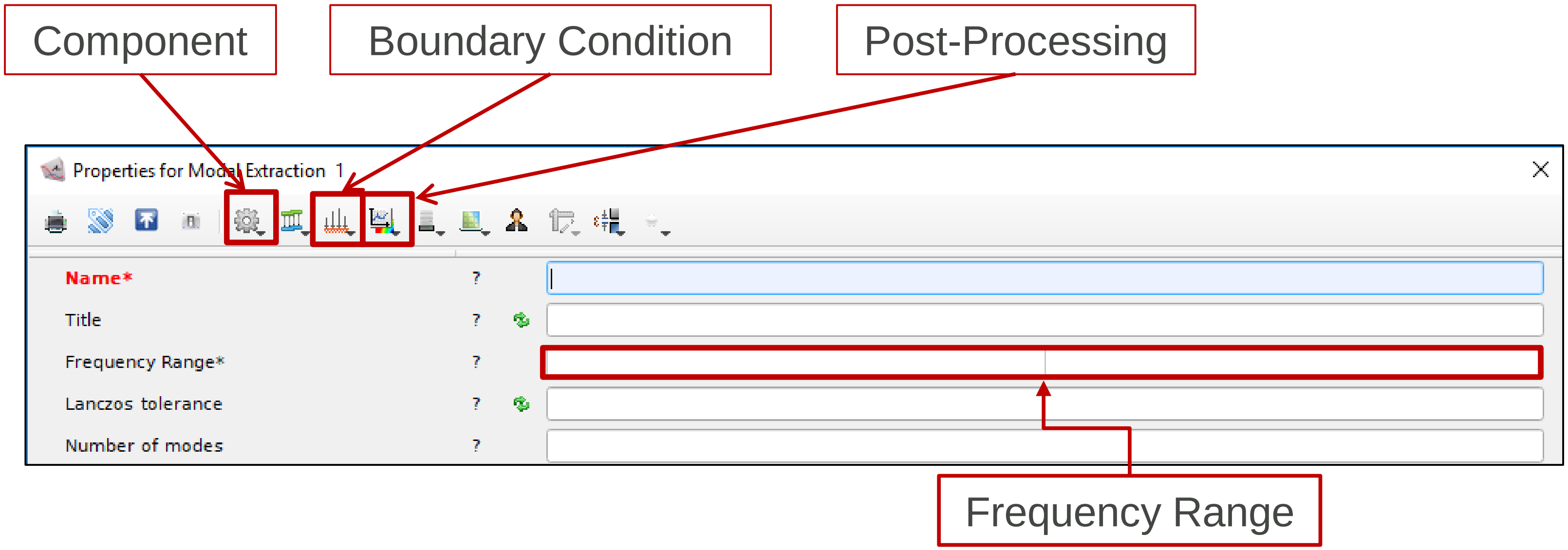


Create the Modal Extraction Analysis

- Create a Modal Extraction 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

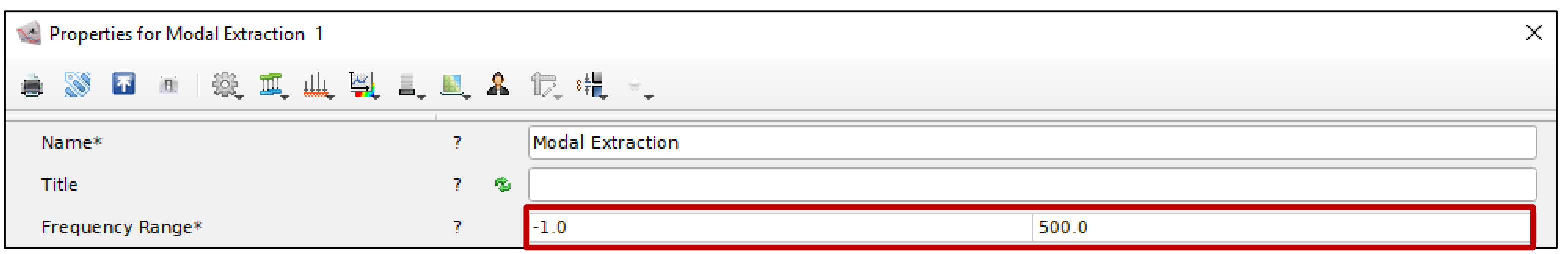


Specify the Frequency Range

- 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

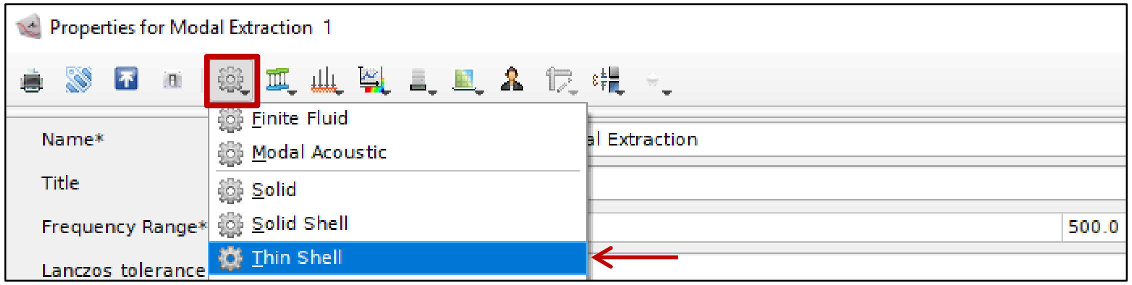


* Remark : The modal extraction starts at -1Hz to make sure that rigid body modes (numerically close to 0Hz) are calculated

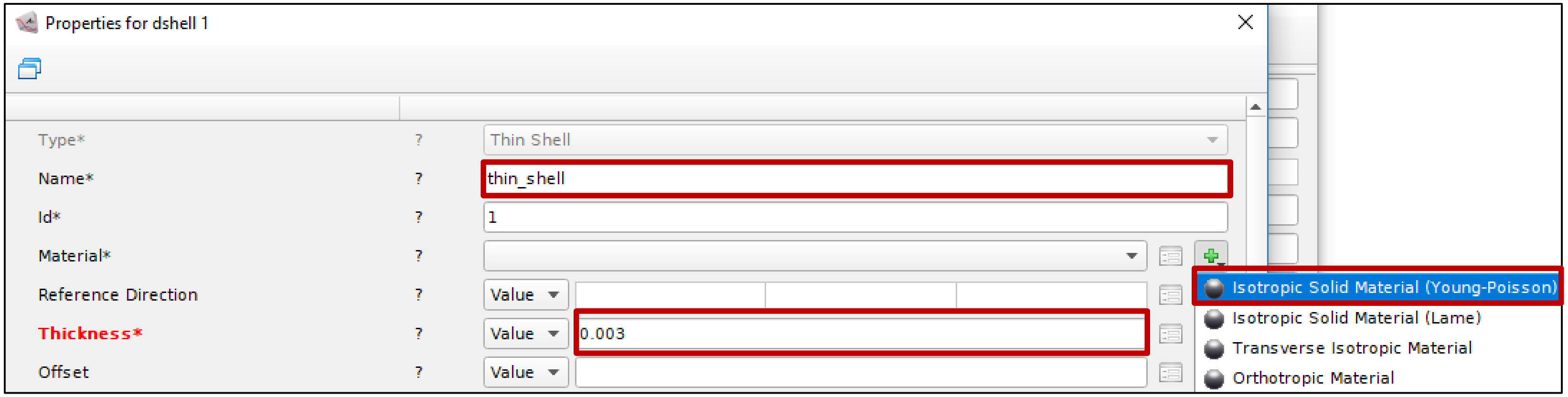
Create the Thin shell Component

1 – Add a Component

- Add a Thin shell component



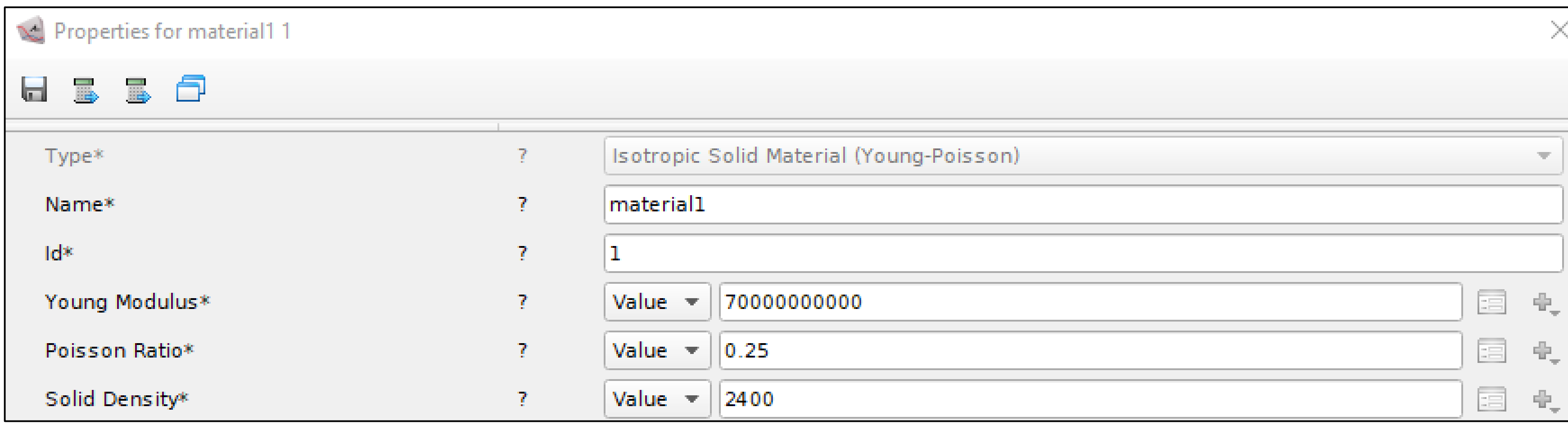
- Component properties:
  – Specify the name of the Thin Shell component: thin_shell
  – Specify the thickness value: 0.003 m
  – Create a new Isotropic Solid Material (Defined by its Young modulus and Poisson ratio)



Create the Thin shell Component

2 – Set up the Isotropic solid Material

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

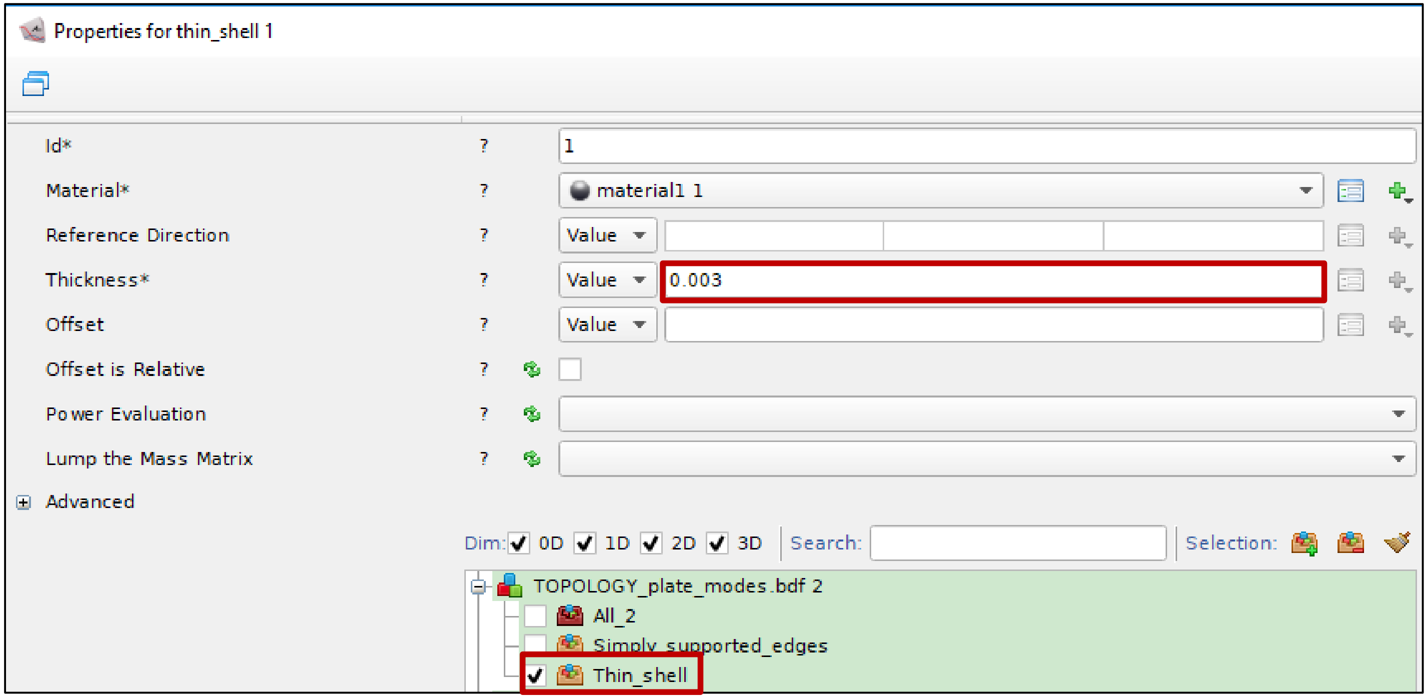


- Close the material properties window

Create the Thin shell Component

3 – Assign the Domain

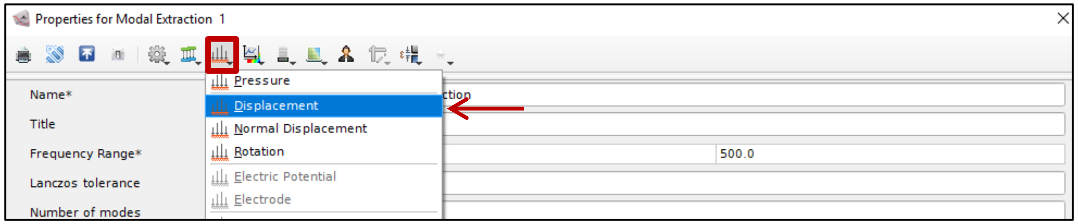
- With the Scope selector, assign the *Thin_shell* domain to the *Thin Shell* component

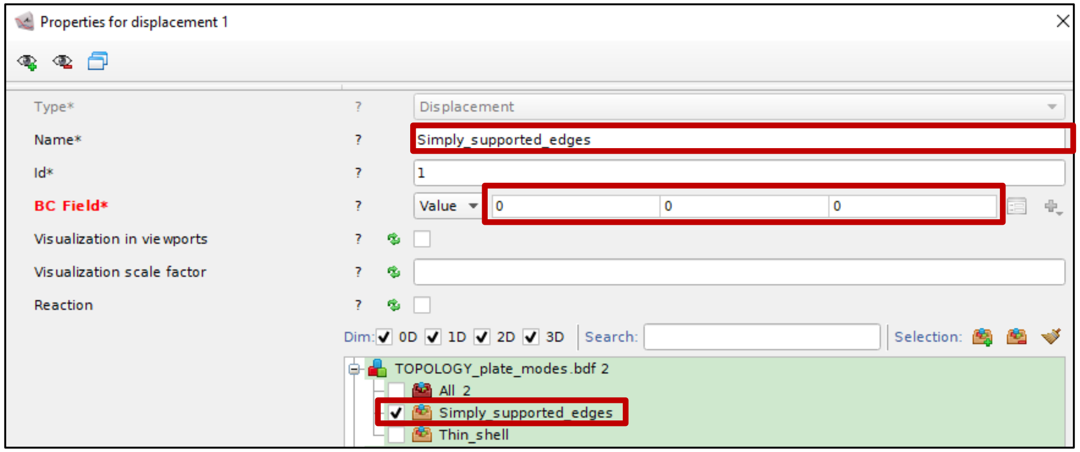


- Close the component properties window

Create the Simply Supported Boundary Condition

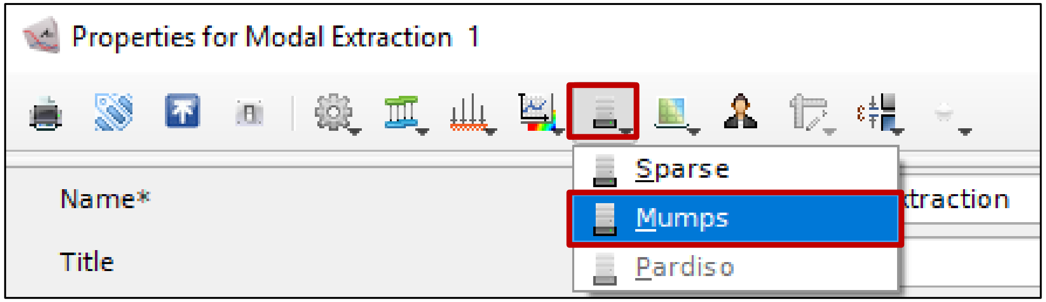
- Add a Displacement Boundary condition
- Set the following properties
  - Name: Simply_supported_edges
  - BC Field: [0,0,0] (X and Y displacements are constrained as well to avoid rigid body modes)
  - Domain: Simply_supported_edges
- Close the boundary condition properties window





Specify the Solver

- Define the solver of the analysis

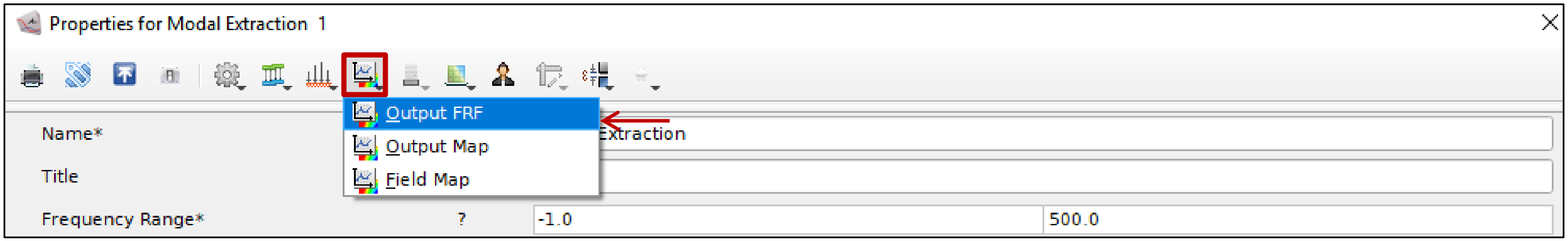


- Set the MUMPS solver
- Close the pop-up window of MUMPS

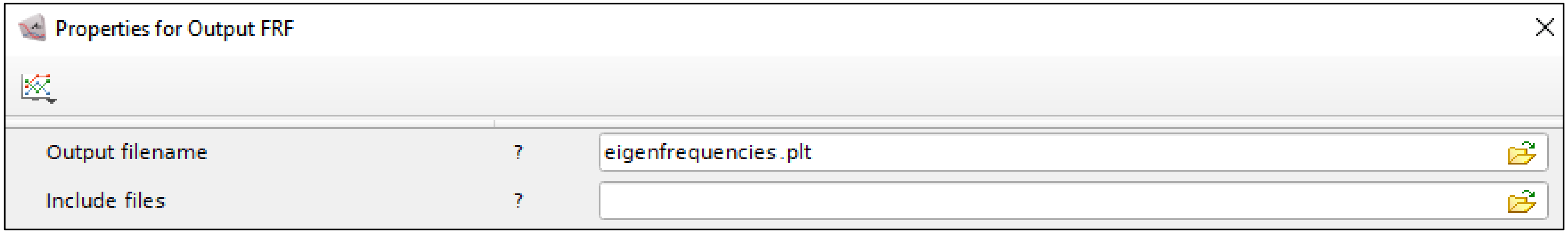
- Close the properties window of the Modal Extraction

Set the Post-processing Parameters

- Add an Output FRF post-processing parameter



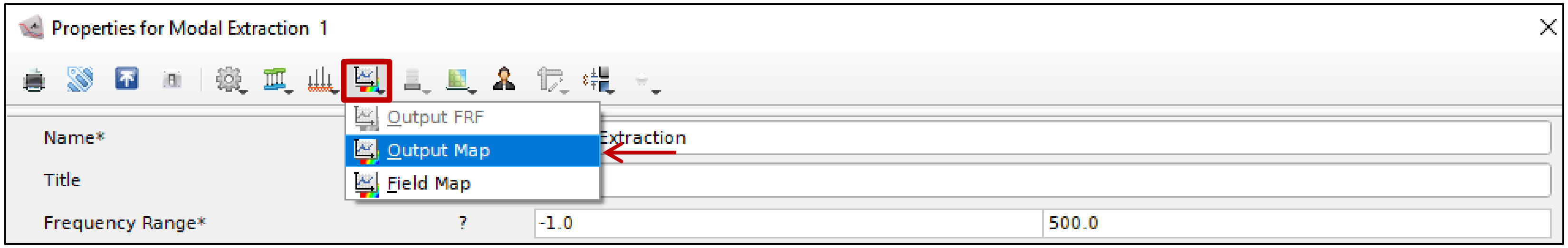
- Set the name of the FRF output file: eigenfrequencies.plt
- This FRF file contains a list of all the eigenfrequencies to be found



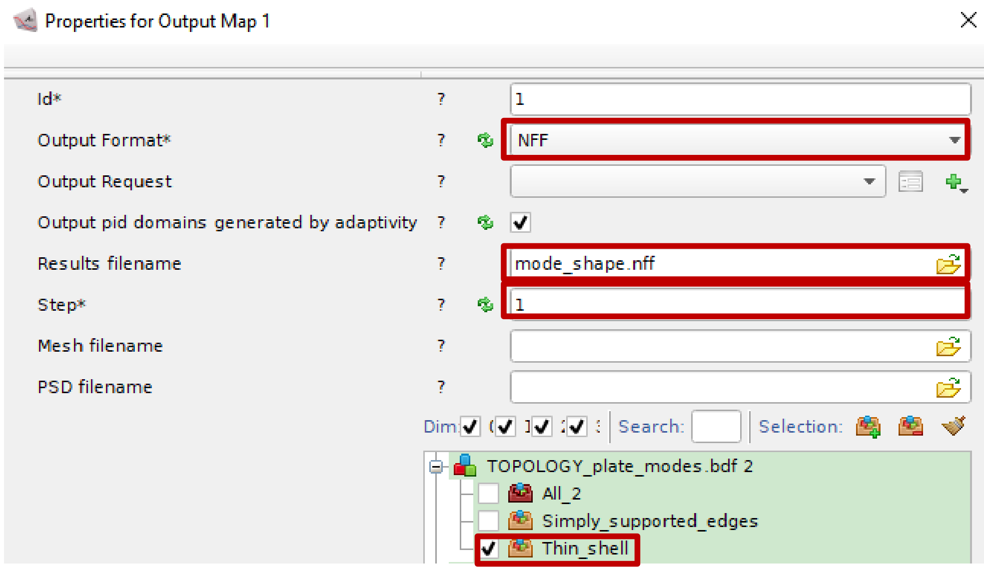
- Close the Output FRF properties window

Set the Post-processing Parameters

- Add an Output Map post-processing parameter

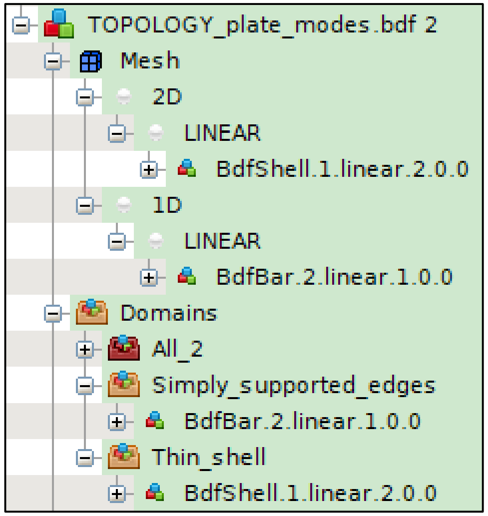


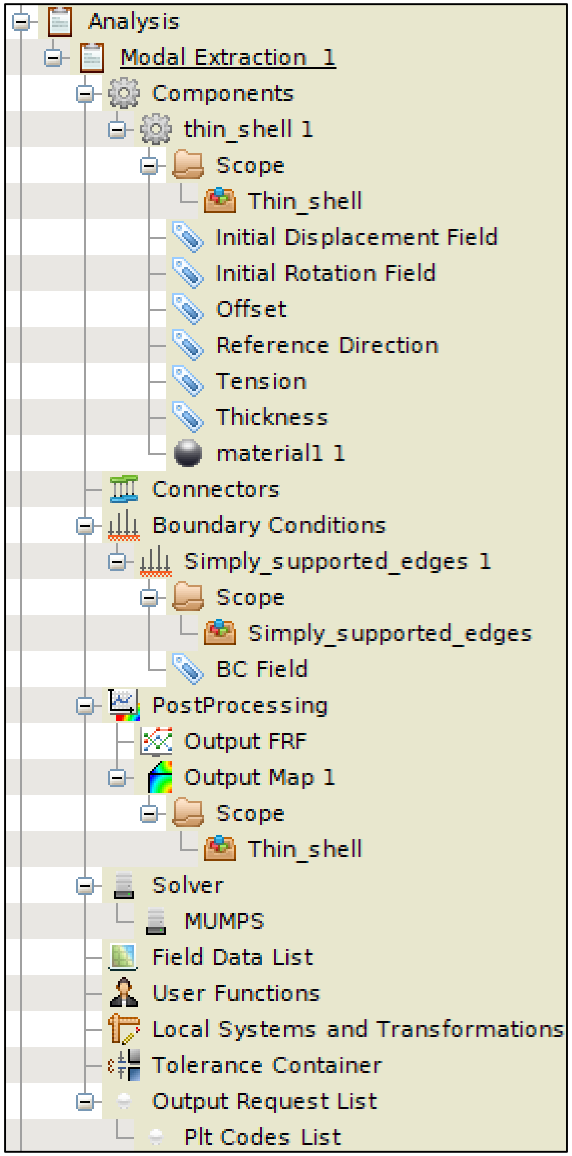
- Set output parameters:
  - Specify the output format NFF
  - Specify the filename mode_shape.nff
  - Output the map for every mode (step: 1)
  - Select the Thin_shell domain
- Close the Output Map properties window



Check the Analysis

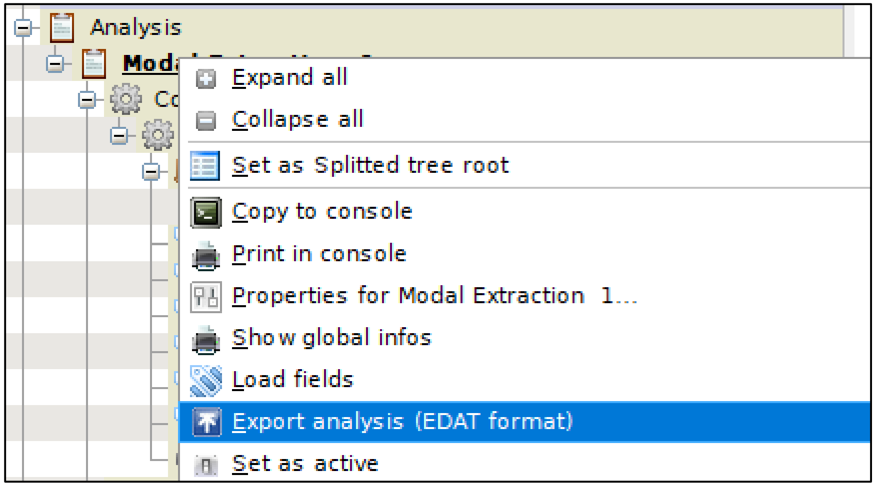
- The analysis is now completely defined
- All the parts of the analysis are available and editable on the data tree panel
- Check if the analysis tree is identical to the one shown





Export the Analysis File

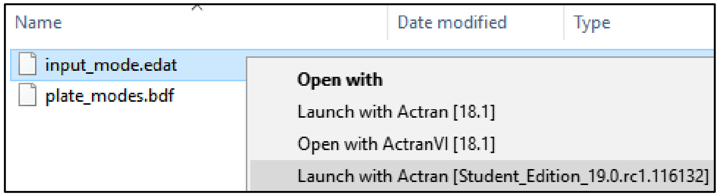
- The analysis can be exported in the EDAT Actran input file
- Right click on the *Modal Extraction*, and choose *Export analysis (EDAT format)*

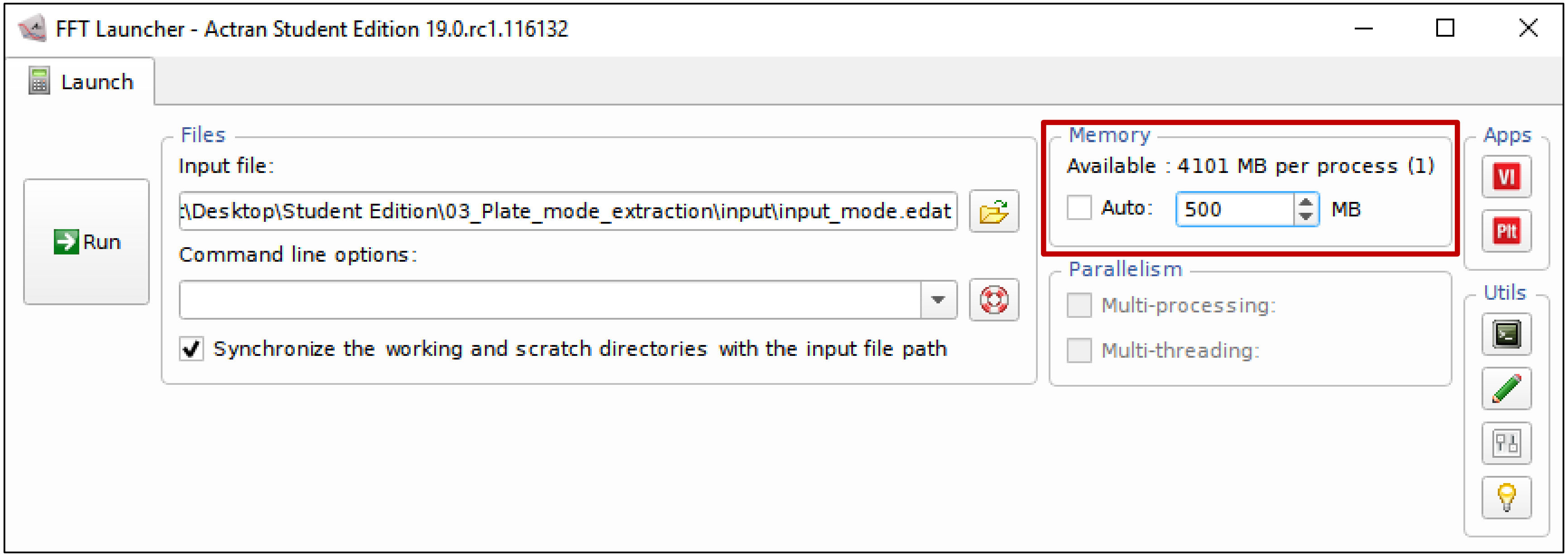


- The mesh has been created in ActranVI. It will be written on the input file
- Export the analysis and name the file “input_mode.edat”

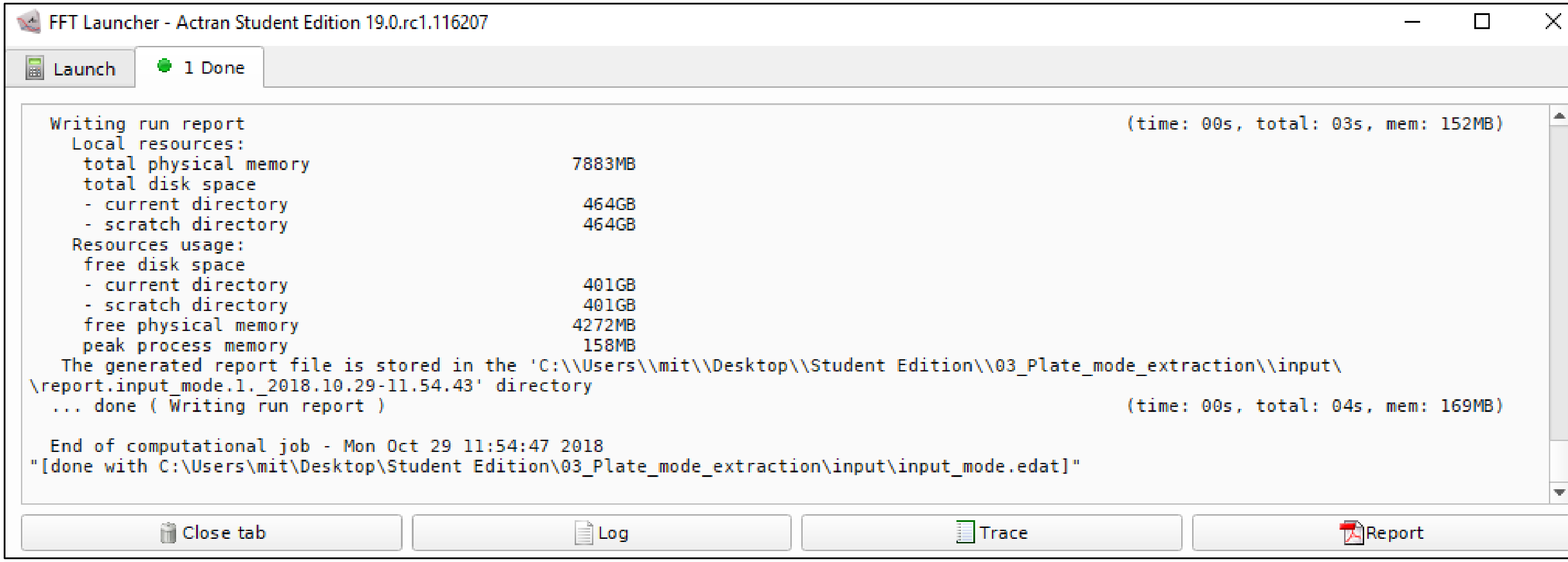
Launch Actran Analysis

- Launch the computation:
  - Open the FFT Launcher by right clicking on the *input_mode.edat* input file and selecting *Launch with ACTRAN [Student Edition]*
  - In Command line options, specify the allocated memory (in MB): -m 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

Examine eigenfrequencies

Visualize Mode shapes in ActranVI

Calculated vs Analytical Eigenfrequencies

- 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 = 7×10^{10} Pa, v = 0.25, ρ = 2400 kg/m^3
  - Plate simply supported along the four edges
- Kirchhoff–Love model for Isotropic thin plate gives

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

- Which leads to eigenmodes $$\mathrm{w_{mn}(x,y)} = A_{mn} \sin\left(\frac{m\pi x}{L_x}\right) \sin\left(\frac{n\pi x}{L_y}\right)$$
- 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 $D = \dfrac{Et^3}{12(1-\upsilon^2)}$ is the bending stiffness

Calculated vs Analytical Eigenfrequencies

- Eigenfrequencies calculated by Actran are stored in the eigenfrequencies.plt file
- This file can be opened using a text editor
- Eigenfrequencies are stored in the first column

```
BEGIN LOADCASE_INDEX
  1   0
END LOADCASE_INDEX
BEGIN OUTPUT_FRF
BEGIN TITLE
Actran Analysis
END TITLE
BEGIN   DOMAIN  Dshell1 "dshell"
 /*     NFreq    NLdCase   NRes                                Code of mass
          11         1         8                                    100
 /*  Freq                  LoadCase                                 Mass
  6.0820870493459e+01          1          {  2.160000000000e+00,  0.000000000000e+00}
  1.0103110745468e+02          1          {  2.160000000000e+00,  0.000000000000e+00}
  1.6893101772489e+02          1          {  2.160000000000e+00,  0.000000000000e+00}
  2.0543138464199e+02          1          {  2.160000000000e+00,  0.000000000000e+00}
  2.4489249472114e+02          1          {  2.160000000000e+00,  0.000000000000e+00}
  2.6530880340537e+02          1          {  2.160000000000e+00,  0.000000000000e+00}
  3.1156051535900e+02          1          {  2.160000000000e+00,  0.000000000000e+00}
  3.9129452066765e+02          1          {  2.160000000000e+00,  0.000000000000e+00}
  4.0642764859024e+02          1          {  2.160000000000e+00,  0.000000000000e+00}
  4.5337254057610e+02          1          {  2.160000000000e+00,  0.000000000000e+00}
  4.9199075014286e+02          1          {  2.160000000000e+00,  0.000000000000e+00}
END DOMAIN  Dshell1
END OUTPUT_FRF
```

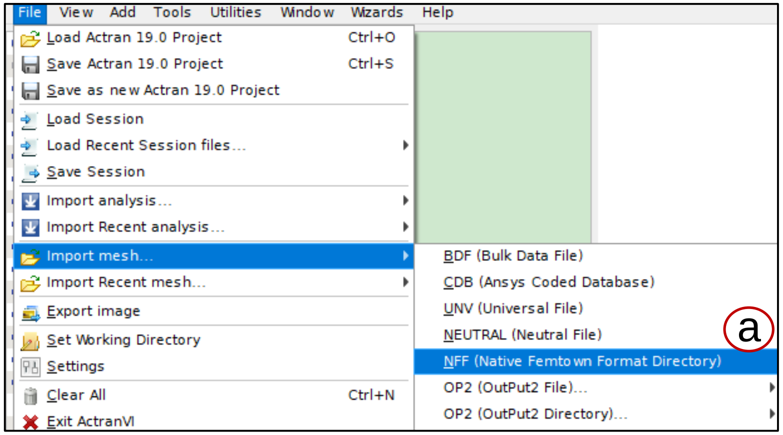
Mode	(m x n)	Analytical	Actran	Error
1	1 x 1	60.912	60.821	0.1%
2	2 x 1	101.380	101.031	0.3%
3	3 x 1	168.826	168.931	0.05 %
4	1 x 2	203.181	205.431	1.1%
5	2 x 2	243.649	244.892	0.5%
6	4 x 1	263.250	265.309	0.8%
7	3 x 2	311.094	311.560	0.1%
8	5 x 1	384.653	391.294	1.7%
9	4 x 2	405.519	406.428	0.2%
10	1 x 3	440.295	453.372	3%

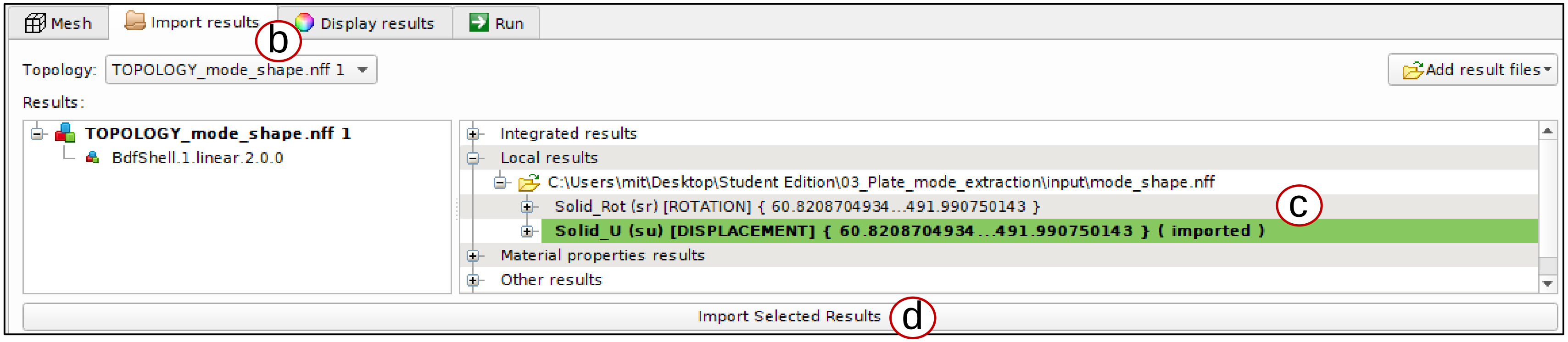
Relative error calculation: $\varepsilon = \frac{\left| f_{analytic} - f_{actran} \right|}{f_{analytic}} * 100$

- Eigenfrequencies found by Actran calculation are close to Analytical values

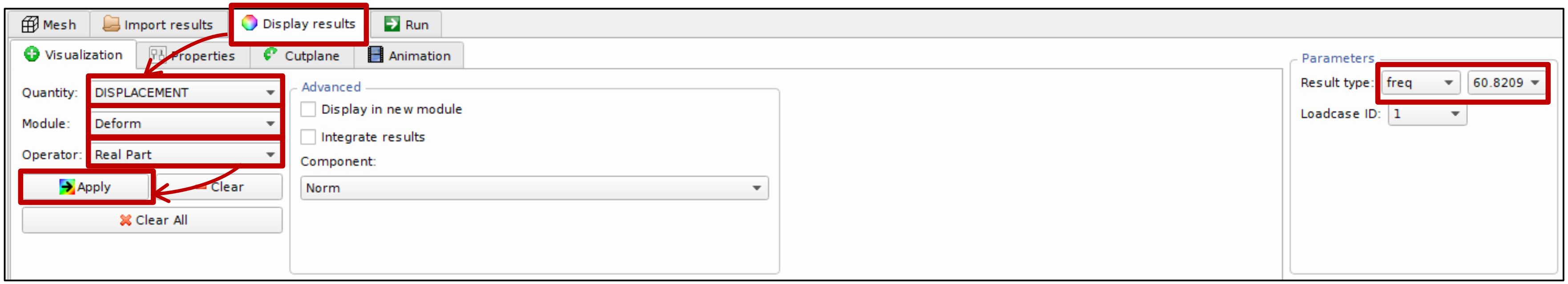
Visualize Field Map in ActranVI

- In ActranVI, import the displacement results on the mesh:
  a) Import the NFF mesh created during the calculation
  b) Open tab: Import Results
  c) Choose Solid_U (su) [DISPLACEMENT] for structure displacement
  d) Import Selection

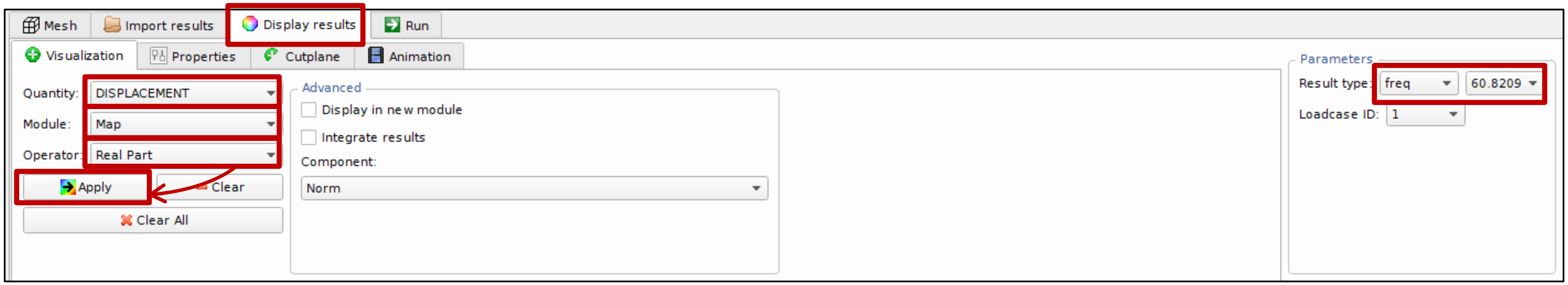




- From the *Display results* tab of the Toolbox, visualize the Displacement Deform

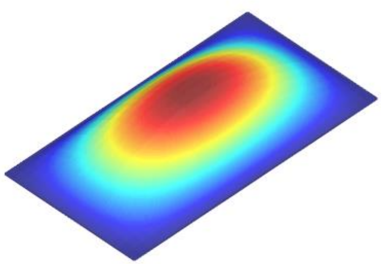


- In order to better observe the plate modes, display the Displacement Map over the Displacement Deform

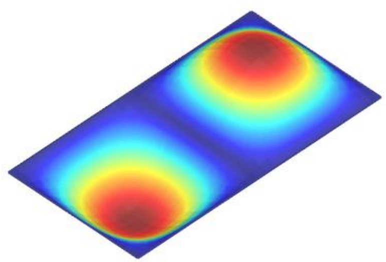


- Adjust the deformation scaling factor to better visualize modes shape

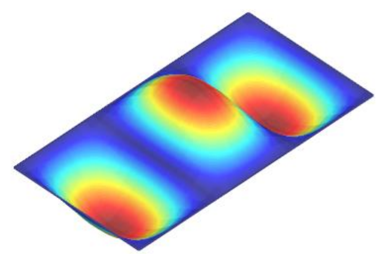
- The higher the frequency, the more complex the mode shape is



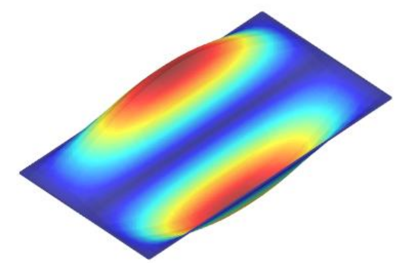
Mode 1 (1x1)
f = 60.821 Hz



Mode 2 (2x1)
f = 101.031 Hz



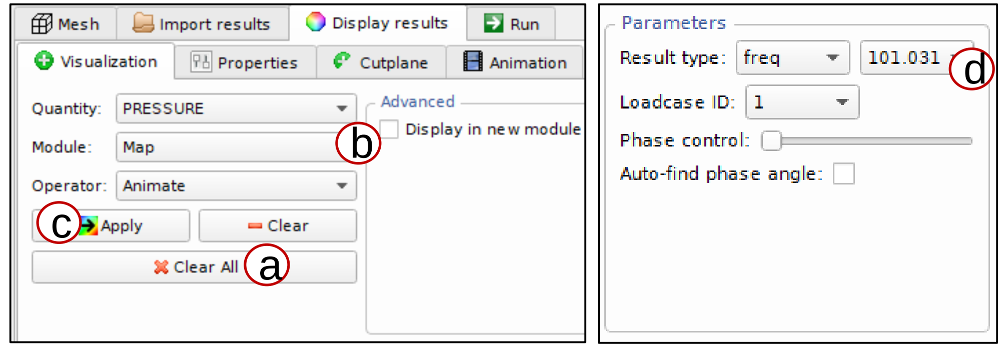
Mode 3 (3x1)
f = 168.830 Hz



Mode 4 (1x2)
f = 205.431 Hz

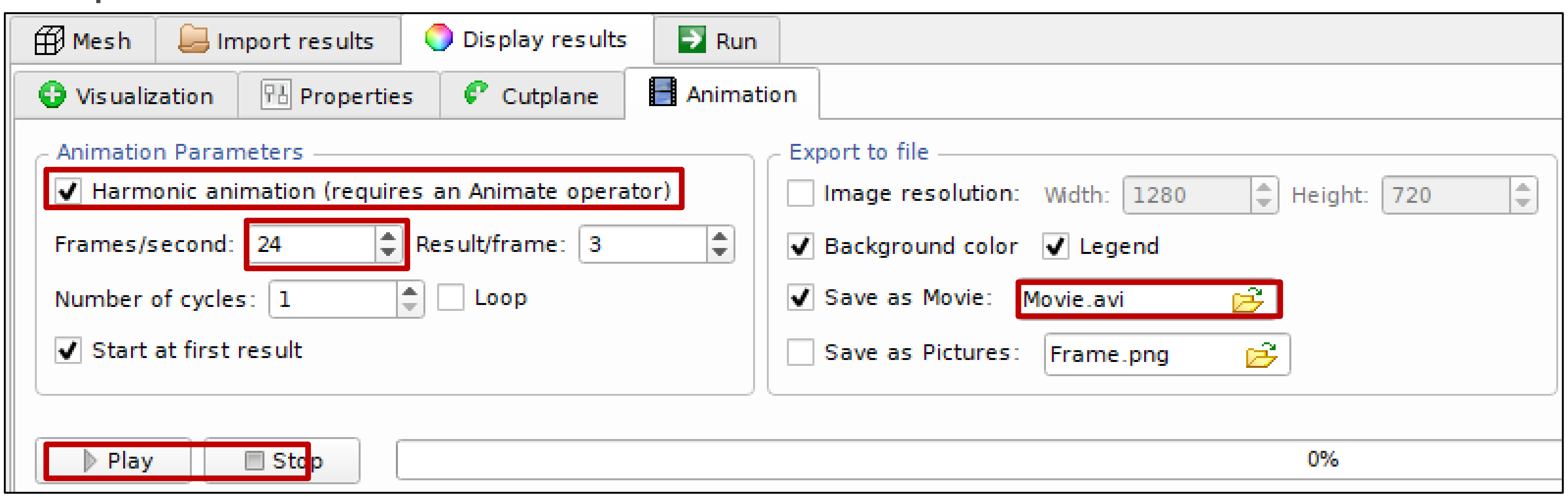
* Remark: the amplitude of the displacement does not have a physical meaning, the eigen vectors are normalized using the mass matrix $\Phi_i^T M \Phi_i = 1$

- To better visualize mode shapes, animations can be done in ActranVI
  a) Click on Clear all
  b) Select Operator: Animate
  c) Click on Apply
  d) Select Frequency 101,031Hz



- From the Display Results → Properties tab, set a custom range for the visualization: from -0.1 to 0.1

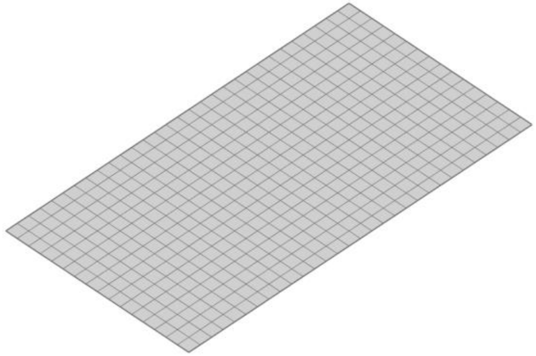
- Visualize an animation of the acoustic pressure real part



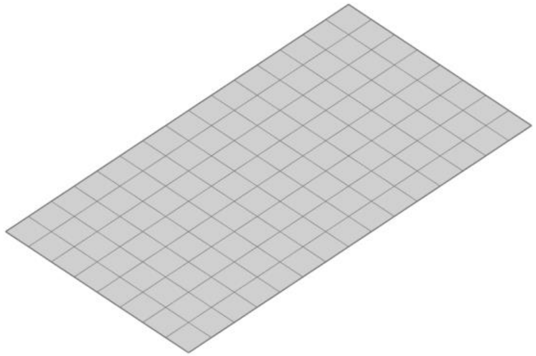
- Animations can be exported as AVI files using the Movie settings panel

Mesh Quality

- 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 input file input_mode_coarse.edat contains the same analysis using a coarser mesh. Eigenfrequencies calculated with this mesh have been output in the file eigenfrequencies_coarse.plt



Refined mesh used in input_mode.edat
Elements size: 0.025

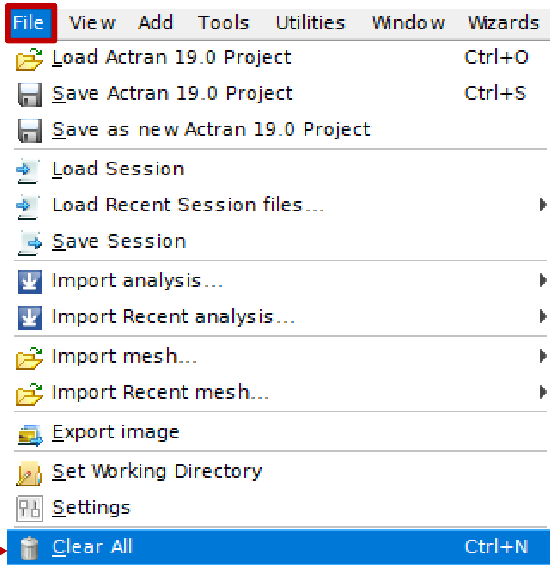


Coarse mesh used in input_mode_coarse.edat
Elements size: 0.05

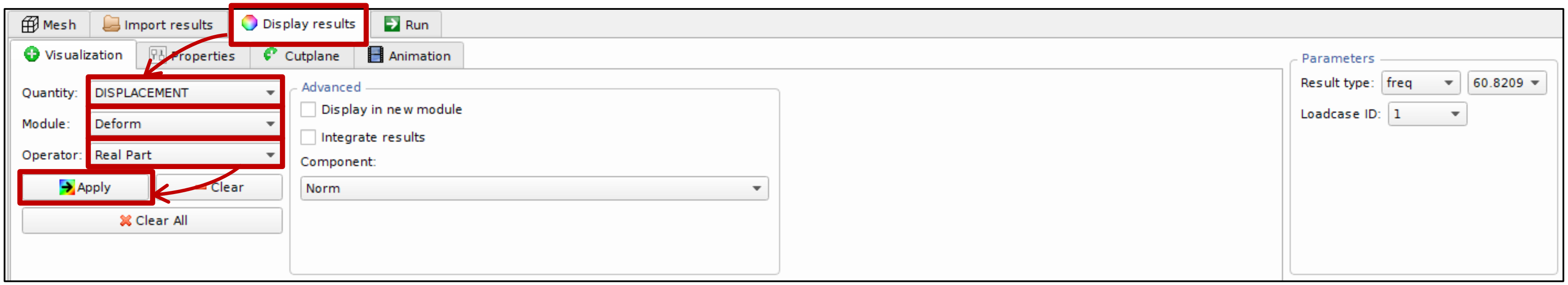
- Based on the 10 linear elements per bending wavelength criterion used to define the frequency range of the modal extraction, the maximum frequency for which the coarse mesh can be used is 120Hz

$$c_{bend} = \sqrt{\omega.t\sqrt{\frac{E}{12.\rho.(1-\nu^2)}}} \qquad \lambda_{bend} = {c_{bend}}/{f} = 0.503 \; for \; f = 120Hz$$

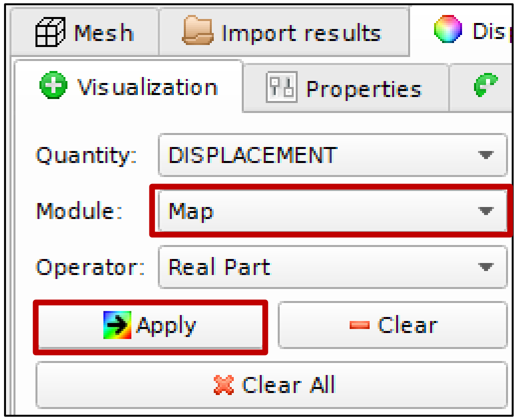
- Run the calculation input_mode_coarse.edat located in the “results” folder
- In ActranVI, reset the workspace by clicking on File → Clear All
- Import the results from the calculation done with the coarse mesh
  - Click File → Import mesh… → NFF
  - Select the folder mode_shape_coarse.nff



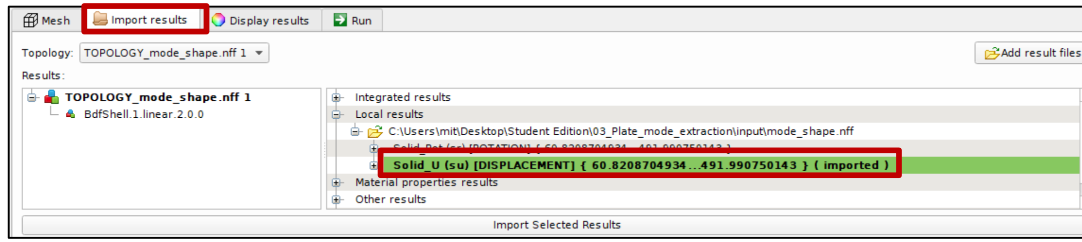
- From the Display results tab of the Toolbox, visualize the displacement map



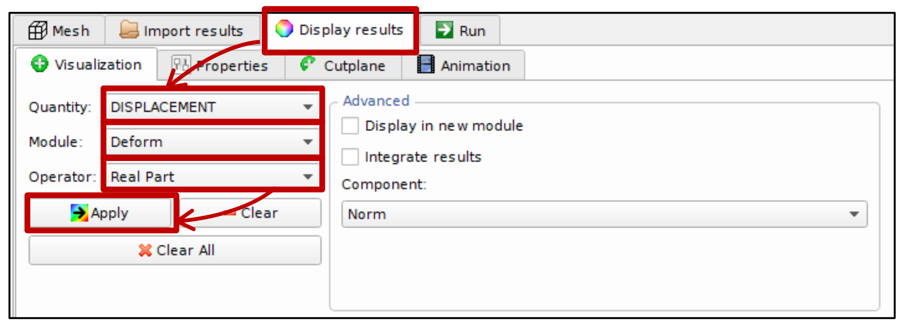
- In order to better observe the plate modes, display the Displacement Map over the Displacement Deform



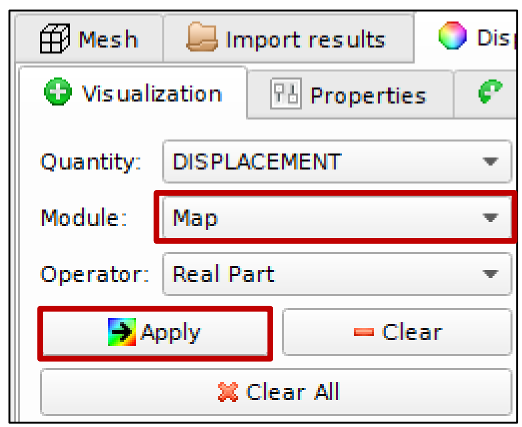
- Import the displacement results on the mesh:
  a) Open tab: Import Results
  b) Select the Topology
  c) Choose Solid_U (su) [DISPLACEMENT]
  d) Click on Import Selection



- From the Display results tab of the Toolbox, visualize the displacement map

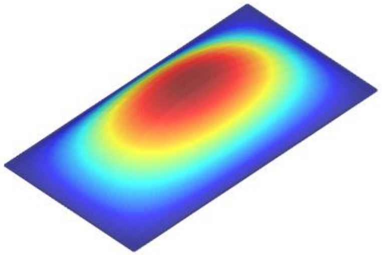


- In order to better observe the plate modes, display the Displacement Map over the Displacement Deform

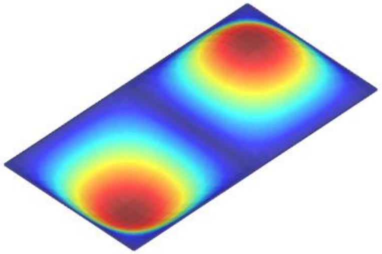


- Adjust the deformation scaling factor to better visualize modes shape

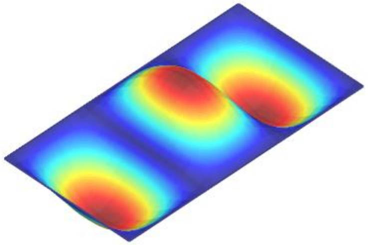
- Visualize all modes and note which mode corresponds to which frequency



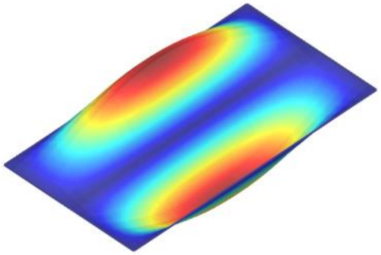
Mode 1 (1x1)
f = 61.04 Hz



Mode 2 (2x1)
f = 101.174 Hz



Mode 3 (3x1)
f = 170.979 Hz



Mode 4 (1x2)
f = 213.212 Hz

- Eigenfrequencies computed using the coarse mesh are accurate up to the second mode (~100Hz)
- For higher frequency modes the eigenfrequencies are not 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

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: $\varepsilon = \frac{\left| f_{analytic} - f_{actran} \right|}{f_{analytic}} * 100$

- Having a sufficiently refined mesh is a prerequisite for finite elements computation

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