Extraction of Acoustic Cavity Modes

Actran Student Edition Tutorial

Introduction

- This workshop introduces the modal extraction analysis for acoustic cavity and proposes an application case on a simple cavity

- The objectives of this workshop are the following :
  - Get introduced to acoustic cavities and resonance
  - Get introduced to the modal extraction analysis

- Software Version:
  - Actran 19 Student Edition

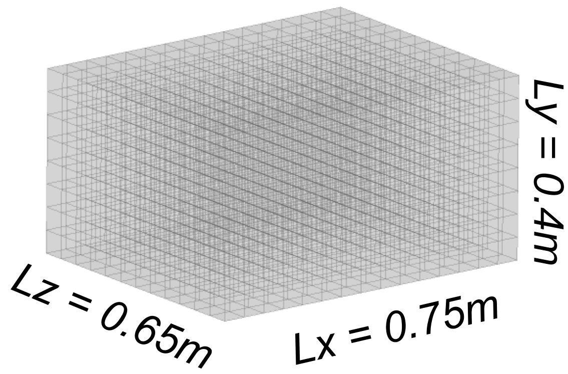
Workshop Description

- The cavity is assumed rigid at its boundary, only the acoustic part is modeled
  - A finite fluid component is defined
- Modal extraction is only suitable for real system (no imaginary part). Therefore, the model can not include damping mechanisms

Finite fluid component

Analytical Solution

- Let us consider a plate with the following properties:
  - Size: $L_x = 0.75$ m, $L_y = 0.4$ m, $L_z = 0.65$ m
  - Fluid properties:
    Speed of sound c = 340 m/s
    Density ρ = 1.225 kg/m^3



- For rectangular cavities, analytical eigen-frequencies can be calculated using the equation below

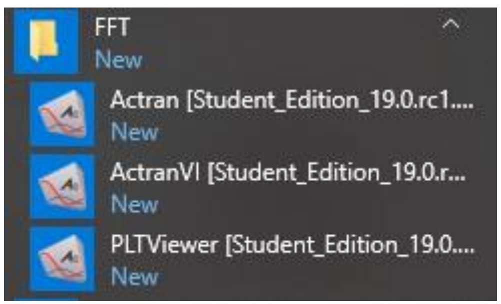
$$f_{ijm} = \frac{c}{2}\sqrt{\left(\frac{i}{L_x}\right)^2 + \left(\frac{j}{L_y}\right)^2 + \left(\frac{m}{L_z}\right)^2}$$

Workshop Pre-Processing

Modal Extraction Analysis

Start ActranVI

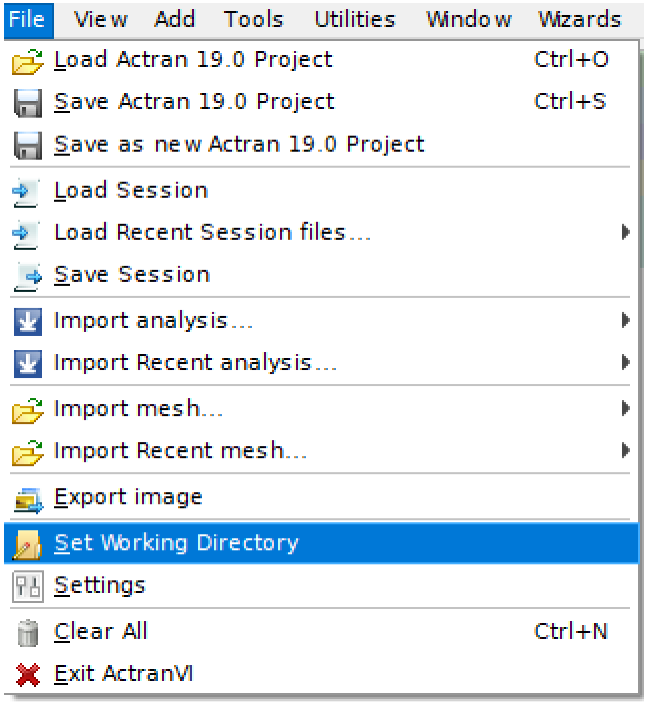
- Start ActranVI:
  - shortcut is available through the Windows Start Menu



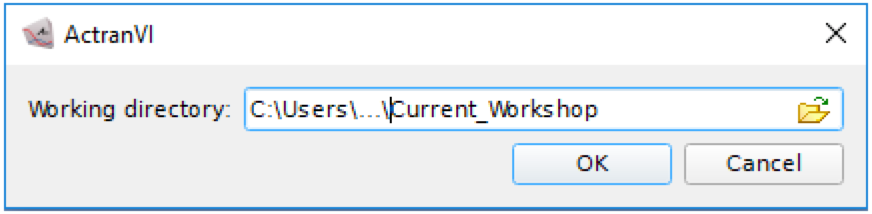
(Windows Start Menu)

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

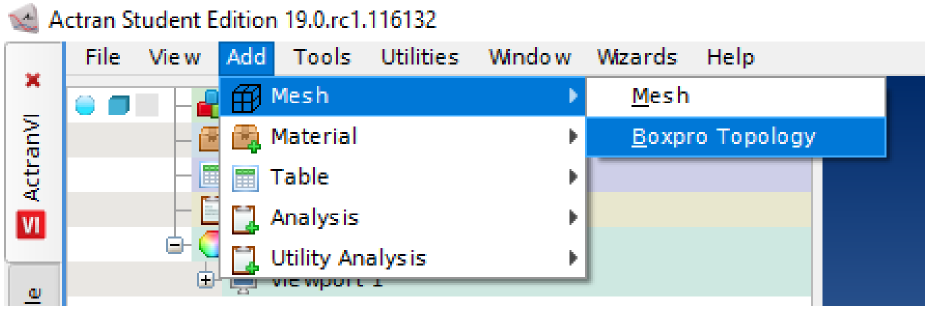


<u>Important:</u> The working directory path should not contain any space or special character

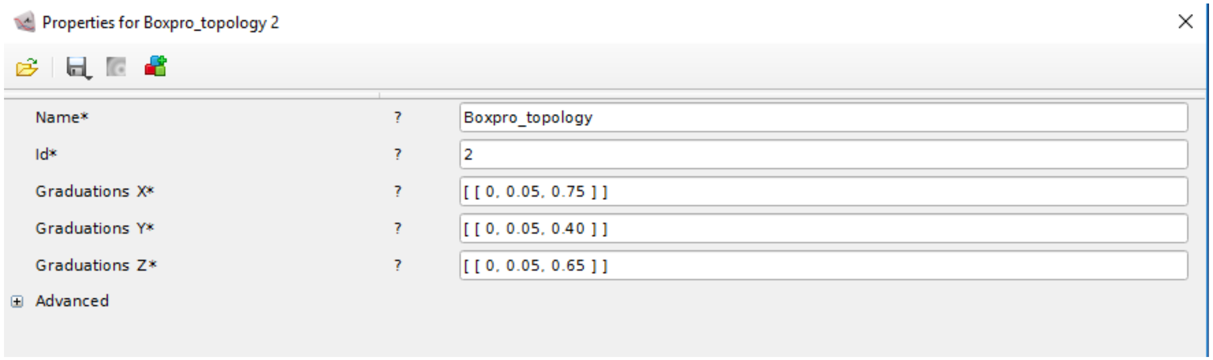


Create the BOXPRO Mesh

- Create a BOXPRO Topology in
  Add → Mesh → Boxpro Topology

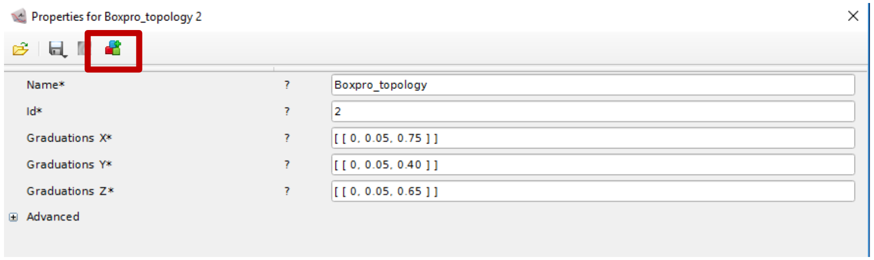


- Enter the X,Y, Z dimensions of the Boxpro topology
  - X: [[0, 0.05, 0.75]]
  - Y: [[0, 0.05, 0.40]]
  - Z: [[0, 0.05, 0.65]]

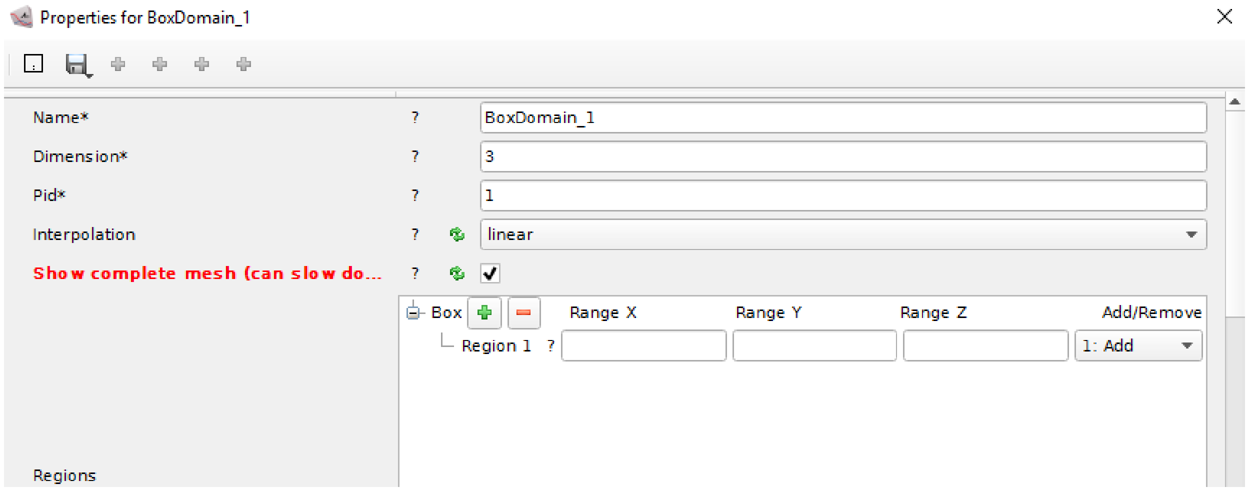


Create the Cavity Domain

- On the topology properties window click on “Add Domains…”



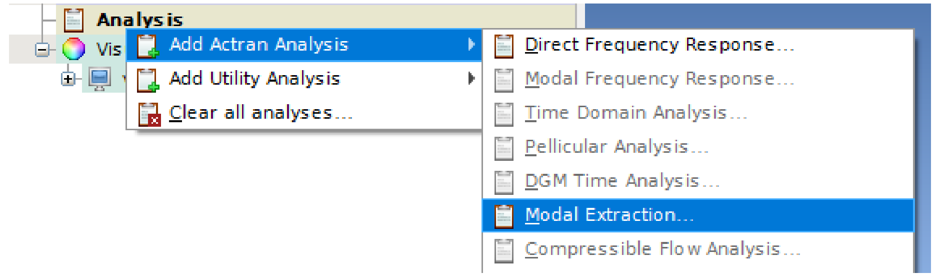
- Then, in the popup window
  a. Set the domain dimension to 3
  b. Click on the “**+**” sign to add a region
  c. Click on the check box of *Show complete mesh* to display the mesh



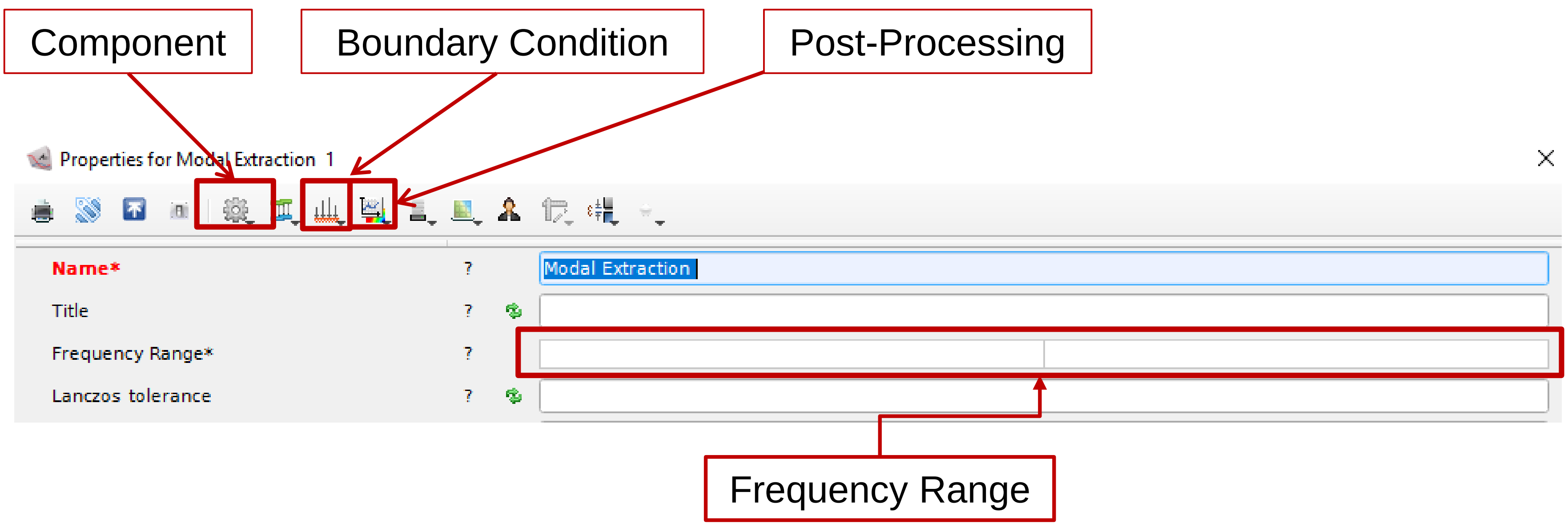
Remark: no specified range means “all”

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

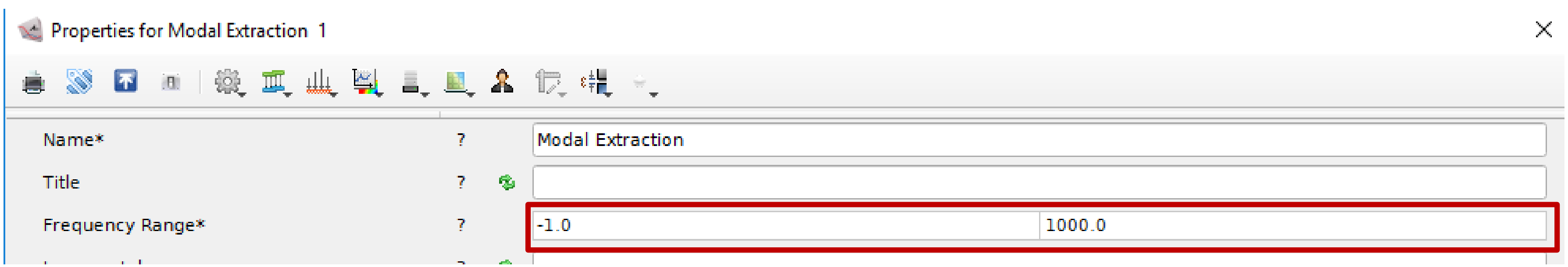


Specify the Frequency Range

- The analysis parameters are specified in the properties of the analysis
- The maximum frequency is driven by the largest element length (i.e. the smallest wavelength) :
  - For linear elements, a rule of thumb is to use 6 linear elements per wavelength to capture the acoustic fluctuation
  - The largest element length in the mesh is 5cm

$$f_{max} = \frac{c}{\lambda_{min}} = \frac{340}{6 * 0.05} = 1133\ Hz$$

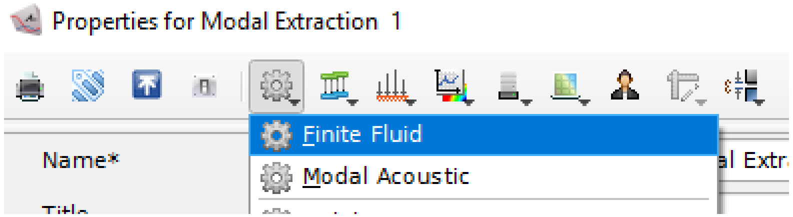
- The mesh can then be considered as valid up to 1133 Hz based on this criterion
- This analysis is performed from -1Hz up to 1000Hz



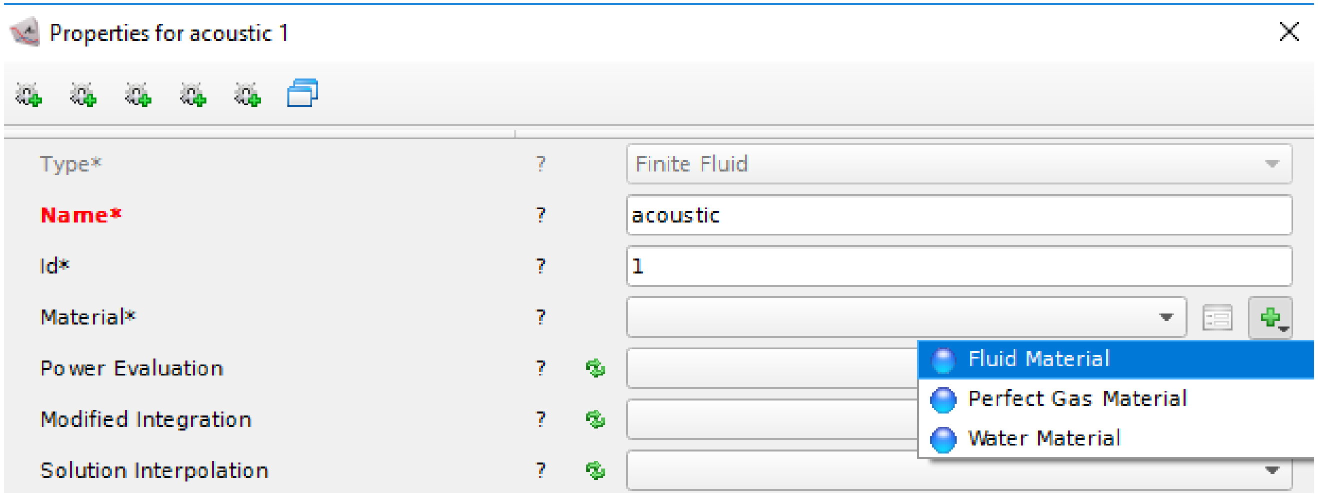
Remark : The modal extraction must start at -1Hz to be sure to take into account the cavity breathing modes

Create the Finite Fluid Component
1 - Add a Component

- Add a Finite Fluid component:



- Component properties :
  - Specify the name of the Finite Fluid component: *Acoustic*
  - Create a new Fluid material

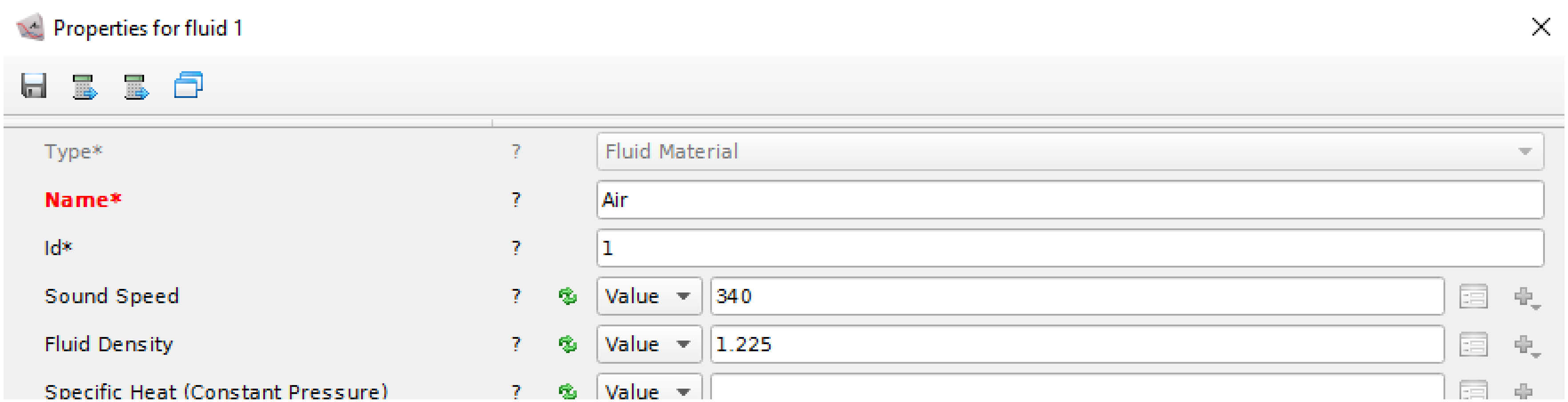


Create the Finite Fluid Component
2 - Set up the Fluid Material

- Name: *Air*
- Standard properties of air:

Speed of sound: 340 m/s; Density : 1.225 kg/m³

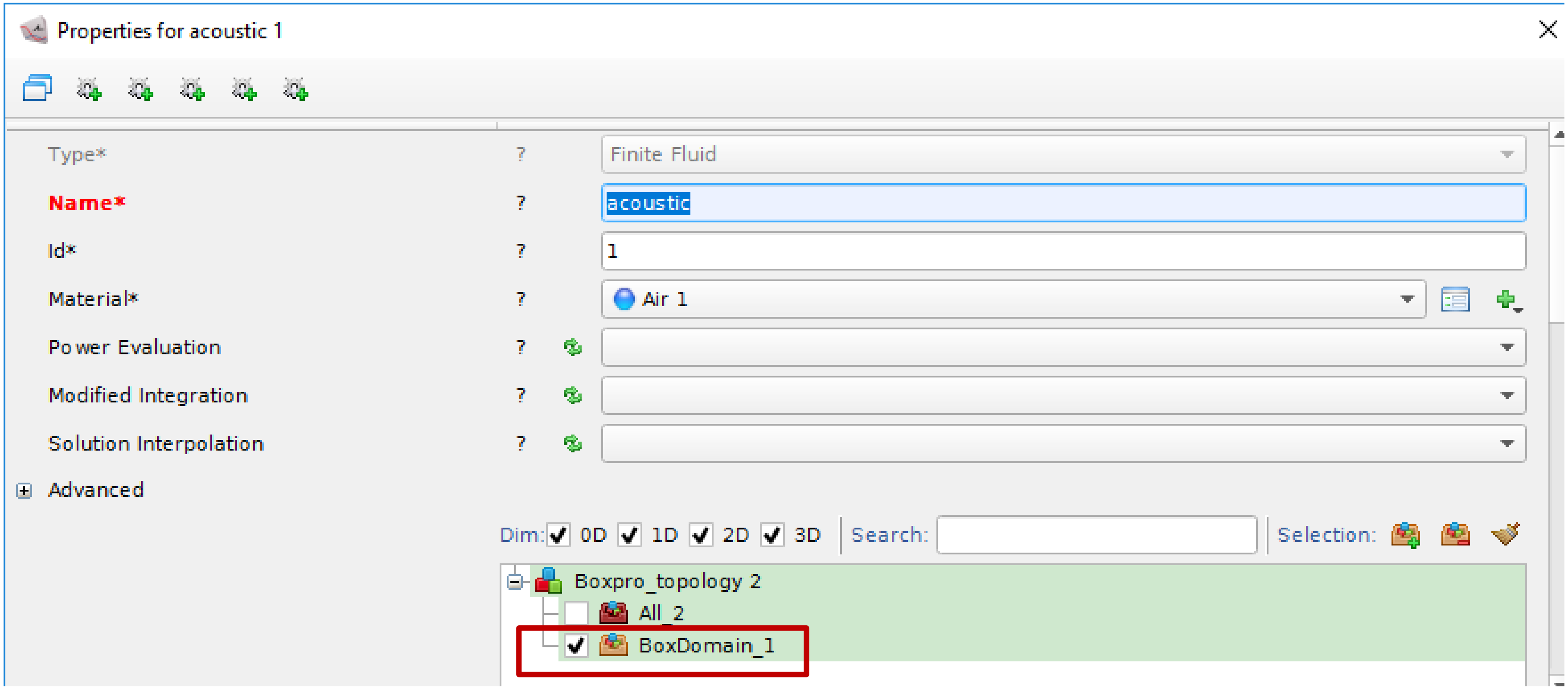
(Remark: These values are default values if they are not specified)



- Close the Properties window

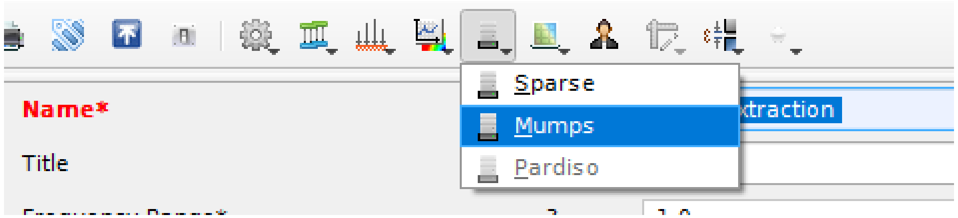
Create the Finite Fluid Component
3 - Assign the Domain

- With the Scope Selector, assign the BoxDomain_1 to the Acoustic component



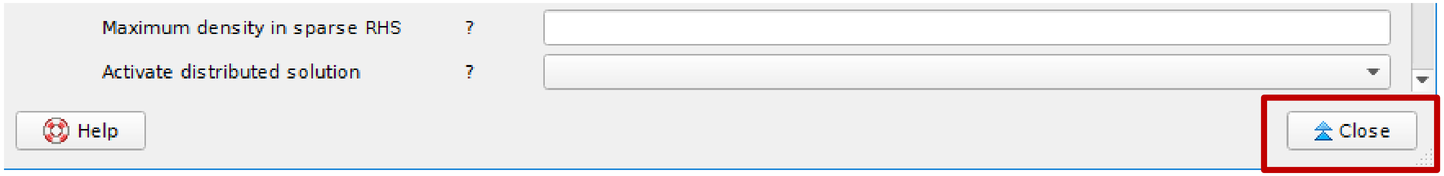
Specify the Solver

- Define the solver of the analysis



- Set the MUMPS solver

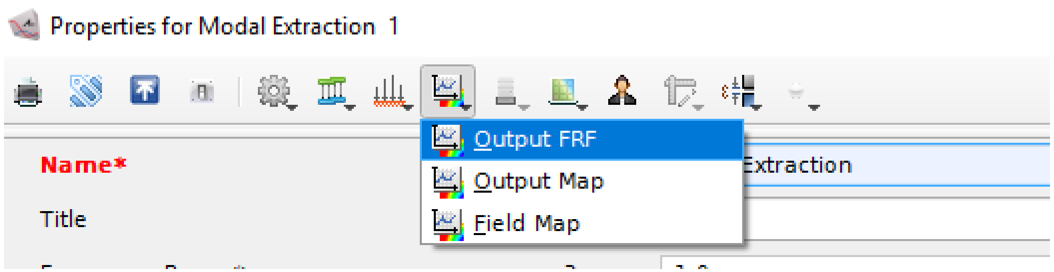
- Close the pop-up window of MUMPS



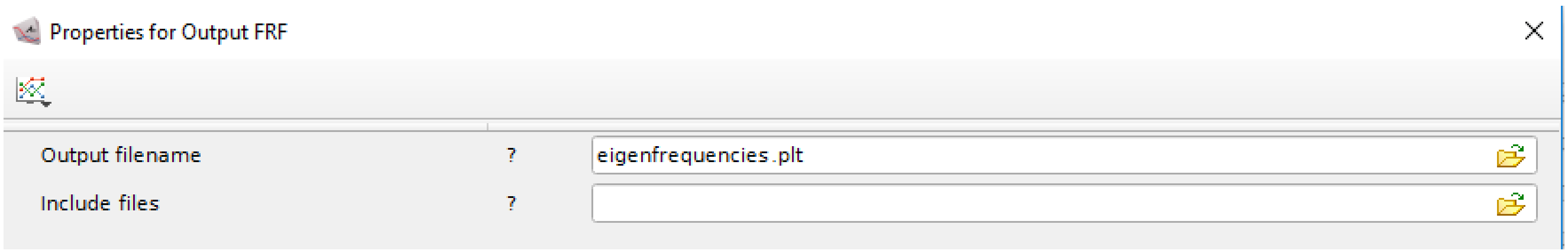
- Close the properties window of the Modal Frequency Response

Set the Post-processing Parameters

- Add an Output FRF post-processing parameter



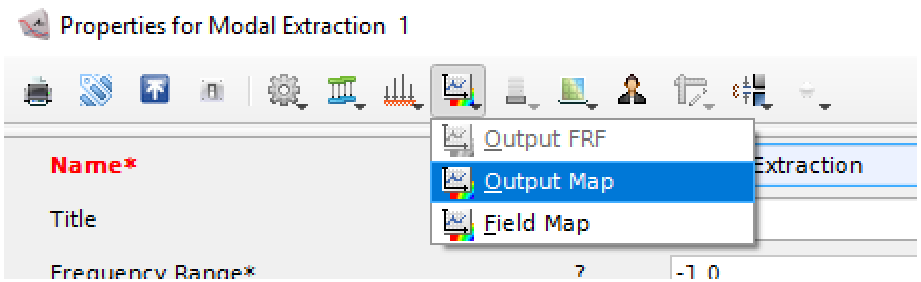
- Set the name of the FRF output file: eigenfrequencies.plt
- This FRF file contains a list of all the eigenfrequencies to be found



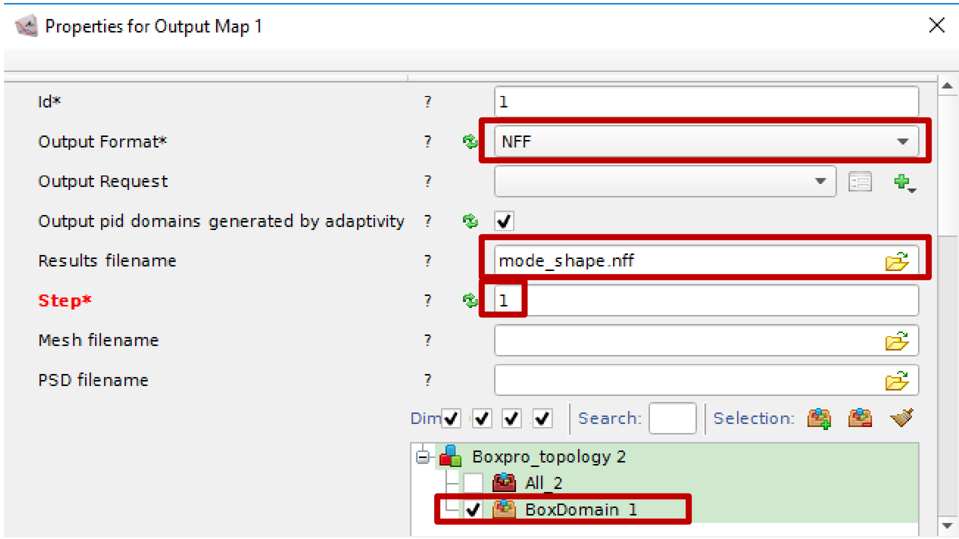
- Close the Output FRF properties window

Set the Post-processing Parameters

- Add an Output Map post-processing parameter

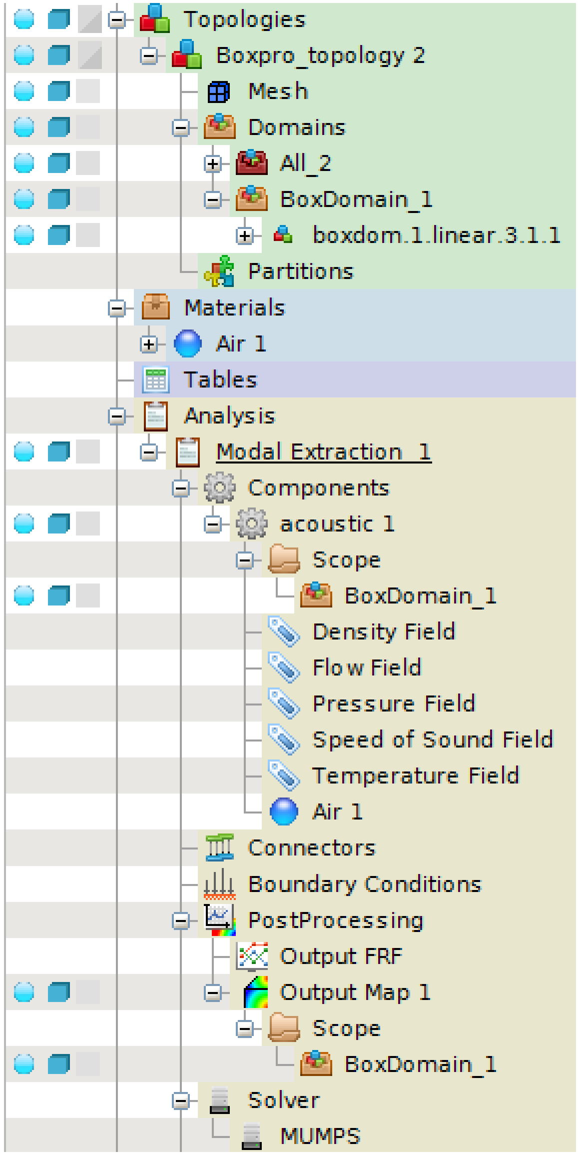


- Set output parameters:
  - Specify the output format NFF
  - Specify the filename mode_shape.nff
  - Output the map for every mode (step: 1)
  - Select the BoxDomain domain
- Close the Output Map properties window

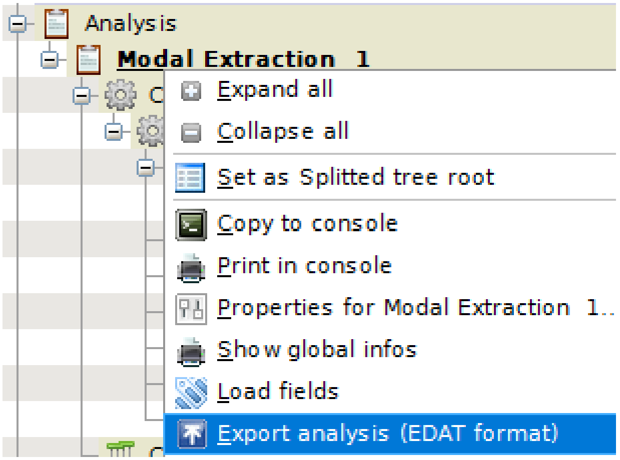


Check the Analysis

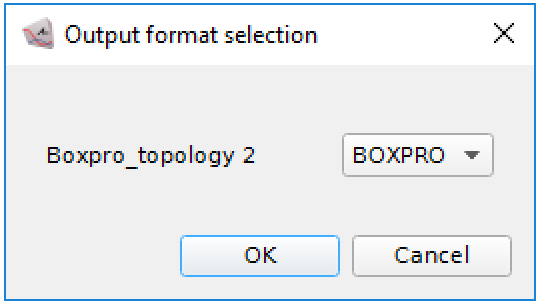
- The analysis is now completely defined
- All the parts of the analysis are available and editable on the data tree panel
- Check if the analysis tree is identical to the one shown



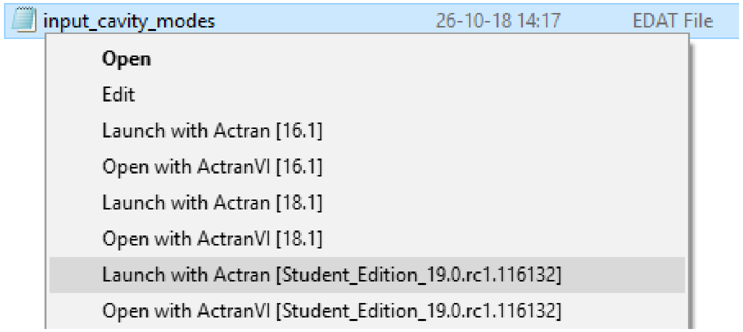
Export the Analysis File



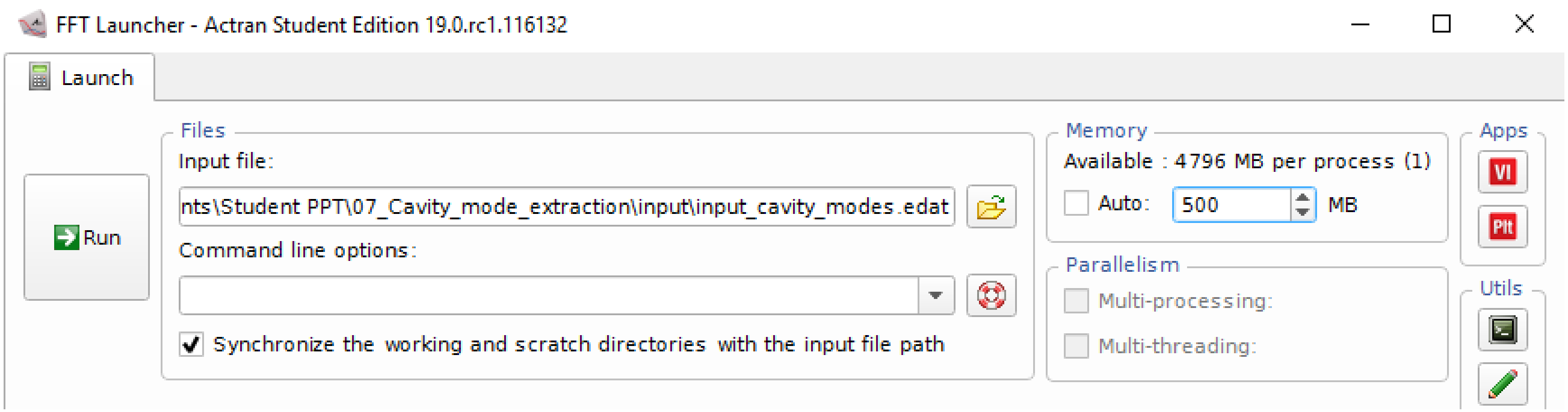
- The analysis can be exported in the EDAT Actran input file
- Right click on the *Modal Extraction*, and choose *Export analysis (EDAT format)*
- The topology was created with BOXPRO, the analysis may be exported in two different formats:
  - The *BOXPRO* format: the mesh is written in the EDAT file the same way as in the topology definition. Nodes coordinates and elements are created at the beginning of the analysis
  - The *ACTRAN* format: the mesh is explicitly written in the EDAT file
- Select BOXPRO as the Output format and name the input file “input_cavity_modes”



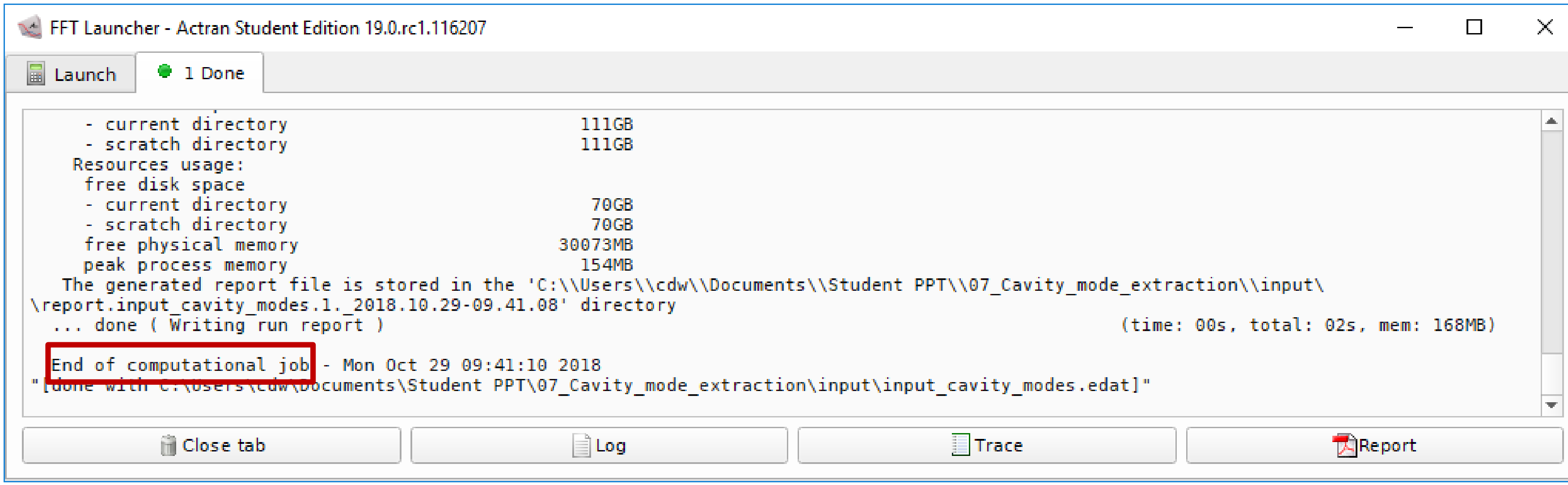
Launch Actran Analysis



- Launch the computation:
  - Open the FFT Launcher by right clicking on the *input_cavity_modes.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



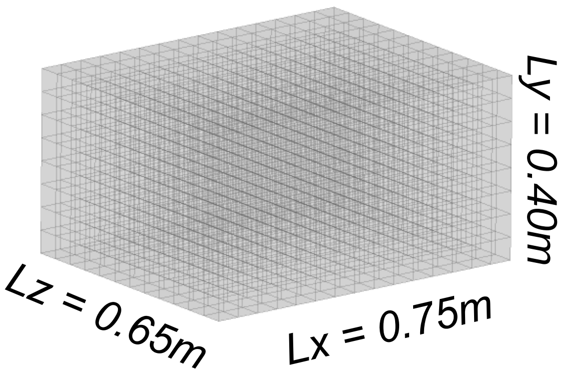
- Close the Launcher window

Post-processing

Examine eigenfrequencies
Visualize Mode shapes in ActranVI

Calculated vs Analytical Eigenfrequencies

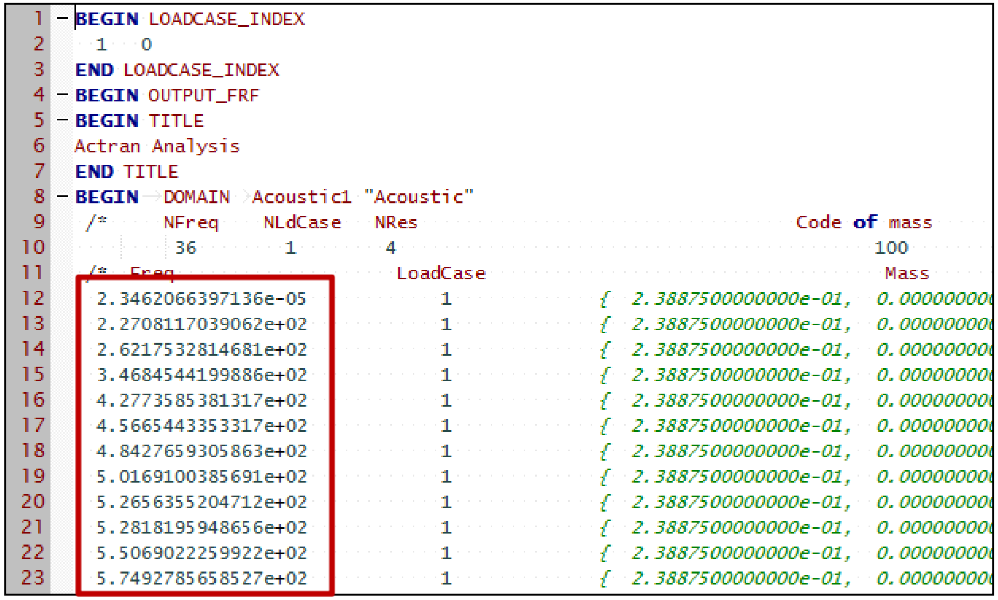
- Let us consider a plate with the following properties:
  - Size: $L_x = 0.75$ m, $L_y = 0.40$ m, $L_z = 0.65$ m
  - Fluid properties:
    Speed of sound c = 340 m/s
    Density ρ = 1.225 kg/m^3



- For rectangular cavities, analytical eigen-frequencies can be calculated using the equation below

$$f_{ijm} = \frac{c}{2}\sqrt{\left(\frac{i}{L_x}\right)^2 + \left(\frac{j}{L_y}\right)^2 + \left(\frac{m}{L_z}\right)^2}$$

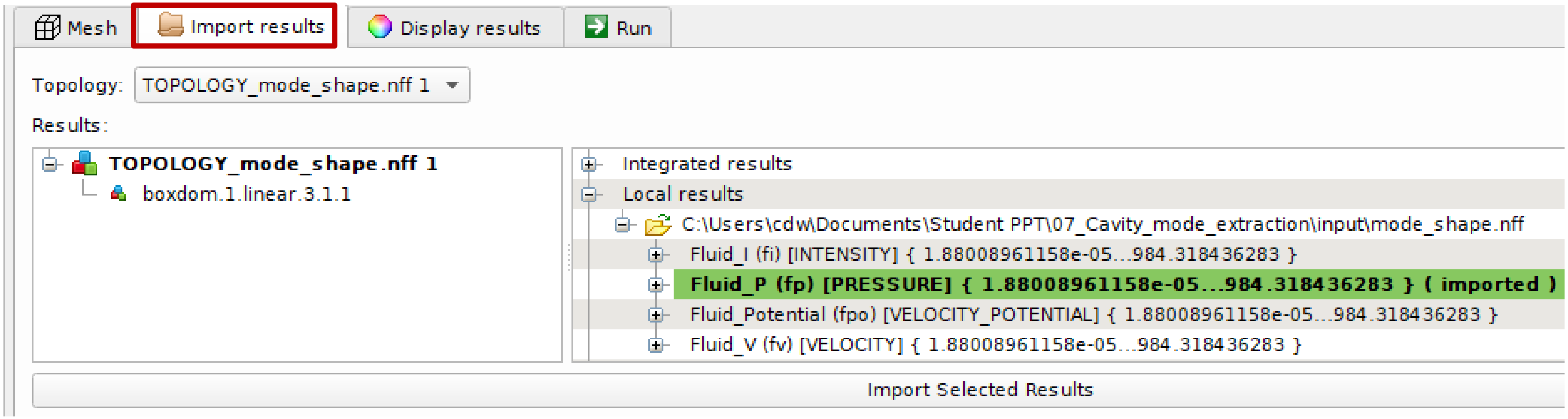
- Eigenfrequencies calculated by Actran are stored in the eigenfrequencies.plt file
- This file can be opened using a text editor
- Eigenfrequencies are stored in the first column



```
BEGIN LOADCASE_INDEX
  1   0
END LOADCASE_INDEX
BEGIN OUTPUT_FRF
BEGIN TITLE
Actran Analysis
END TITLE
BEGIN  DOMAIN  Acoustic1 "Acoustic"
 /*     NFreq     NLdCase    NRes                                    Code of mass
            36          1       4                                           100
 /*  Freq                    LoadCase                                       Mass
  2.3462066397136e-05             1              {  2.3887500000000e-01,  0.00000000
  2.2708117039062e+02             1              {  2.3887500000000e-01,  0.00000000
  2.6217532814681e+02             1              {  2.3887500000000e-01,  0.00000000
  3.4684544199886e+02             1              {  2.3887500000000e-01,  0.00000000
  4.2773585381317e+02             1              {  2.3887500000000e-01,  0.00000000
  4.5665443353317e+02             1              {  2.3887500000000e-01,  0.00000000
  4.8427659305863e+02             1              {  2.3887500000000e-01,  0.00000000
  5.0169100385691e+02             1              {  2.3887500000000e-01,  0.00000000
  5.2656355204712e+02             1              {  2.3887500000000e-01,  0.00000000
  5.2818195948656e+02             1              {  2.3887500000000e-01,  0.00000000
  5.5069022259922e+02             1              {  2.3887500000000e-01,  0.00000000
  5.7492785658527e+02             1              {  2.3887500000000e-01,  0.00000000
```

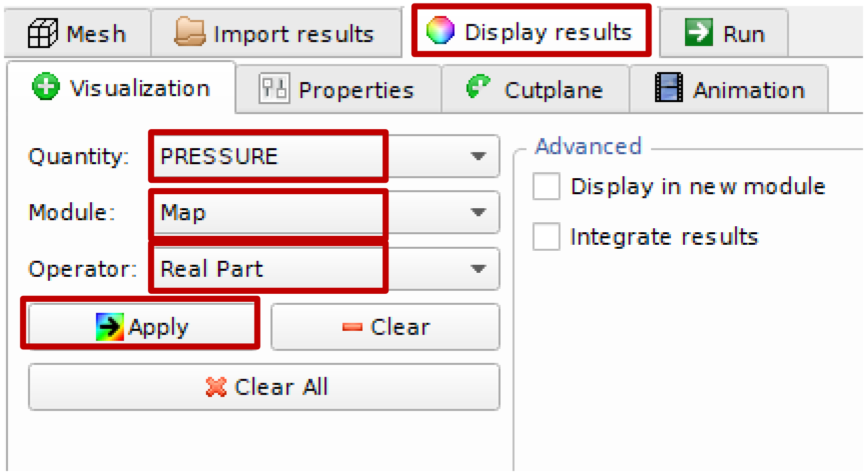
Visualize Field Map in ActranVI

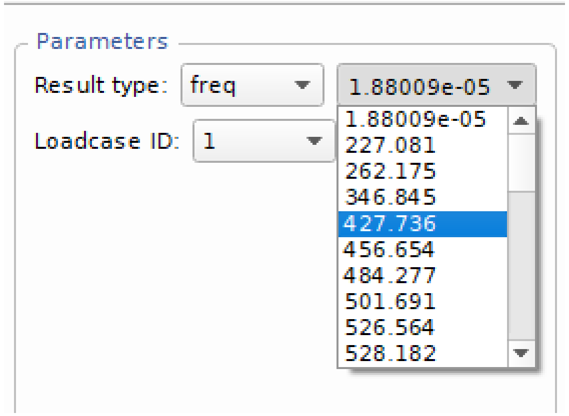
- In ActranVI, import the displacement results on the mesh:
  a) Open tab: Import Results
  b) Select the topology TOPOLOGY 1
  c) Add NFF: mode_shape.nff (all available quantities are listed in the Database Content)
  d) Choose Fluid_P (fp) [PRESSURE]
  e) Import Selected Results



Visualize Field Map in ActranVI

- From the *Display results* tab of the Toolbox, visualize the real part of pressure map
    a. Change to “Display results” tab
    b. Choose the indicated combination of Module, Quantity and Operator
    c. Click on “Apply”
    d. Click on “Frequency” on the right
    e. Change frequencies for visualization

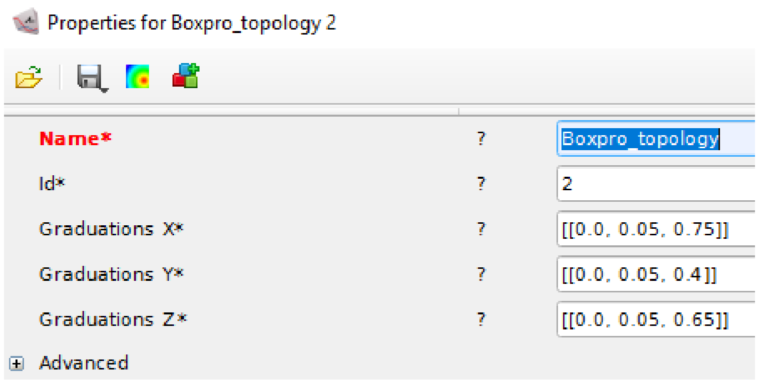


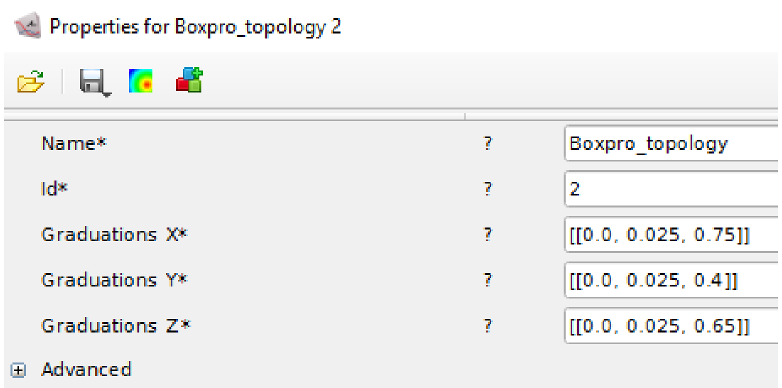


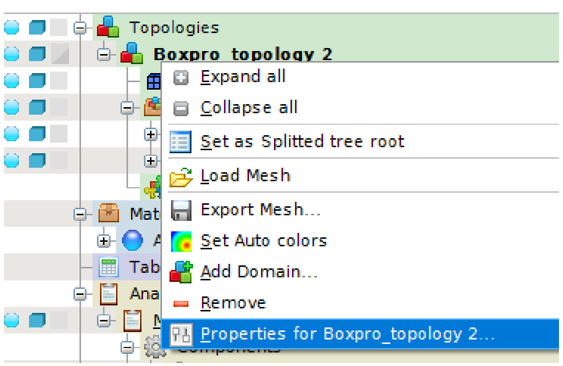
Going Further (Optional)

Refining Mesh to Increase Model Accuracy

- Refining the cavity mesh can increase the accuracy of numerical prediction of the eigen-frequencies. To refine the mesh, two methods can be used
- Method 1: decrease the mesh step, for example, from 0.05m to 0.025m

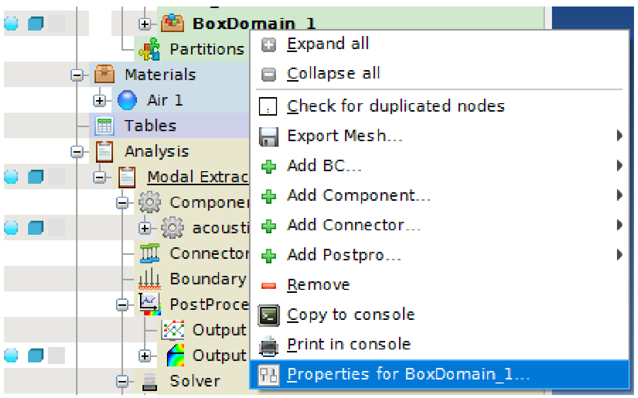


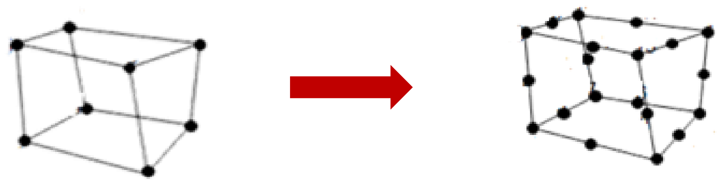




Refining Mesh to Increase Model Accuracy

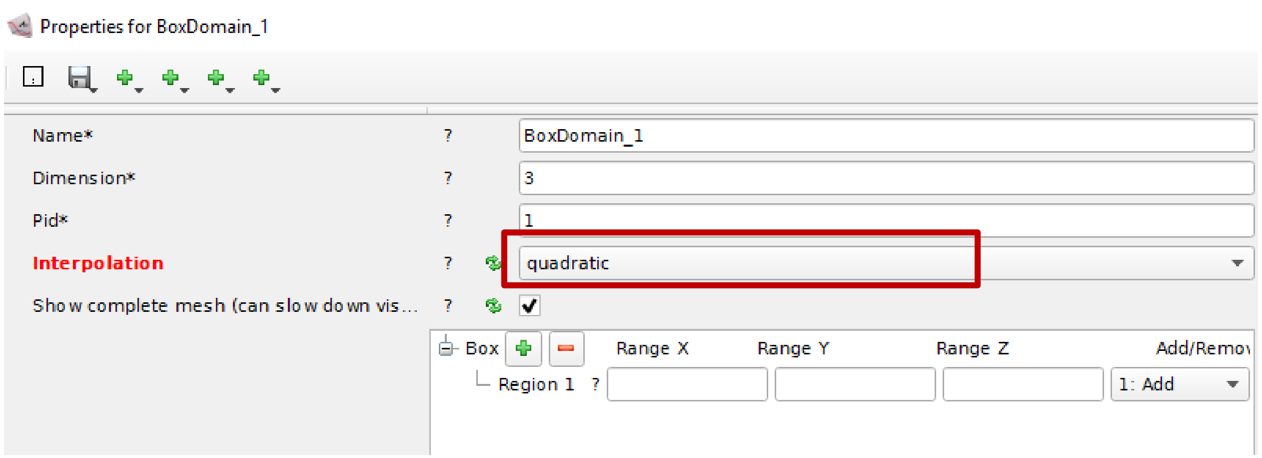
- Method 2: Change the mesh from linear interpolation from quadratic interpolation





Linear element

Quadratic element



Going Further

- Visualize how cavity modes are modified if the cavity dimensions are changed

- Compute modes of cavities of complex shapes (external mesh is needed)

  Example ➔ computing the modes of a car compartment is a frequently performed task in automotive NVH (Noise, Vibration, Harshness) analysis