Transmission Loss of a Muffler

Actran Student Edition Tutorial

Introduction

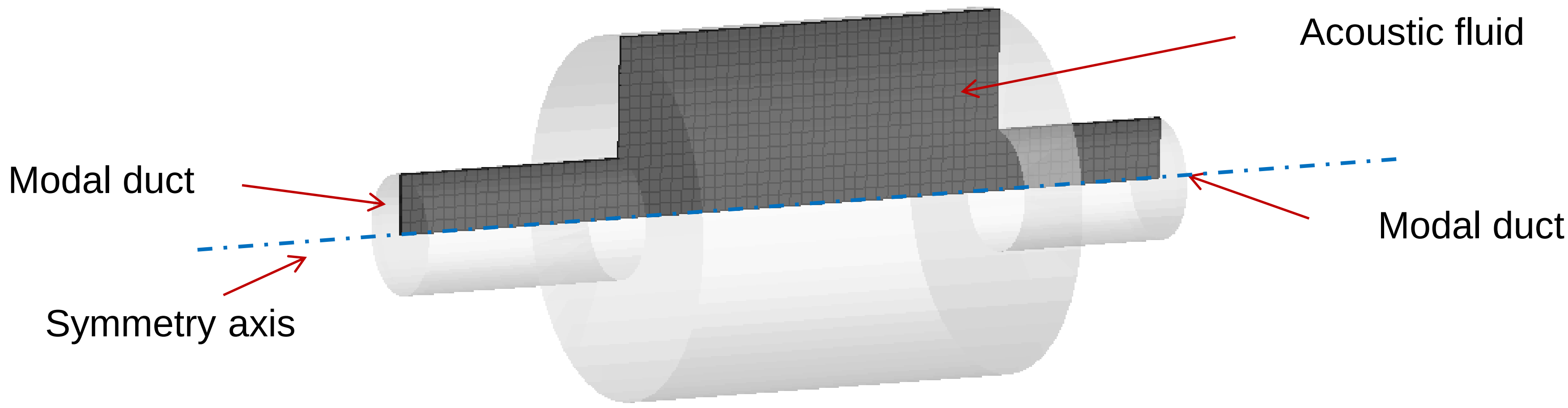
- This workshop demonstrates Actran capabilities to model an expansion chamber (simple muffler) and calculate its transmission loss

- The objectives of this workshop are the following :
  – Get introduced to muffler transmission loss
  – Use 2D geometry to model axi-symmetric 3D problem
  – Distinguish plane wave duct propagation and non-plane wave propagation

- Software Version:
  – Actran 19 Student Edition

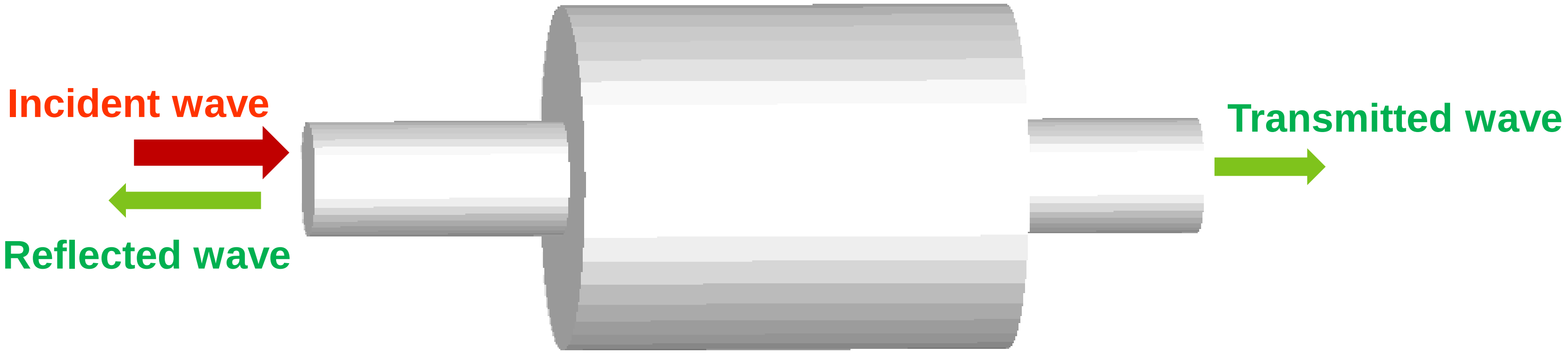
Workshop Description

- Through this workshop, we will model an expansion chamber using 2D axisymmetric modeling technique and calculate its transmission loss
- The muffler is modeled in 2D
  - A finite fluid component is defined
- Muffler inlet and outlet are modeled by modal ducts
  - Modal basis components are defined



Transmission Loss of Muffler

- Incident wave is partially transmitted and partially reflected by the muffler



- The Transmission Loss (TL) is the ratio between the incident power and transmitted power

$$TL = 10 \cdot \log10(Winc/Wtrans)$$

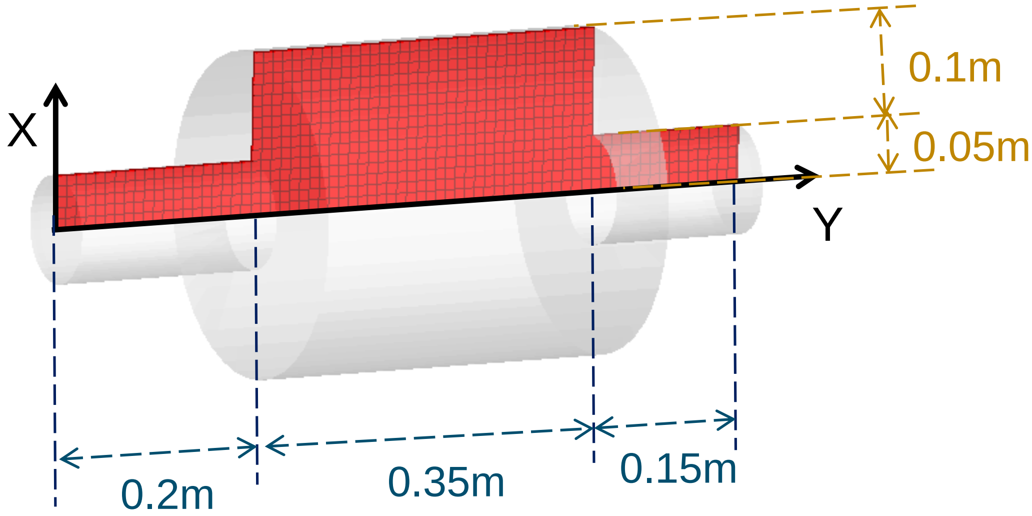
- Acoustic anechoic condition is applied
  - at outlet for the transmitted wave
  - at inlet for the reflected wave

Analytical solution

- For an expansion chamber, the analytical TL can be calculated using the equation

$$TL = 10\log\left[1+\left(\frac{m^2-1}{2m}\sin kl\right)^2\right]$$

  - m: cross section area ratio between expansion chamber and inlet (outlet) tube $(0.15 / 0.05)^2 = 9$
  - l: length of expansion part of the chamber = 0.35
  - k: wave number $2 \cdot \pi \cdot$ freq / speed of sound



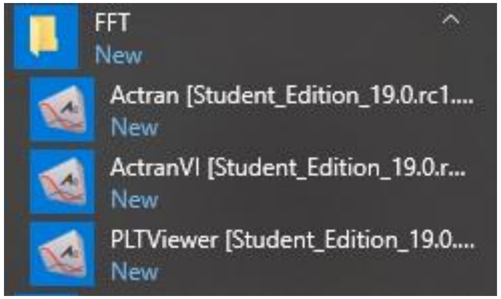
- This analytical solution is calculated with the assumption of plane wave propagation in the muffler

Workshop Pre-Processing

Direct Frequency Analysis

Start ActranVI

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



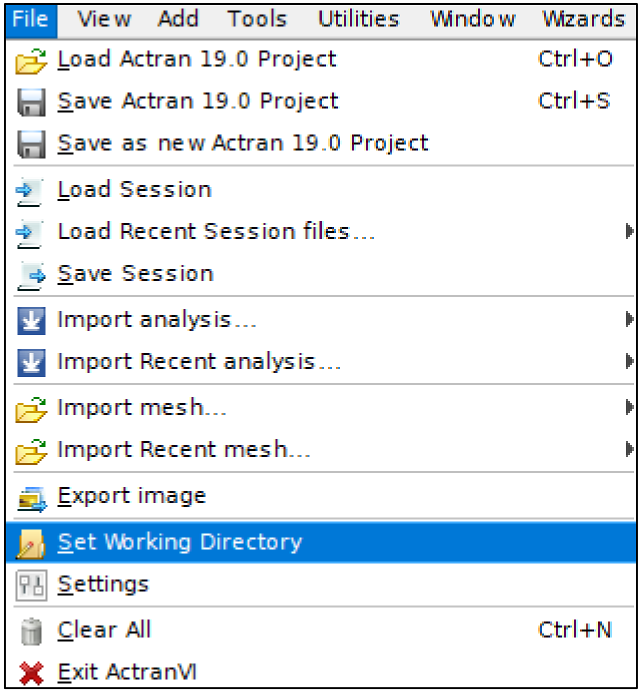
(Windows Start Menu)

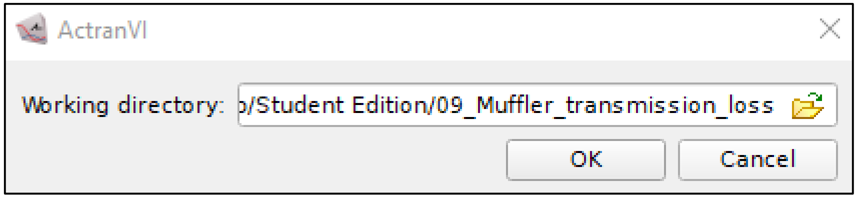
Set the Working Directory

- 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

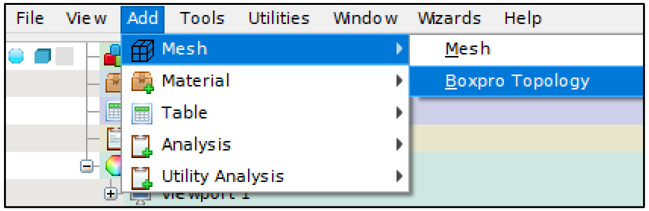




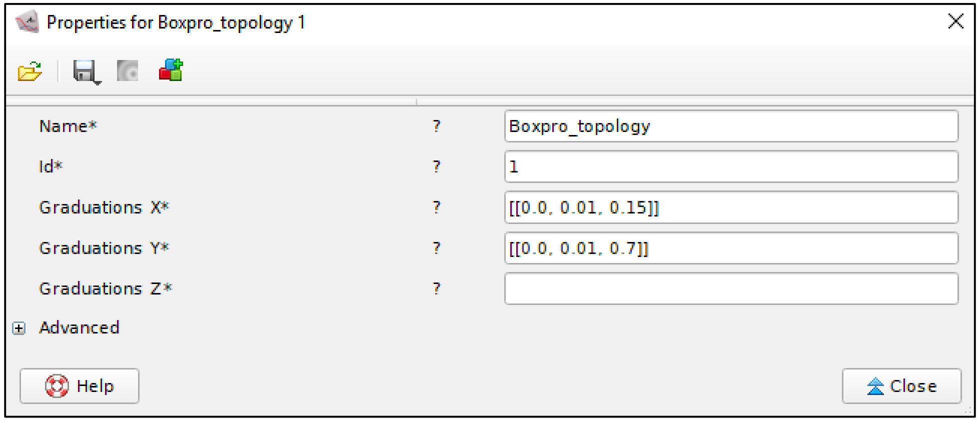
* Important: The working directory path should not contain any space or special character

Create the mesh using BOXPRO

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

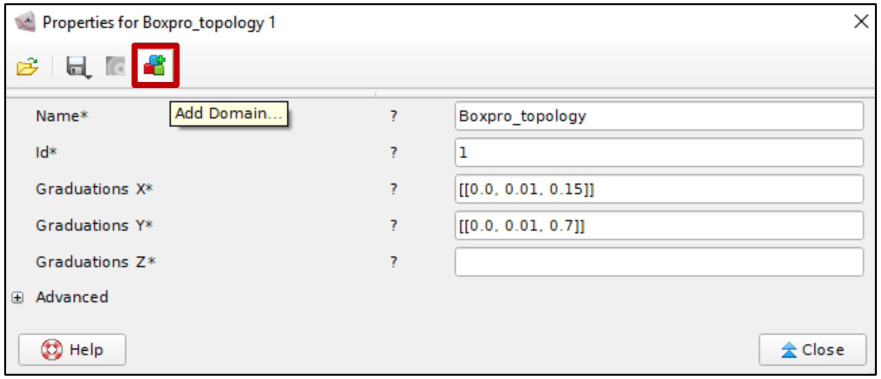


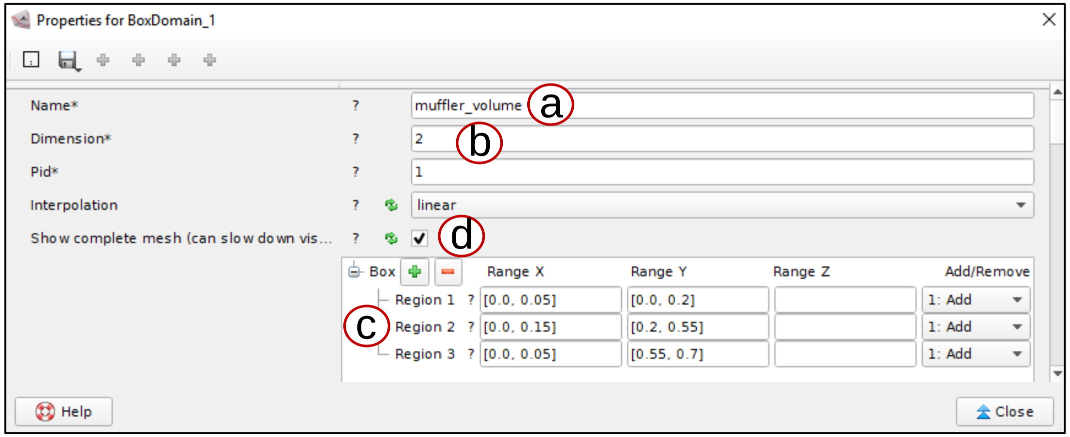
- Enter the X,Y, Z dimensions of the Boxpro topology
  - X: [[0, 0.01, 0.15]]
  - Y: [[0, 0.01, 0.7]]
  - Z: EMPTY (z=0 for entire mesh)



Create the mesh using BOXPRO

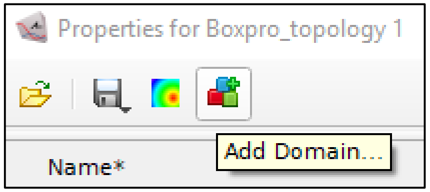
- Based on the definition of the global box, the domains may be created
- On the topology properties window click on “Add Domains…”
- Then, in the popup window
    a. Set the name of the domain
    b. Set the domain dimension to 2
    c. Create the three regions
    d. Click on the check box of *Show complete mesh* to display the mesh

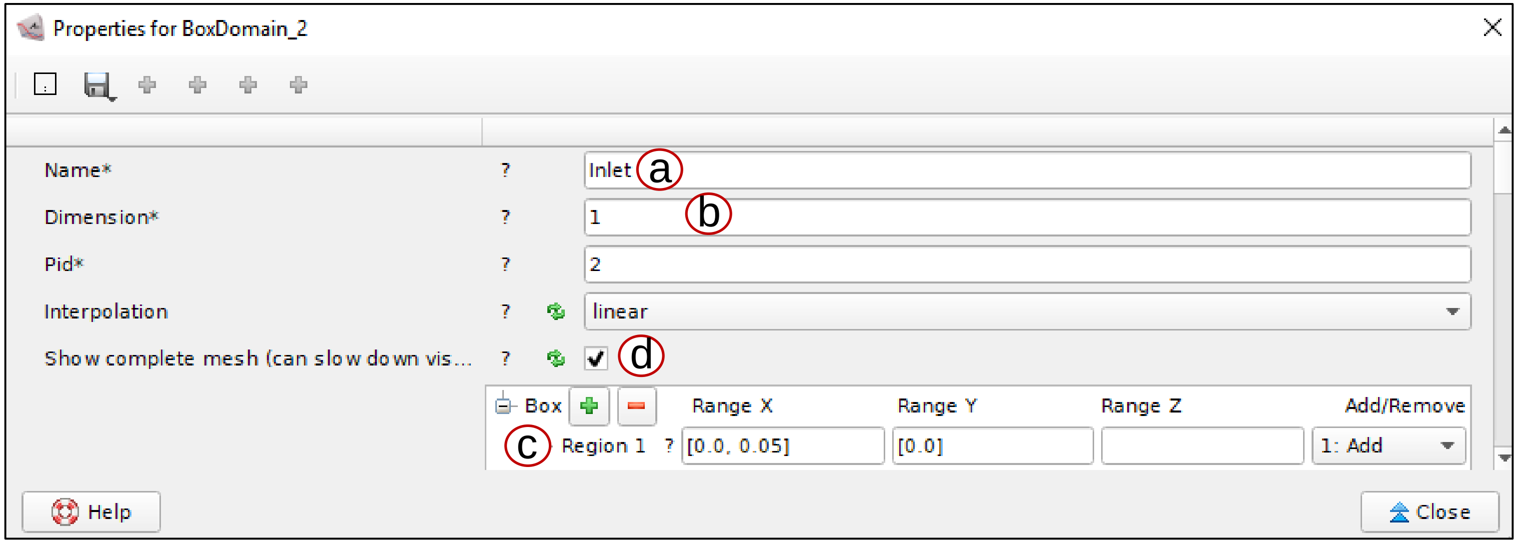




	Range X	Range Y
Region 1	0.0 , 0.05	0.0 , 0.2
Region 2	0.0 , 0.15	0.2 , 0.55
Region 3	0.0 , 0.05	0.55 , 0.7

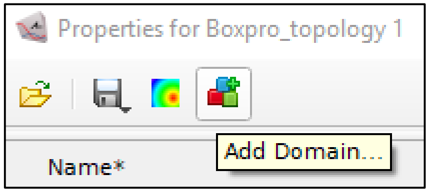
- Add a new domain for the Inlet
- Fill the properties of the Domain
  a. Change domain Name to “Inlet”
  b. Enter “1” in the Dimension of the domain
  c. Create a region.   X: 0, 0.05   Y: 0   Z: empty
  d. Click on the check box of Show complete mesh

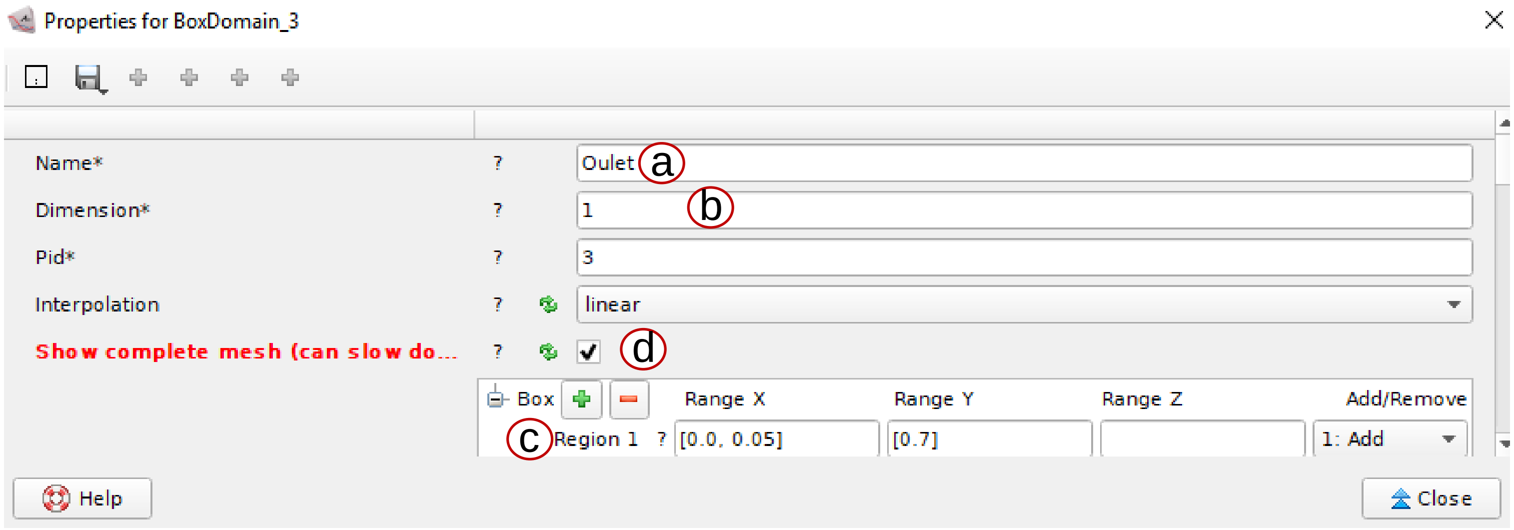




- The line supporting the inlet is now created

- Add a new domain for the Outlet
- Fill the properties of the Domain
  a. Change domain Name to “Outlet”
  b. Enter “1” in the Dimension of the domain
  c. Create a region. X: 0, 0.05 Y: 0.7 Z: empty
  d. Click on the check box of Show complete mesh

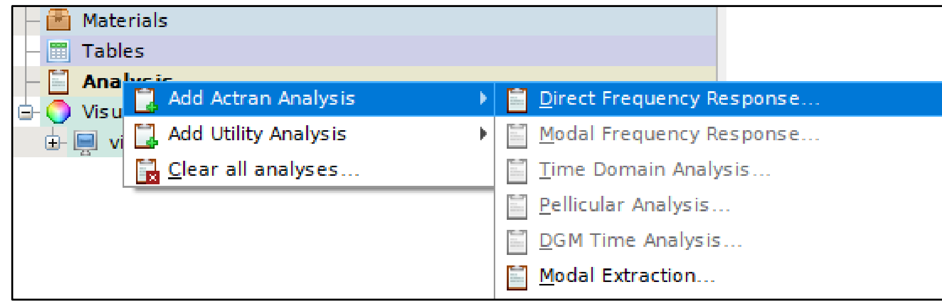




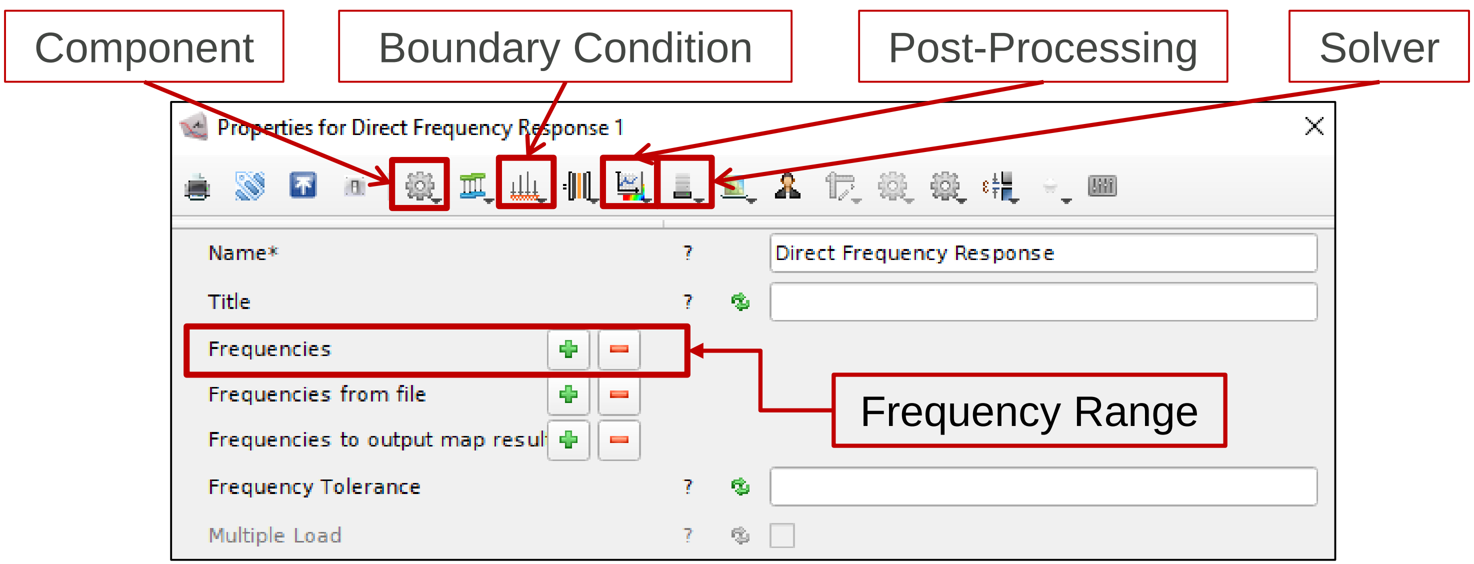
- The line supporting the outlet is now created

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

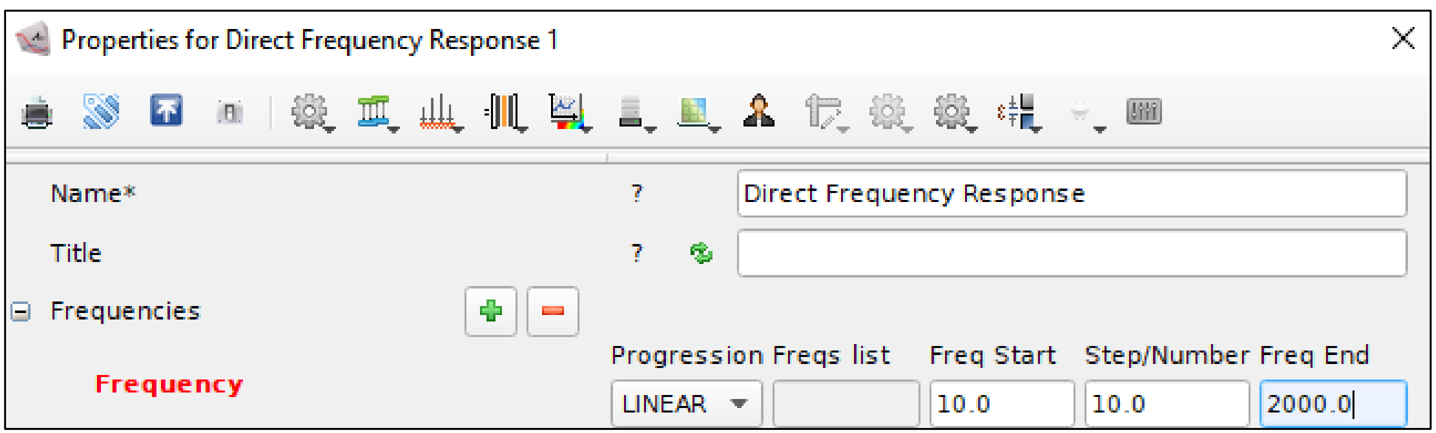
- The frequency range of computation is specified through the properties of the analysis

- The maximum frequency is driven by the smallest wave length :
  - For linear elements, a rule of thumb is to use 8 linear elements per wavelength to capture the acoustic fluctuation
  - The largest element length in the mesh is 10 mm

$$f_{max} = \frac{c}{\lambda_{min}} \quad with \;\; L_{max} = \frac{\lambda_{min}}{8} = 0.01m$$

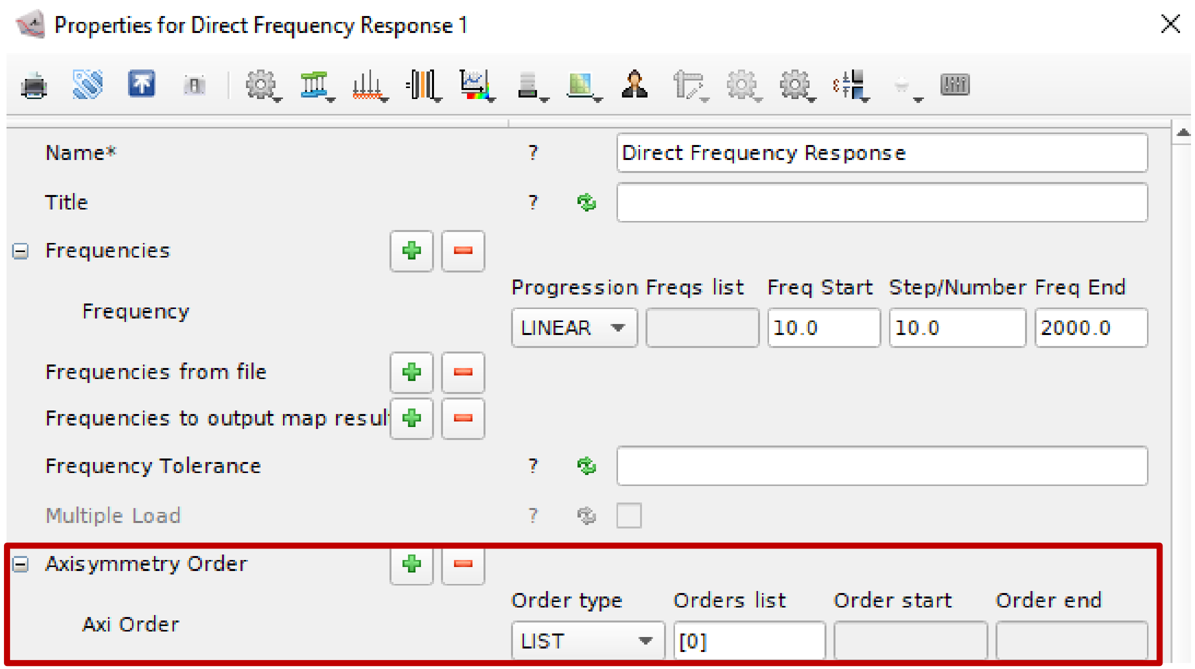
$$gives \; f_{max} = \frac{340}{8 * 0.01} = 4250Hz$$

- This analysis is performed from 10Hz up to 2000 Hz with a step of 10 Hz



Axi-symmetry Definition

- The Actran model is a 2D axi-symmetric model. This should be defined in the Direct Frequency Analysis properties → Set the Axisymmetry Order to 0. This specifies a constant solution with varied azimuthal angle in the duct cross section

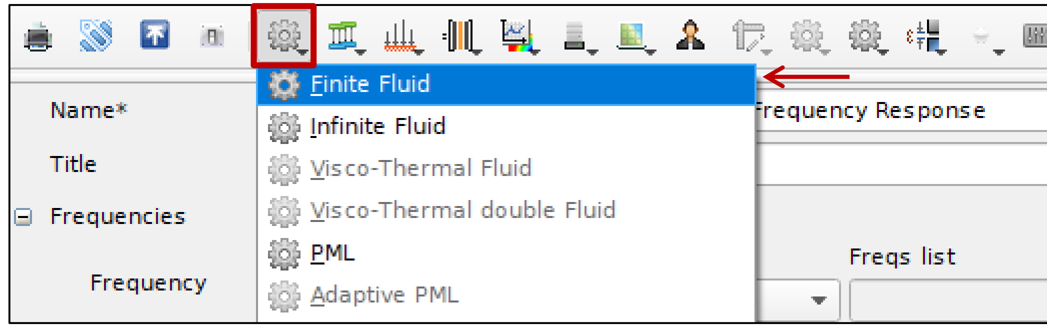


- By default, the Y axis is always the axis of revolution. The mesh of axisymmetric problems must locate in the right half (X > 0) of the XY plane

Create the Finite Fluid Component

1 – Add a Component

- Add a Finite Fluid component



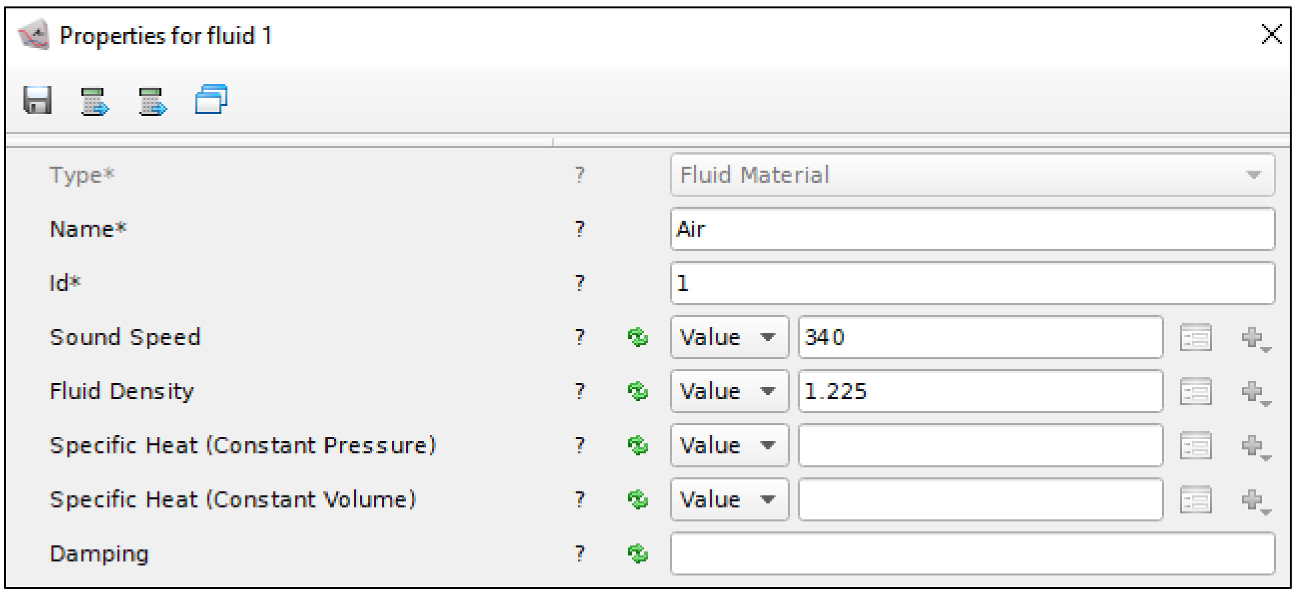
- Component properties:
  - Specify the name of the Finite Fluid component: Muffler_volume
  - Create a new Fluid material

Create the Finite Fluid Component

2 – Set up the Fluid Material

- Name: *Air*
- Standard properties of air:
  - Speed of sound: 340 m/s
  - Density: 1.255 kg/m^3

Remark: These values are defaults values if they are not specified

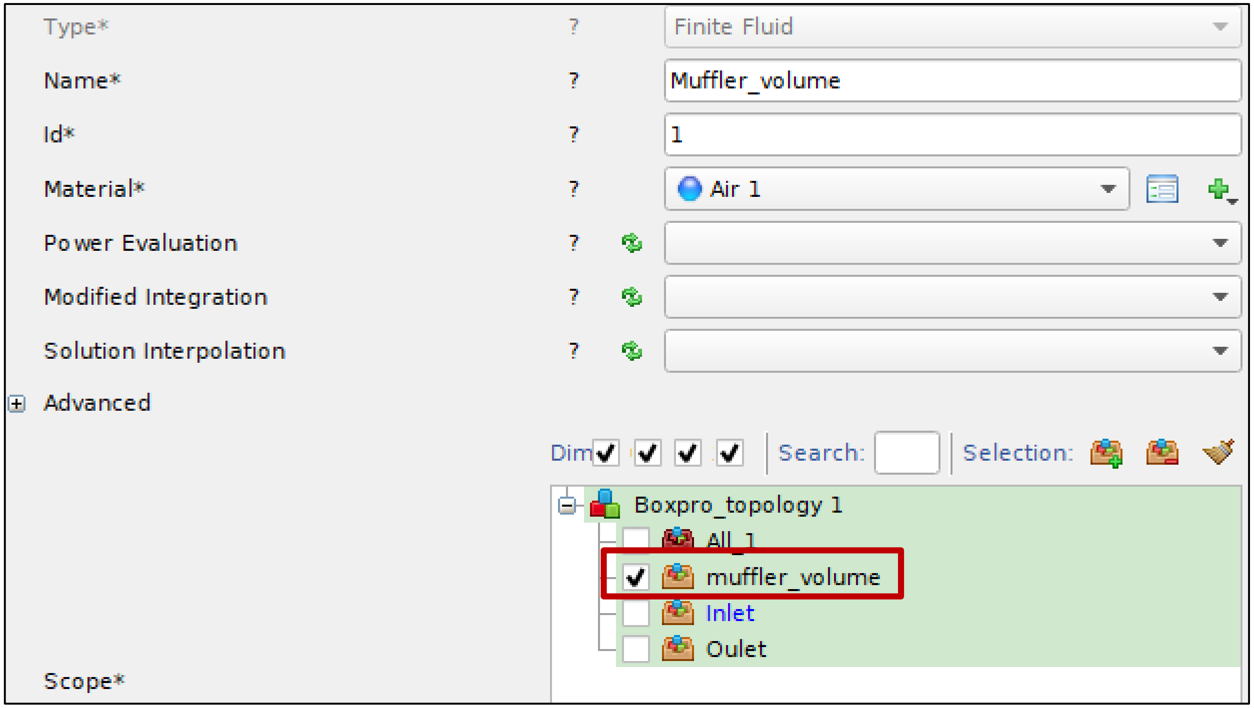


- Close the material properties window

Create the Finite Fluid Component

3 – Assign the Domain

- With the Scope selector, assign the *Muffler_volume* domain to the *Muffler_volume* component



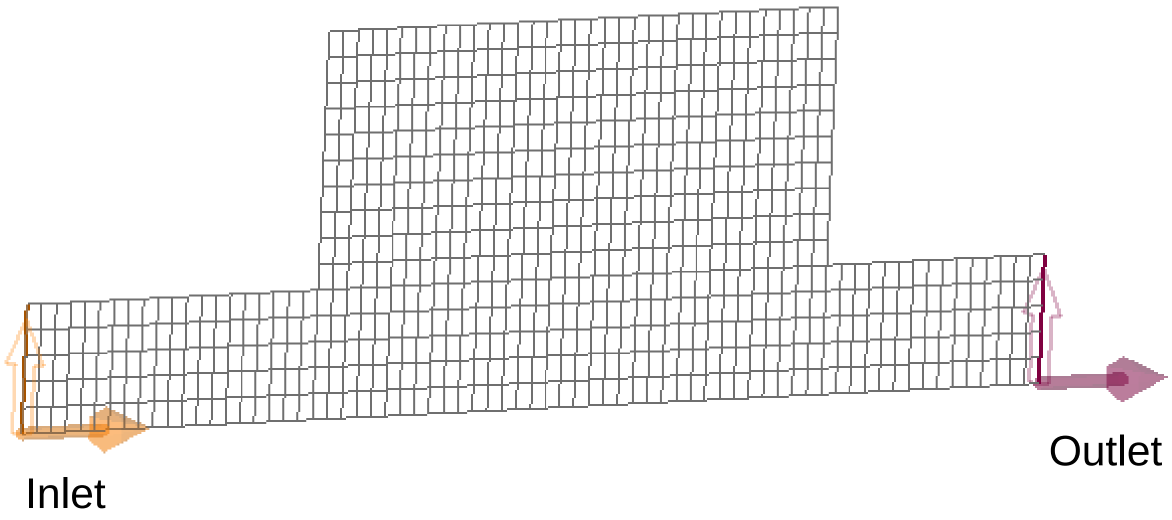
- Close the component properties window

Modal Ducts Components

- For in-duct propagation problems, acoustic wave can be seen as a mathematical superposition of duct modes
- In Actran, such an analytical representation is used in order to specify non-reflecting BC as well as to inject energy through a given system assuming a connection to semi-infinite ducts
- Two types of modes can be defined:
  - Constrained : allows injecting energy in the system and must be defined in the +1 direction (see further slides) if the first axis points inside the system
  - Free : allows representing a non-reflecting BC and must be defined in the -1 direction (see further slides) if the first axis points inside the system

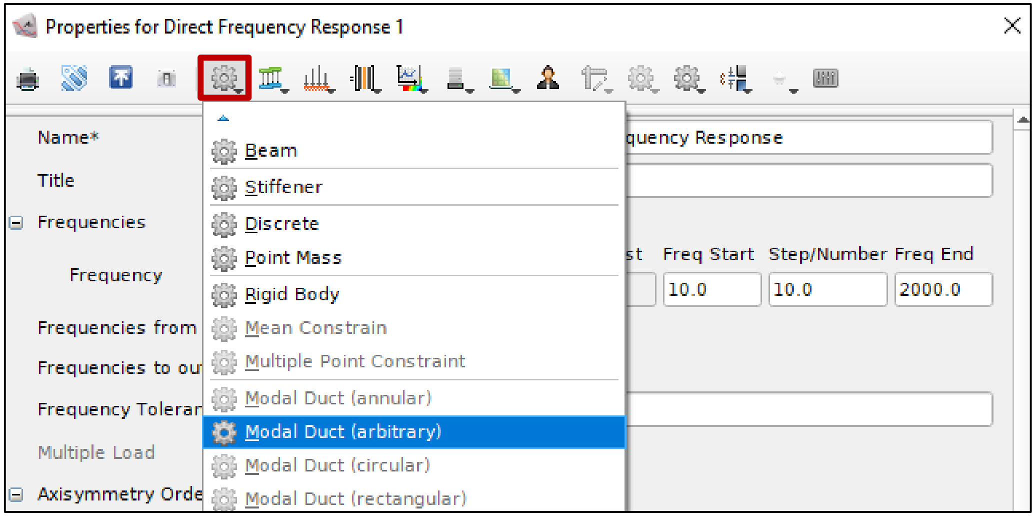
Power Quantities of Duct Modes

- In the PLT result, a duct modal basis contains two power quantities.
  – Incident power: power along the positive direction (indicated by the thick arrow) of the duct mode
  – Transmitted power: power opposite the positive direction

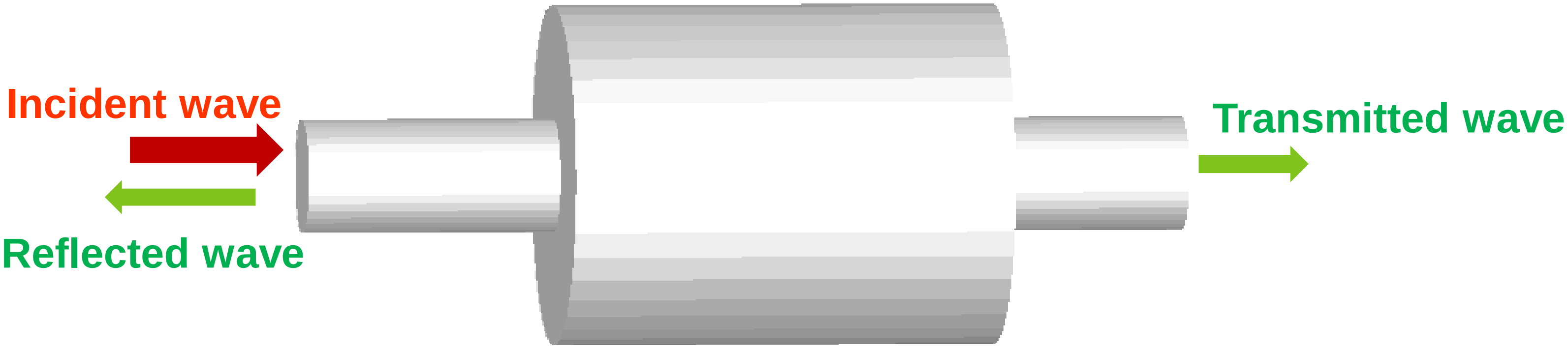


Create a Modal Duct Component for the Inlet

- Add a Modal Duct component



- Incident wave is injected through this inlet modal duct component
- Reflected wave should be free to go through the inlet modal duct component



a. Set duct name: Inlet

b. Choose material: Air 1

c. Set cross section parameters:

Center at [0,0]

Axes: [0,1],[1,0]

d. Cross section surface:

0.007854 $(\pi \cdot 0.05^2)$

e. Create incident plane mode

Direction: 1

Format: Amplitude

Value: 1

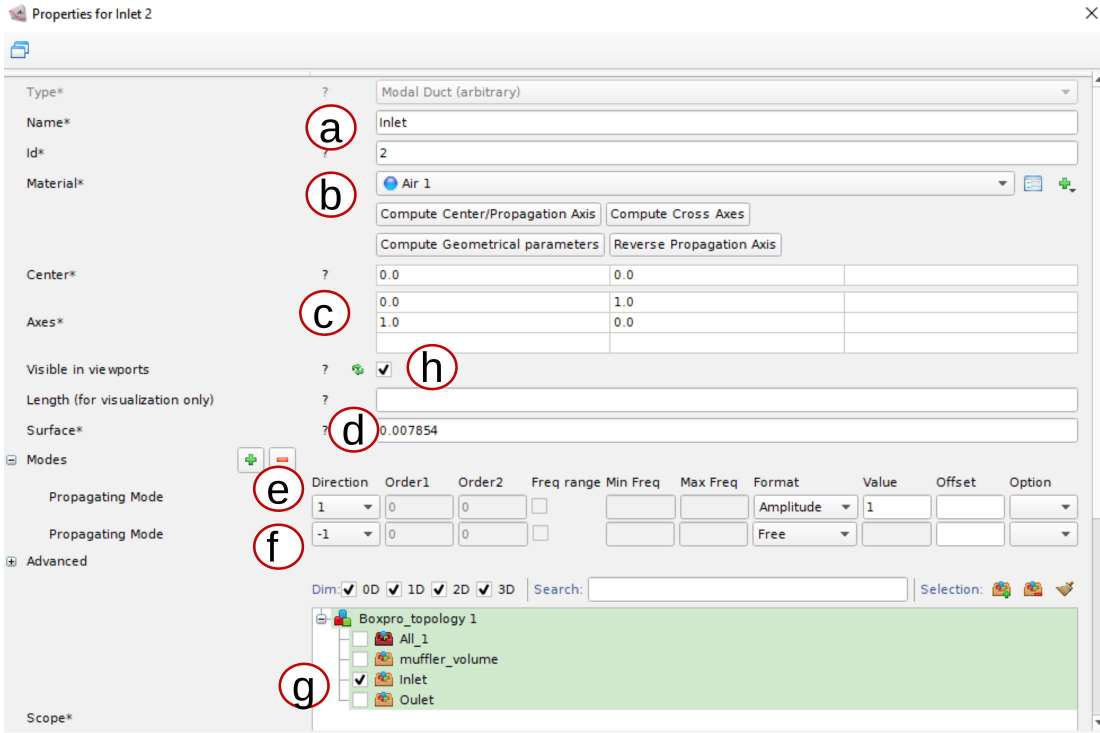
f. Create anechoic condition for reflected wave

Direction: -1

Format: Free

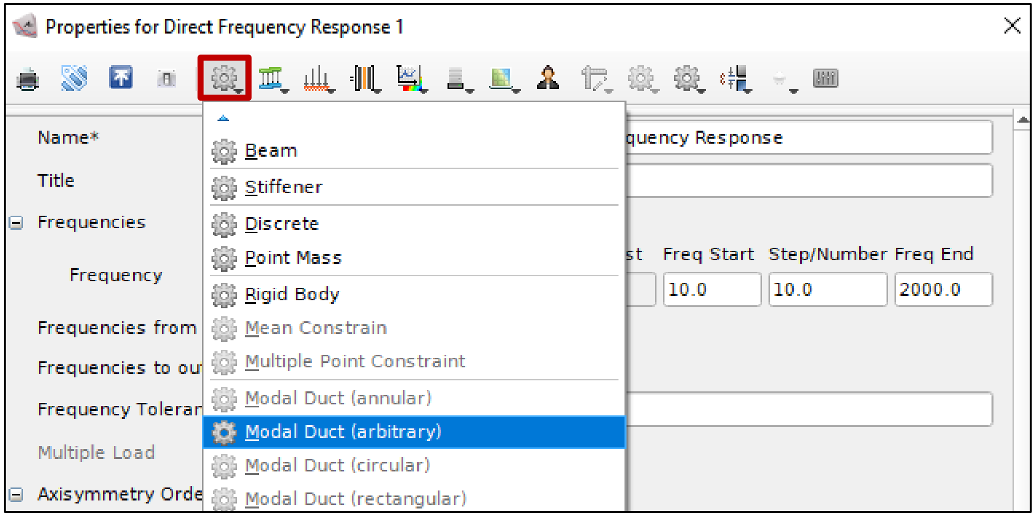
g. Select the domain “inlet”

h. Click “View Geometry”

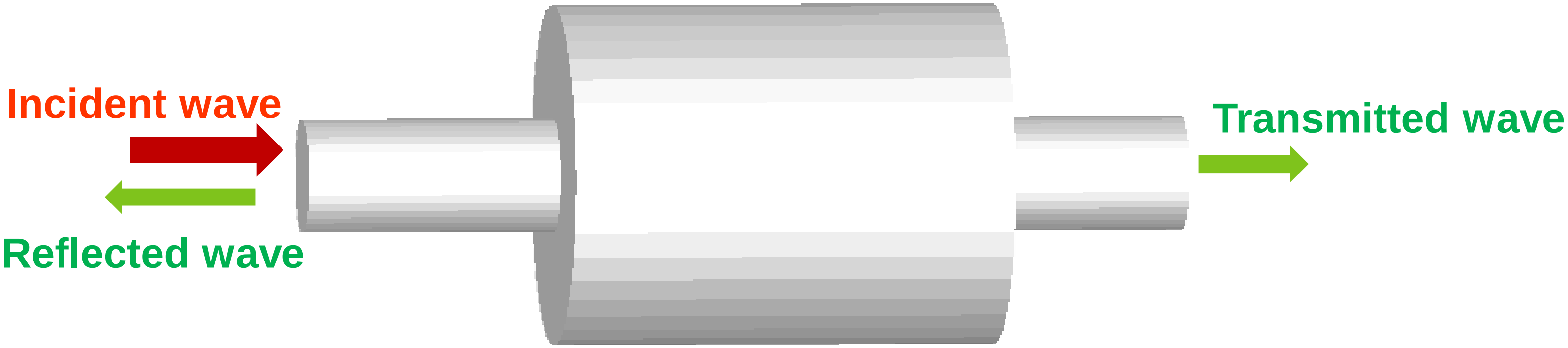


Create a Modal Duct Component for the Outlet

- Add a Modal Duct component



- Transmitted wave should be free to go through the outlet modal duct component



a. Set duct name: outlet

b. Choose material: air

c. Set cross section parameters:

   Center at [0, 0.7]

   Axes: [0,1] , [1,0]

d. Cross section surface:

   0.007854 $(\pi \cdot 0.05^2)$

e. Create anechoic condition for reflected wave

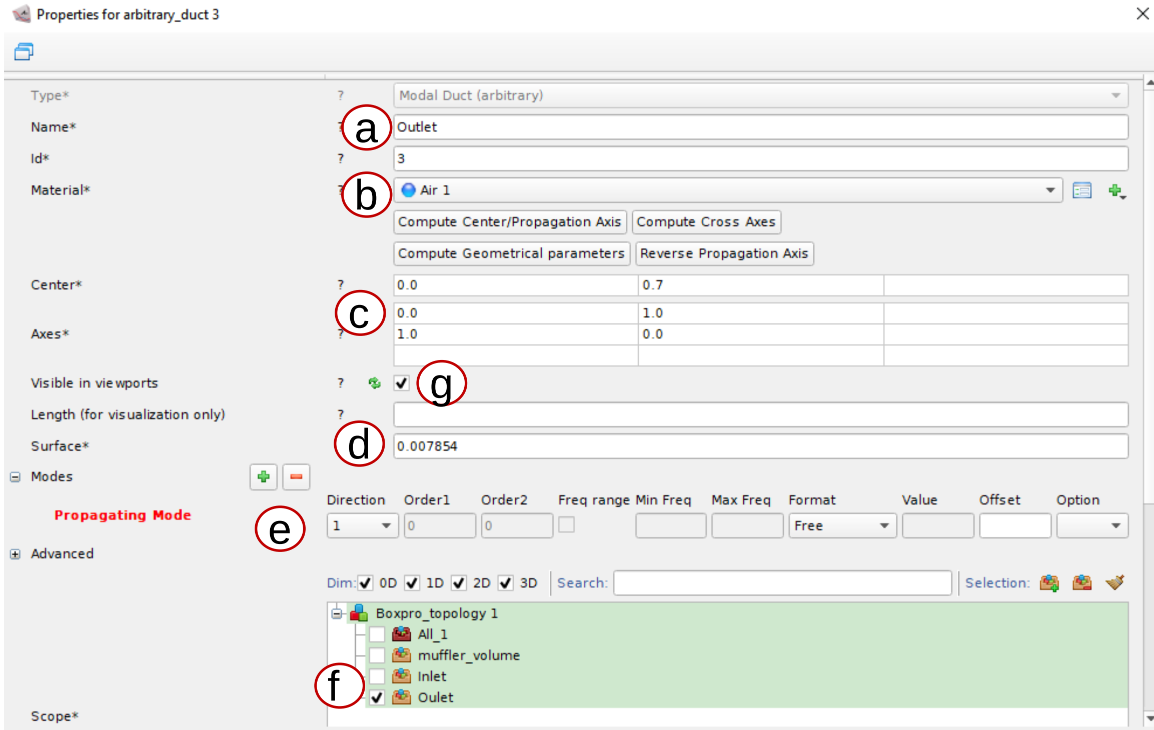
   Direction: 1

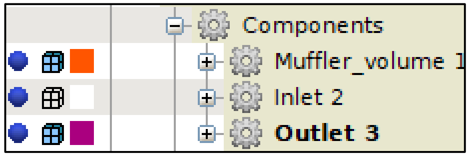
   Format: Free

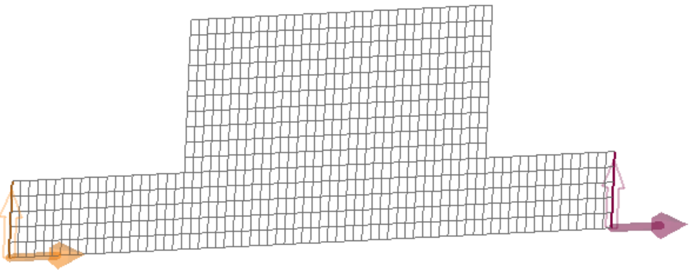
f. Select the domain “outlet”

g. Click “View Geometry”

h. Adjust the components visualization



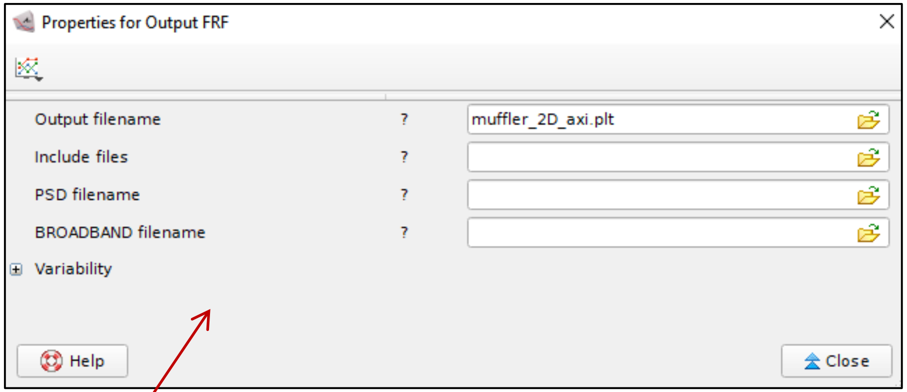


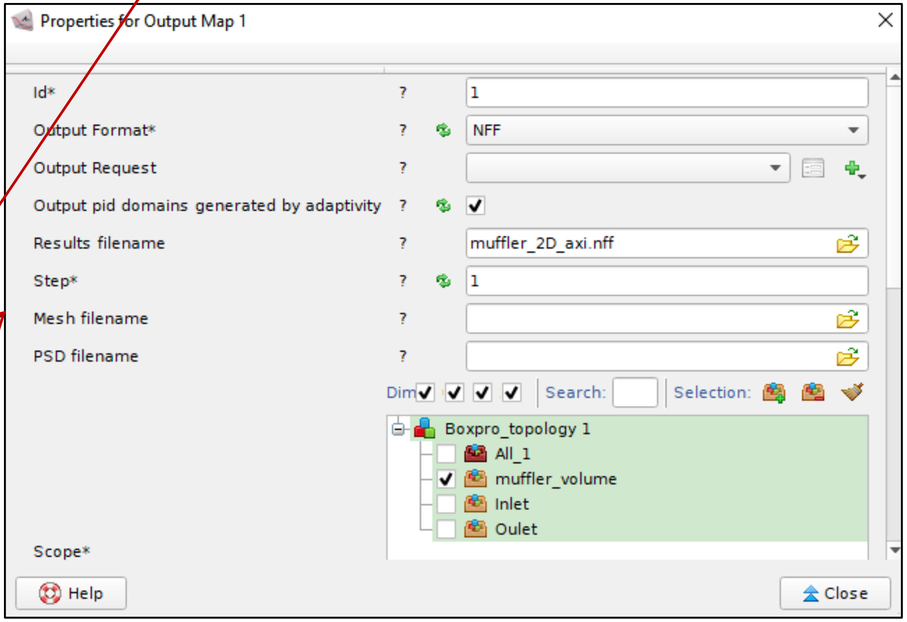


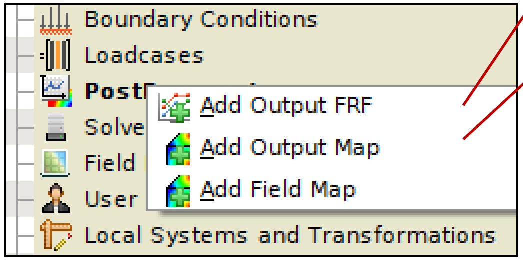
Local duct systems

Create Post-Processing Requests

- Create an “Output FRF”
  - Right Click on *Postprocessing*
  - Select *Add Output FRF*
  - Specify PLT file name: *muffler_2D_axi.plt*
- Create an “Output Map”
  - Right Click on *Postprocessing*
  - Select *Add Output Map*
  - Choose “NFF” as map output format
  - NFF name: *muffler_2D_axi.nff*
  - Type “1” in Step for map results

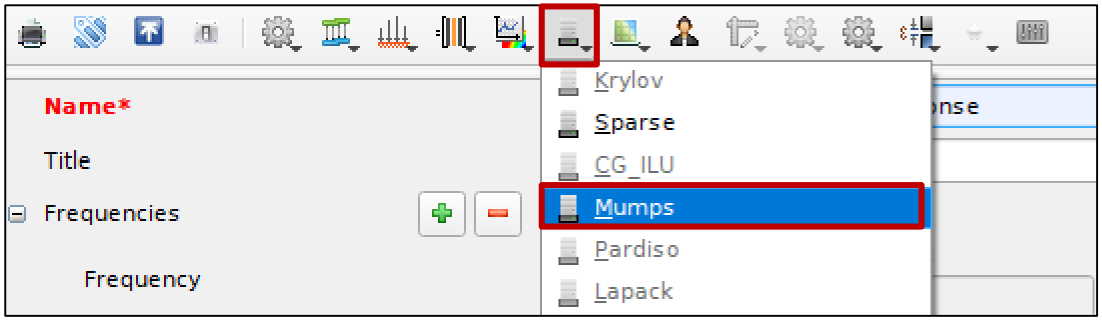






Specify the Solver

- Define the solver of the analysis



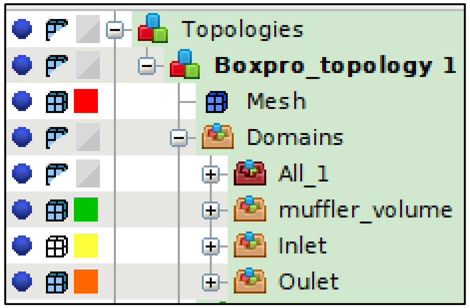
- Set the MUMPS solver

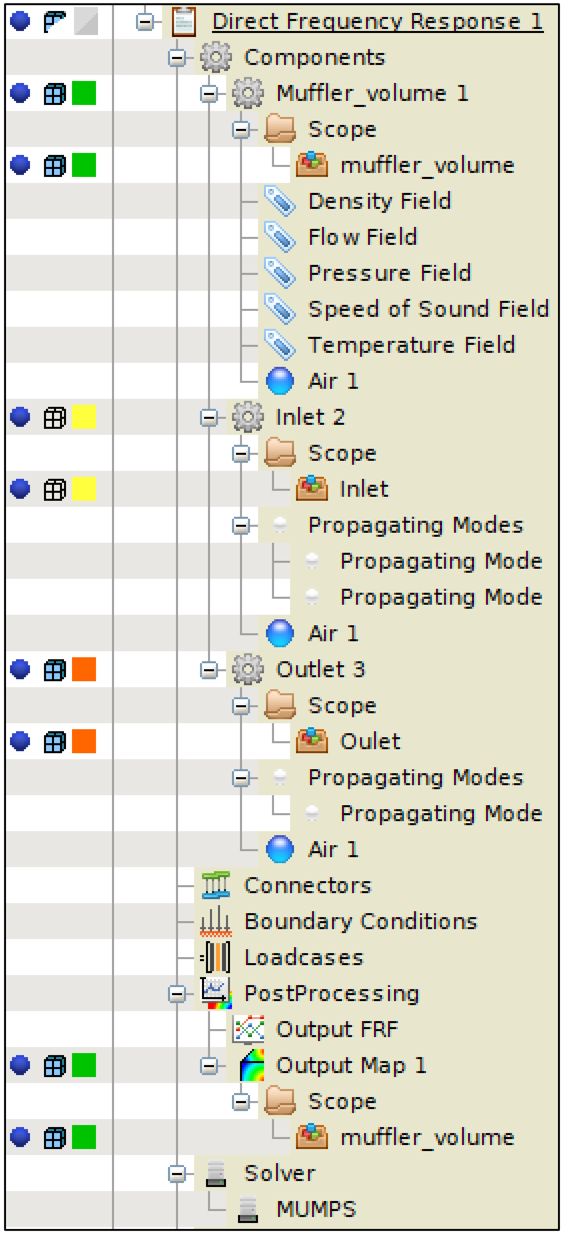
- Close the pop-up window of MUMPS

- Close the properties window of the Direct Frequency Response

Check the Analysis

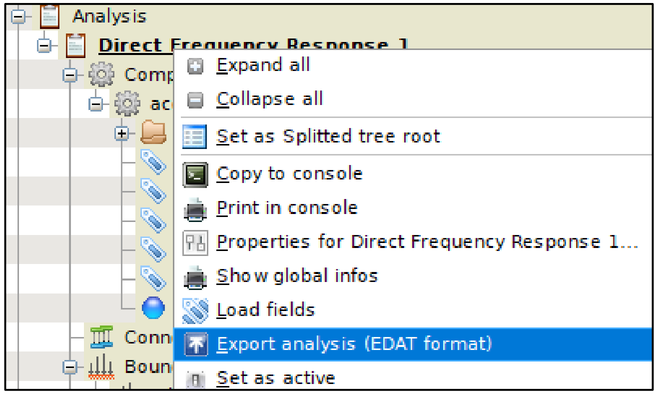
- The analysis is now completely defined
- All the parts of the analysis are available and editable on the data tree panel
- Check if the analysis tree is identical to the one shown here



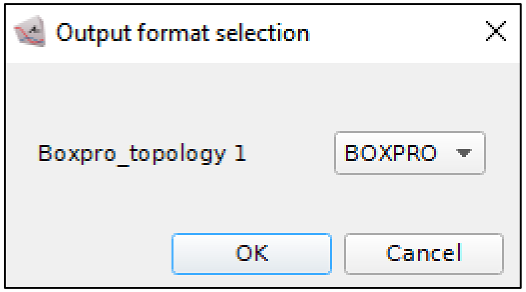


Export the Analysis File

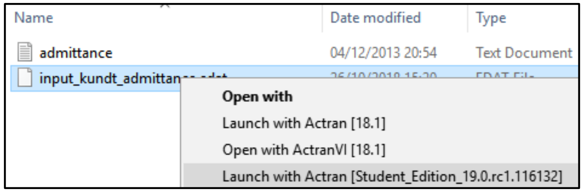
- The analysis can be exported in the EDAT Actran input file
- Right click on the *Direct Frequency Response*, and choose *Export analysis (EDAT format)*



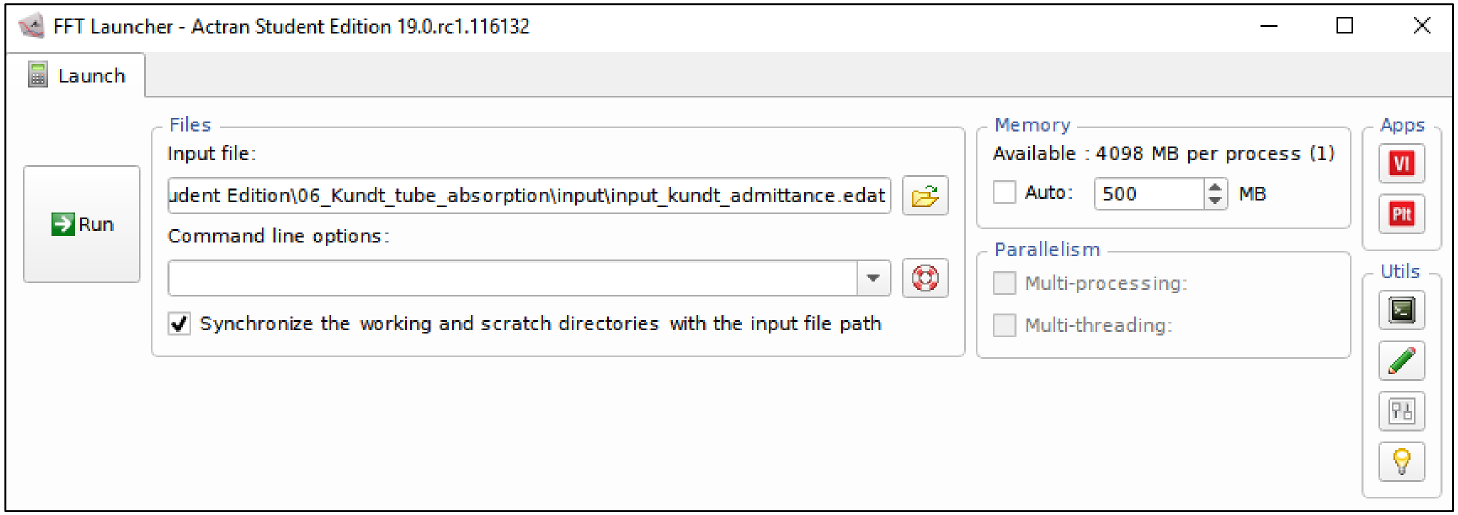
- The topology was created with BOXPRO, the analysis may be exported in two different formats:
  – The *BOXPRO* format: the mesh is written in the EDAT file the same way as in the topology definition. Nodes coordinates and elements are created at the beginning of the analysis
  – The *ACTRAN* format: the mesh is explicitly written in the EDAT file
- Select BOXPRO as the Output format and name the input file “input_muffler.edat”



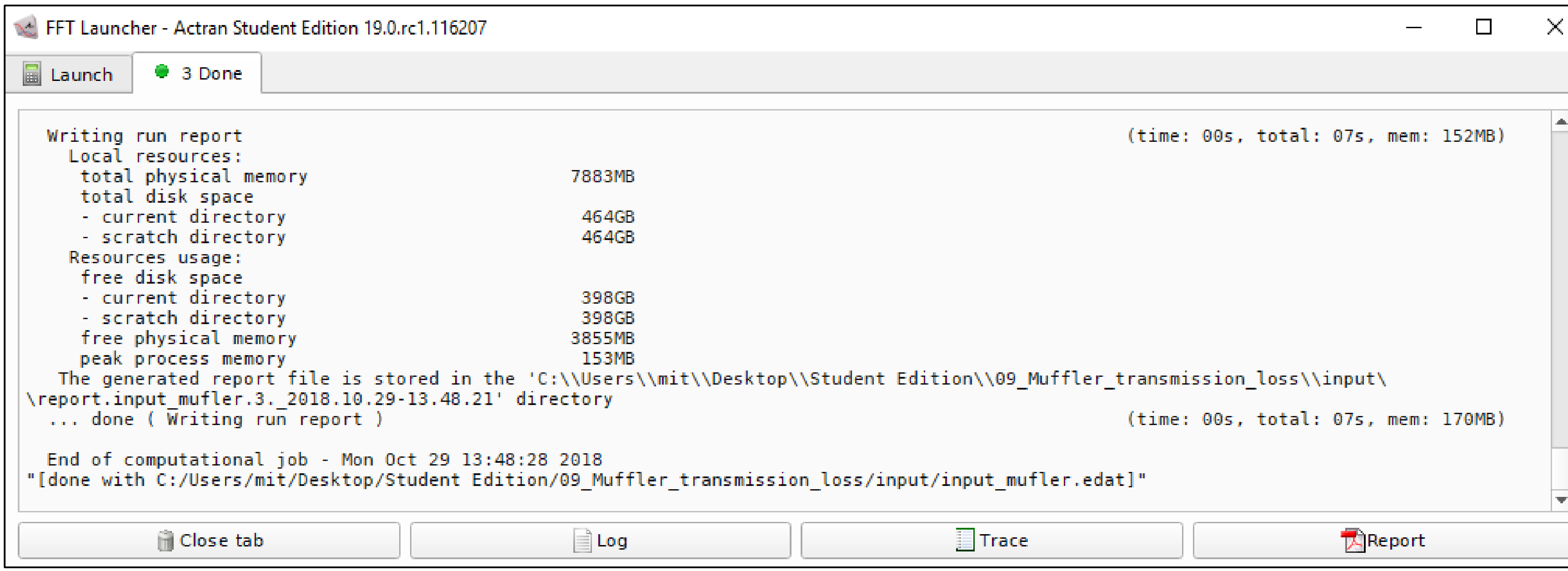
Launch Actran Analysis



- Launch the computation:
  - Open the FFT Launcher by right clicking on the *input_muffler.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 computation job” indicates the computation has finished**



- **Close the Launcher window**

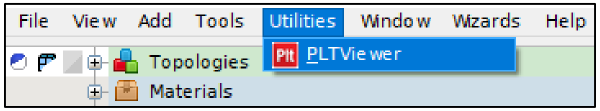
Post-processing

Compute the transmission loss in PLTViewer
Visualize pressure maps in ActranVI

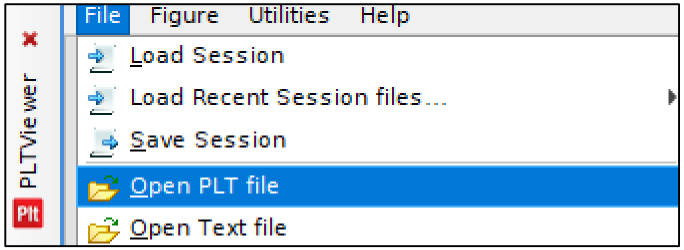
Open PLTViewer and import field point results

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

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

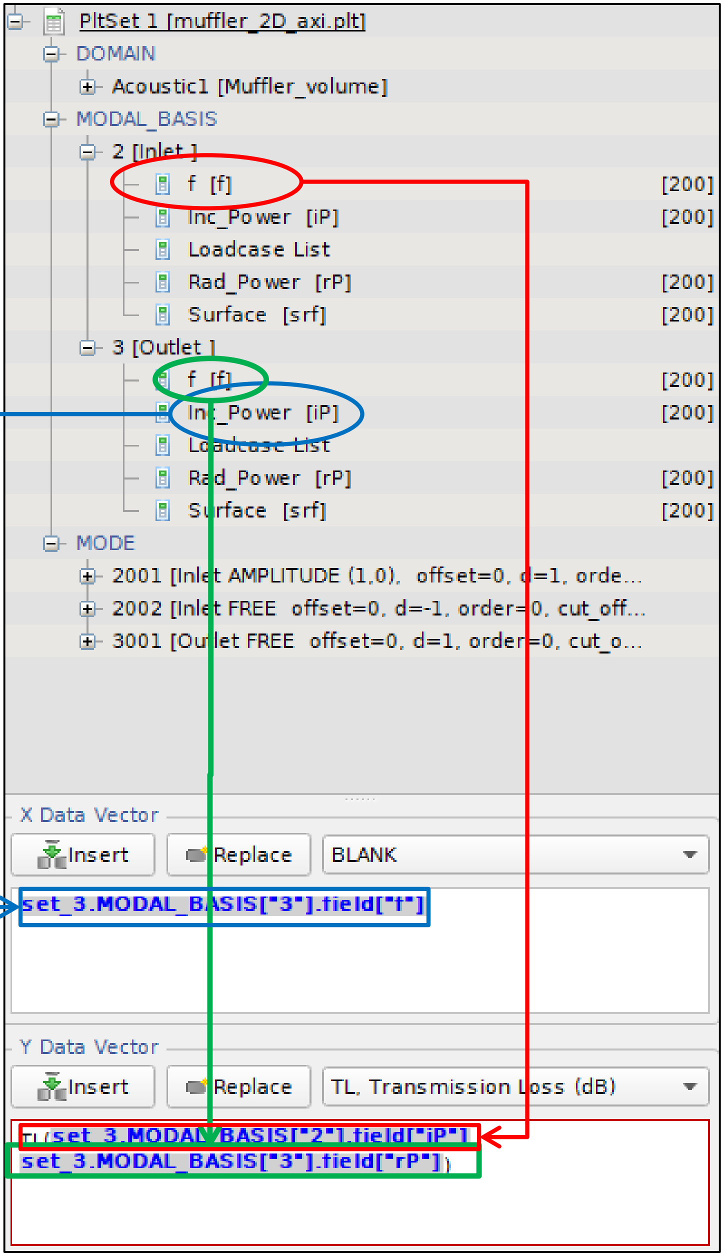


- Import the file *muffler_2D_axi.plt*
  – Under *File* select *Open PLT file*
  – Select the file *muffler_2D_axi.plt*



Plot the acoustic pressure

- The transmission loss (TL) index is an indicator of the acoustic efficiency of the muffler
- Click on insert in X data vector sub-window
- Choose "TL" for the Y Data vector and click on "Insert" in this sub-window
- Drag and drop the selected outputs at the appropriate locations
  - Inc-Power of “inlet” ➔ “incident” in the TL operator
  - Inc-Power of “outlet” ➔ “transmitted” in the TL operator

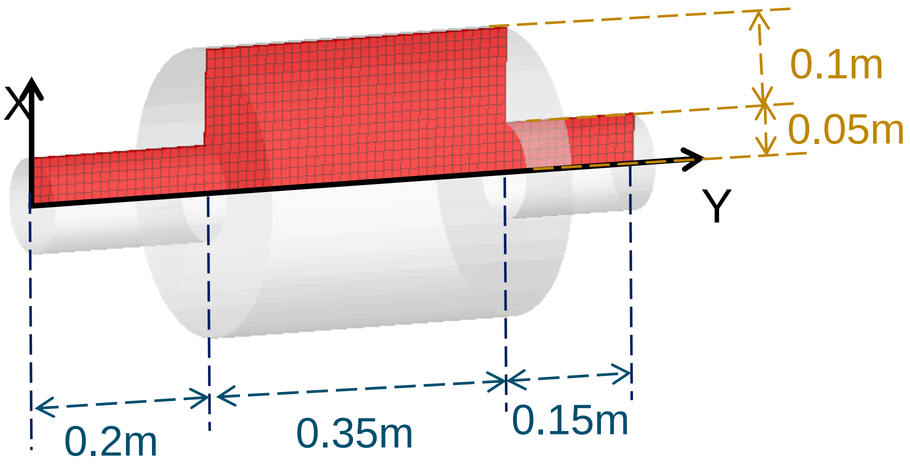


Comparison with Analytical Solution

- For an expansion chamber, the analytical TL can be calculated using the equation

$$TL = 10\log\left[1 + \left(\frac{m^2-1}{2m}\sin kl\right)^2\right]$$

  - m: cross section area ratio between expansion chamber and inlet (outlet) tube $(0.15 / 0.05)^2 = 9$
  - l: length of expansion part of the chamber = 0.35
  - k: wave number $2 \cdot \pi \cdot$ freq / speed of sound



- This analytical solution is calculated with the assumption of plane wave propagation in the muffler
- The analytical solution has been calculated and is available in the file *TL_analytical.txt*

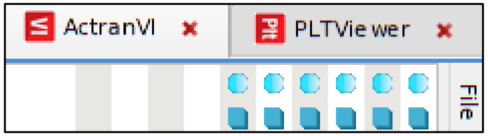
Comparison with Analytical Solution

- Plot the analytical solution
  - Open the results file
    File menu ➔ Open Text file ➔ Choose file “TL_analytical.txt”
  - Add a new Function: Function 2
  - In Function 2, choose “BLANK” for Y Data vector
  - Click “Replace” under Y Data vector
  - Plot the analytical curve

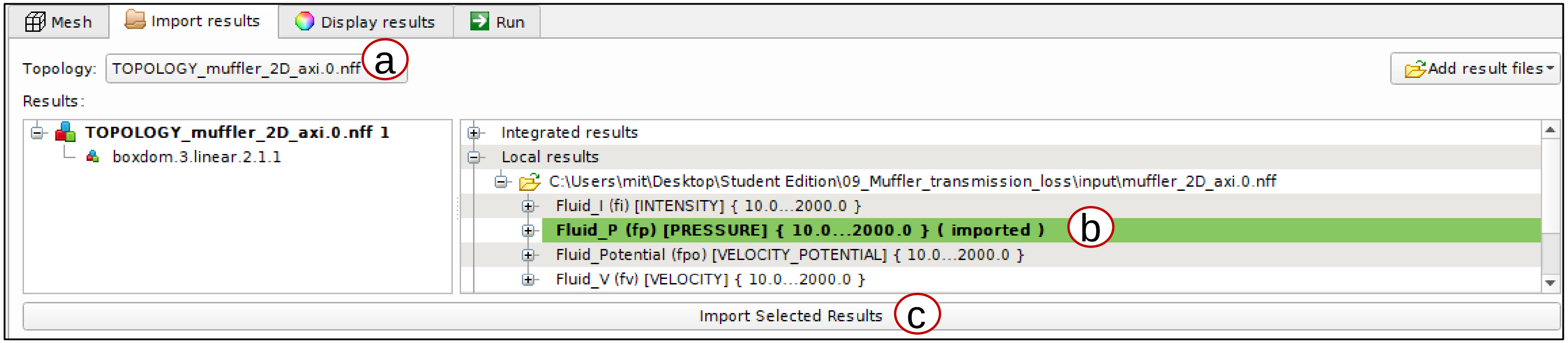
- Adjust the curves parameters to plot the analytical solution with a dashed red line

Visualize Field Map in ActranVI

- Switch back to tab ActranVI:

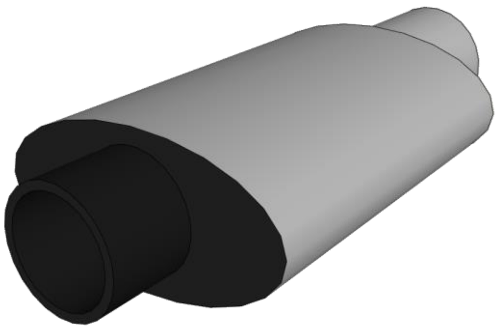


- Import the pressure results on the field mesh
  a) Import the NFF: muffler_2D_axi.0.nff (the ".0" indicates that the calculation was run with axisymmetrical order 0)
  b) Choose Fluid_P(fp) for acoustic pressure
  c) Import Selection

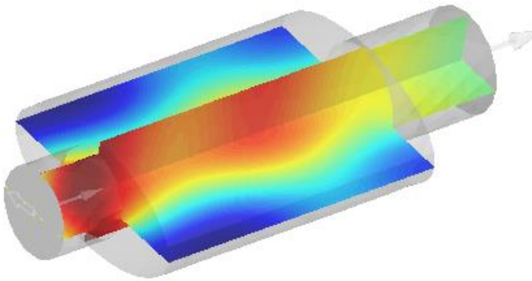


Going Further – Modeling Muffler in 3D

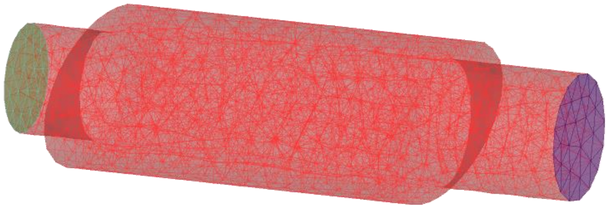
- A muffler model in 3D is contained in the folder “3D muffler”
- Import the analysis file in ActranVI
  - visualize the mesh
  - view the model set up
- Run the analysis
- Perform post processing in ActranVI
  - Plot TL curve
  - Display pressure maps



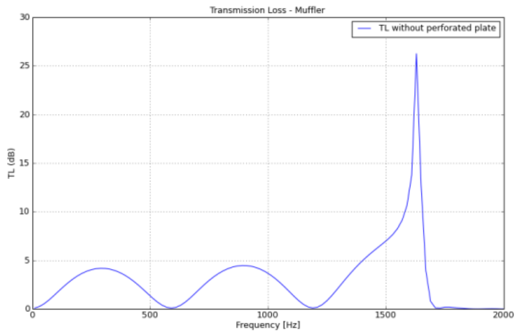
Geometry



Pressure map with cut planes



Mesh



Transmission loss