

Transmission Loss of a Muffler

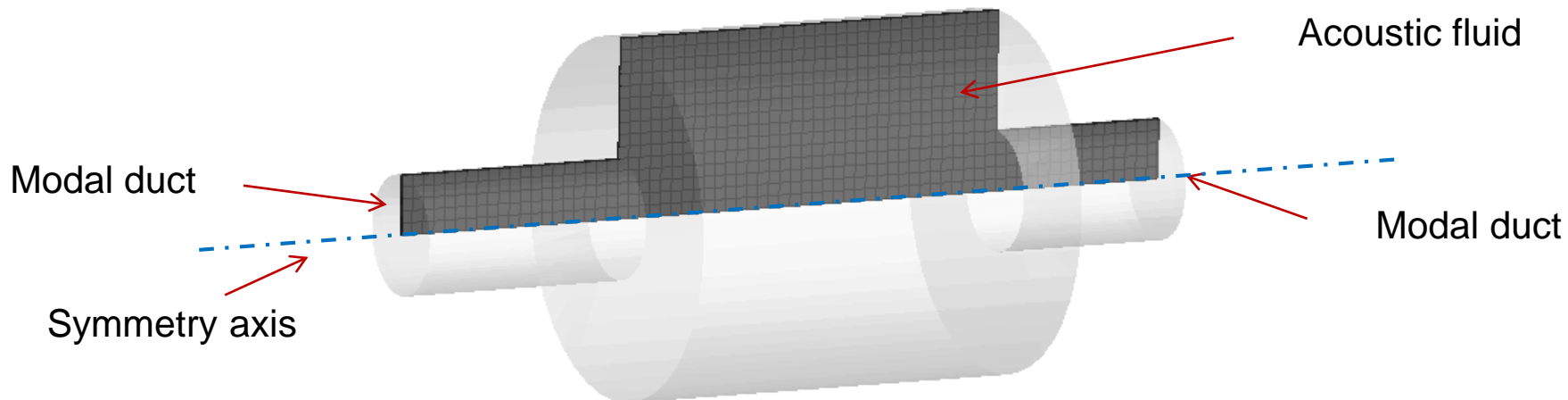
Actran Student Edition Tutorial

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

$$TL = 10 \cdot \log_{10}(W_{inc}/W_{trans})$$

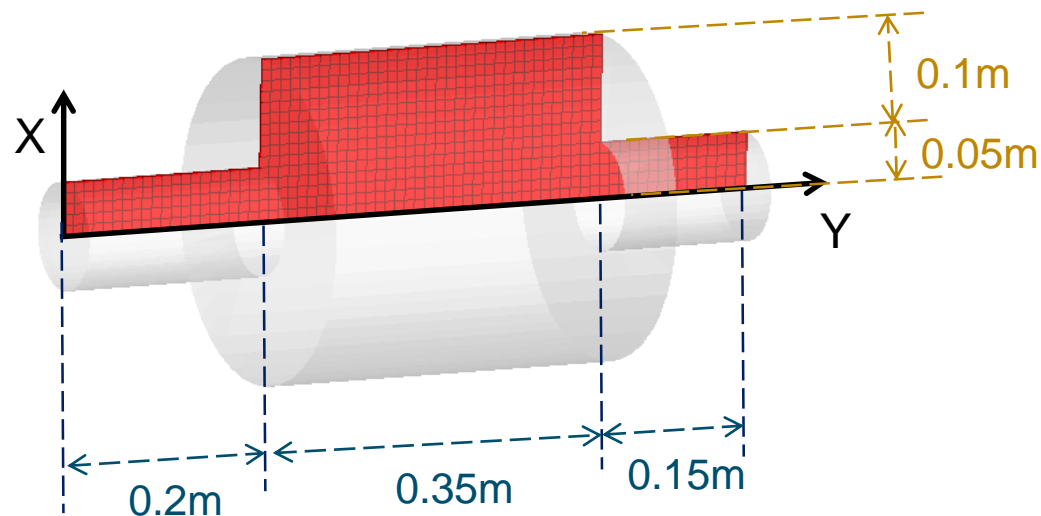
- Acoustic anechoic condition is applied
 - at outlet for the transmitted wave
 - at inlet for the reflected wave

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
 $2 \cdot \pi \cdot \text{freq} / \text{speed of sound}$



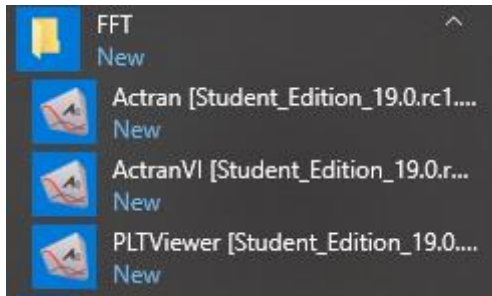
- This analytical solution is calculated with the assumption of plane wave propagation in the muffler

Workshop Pre-Processing

Direct Frequency Analysis

Start ActranVI

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

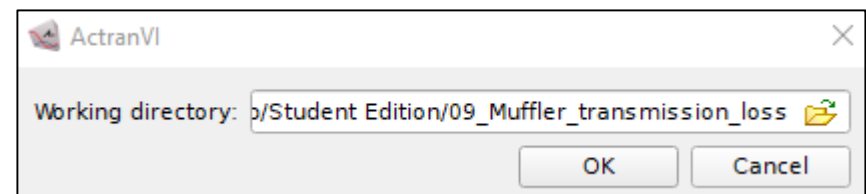
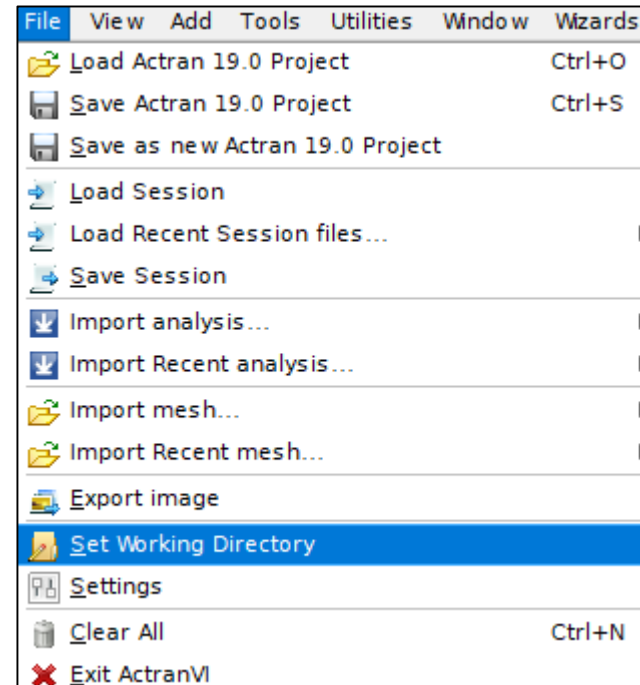


(Windows Start Menu)



Set the Working Directory

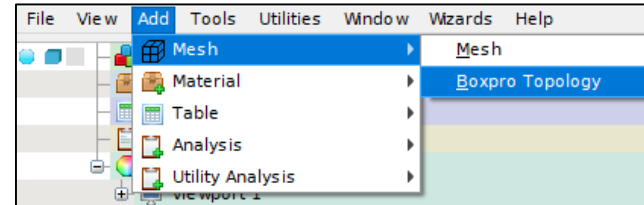
- The working directory is the default directory where all the files are output
- Click on :
 - File → Set Working Directory...
- Select the workshop directory as the working directory



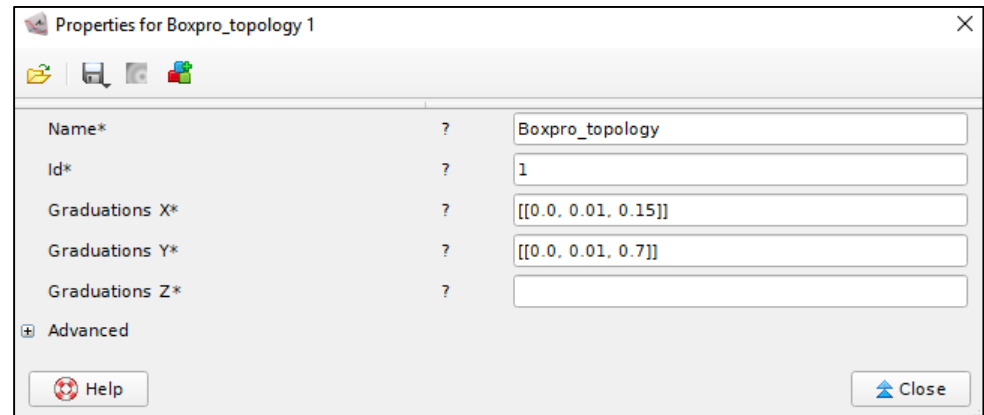
* Important: The working directory path should not contain any space or special character

Create the mesh using BOXPRO

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



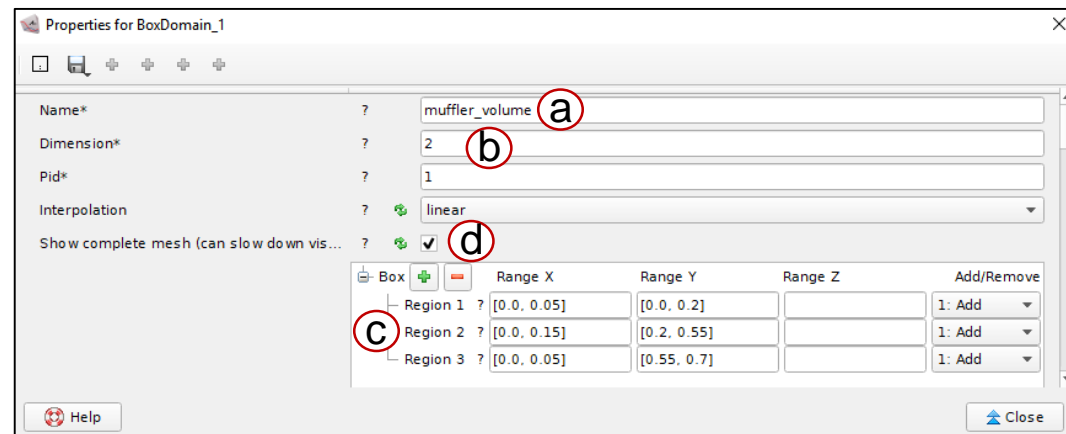
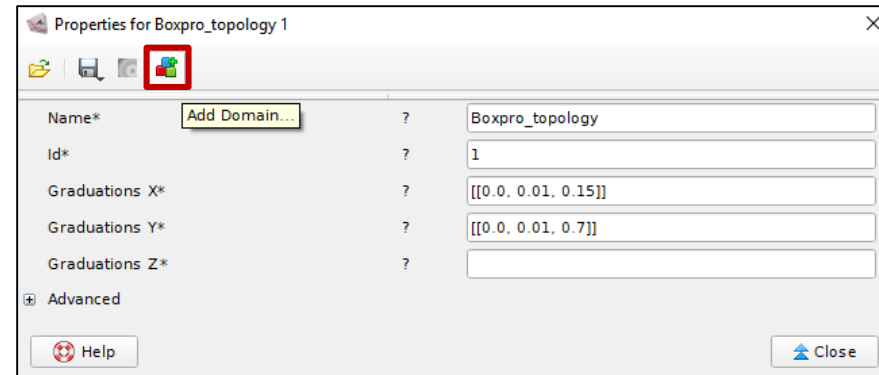
- Enter the X,Y, Z dimensions of the Boxpro topology
 - X: $[[0, 0.01, 0.15]]$
 - Y: $[[0, 0.01, 0.7]]$
 - Z: EMPTY (z=0 for entire mesh)



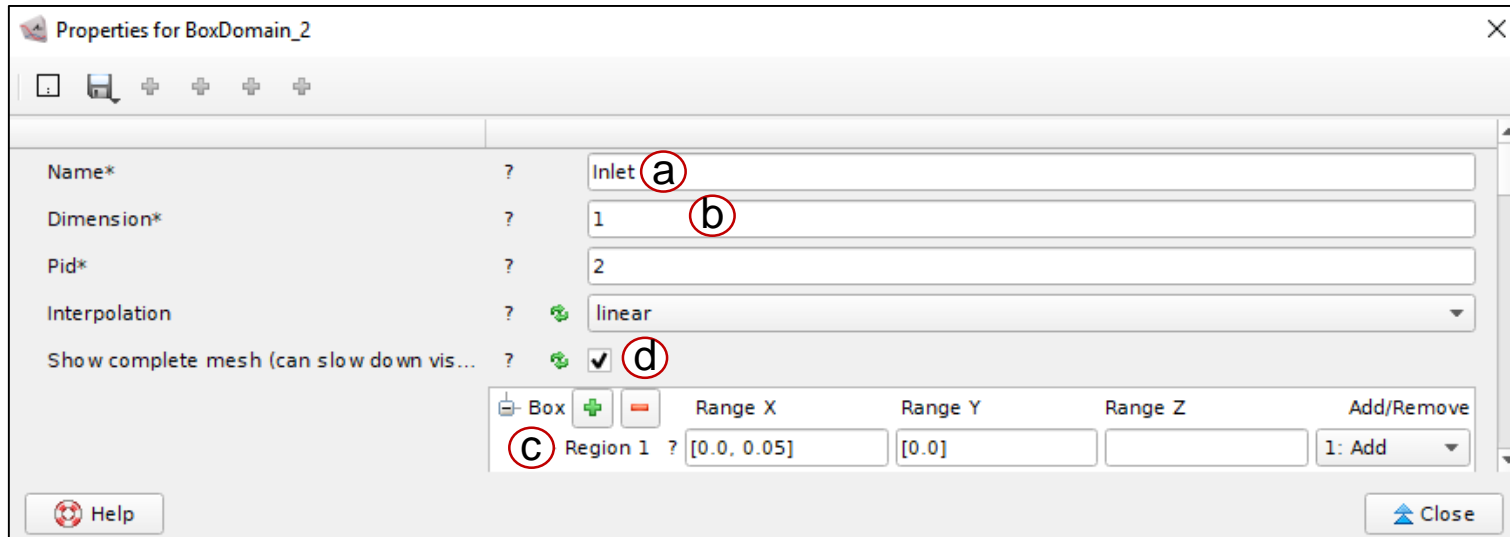
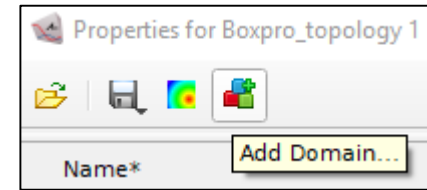
Create the mesh using BOXPRO

- 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

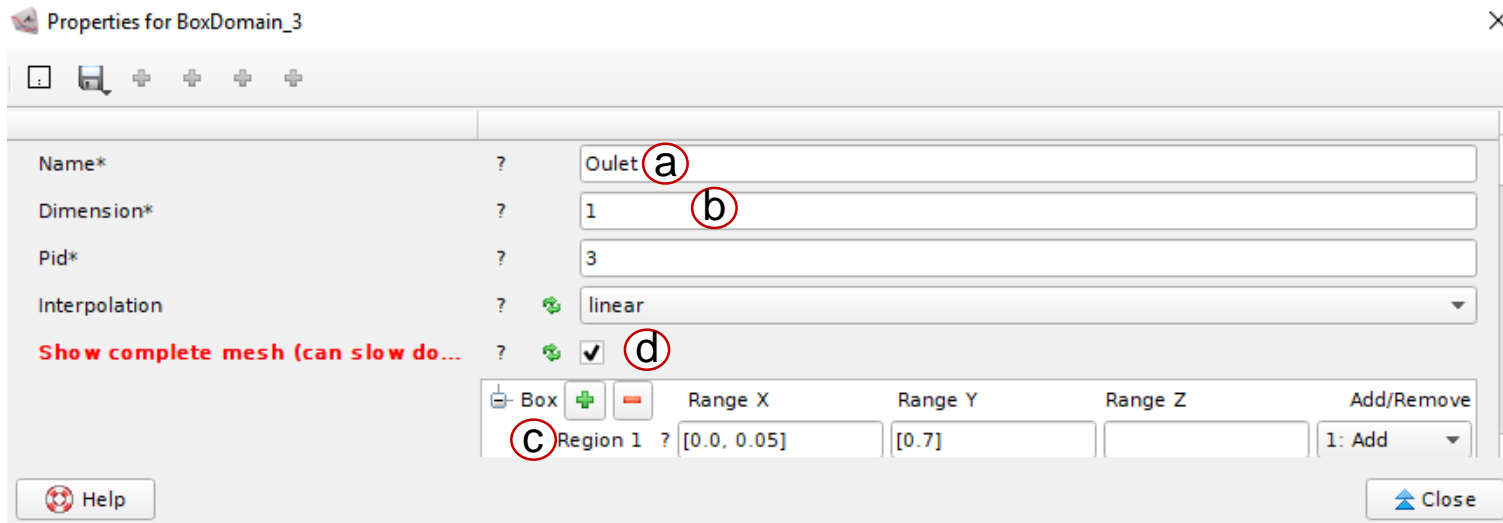
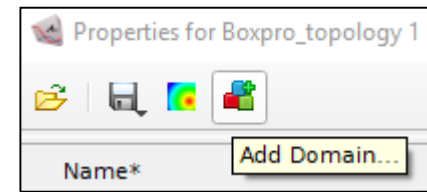


- Add a new domain for the Inlet
- Fill the properties of the Domain
 - a. Change domain Name to “Inlet”
 - b. Enter “1” in the Dimension of the domain
 - c. Create a region. X: 0, 0.05 Y: 0 Z: empty
 - d. Click on the check box of Show complete mesh



- The line supporting the inlet is now created

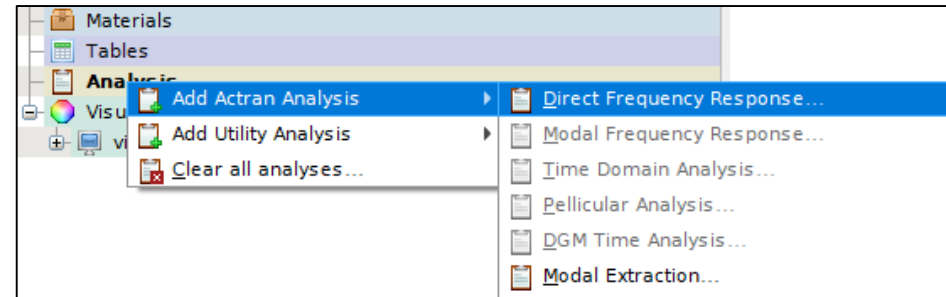
- Add a new domain for the Outlet
- Fill the properties of the Domain
 - a. Change domain Name to “Outlet”
 - b. Enter “1” in the Dimension of the domain
 - c. Create a region. X: 0, 0.05 Y: 0.7 Z: empty
 - d. Click on the check box of Show complete mesh



- The line supporting the outlet is now created

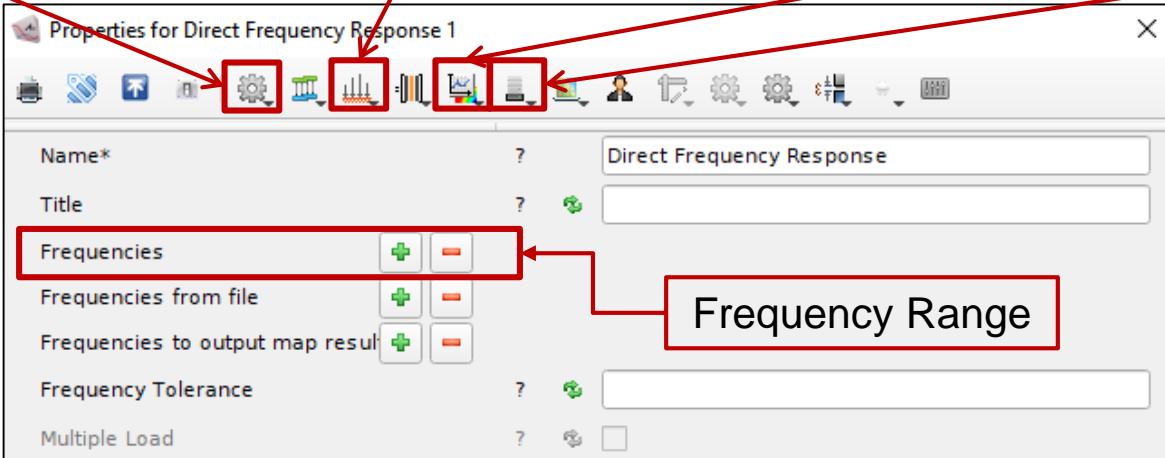
Create the Direct Frequency Response

- 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

Component Boundary Condition Post-Processing Solver



The screenshot shows the 'Properties for Direct Frequency Response 1' window. The window title is 'Properties for Direct Frequency Response 1'. The window contains a toolbar with icons for Component, Boundary Condition, Post-Processing, and Solver. Below the toolbar, there are several fields and controls:

- Name*: Direct Frequency Response
- Title: [Empty field]
- Frequencies: [Field with '+' and '-' buttons]
- Frequencies from file: [Field with '+' and '-' buttons]
- Frequencies to output map result: [Field with '+' and '-' buttons]
- Frequency Tolerance: [Field with '?' and refresh icon]
- Multiple Load: [Field with '?' and refresh icon]

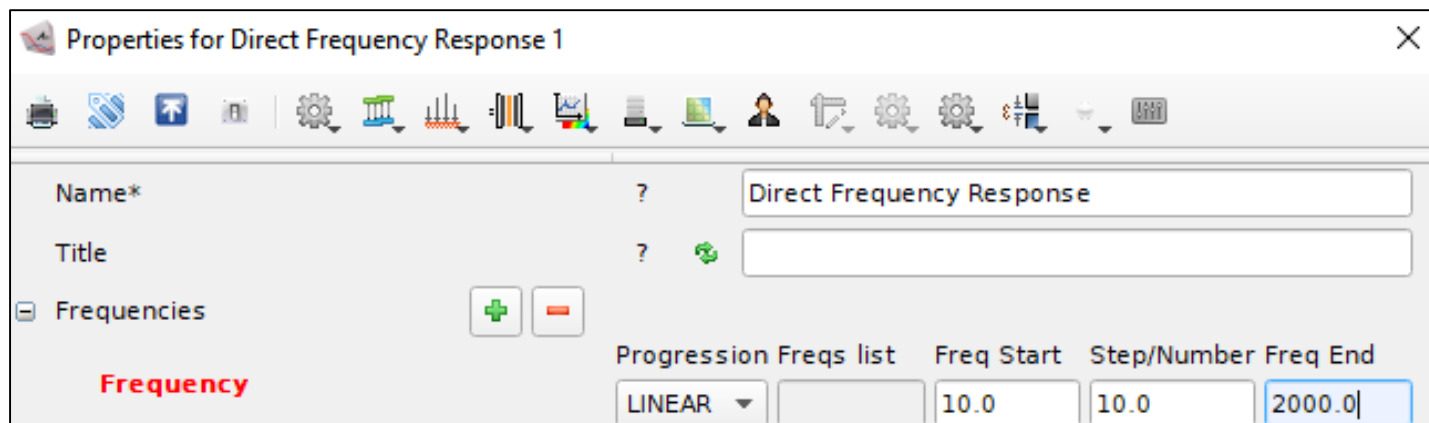
Red boxes and arrows highlight the following elements:

- Component: Points to the gear icon in the toolbar.
- Boundary Condition: Points to the boundary condition icon in the toolbar.
- Post-Processing: Points to the post-processing icon in the toolbar.
- Solver: Points to the solver icon in the toolbar.
- Frequency Range: Points to the 'Frequencies' field.

Specify the Frequency Range

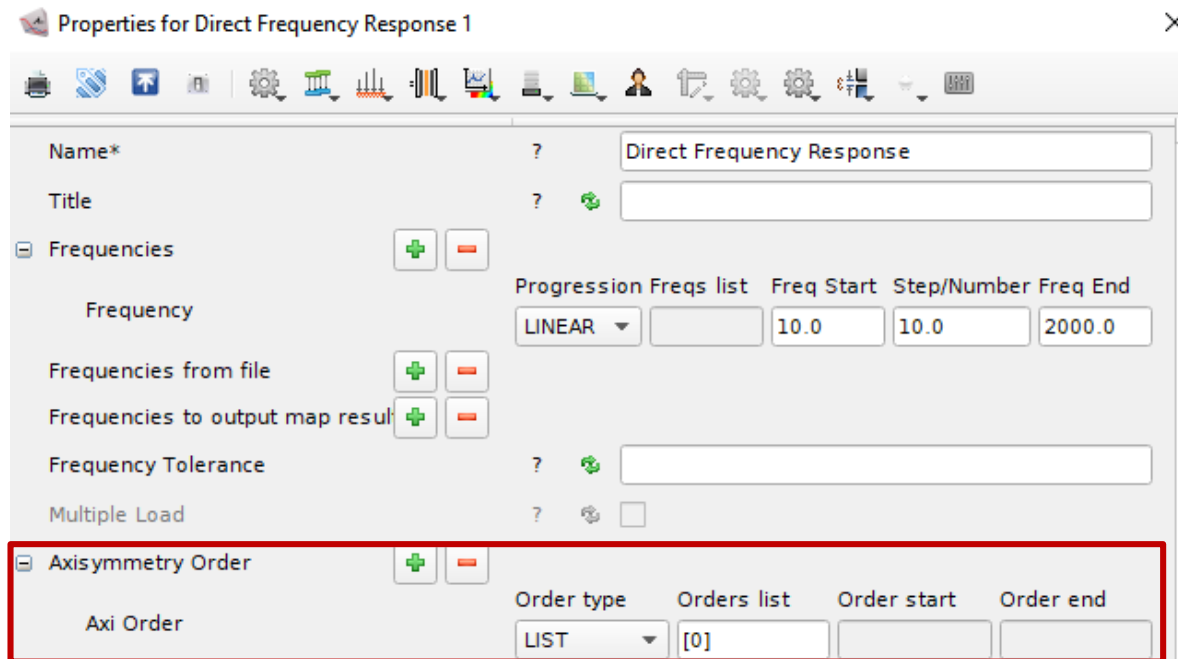
- The frequency range of computation is specified through the properties of the analysis
- The maximum frequency is driven by the smallest wave length :
 - For linear elements, a rule of thumb is to use 8 linear elements per wavelength to capture the acoustic fluctuation
 - The largest element length in the mesh is 10 mm
- This analysis is performed from 10Hz up to 2000 Hz with a step of 10 Hz

$$f_{max} = \frac{c}{\lambda_{min}} \text{ with } L_{max} = \frac{\lambda_{min}}{8} = 0.01m$$
$$\text{gives } f_{max} = \frac{340}{8 * 0.01} = 4250Hz$$



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

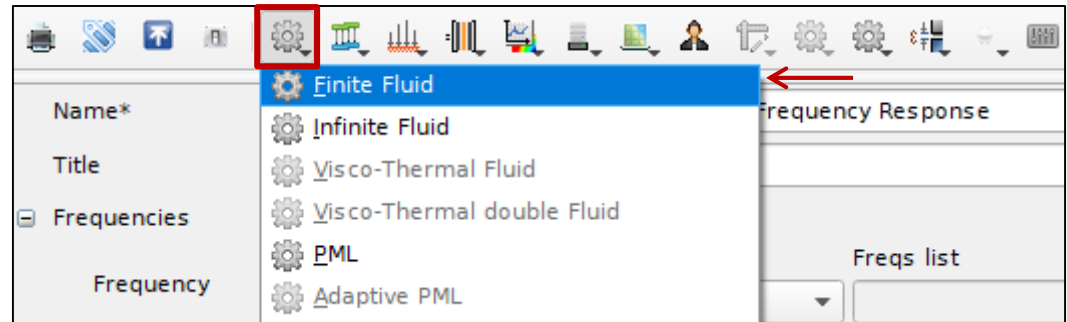


- By default, the Y axis is always the axis of revolution. The mesh of axisymmetric problems must locate in the right half ($X > 0$) of the XY plane

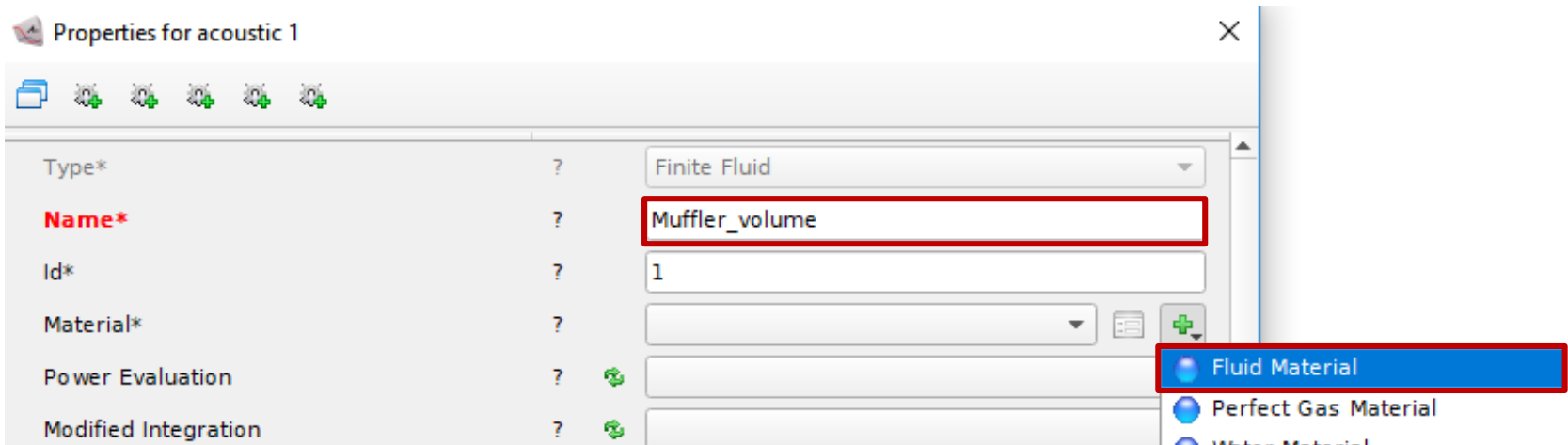
Create the Finite Fluid Component

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

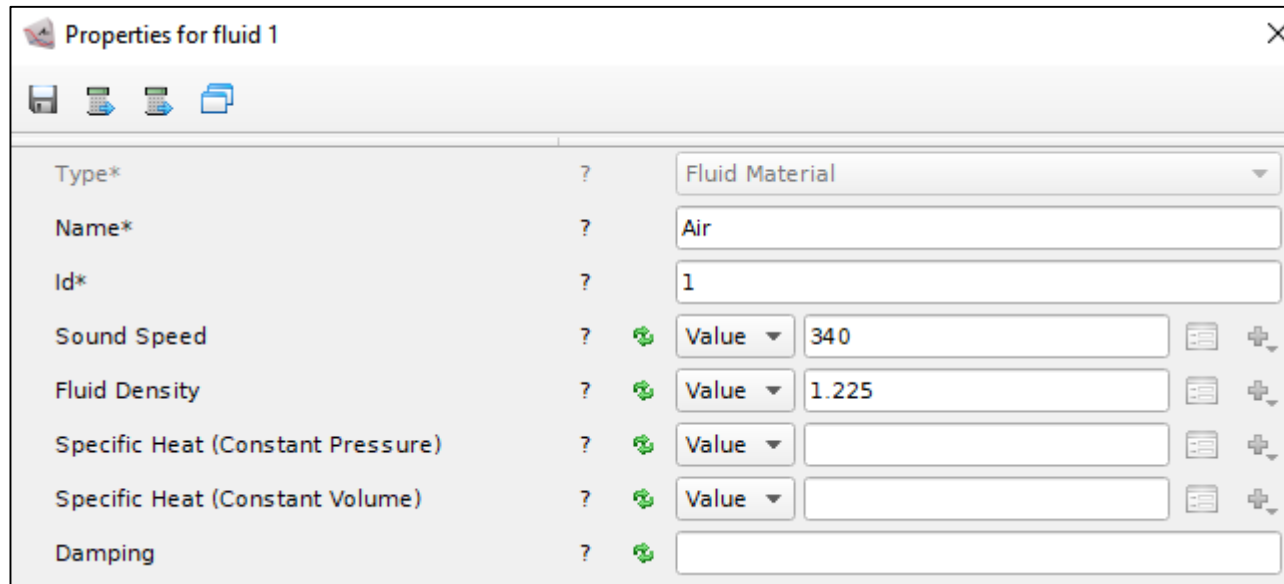


Create the Finite Fluid Component

2 – Set up the Fluid Material

- Name: *Air*
- Standard properties of air:
 - Speed of sound: 340 m/s
 - Density: 1.255 kg/m³

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



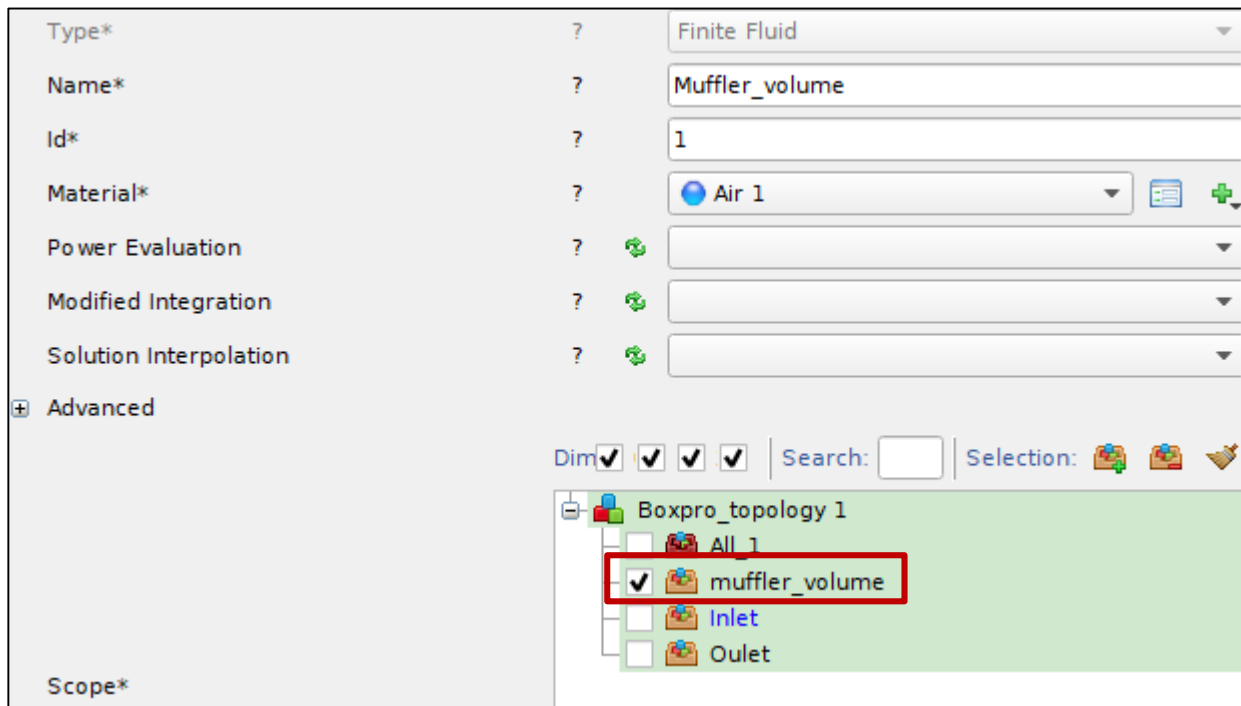
Property	Value
Type*	Fluid Material
Name*	Air
Id*	1
Sound Speed	340
Fluid Density	1.225
Specific Heat (Constant Pressure)	Value
Specific Heat (Constant Volume)	Value
Damping	

- 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



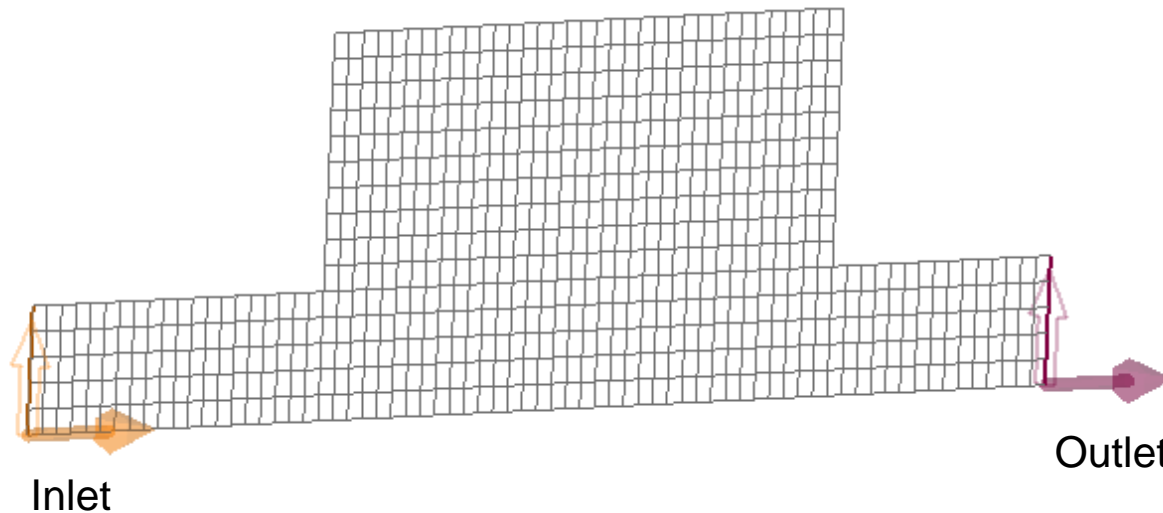
- Close the component properties window

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

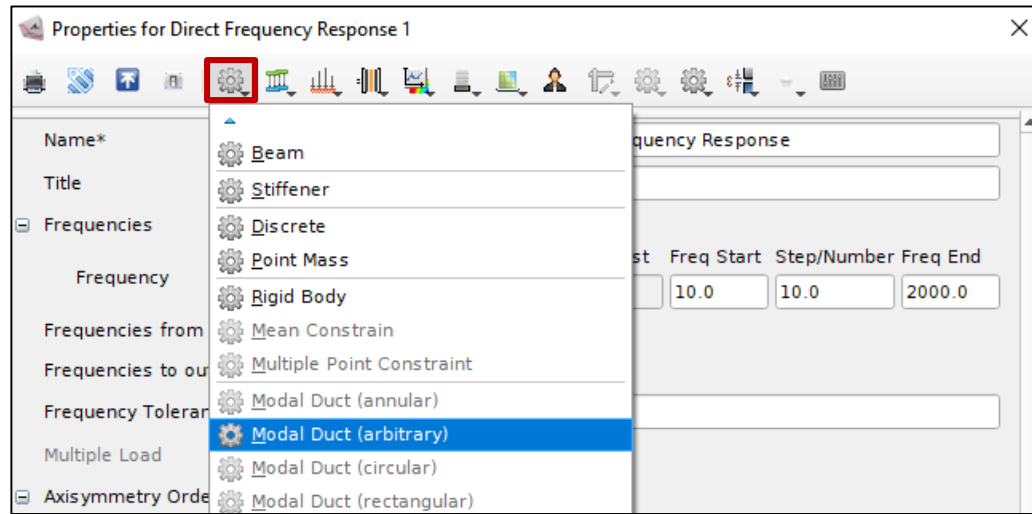
Power Quantities of Duct Modes

- 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

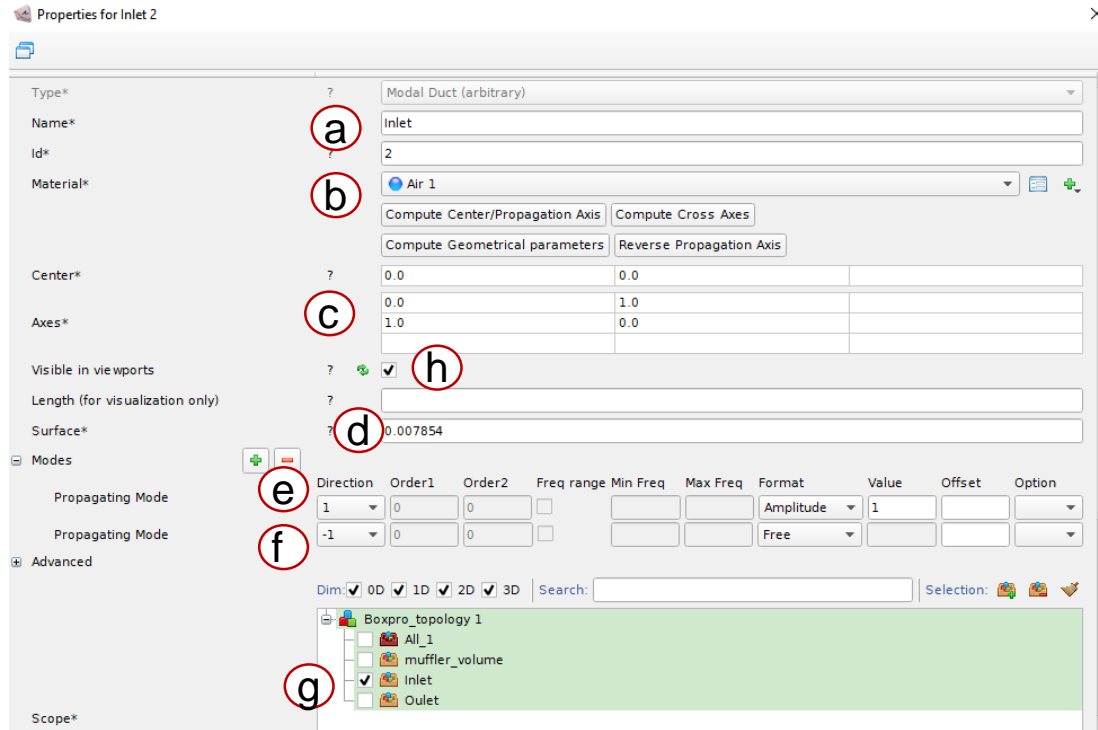
- Add a Modal Duct component



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

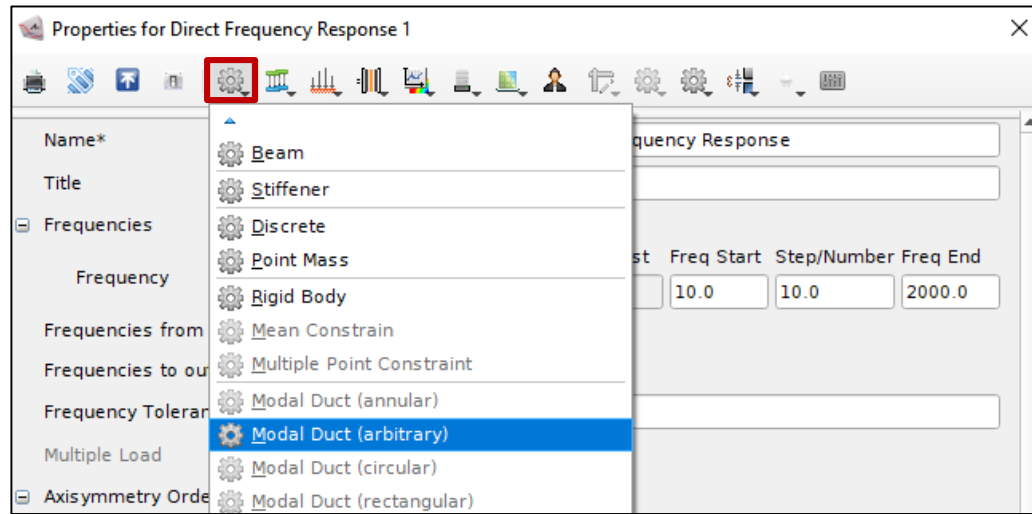


- a. Set duct name: Inlet
- b. Choose material: Air 1
- c. Set cross section parameters:
Center at [0,0]
Axes: [0,1],[1,0]
- d. Cross section surface:
 $0.007854 \quad (\pi \cdot 0.05^2)$
- e. Create incident plane mode
Direction: 1
Format: Amplitude
Value: 1
- f. Create anechoic condition for reflected wave
Direction: -1
Format: Free
- g. Select the domain "inlet"
- h. Click "View Geometry"



Create a Modal Duct Component for the Outlet

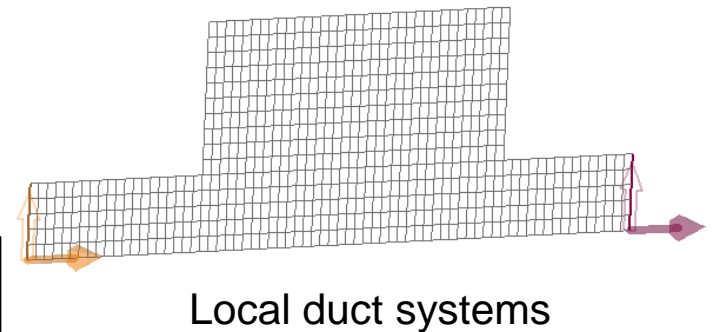
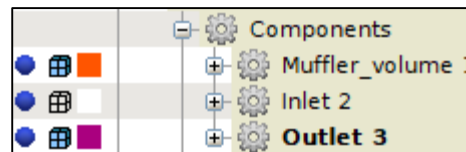
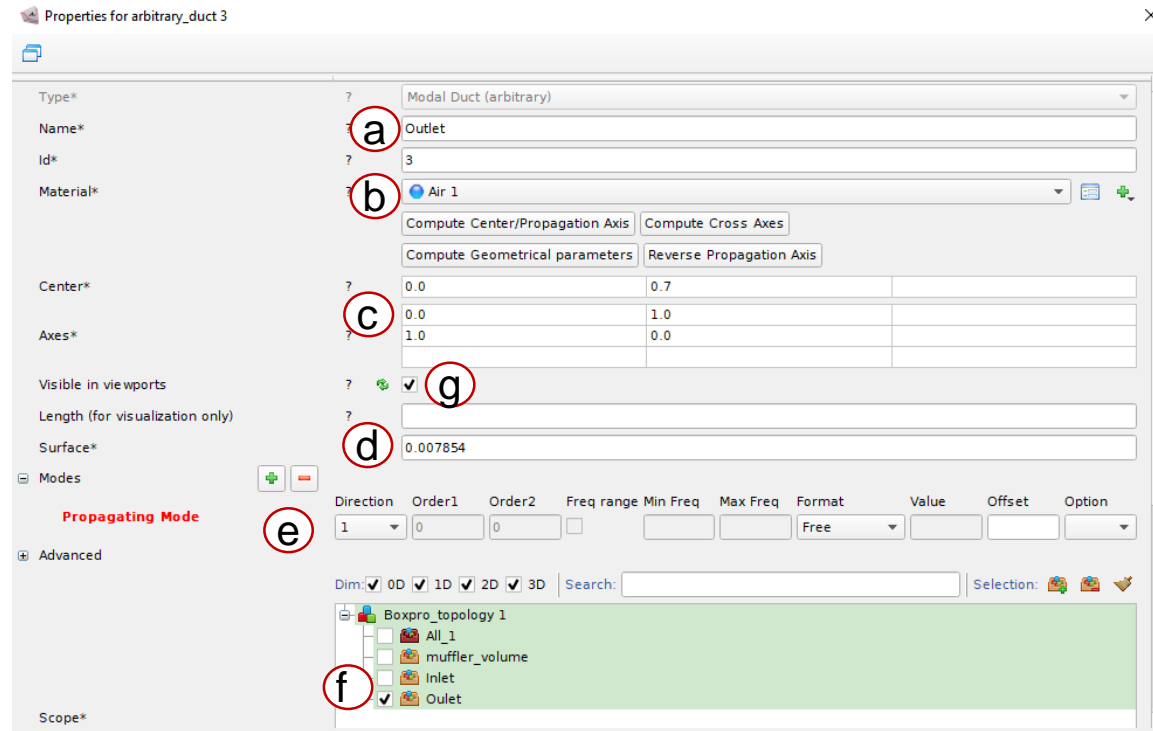
- Add a Modal Duct component



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

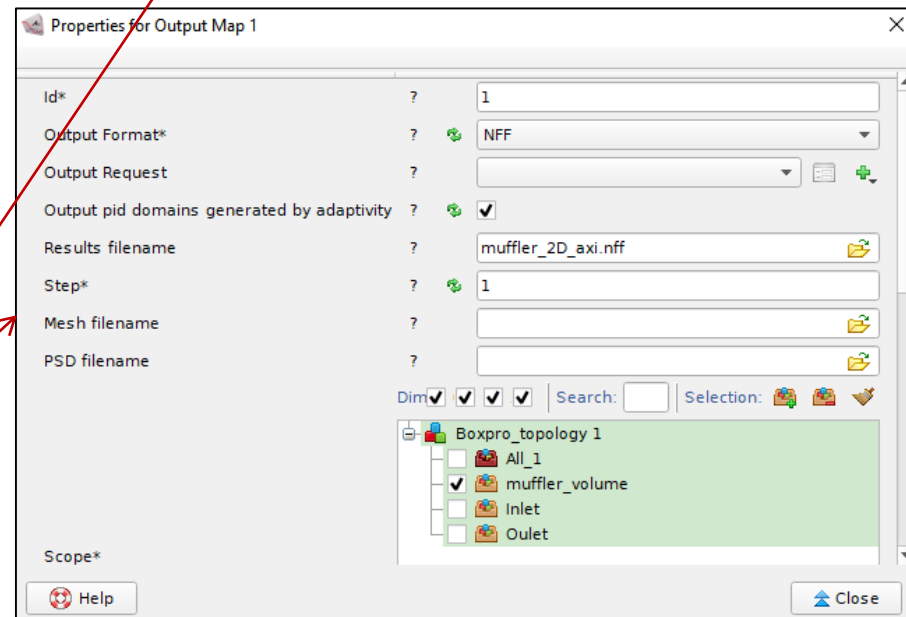
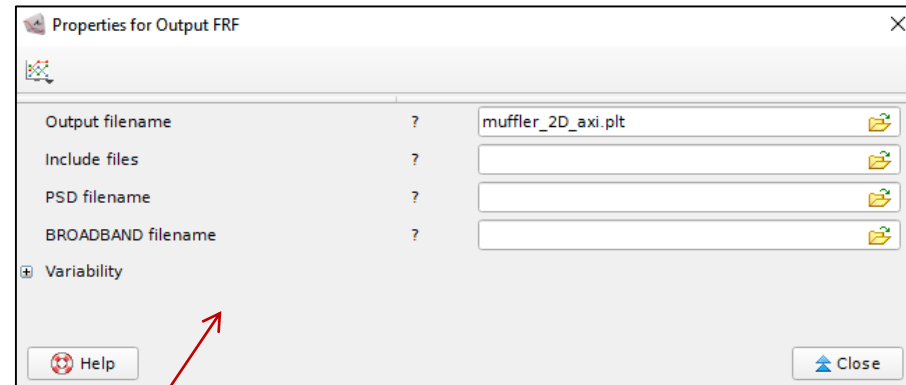
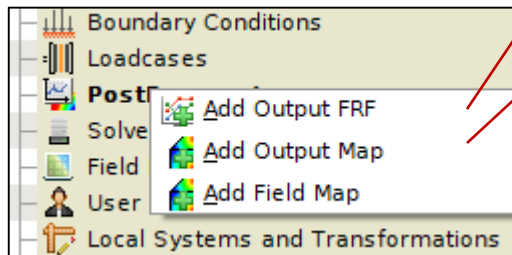


- a. Set duct name: outlet
- b. Choose material: air
- c. Set cross section parameters:
Center at [0, 0.7]
Axes: [0,1] , [1,0]
- d. Cross section surface:
 $0.007854 \quad (\pi \cdot 0.05^2)$
- e. Create anechoic condition for reflected wave
Direction: 1
Format: Free
- f. Select the domain "outlet"
- g. Click "View Geometry"
- h. Adjust the components visualization



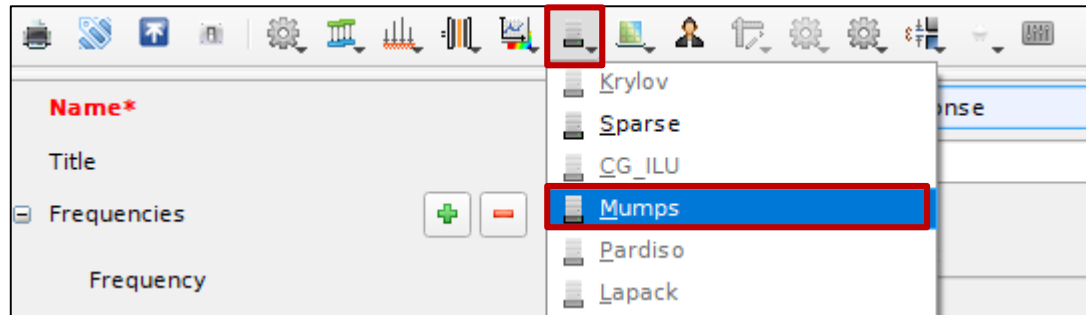
Create Post-Processing Requests

- 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



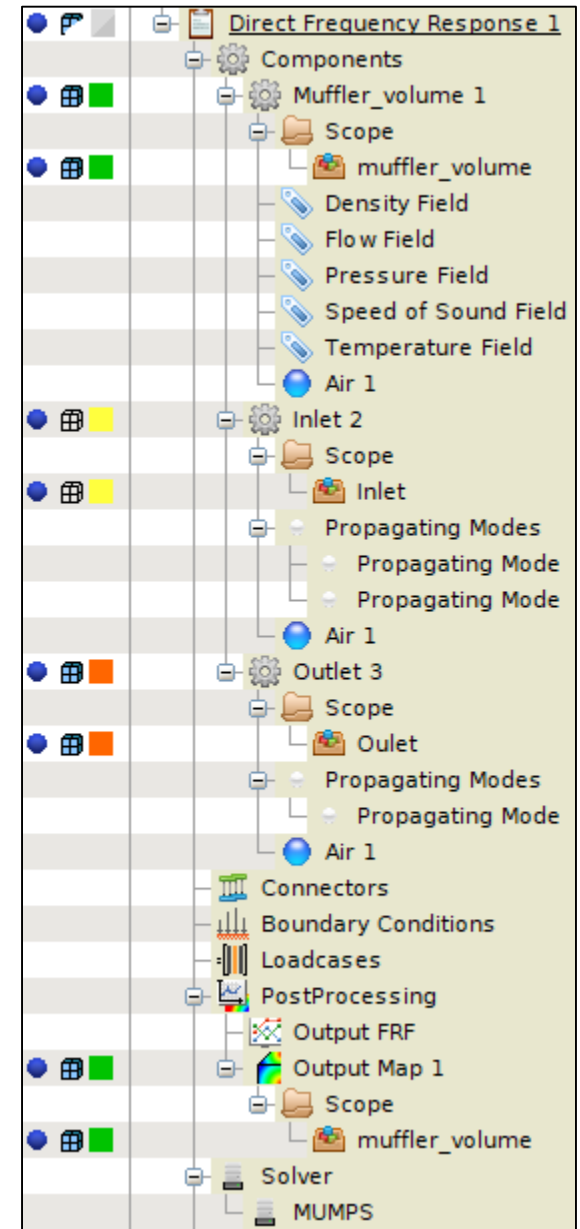
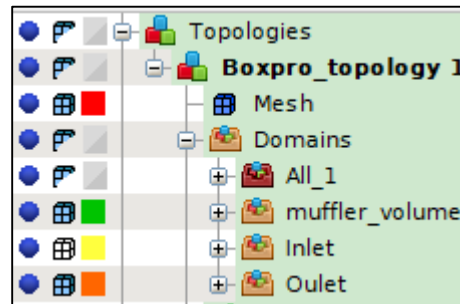
- Set the MUMPS solver
- Close the pop-up window of MUMPS



- Close the properties window of the Direct Frequency Response

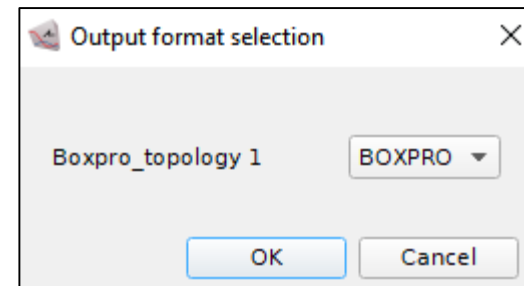
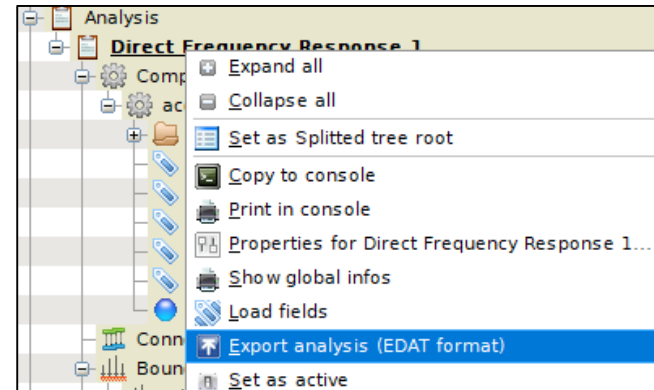
Check the Analysis

- 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



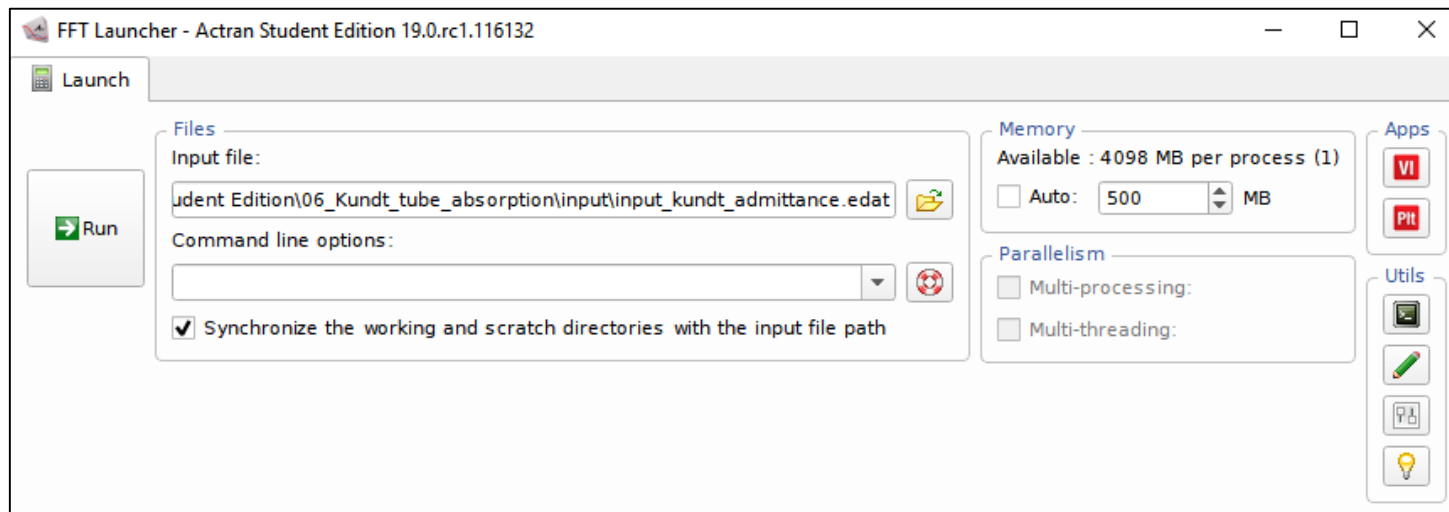
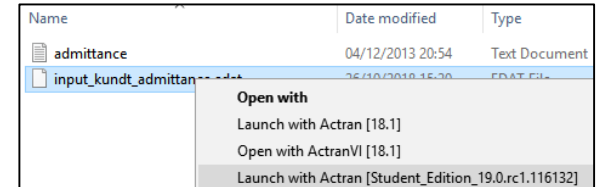
Export the Analysis File

- 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”

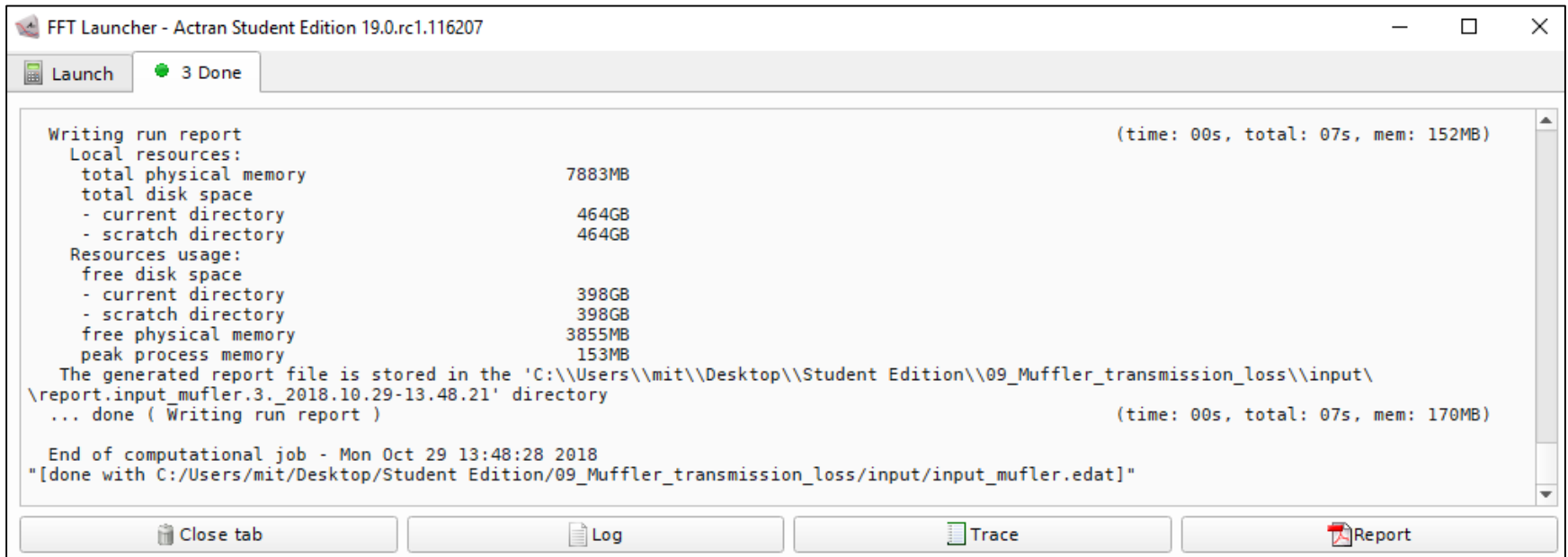


Launch Actran Analysis

- 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



The screenshot shows the FFT Launcher window with a tab titled "Launch" and a status bar indicating "3 Done". The main text area displays the following output:

```
Writing run report (time: 00s, total: 07s, mem: 152MB)
Local resources:
total physical memory          7883MB
total disk space
- current directory           464GB
- scratch directory           464GB
Resources usage:
free disk space
- current directory           398GB
- scratch directory           398GB
free physical memory          3855MB
peak process memory           153MB
The generated report file is stored in the 'C:\Users\mit\Desktop\Student Edition\09_Muffler_transmission_loss\input\
\report.input_mufler.3_2018.10.29-13.48.21' directory
... done ( Writing run report ) (time: 00s, total: 07s, mem: 170MB)

End of computational job - Mon Oct 29 13:48:28 2018
"[done with C:/Users/mit/Desktop/Student Edition/09_Muffler_transmission_loss/input/input_mufler.edat]"
```

At the bottom of the window, there are four buttons: "Close tab", "Log", "Trace", and "Report".

- Close the Launcher window

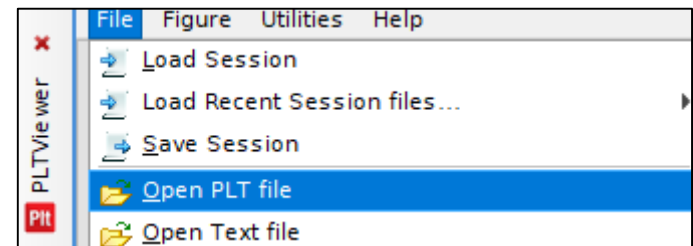
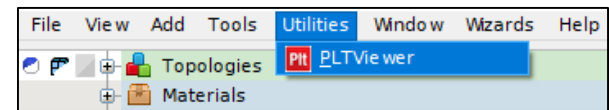
Post-processing

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

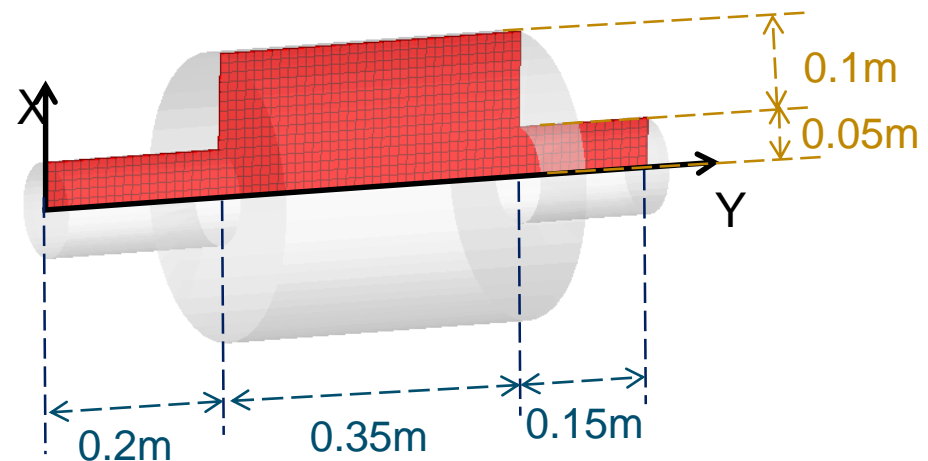
The screenshot displays a software interface for configuring a simulation. The main tree view shows a hierarchy: PitSet 1 [muffler_2D_axi.plt] > DOMAIN > Acoustic1 [Muffler_volume] > MODAL_BASIS > 2 [Inlet] > f [f] [200]. Below this, 3 [Outlet] > f [f] [200] is also visible. The X Data Vector panel has an 'Insert' button and a text field containing 'set_3.MODAL_BASIS[*3*].field[*f*]'. The Y Data Vector panel has an 'Insert' button and a dropdown menu set to 'TL, Transmission Loss (dB)'. Below the dropdown, two text fields are visible: 'set_3.MODAL_BASIS[*2*].field[*iP*]' and 'set_3.MODAL_BASIS[*3*].field[*rP*]'. Red and green circles highlight the 'f [f]' outputs in the tree view, and red and green arrows point from these circles to the corresponding text fields in the Y Data Vector panel.

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
 $2 \cdot \pi \cdot \text{freq} / \text{speed of sound}$



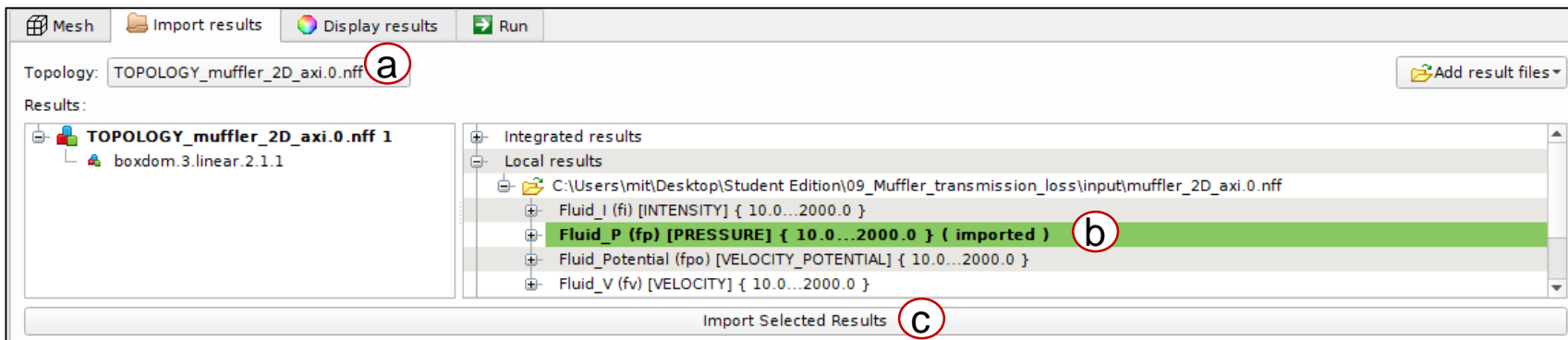
- This analytical solution is calculated with the assumption of plane wave propagation in the muffler
- The analytical solution has been calculated and is available in the file *TL_analytical.txt*

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

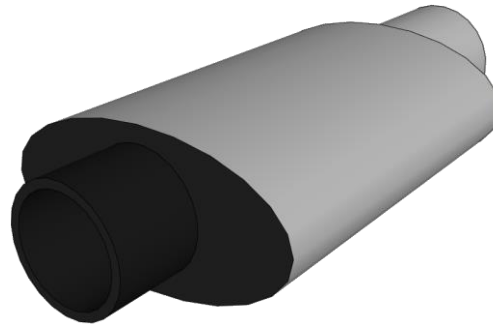
Visualize Field Map in ActranVI

- 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

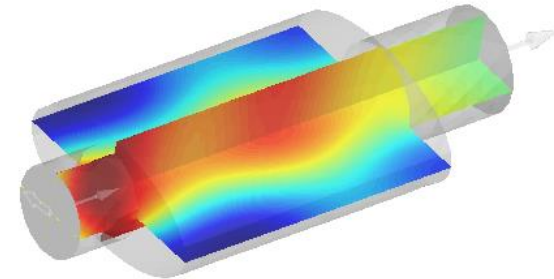


Going Further – Modeling Muffler in 3D

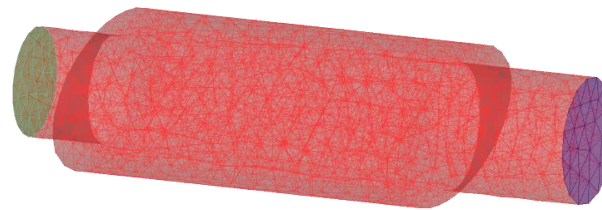
- A muffler model in 3D is contained in the folder “3D muffler”
- Import the analysis file in ActranVI
 - visualize the mesh
 - view the model set up
- Run the analysis
- Perform post processing in ActranVI
 - Plot TL curve
 - Display pressure maps



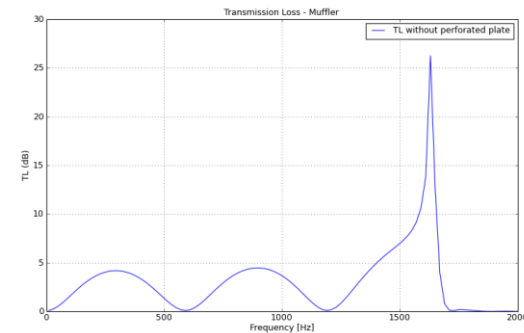
Geometry



Pressure map with cut planes



Mesh



Transmission loss