



# OpenPulse

## Alpha Version

### Getting Started Tutorial

2 Apr 2021

## 1 About *OpenPulse*

*OpenPulse* is a software written in Python for numerical modeling of low-frequency acoustically induced vibration in gas pipeline systems. It allows to import the geometry of the pipe system (lines in IGES), insert materials properties, set sections, and import pressure/acceleration/force loads (from measurements or theory). *OpenPulse* performs an acoustic time harmonic response analysis of the respective 1D acoustic domain using the Finite Element Transfer Method (FETM). The resulting pressure field is applied as a distributed load over the respective structural piping system, modeled with the Timoshenko beam theory and the Finite Element Method (FEM), in order to run a structural time harmonic response analysis. In addition to simply boundary conditions as constraints on displacements, *OpenPulse* allows to insert lumped springs, masses and dampers along the domain.

After defining the FEM mesh for the model, you can plot the piping system geometry and run simulations such as modal analysis and harmonic analysis. It is possible to plot deformed shapes, frequency plots of acoustical and structural responses, stress fields and local stresses of desired sections.

## 2 Installing

The complete procedure can be found in <https://github.com/open-pulse/OpenPulse>. Main steps:

- install Python 3.7.7 (<https://www.python.org/downloads/release/python-377/>).
- install *OpenPulse*. Clone or download *OpenPulse* files (GitHub link above). In the case of download, unzip the received file and open a terminal in the main folder (an easy way to do this is to enter the folder, press shift and right click, then "Open PowerShell here"). So, enter the command: `pip install -r requirements.txt`.
- Run *OpenPulse*. In the same folder, enter the following command in the terminal: `python pulse.py`.

### 3 Simple example: time-harmonic coupled analysis

Open *OpenPulse* and create a new Project:

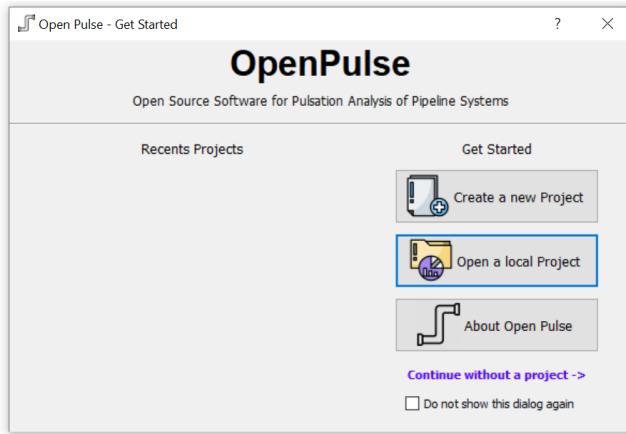


Figure 1: New Project.

Choose the *Project name* and the *Project directory* (working directory). Thus, import the desired geometry (only lines in IGES format. You cannot build or edit geometry in *OpenPulse*). The IGES file for this exemple can be found in the program folder:

/examples/iges\_files/new\_geometries/example\_2\_noBeam.iges.

Set the element size according to the geometry and to the physical parameters. *OpenPulse* is like a Finite Element software, and you need to know the problem to choose the refinement of your mesh.

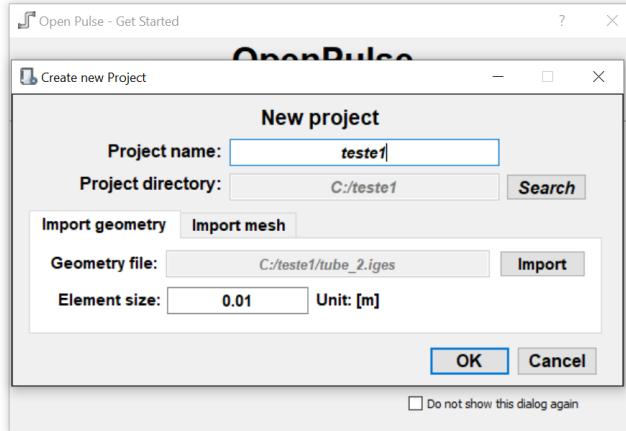


Figure 2: Project name, work directory, importing IGES, and element size.

After this, the imported geometry appears on the screen.

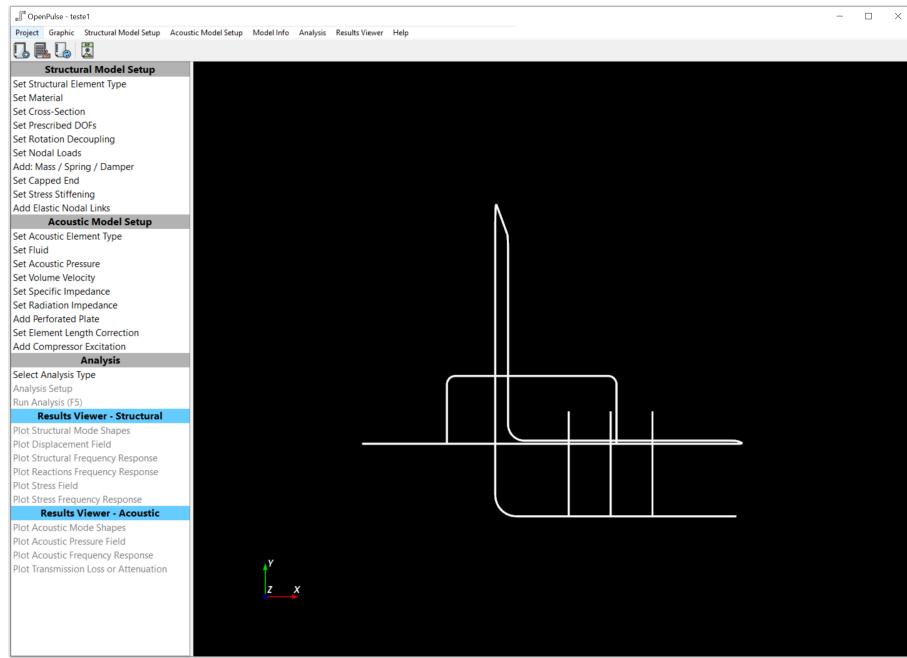


Figure 3: Imported geometry (view of Entities).

## Structural model setup:

- Defining structural element type, sections and structural material properties:

*OpenPulse* has three structural element types. In this introductory example, we use the *Pipe 1* element: Timoshenko-beam finite element, with 6 DOFs per node, with hollow circular section. This is the default element to simulate piping systems with our software. In the left menu, in *Structural Model Setup*, click on *Set Structural Element Type*. So, click on *All lines*, select *Pipe 1* and proceed.

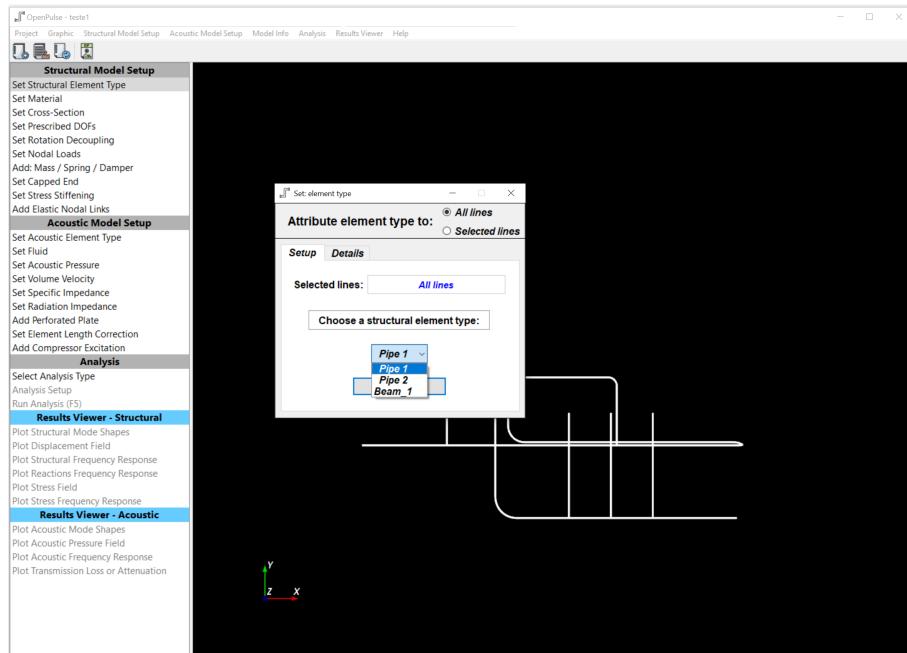


Figure 4: Structural element type.

After, a new window automatically appears to set the cross section parameters. Enter the *Outer diameter* and *Thickness* for *All lines*. If you want, the same procedure can be done by clicking on *Set Cross-Section* at the left menu, and you can set different sections for different lines clicking on *Selected lines*.

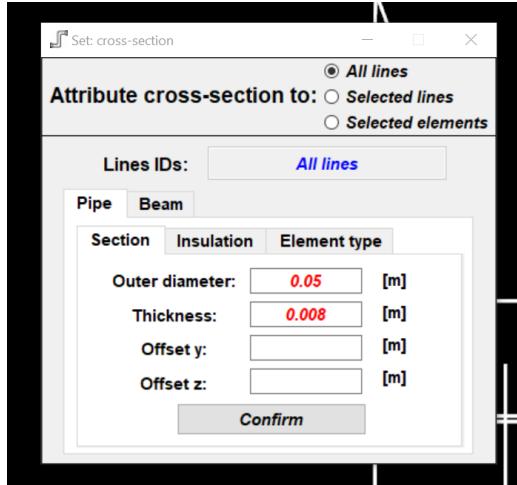


Figure 5: Cross-section setup.

Then, a new window automatically appears to set the structural material parameters. *OpenPulse* has a list of predefined materials. You can choose one by clicking on the list, or you can enter all the properties for a new material (also, you can name it and insert in the list by *Add New Material*).

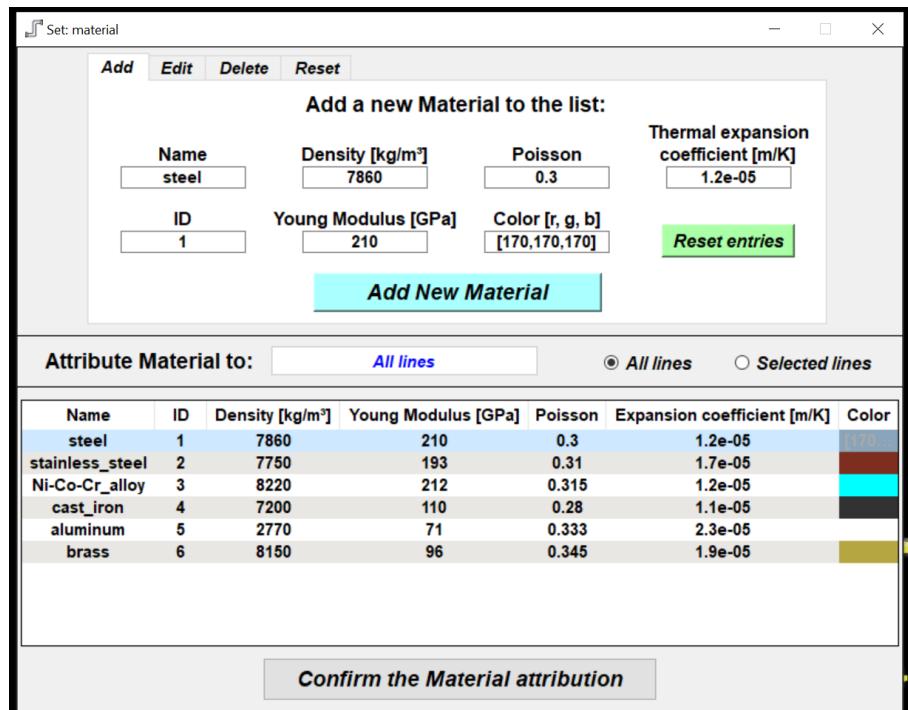


Figure 6: Structural material setup.

The pipe system will then appear on the screen with the representation of sections and materials.

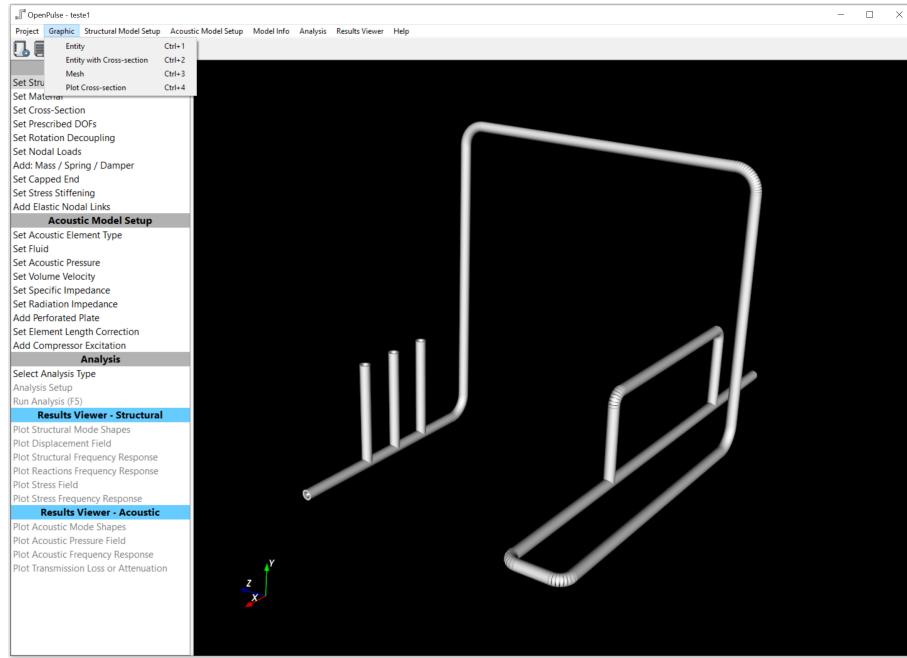


Figure 7: Visualization of entities with Cross-sections.

## - Entity, sections and mesh visualization:

Now, you can observe/verify the details of your system. In the top menu, click on *Graphic*, and *Plot cross-section*. So, choose one line to plot the related cross-section, as in the figure below.

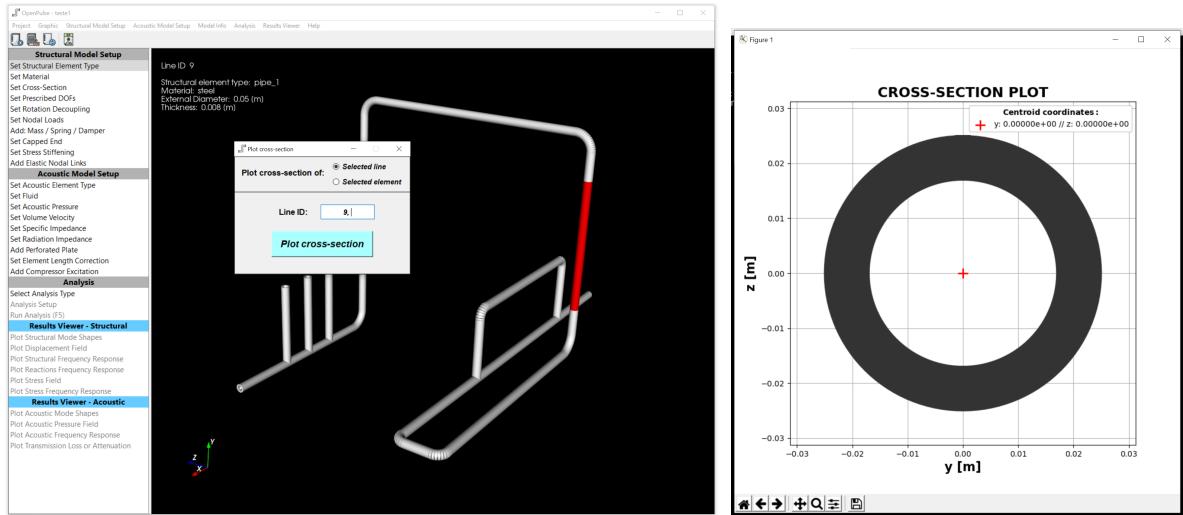


Figure 8: Plot cross-section.

In *OpenPulse*, depending on the application, some properties can be defined on entities (lines) or on elements. You can choose the type of graphic visualization in top menu, clicking on *Graphic* and after: *Entity* to see the lines; *Entity with Cross-section* to see the lines and the respective Cross-sections and materials; and *Mesh* to see the finite element mesh (nodes and elements), which are used to solve the structural vibration problem. For mesh visualization, the nodes are represented by yellow points and the cross-sections are shown as translucent solids.

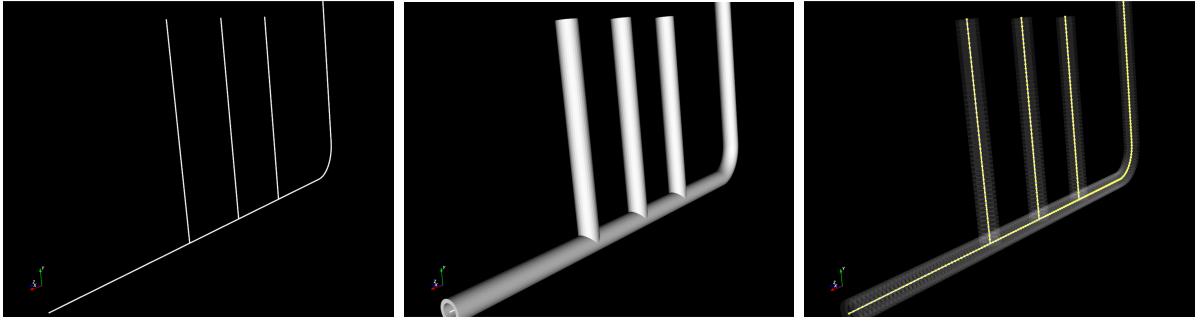


Figure 9: Plot entities (lines), entities with cross-sections and mesh.

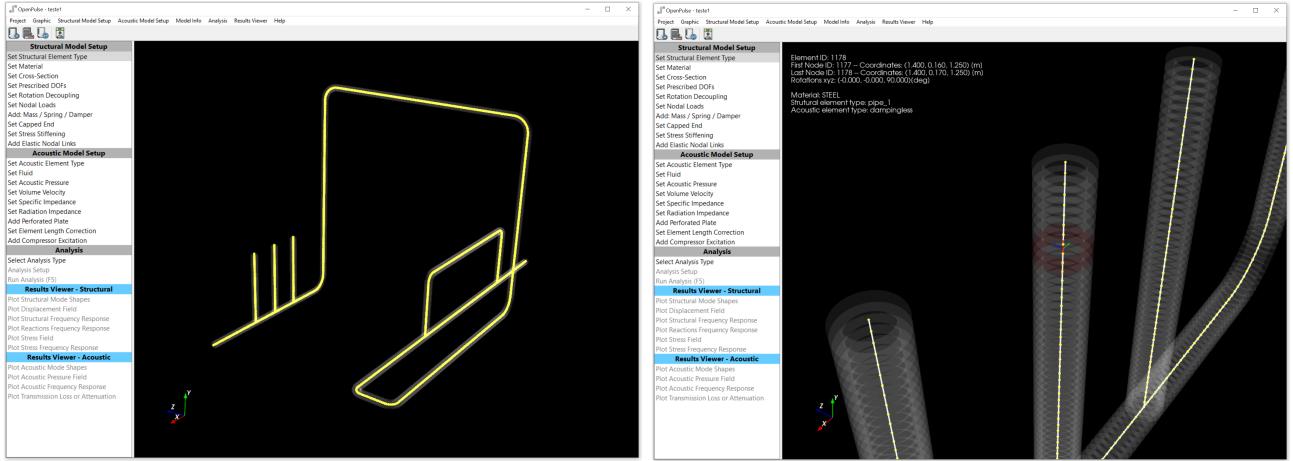


Figure 10: Mesh details.

## - Applying structural boundary conditions:

### Set prescribed DOFs:

To set "zero displacements" for regions and directions of interest, you need to apply prescribed DOFs to the correspondent nodes of the structure (*OpenPulse* is a Finite Element based software). In the left menu, click on *Set Prescribed DOFs*, than click on the nodes (on the screen) you want to constrain. Use the Ctrl button to select more than one node. In this example, we first select the nodes 10 and 998, setting all DOFs to zero. After confirmation, the mesh representation is updated with the symbols for the related prescribed DOFs (figures below).

Next, click again on *Set Prescribed DOFs*, than click on other nodes (on the screen) you want to constrain in other directions. In our example, we select the nodes 438, 491 and 895 to set the DOF UY to zero (the global coordinate system is shown in the lower left corner). After confirmation, the mesh representation is updated with the new symbols for the new prescribed DOFs (figures below).

For applying dynamic displacements (or velocities, or accelerations) with frequency obtained from an experimental test, the tab *Load tables* allows the user to import external files. This is the case when we have a large machine with high vibration levels exciting the piping system and we want to insert this excitation into the model. This procedure will be explained in the *Advanced Tutorial*.

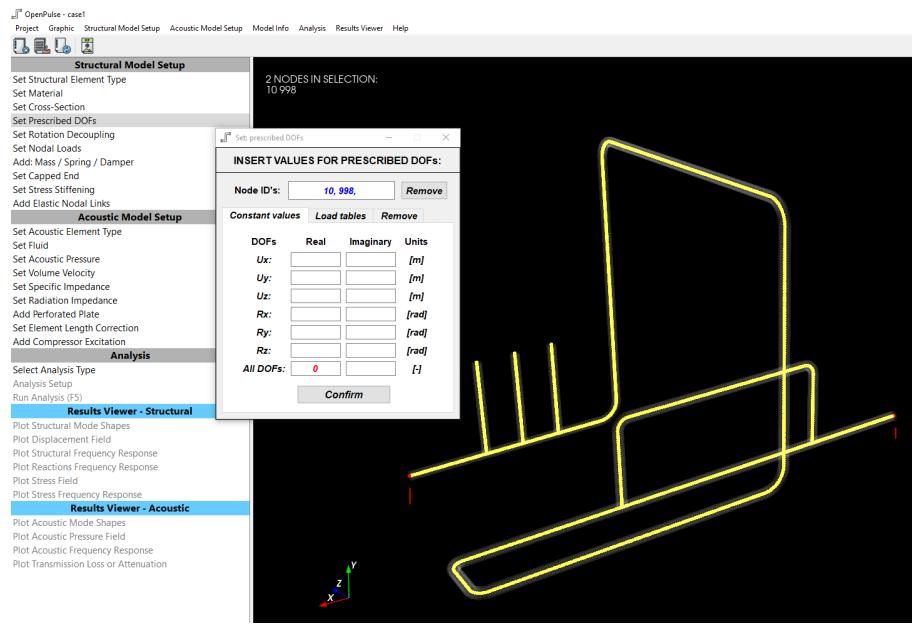


Figure 11: Prescribed DOFs - constraint on all DOFs.

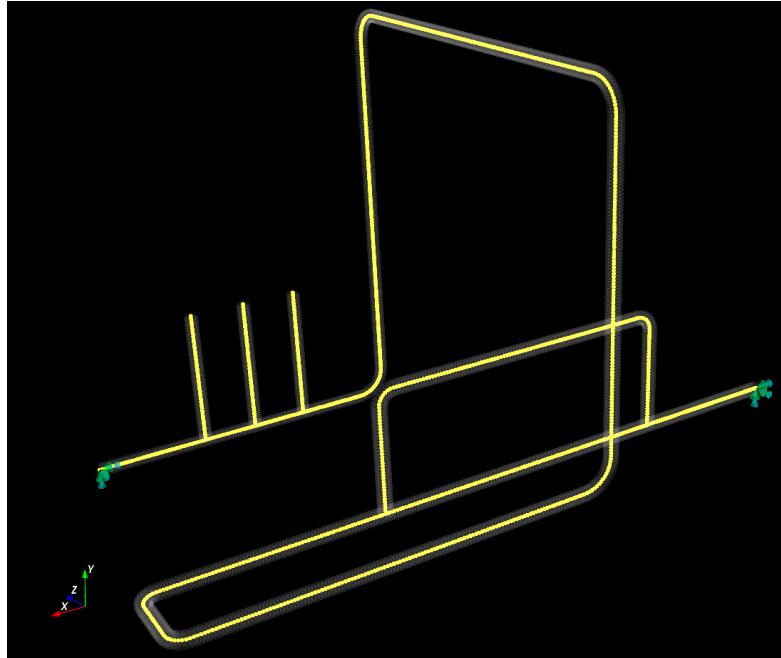


Figure 12: Prescribed DOFs - representation.

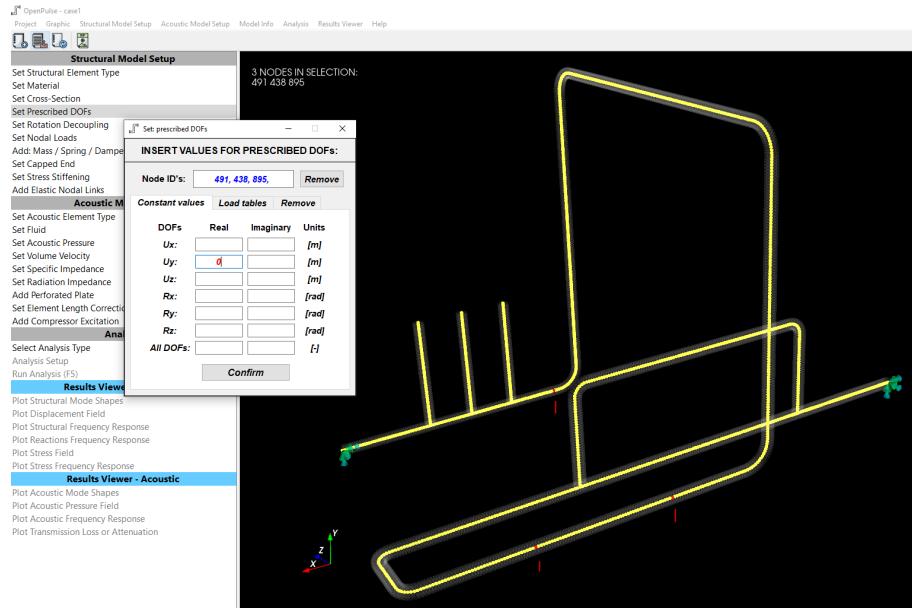


Figure 13: Prescribed DOFs - constraint on UY direction.

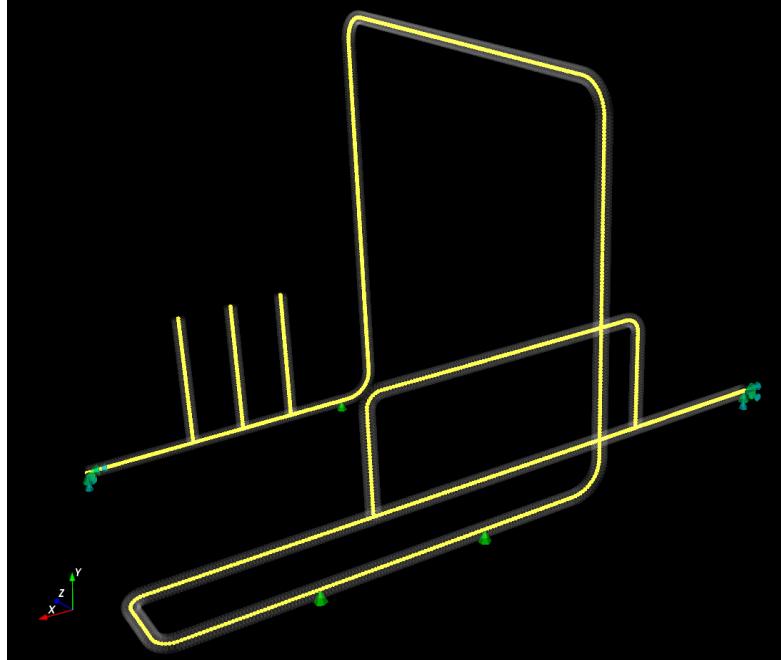


Figure 14: Prescribed DOFs - representation.

### Set Nodal Loads (external loads):

To set loads at regions and directions of interest, you need to apply structural forces to the correspondent nodes of the structure (*OpenPulse* is a Finite Element based software). In the left menu, click on *Set Nodal Loads*, than click on the nodes (on the screen) you want to constrain. Use the Ctrl button to select more than one node. In this example, we select the nodes 672 and 708, applying nodal forces of 10 N in the Y direction (constant with frequency). After confirmation, the mesh representation is updated with the symbols for the related nodal forces (figures below).

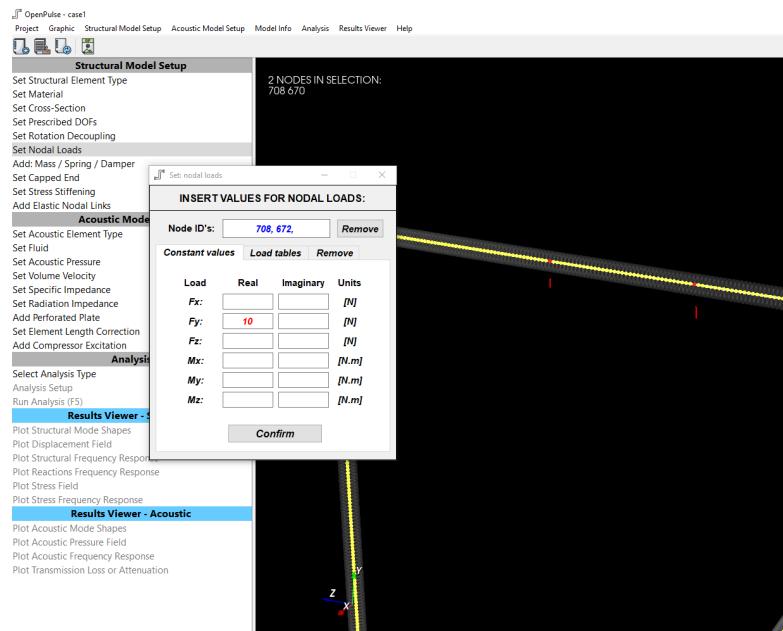


Figure 15: Nodal Loads.

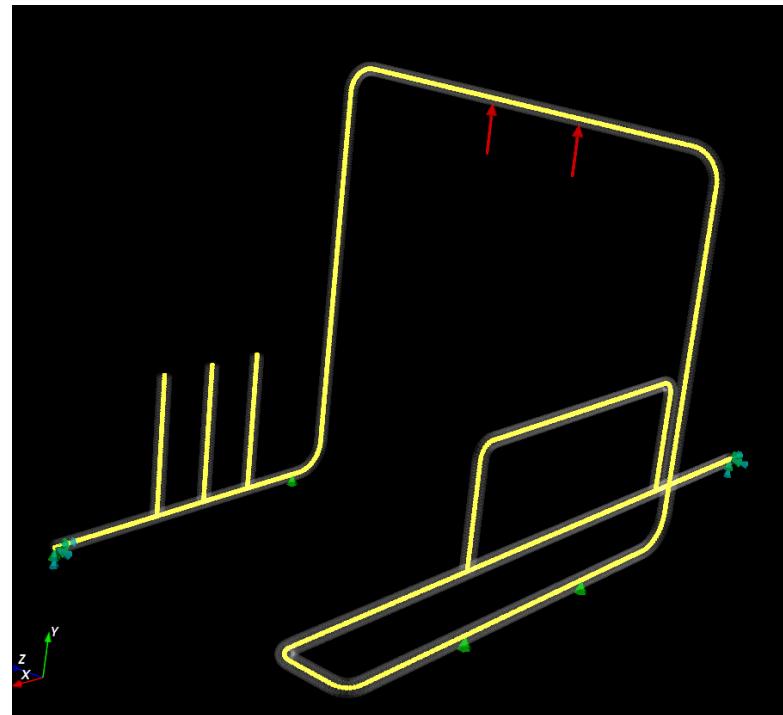


Figure 16: Nodal Load - representation.

## Add Mass / Spring / Damper:

To add mass, springs and/or dampers at regions and directions of interest, click on *Add Mass / Spring / Damper* in the left menu, than click on the nodes (on the screen) you want to add these lumped parameters. Use the Ctrl button to select more than one node. In this example, we select the node 136. The mass is defined in tab *Mass* with 50 kg, and a spring is defined at the same node in tab *Spring* with stiffness 1000 N/m.

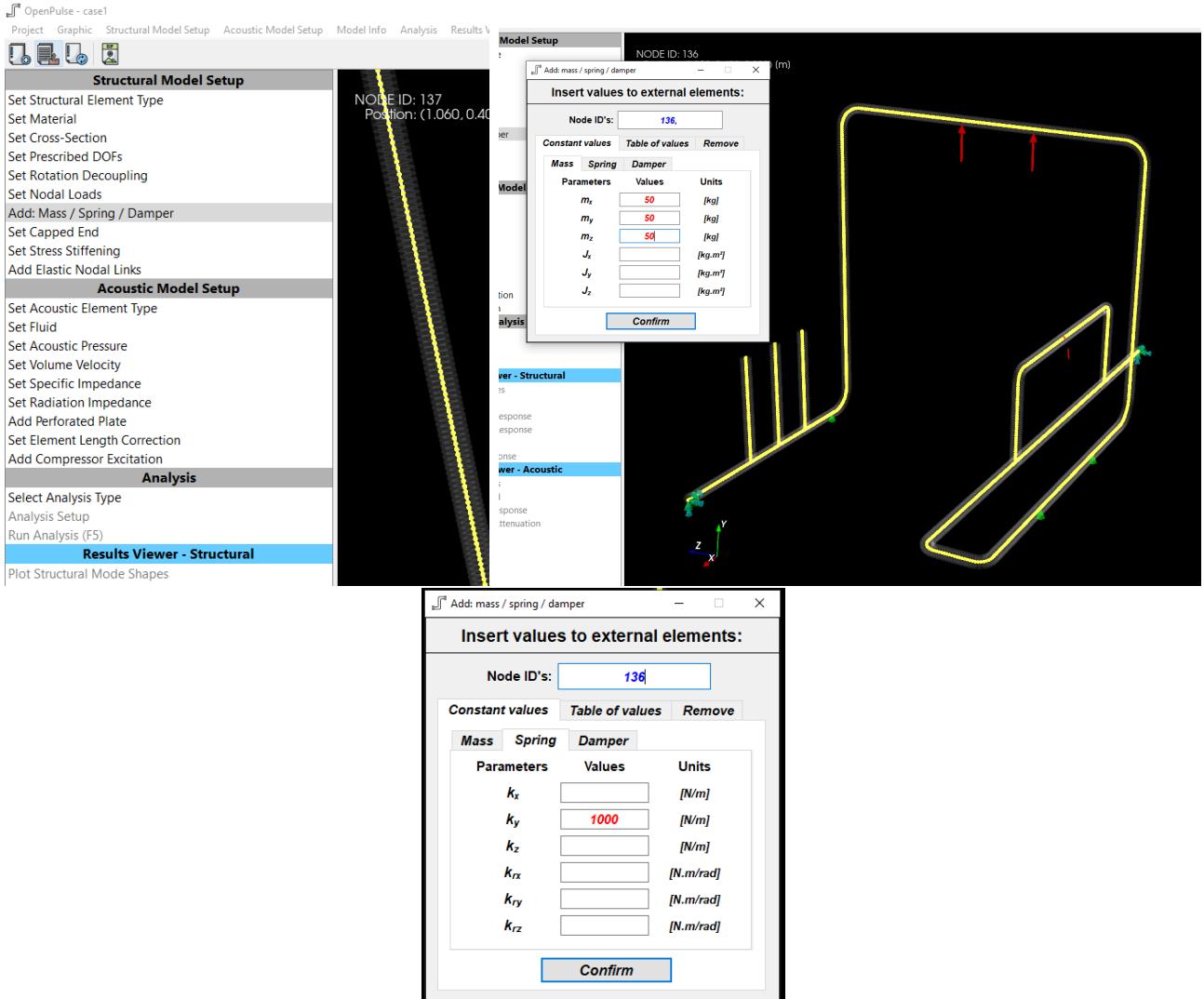


Figure 17: Add mass and spring.

After confirmation, the mesh representation is updated with the symbols for the related nodal forces (figure below). Now, all structural boundary conditions are set for this example.

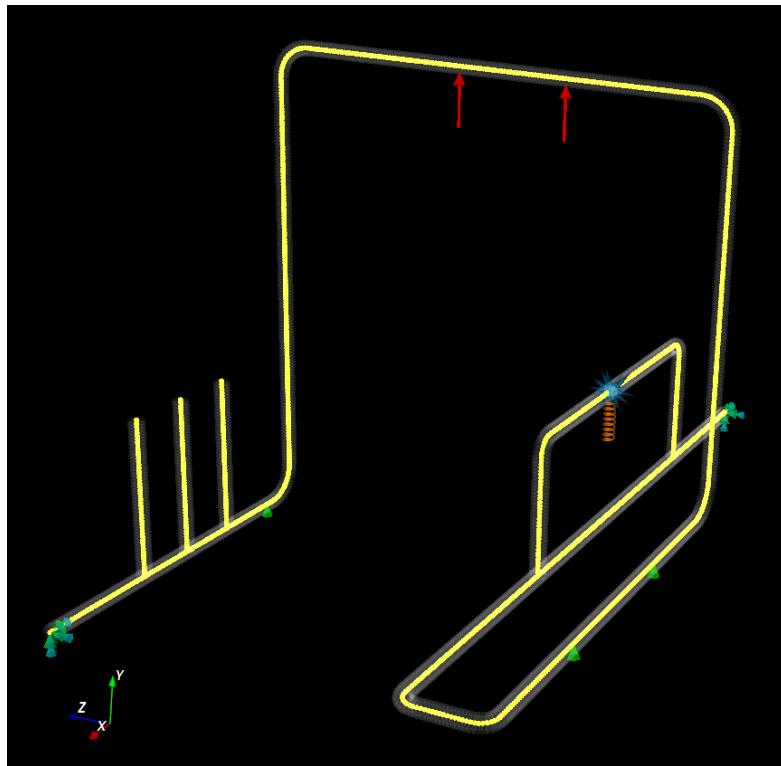


Figure 18: Representation of all boundary conditions of this example.

## Acoustic model setup:

### - Defining acoustic element type and acoustic material properties:

*OpenPulse* has five acoustic element types. In this introductory example, we use the *Dampingless* element: undamped 1D linear acoustics, with 1 DOF per node (Finite Element Transfer Matrix Method). This is the default element to simulate piping systems with our software. In the left menu, in *Acoustic Model Setup*, click on *Set Acoustic Element Type*. So, click on *All lines* and select *Dampingless* and proceed.

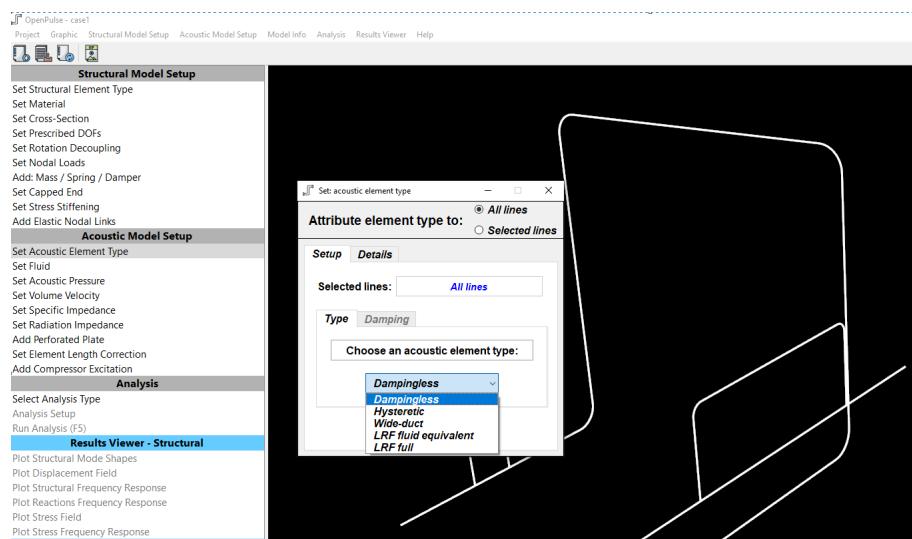


Figure 19: Acoustic element type.

OBS.: The acoustic domain is defined by the internal diameters of the piping system (defined with structural element *Pipe 1*).

Then, a new window automatically appears to set the acoustic material parameters. *OpenPulse* has a list of predefined gas materials. You can choose one by clicking on the list, or you can enter all the properties for a new material (also, you can name it and insert in the list by *Add New Material*).

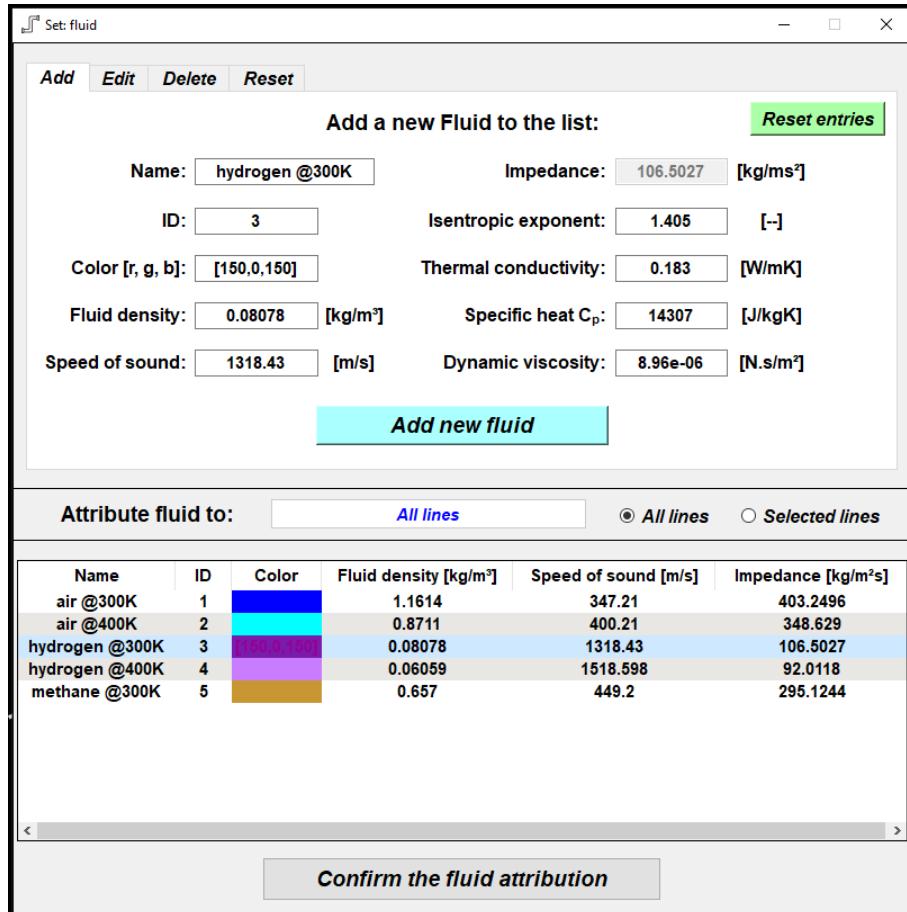


Figure 20: Gas properties.

## - Applying acoustic boundary conditions:

### Set acoustic pressure (dynamic):

To set acoustic dynamic pressure for regions of interest, click on *Set Acoustic Pressure* in the left menu, than click on the nodes (on the screen) you want to apply this boundary condition. Use the Ctrl button to select more than one node. In this example, we select the nodes 1048, 1098 and 1148 applying a pressure of 0.1 Pa (constant with frequency). After confirmation, the mesh representation is updated with the symbols for the related prescribed pressure (figures below).

For applying pressure varying with frequency obtained from an experimental test, the tab *Import table* allows the user to import external files. This is the case when we have information about high dynamic pressure exciting the piping system and we want to insert this excitation into the model. This procedure will be explained in the *Advanced Tutorial*.

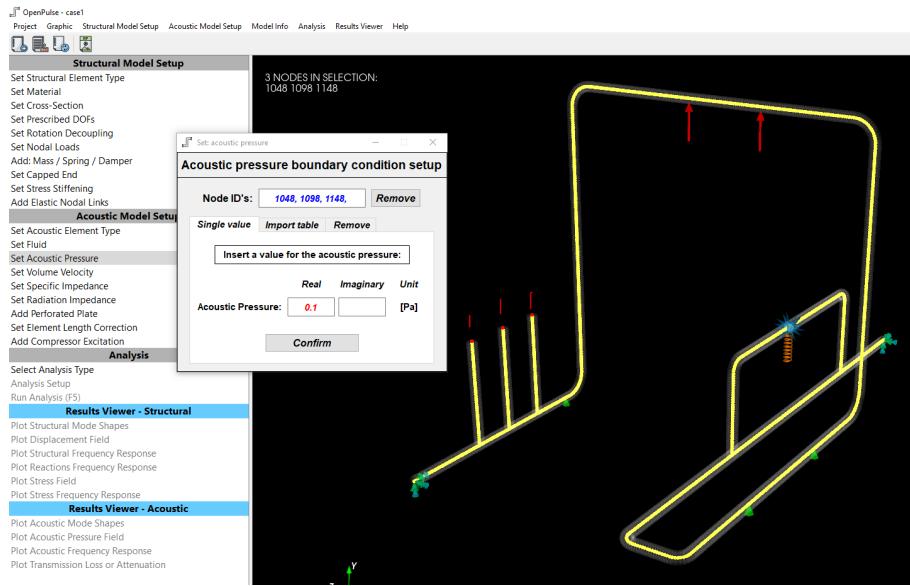


Figure 21: Apply acoustic pressure.

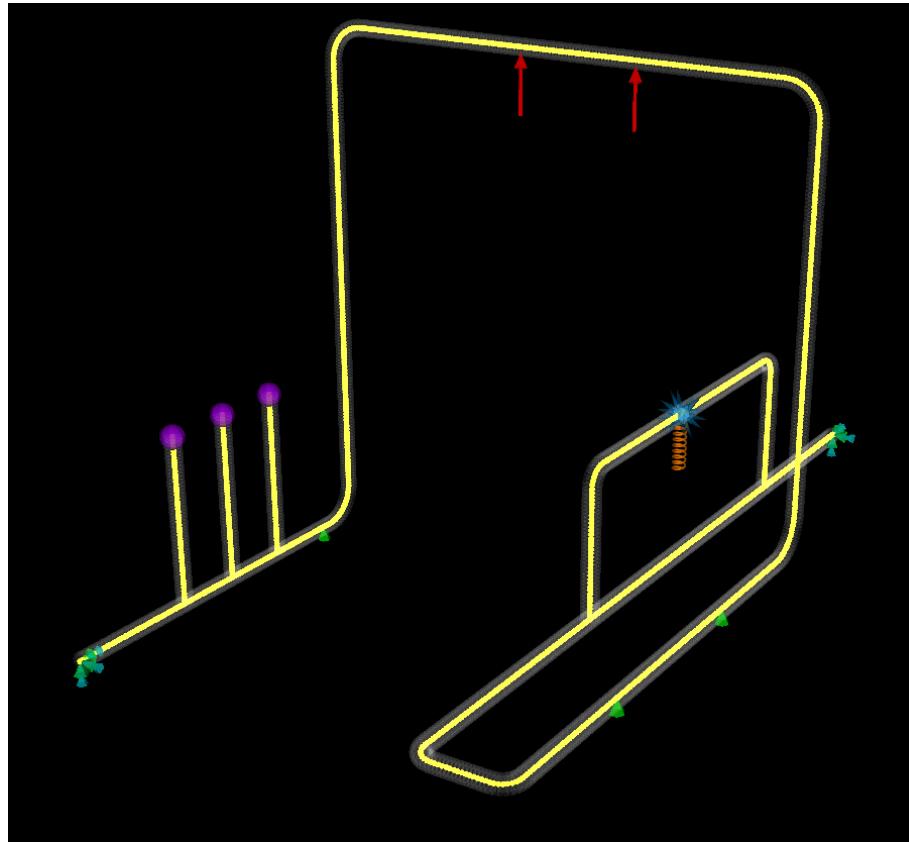


Figure 22: Acoustic pressure as boundary condition - representation.

### Set volume velocity (dynamic):

To set acoustic volume velocity for points of interest, click on *Set Volume Velocity* in the left menu, than click on the nodes (on the screen) you want to apply this boundary condition. Use the Ctrl button to select more than one node. In this example, we select the node 10 applying a volume velocity of  $0.1 \text{ m}^3/\text{s}$  (constant with frequency). After confirmation, the mesh representation is updated with the symbols for the related prescribed pressure (figures below).

For applying acoustic volume velocity with frequency obtained from an experimental test,

the tab *Import table* allows the user to import external files. This is the case when we have information about the compressor pulsation, in the form of variation of the flow rate. This procedure will be explained in the *Advanced Tutorial*. *OpenPulse* has the *Compressor Excitation* boundary conditions, also explained in the *Advanced Tutorial*.

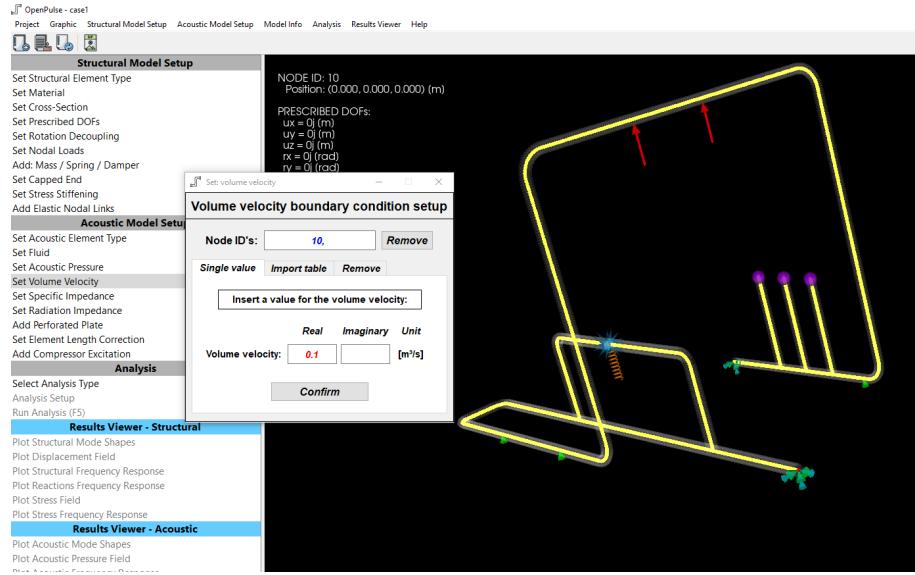


Figure 23: Apply volume velocity.

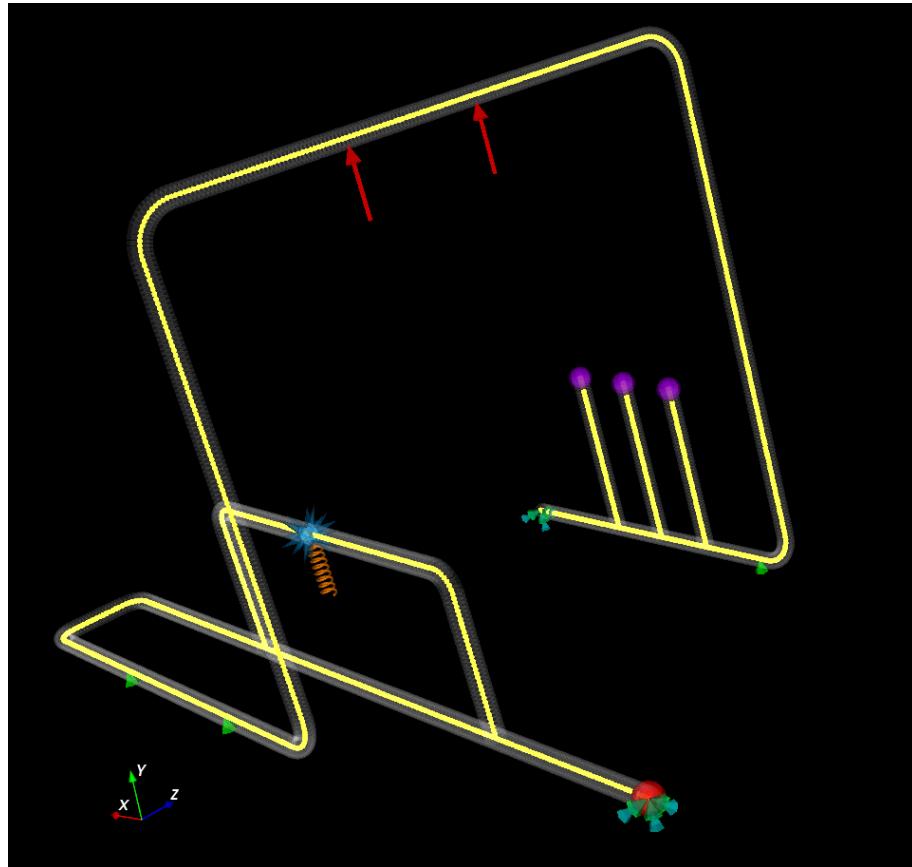


Figure 24: Volume velocity as boundary condition - representation.

## Other important setups for acoustic analysis:

For the sake of simplicity, in the first contact with the software we are not inserting *Impedances*, *Element Length Correction* nor *Capped End Conditions*. These are important parameters/conditions to be defined in a numerical problem to represent the physical phenomena with quality.

## Running the coupled analysis:

Now, its time to run the coupled analysis. To do this, go to *Analysis* in the left menu, and click on *Select Analysis Type*. Thus, choose *Harmonic Analysis - Coupled*. A new window automatically appears to set the *Method*, which is used here as *Direct*. Next, a new window also appears to set the minimum and maximum frequencies of analysis, and the discretization of the frequency range. In this example, the analysis is performed from 0 to 400 Hz, with steps of 5 Hz.

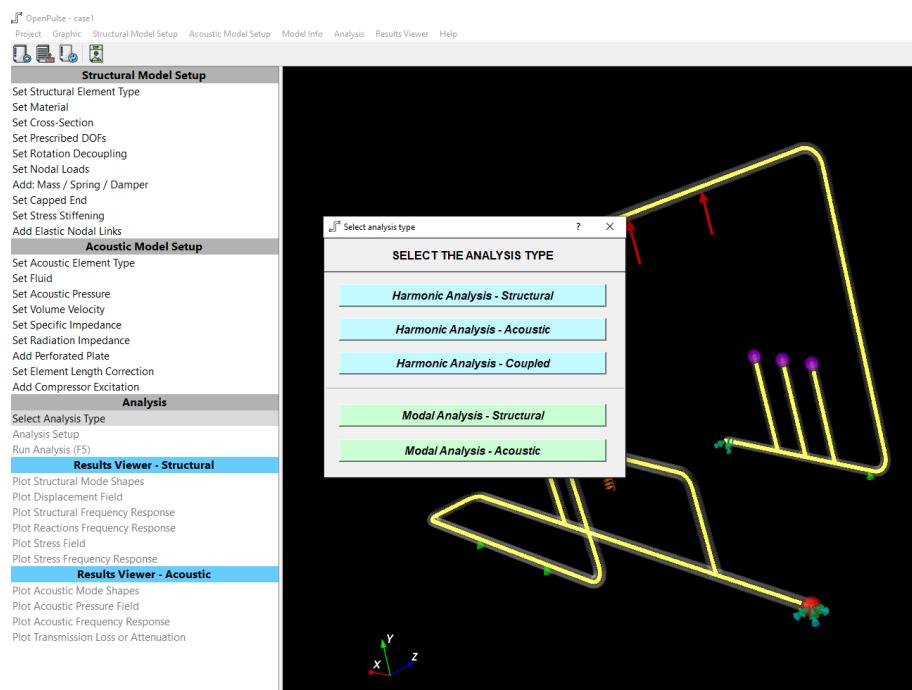


Figure 25: Analysis types.

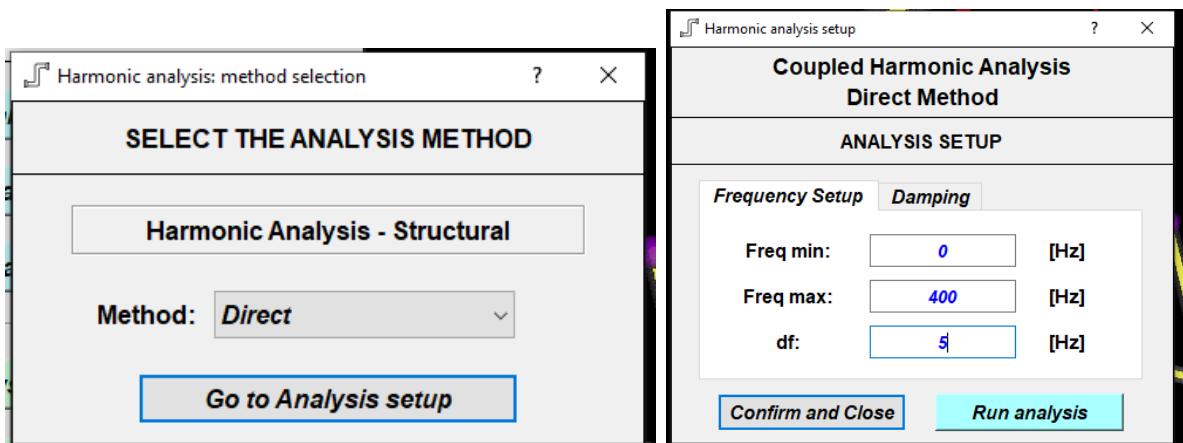


Figure 26: Analysis setup.

Clicking on *Run analysis* the information about the solution is shown on the screen:

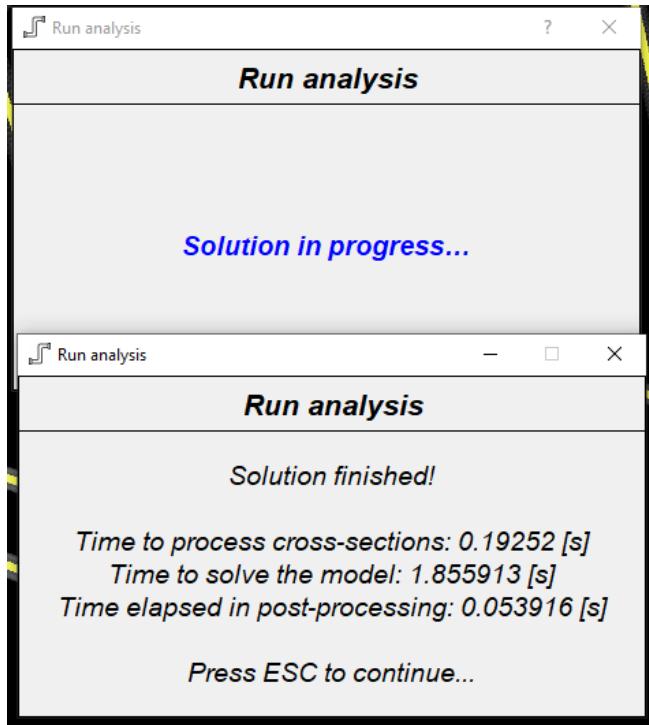


Figure 27: Solution information.

## Viewing the results:

Now we can see the results. For example, from *Results Viewer - Structural* in the left menu, click on *Plot Displacement Field* and choose a frequency of interest. The displaced structure is thus presented on the screen.

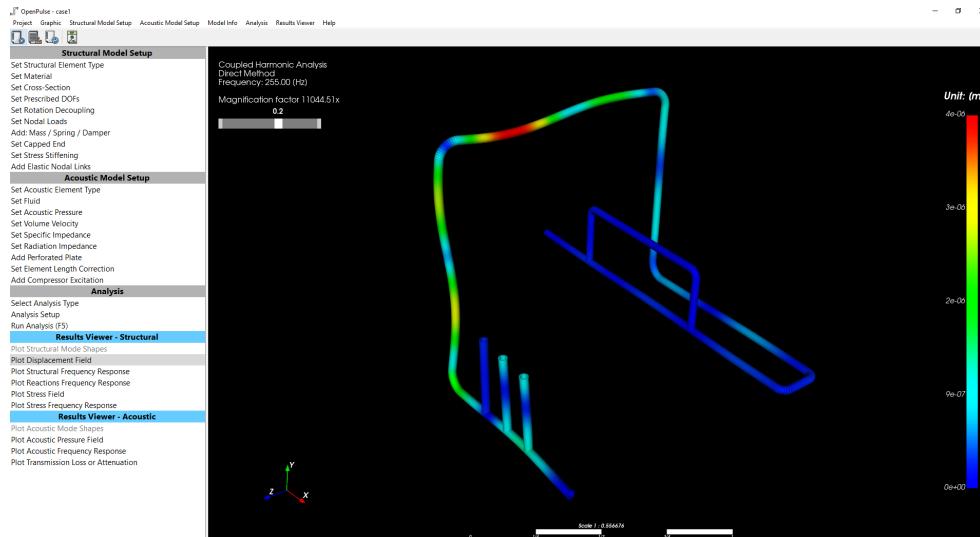


Figure 28: Plot displacement field.

We can also show the frequency plot of displacement, velocity or acceleration of a given point in the structure. From *Results Viewer - Structural* in the left menu, click on *Plot Structural Frequency Response*. In this example, we will plot the displacement in Y direction for node 103 (as shown in the figures below).

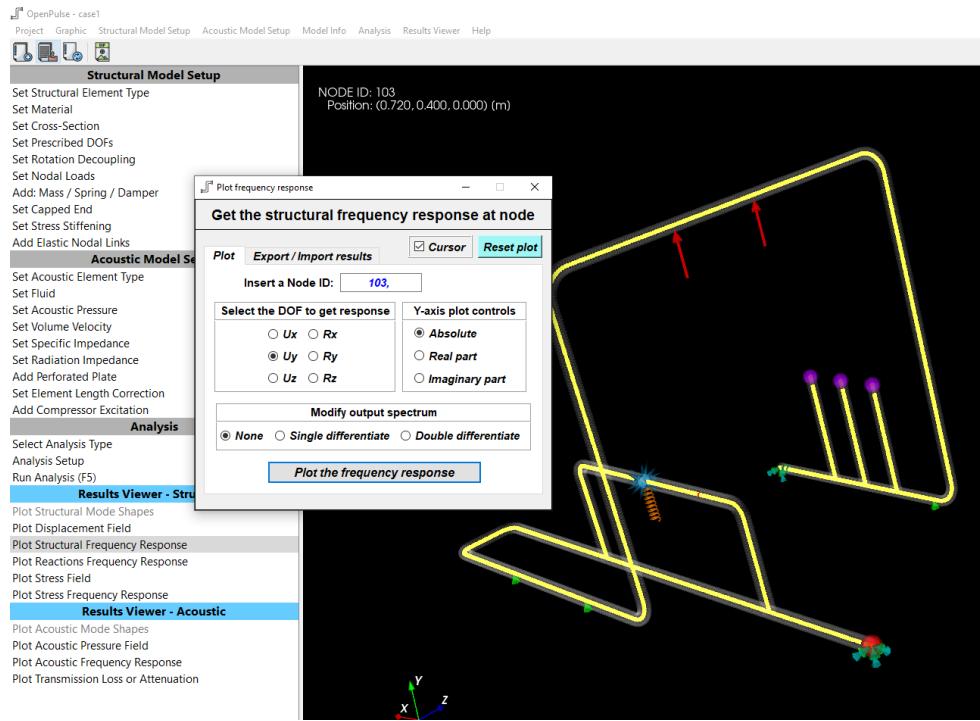


Figure 29: Plot displacement frequency response - setup.

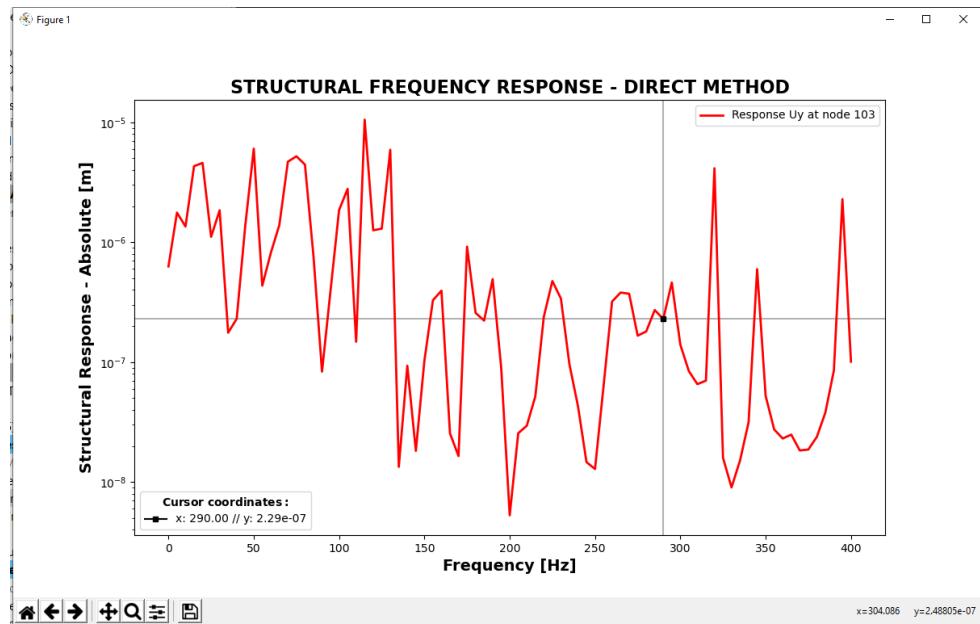


Figure 30: Plot displacement frequency response.

We can also see the acoustic results. For example, from *Results Viewer - Acoustic* in the left menu, click on *Plot Acoustic Pressure Field* and choose a frequency of interest. The distribution of dynamic pressure is thus presented on the screen.

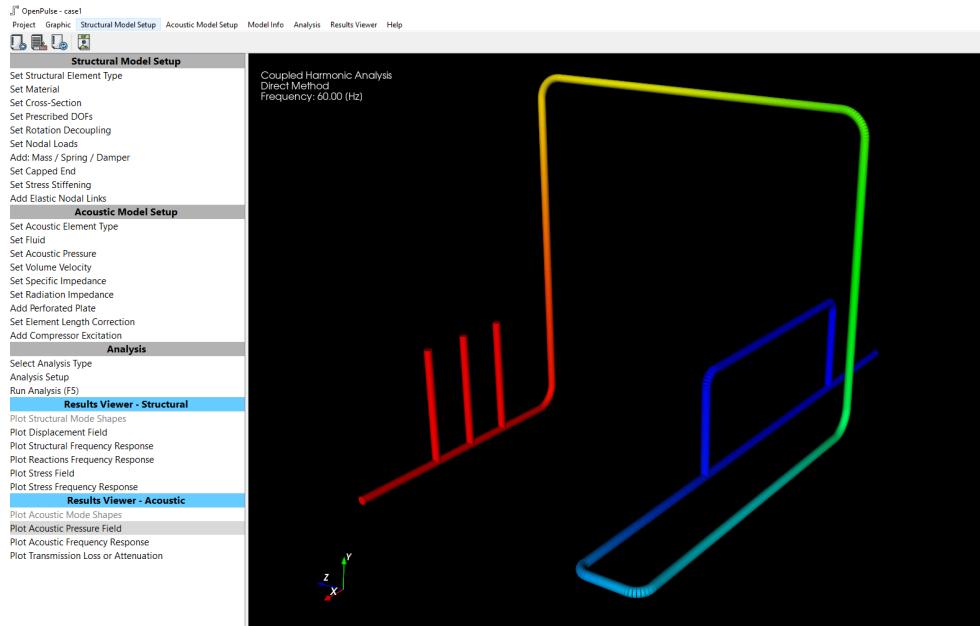


Figure 31: Plot acoustic pressure field.

Now, observe the frequency plot of the acoustic pressure of a given point in the piping system. From *Results Viewer - Acoustic* in the left menu, click on *Plot Acoustic Frequency Response*. In this example, we will plot the displacement in Y direction for node 69 (as shown in the figures below).

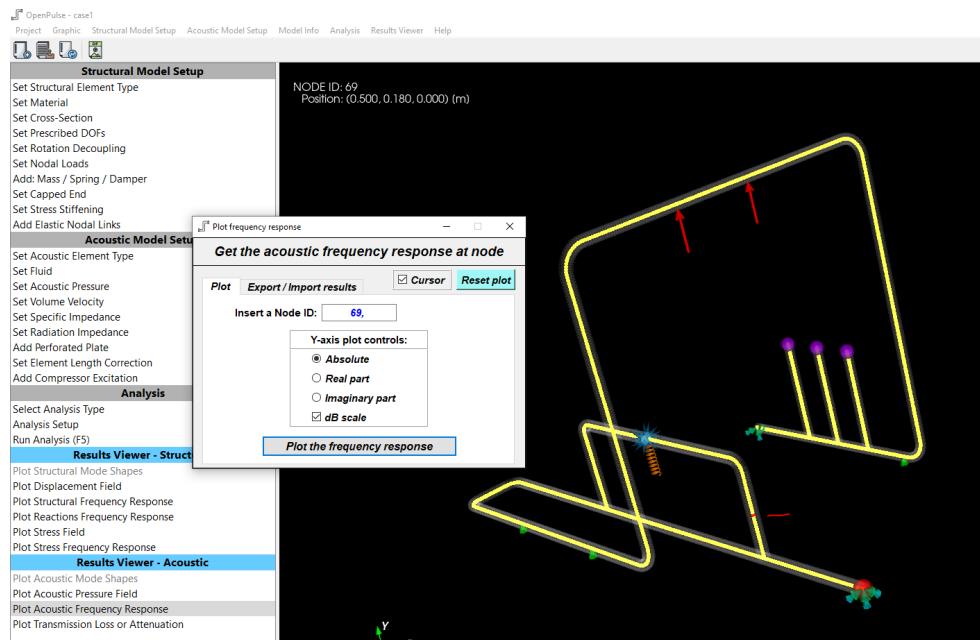


Figure 32: Plot acoustic pressure frequency response - setup.

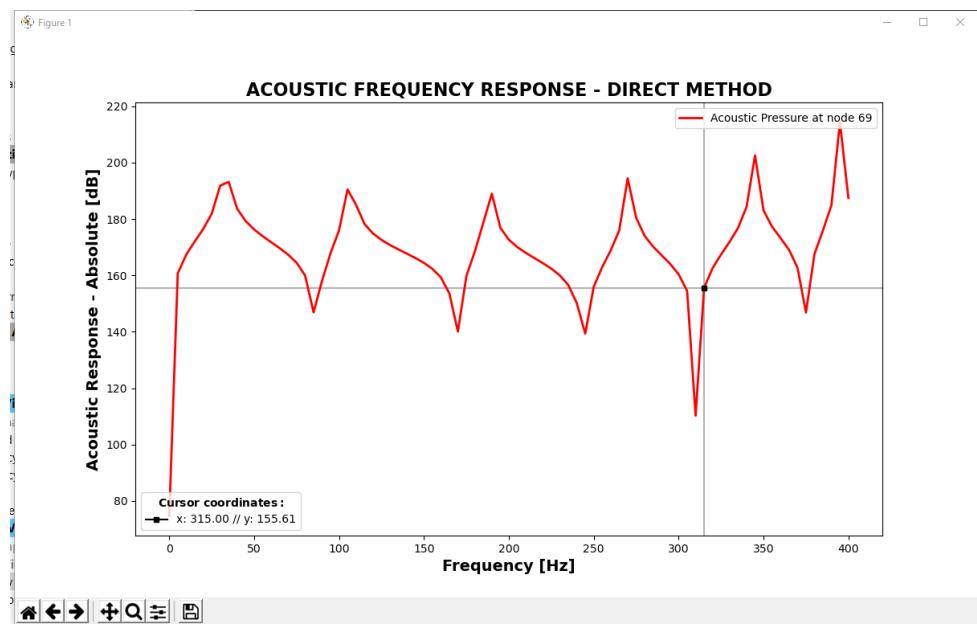


Figure 33: Plot acoustic pressure frequency response.