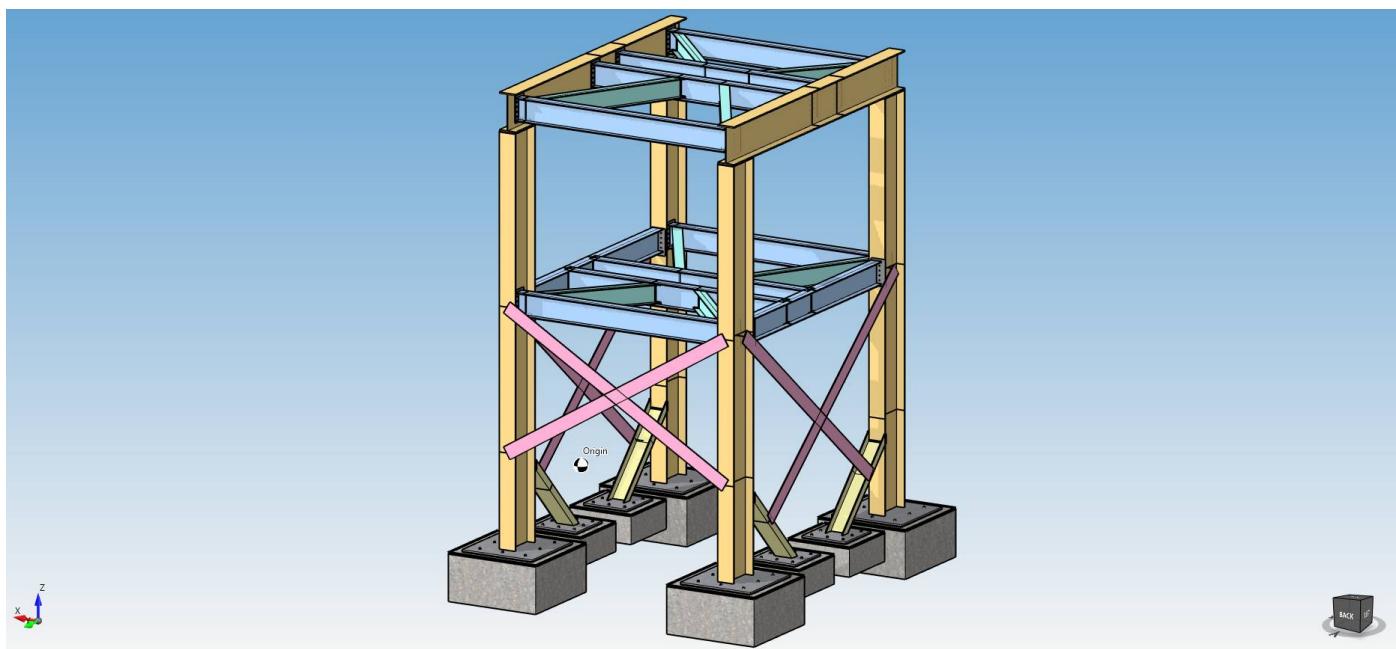
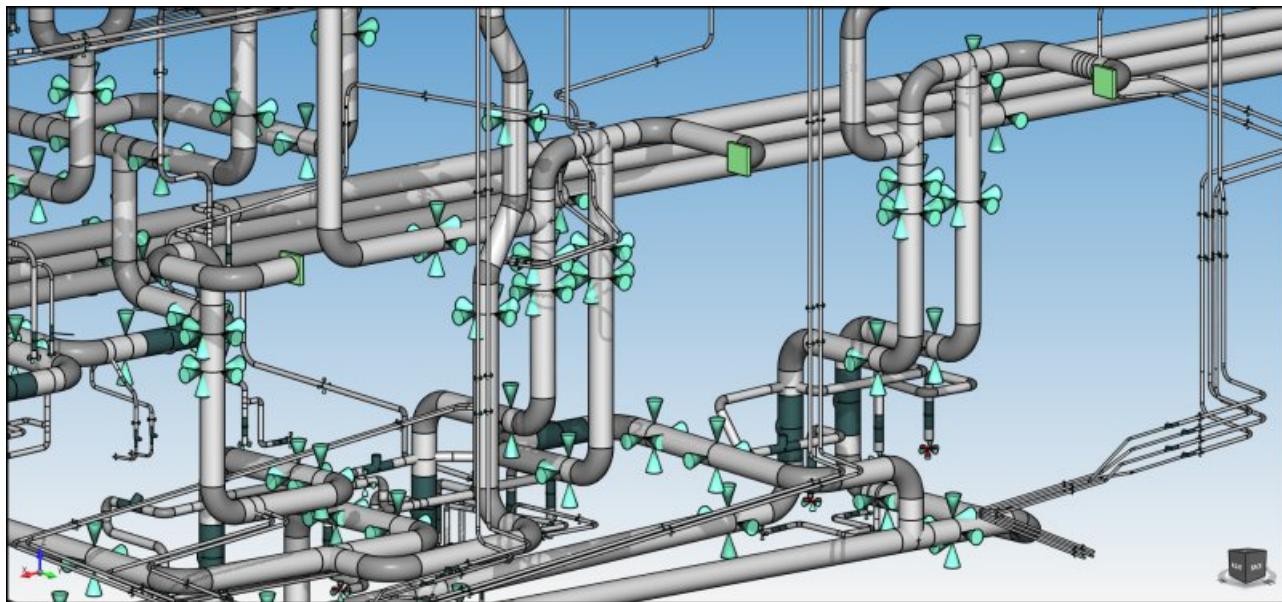


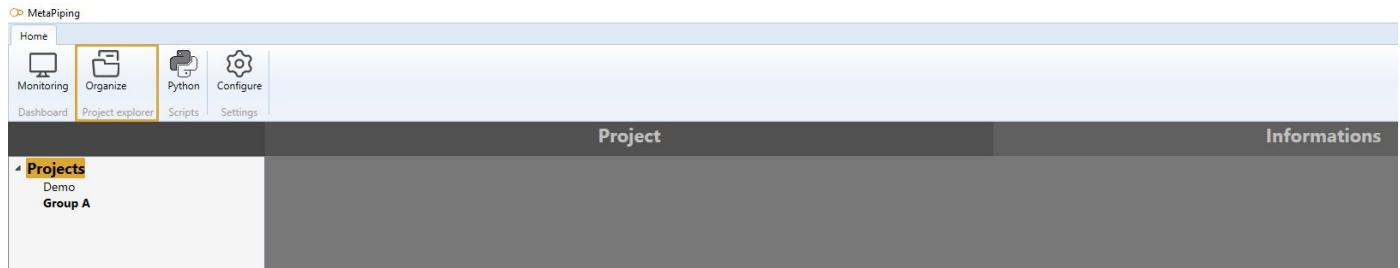
Welcome to MetaPiping User Manual

The new modeling and analysis software for NUCLEAR and INDUSTRIAL piping systems and supports.



Navigate the menu on the left or use the top search bar.

Explorer

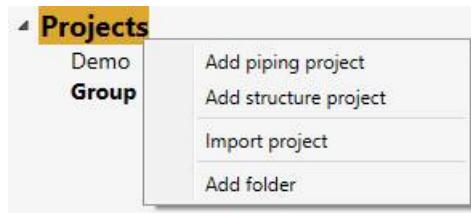


Click on **Organize** button to access the explorer panel.

1. Organizer

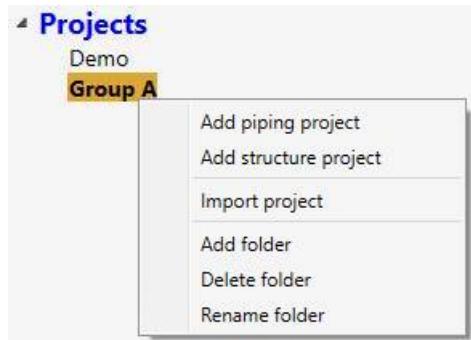
You can organize your projects in the left panel.

Right click on **Projects** lets you :



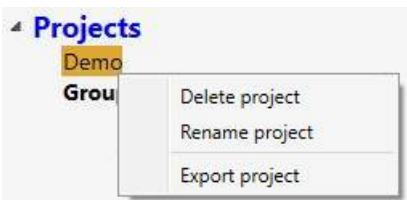
- Add a new piping project (if MetaPiping active on your licence)
- Add a new structure project (if MetaStructure active on your licence)
- Import a project (file with extension *.prockage)
- Add a new folder

Select a **folder** (Ex : Group A). Right click on it lets you :



- Add a new piping project (if MetaPiping active on your licence)
- Add a new structure project (if MetaStructure active on your licence)
- Import a project (file with extension *.prockage)
- Add a new sub-folder
- Delete the current folder with all projects
- Rename the current folder

Select a **project** (Ex : Demo). Right click on it lets you :



- Delete the current project
- Rename the current project
- Export the project (file with extension *.prockage)

WARNING

The EXPORT project command (from source computer) will copy all studies but also all external files, the used python scripts... and will paste it (or install scripts if not exist on target computer) during IMPORT command.

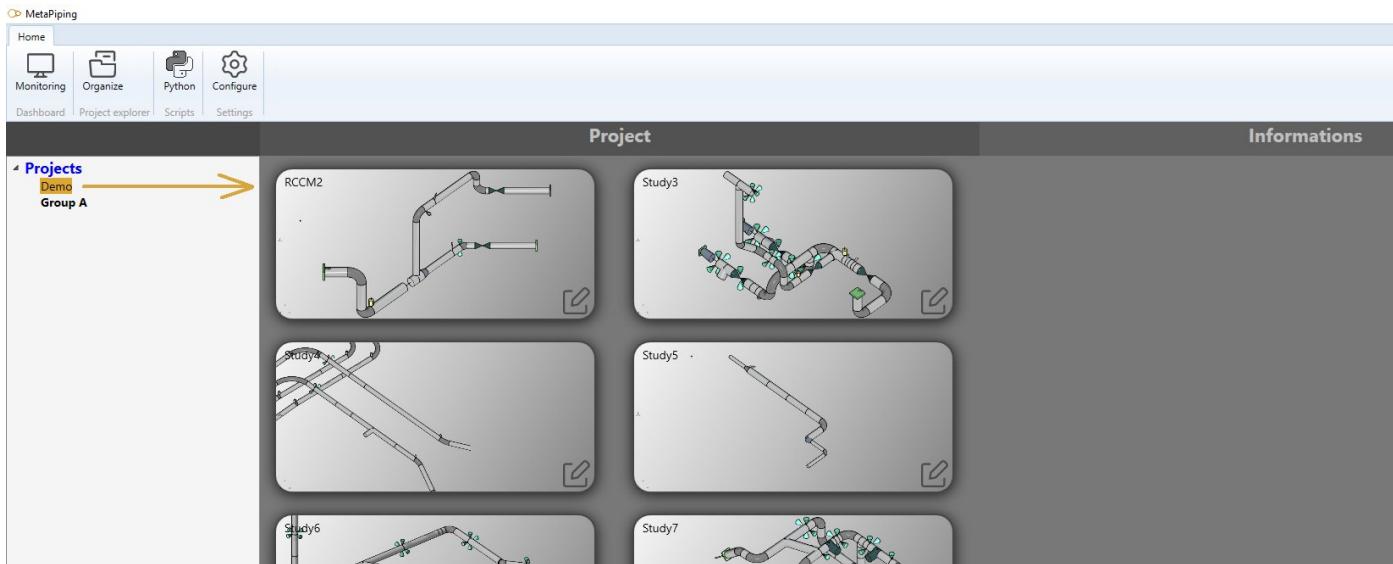
WARNING

The EXPORT/IMPORT project will not work for **distant studies** (those created by opening a FRE file directly from windows explorer) !

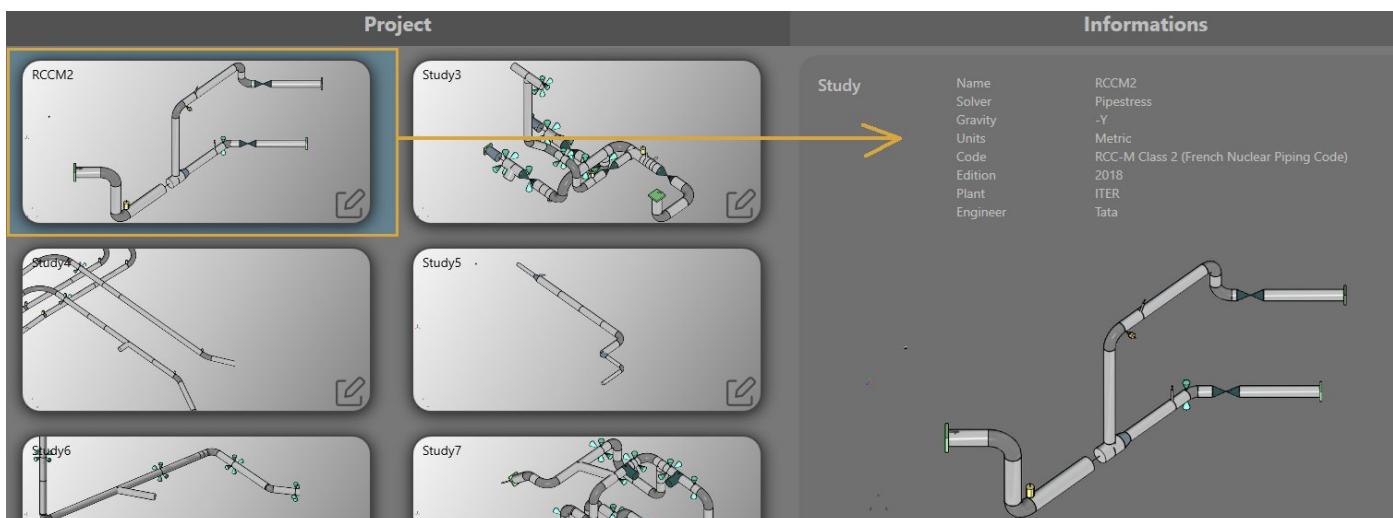
2. Project preview

A **project** is a container of **studies**.

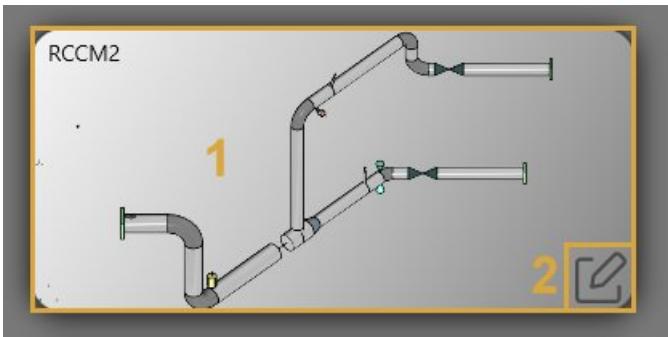
When selecting a project, a preview of all studies is shown :



You can have general information about one study by selecting a thumbnail :



2.1 Project/study edition



To edit a project or a study, just **double click** on the thumbnail of any study (1).

To edit a specific study, **click** on button (2).

3. Python script

A **Python script** can be created to explore the current **MetaL** and **results** of the study and show your own information.

Study	Name	RCCM2
Solver	Pipestress	
Gravity	-Y	
Units	Metric	
Code	RCC-M Class 2 (French Nuclear Piping Code)	
Edition	2018	
Plant	ITER	
Engineer	Tata	

Model	====	
Nb noeuds	41	
Solution	=====	
Max stress ratio	1.928	

Click [here](#) to have more information about python script creation for a project.

Click [here](#) to see how to define a python script as default "information script" in the settings.

4. External opening

User can **Right click** a *.fre file in Windows explorer and ask to open it with MetaPiping.

A project will be created with a study and a complete connection of the internal MetaL to this file + solver PIPESTRESS.

Click [here](#) to have more information about PIPESTRESS **bricks**.

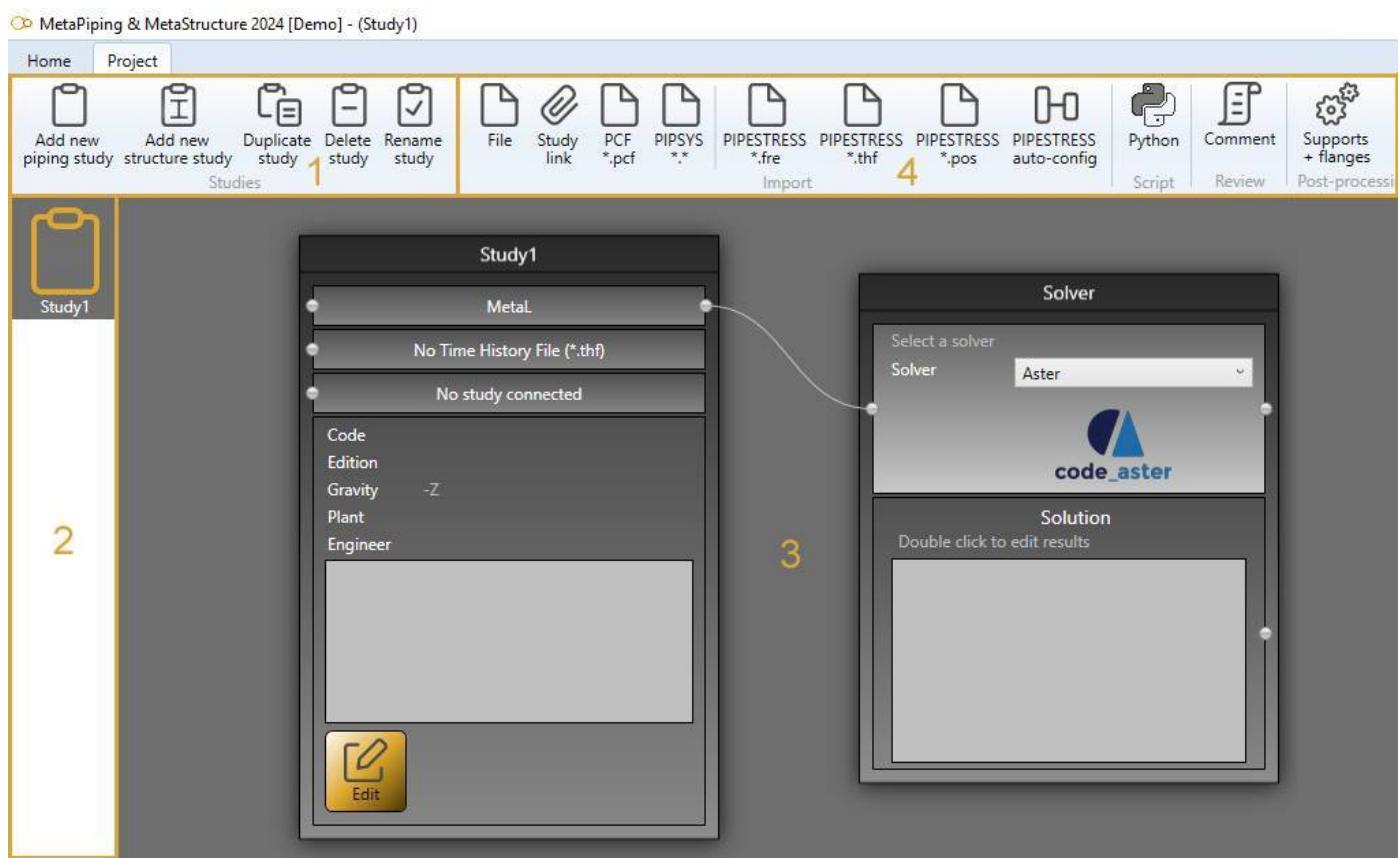
TABLE OF CONTENTS

- [Project](#)
- [Study](#)

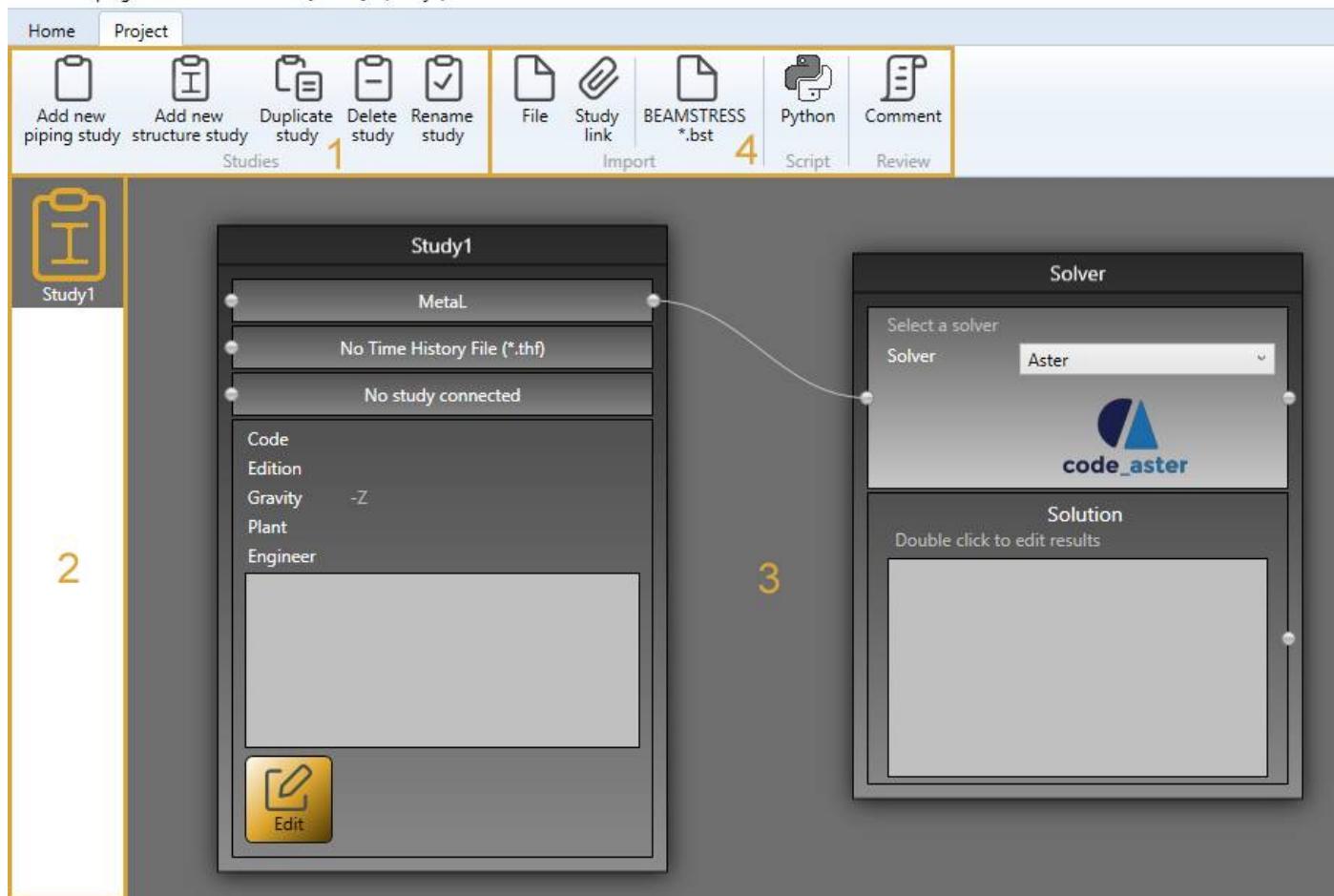
Project

A project is a container of **studies**.

If you create a new **piping project**, the first study will be a piping study :



If you create a new **structure project**, the first study will be a structure study :



The **project** screen consists of 4 zones :

- 1 The **Tools** to manage the studies of the current project
- 2 The **Container** of studies
- 3 The current study **Diagram** - see [study](#) for more information
- 4 The **Bricks** to manage the current study - see [study](#) for more information

1. Tools

1.1 Selection

Select a study in the **container** (2).

This study becomes the **current study** shown in the main panel (3).

1.2 Add

Click on **Add new piping study** to create an empty piping study that will be added to the **container** (2).

Click on **Add new structure study** to create an empty structure study that will be added to the **container** (2).

This new study becomes the **current study** shown in the main panel (3).

1.3 Duplicate

Select a study in the **container** (2) and click to **Duplicate study** to have a full copy of the selected study.

This new study becomes the **current study** shown in the main panel (3).

1.4 Delete

Select a study in the **container** (2) and click to **Delete study** to remove the selected study (and all it contains).

WARNING

ATTENTION, this action is irreversible

WARNING

ATTENTION, You cannot remove the current study if it is the only one

After deleting, the first study of the project becomes the current study shown in the main panel (3).

1.5 Rename

Select a study in the **container** (2) and click to **Rename study** to modify the name of the selected study.

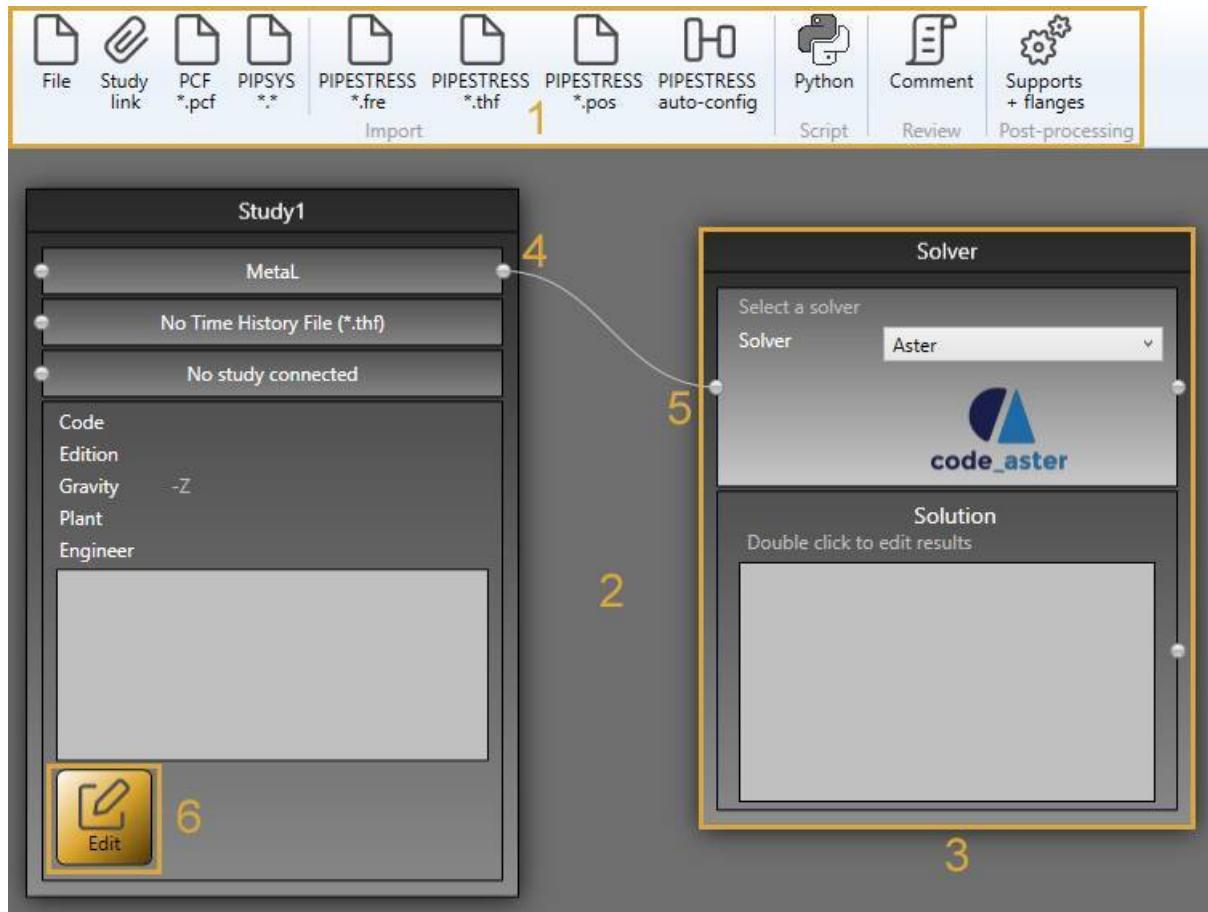
2. Study

Click [here](#) for more information about studies.

Study

A study consists of a **Diagram** of bricks and connections.

It is a folder that contains at least a 3D model and the calculation results.



The **study** screen consists of :

- 1 The **Bricks**
- 2 The main panel that shows the **Study Diagram**
- 3 The **Solver brick**
- 4 A **Connector**
- 5 A **Connector**
- 6 The **Edit** button to open the 3D model

An empty study contains at least one **Study brick** and one **Solver brick**.

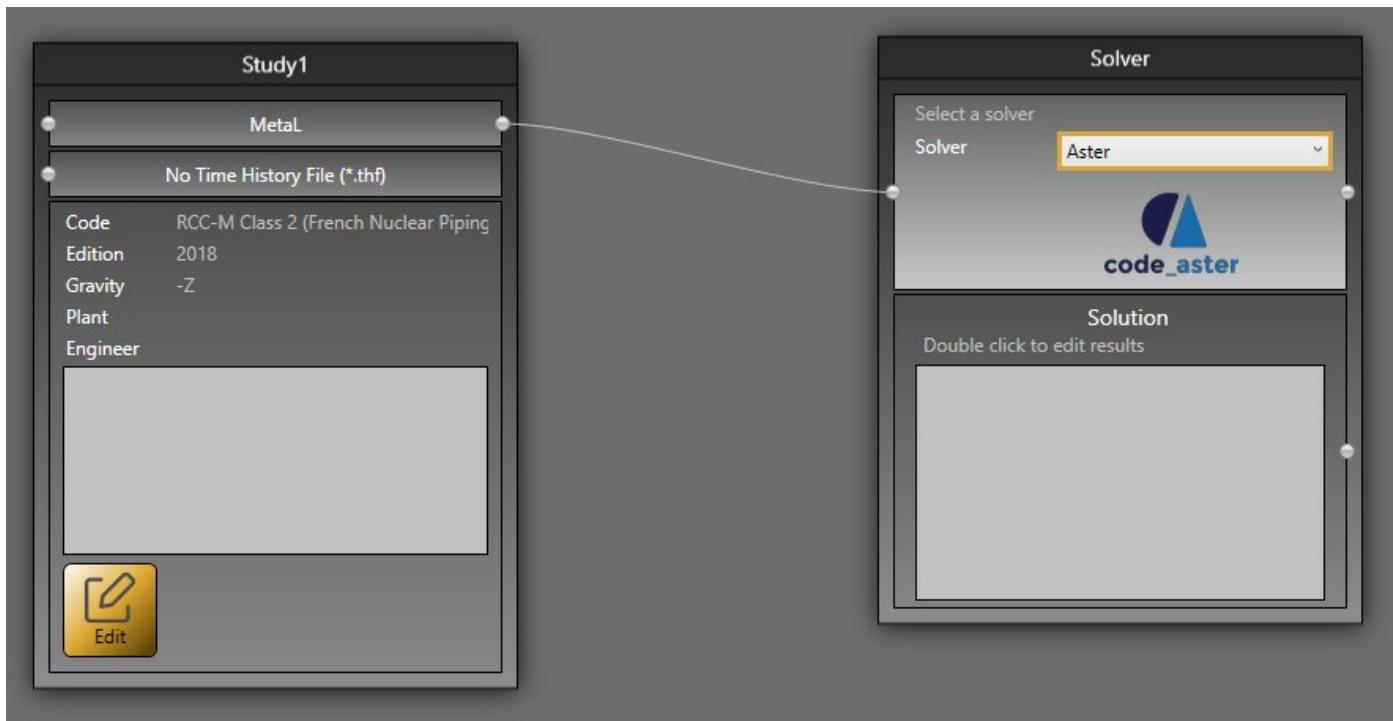
1. Connections

Connect the connectors 4 and 5 to define the current **Solver**.

```
Press left mouse button above connector 4 -> move the mouse to connector 5 -> release the left button
```

```
To remove the connection : select the connection and press DELETE on the keyboard.
```

Select **Aster** for example :



Now, the project will be calculated with **Code_Aster Solver**

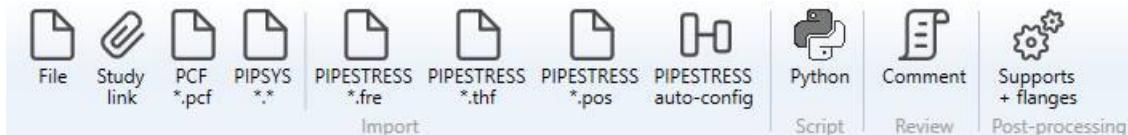
Click [here](#) to have more information about Code_Aster.

2. Empty model

Click **Edit** (6) to start a new design.

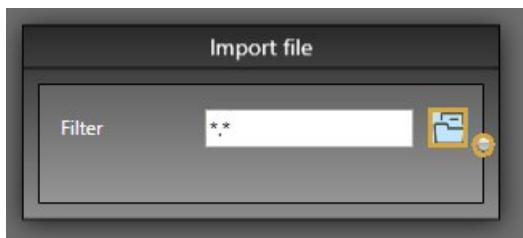
Click [here](#) to have more information about the design.

3. Piping bricks



3.1 File

This creates a brick that lets you import a file of any extension.



This is useful only for Python scripts that need a file as input (Ex : Excel file or Text file).

Click on the button to open a search file dialog.

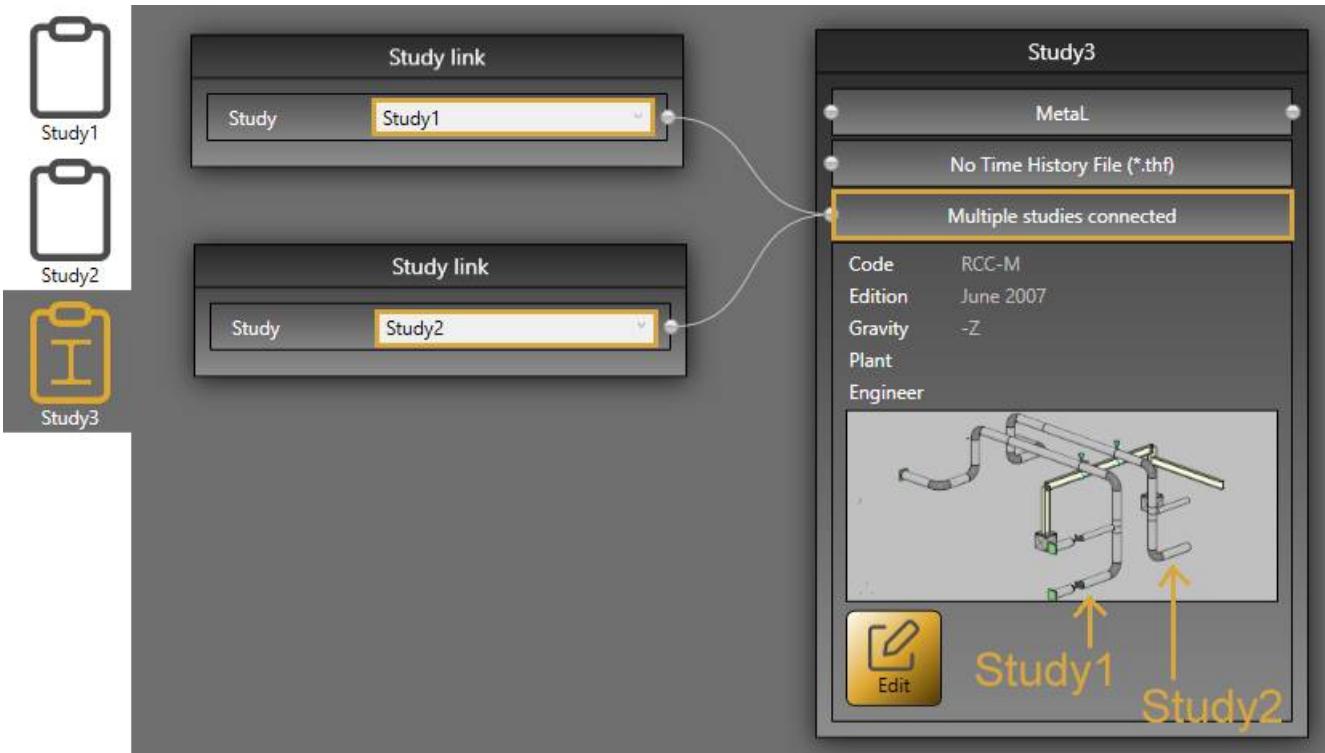
Use the connector to send the file to another brick as an input.

WARNING

You can import another **Metal** file and connect it to the **Metal** connector of the **Study brick**.

3.2 Study link

This creates a brick that virtually replicates existing studies inside another one. This is useful to create new nodes based on those of the linked studies.



WARNING

You can connect multiple studies. All must be defined in the same coordinate system XYZ.

WARNING

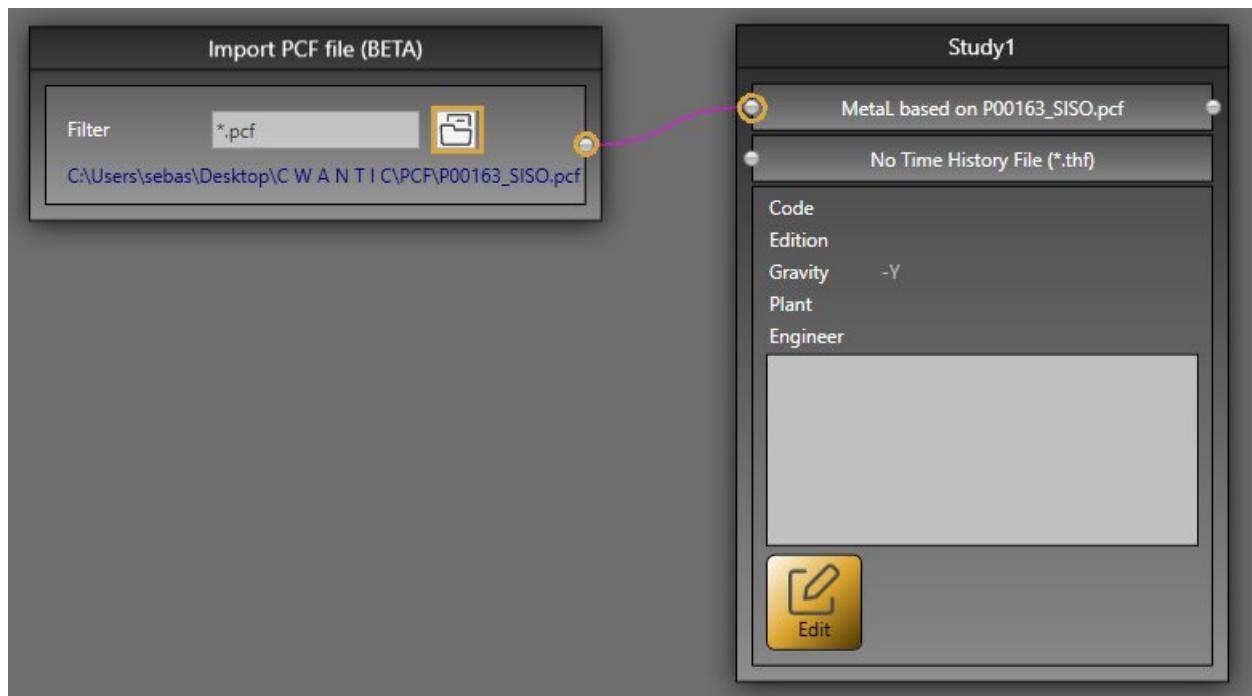
In design, you can hide the linked studies.

WARNING

In design, you cannot modify elements of linked studies.

3.3 PCF

The **plugin** PCF lets you import *.pcf files.

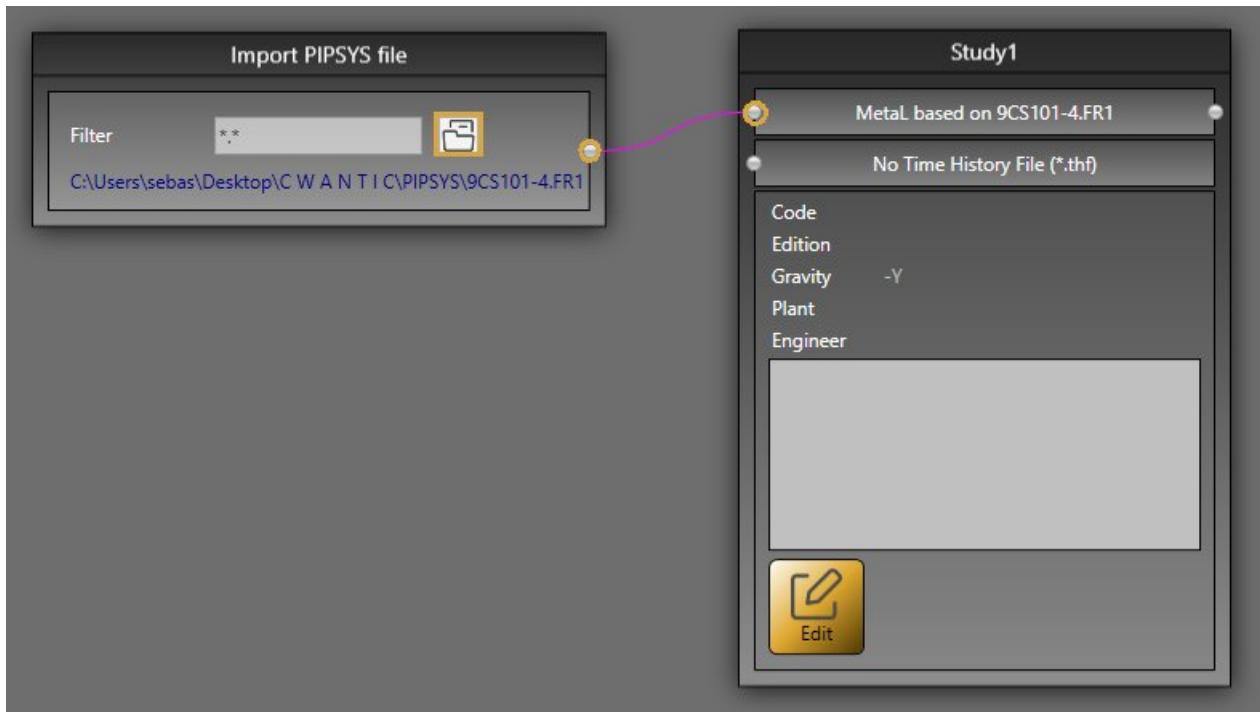


Open a file and connect it to the **MetalL** connector of the **Study** brick.

This will convert the PCF to the MetalL file format.

3.4 PIPSYS

The **plugin** PIPSYS lets you import files with several extensions.



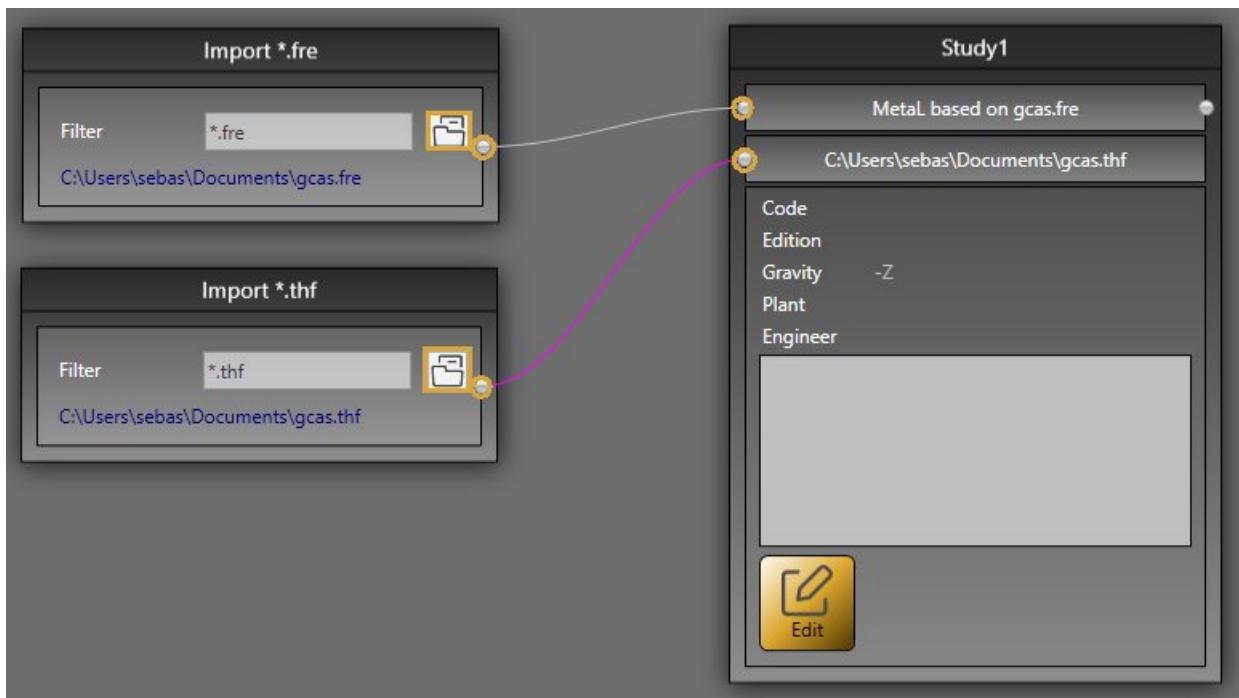
Open a file and connect it to the **Metal** connector of the **Study brick**.

This will convert the PIPSYS to the MetaL file format.

3.5 PIPESTRESS

The **plugin** PIPESTRESS lets you import *.fre (and *.thf files).

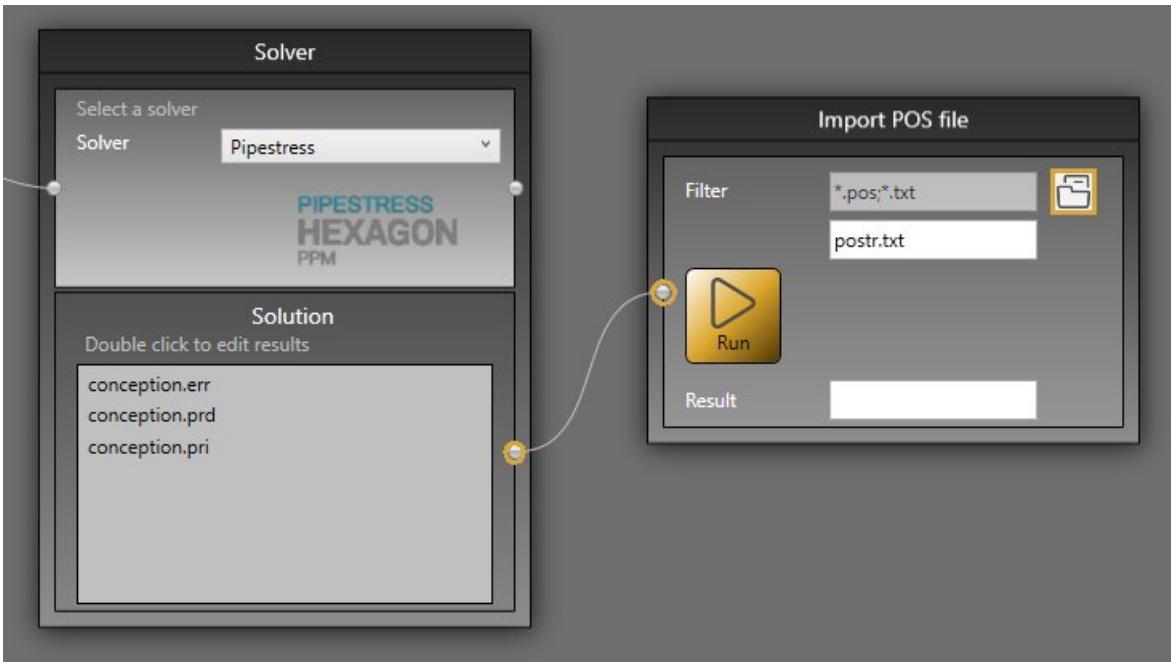
The Time History File is not mandatory and depends on the corresponding FRE file.



Open a file and connect it to the **Metal** connector of the **Study brick**.

This will convert the FRE to the MetaL file format.

The **plugin** PIPESTRESS also lets you import POSTR files for postprocessing.

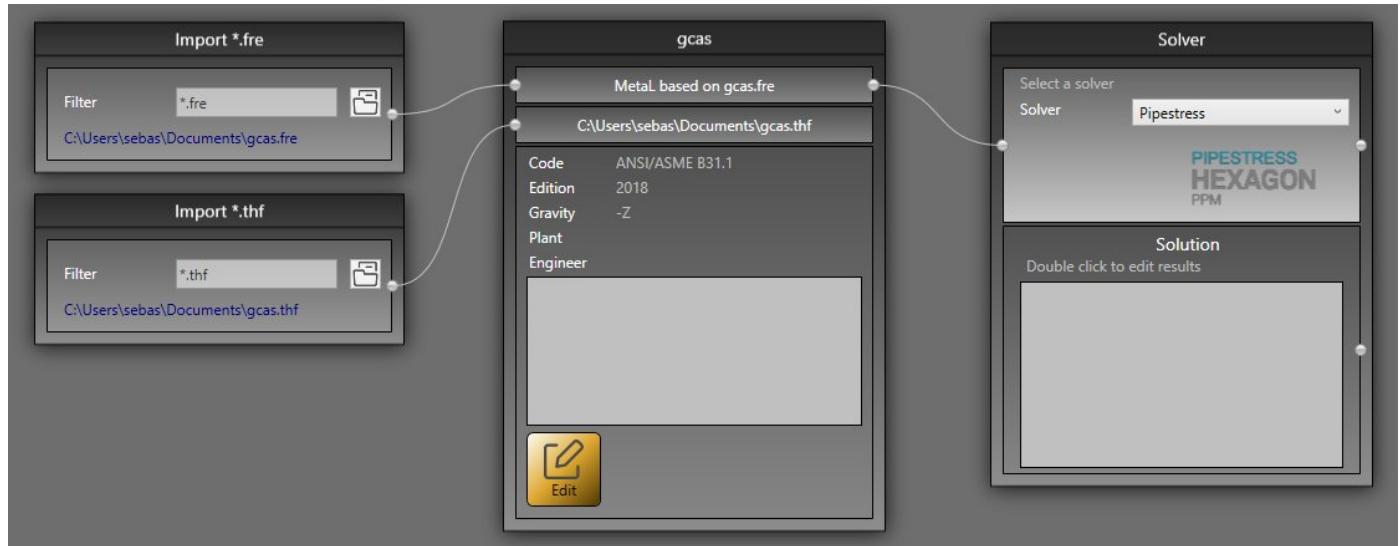


Open a file and connect it to the **Solution** connector of the **Solver brick**.

It will produce a text file (in the result cell) that can be edited by **double click**.

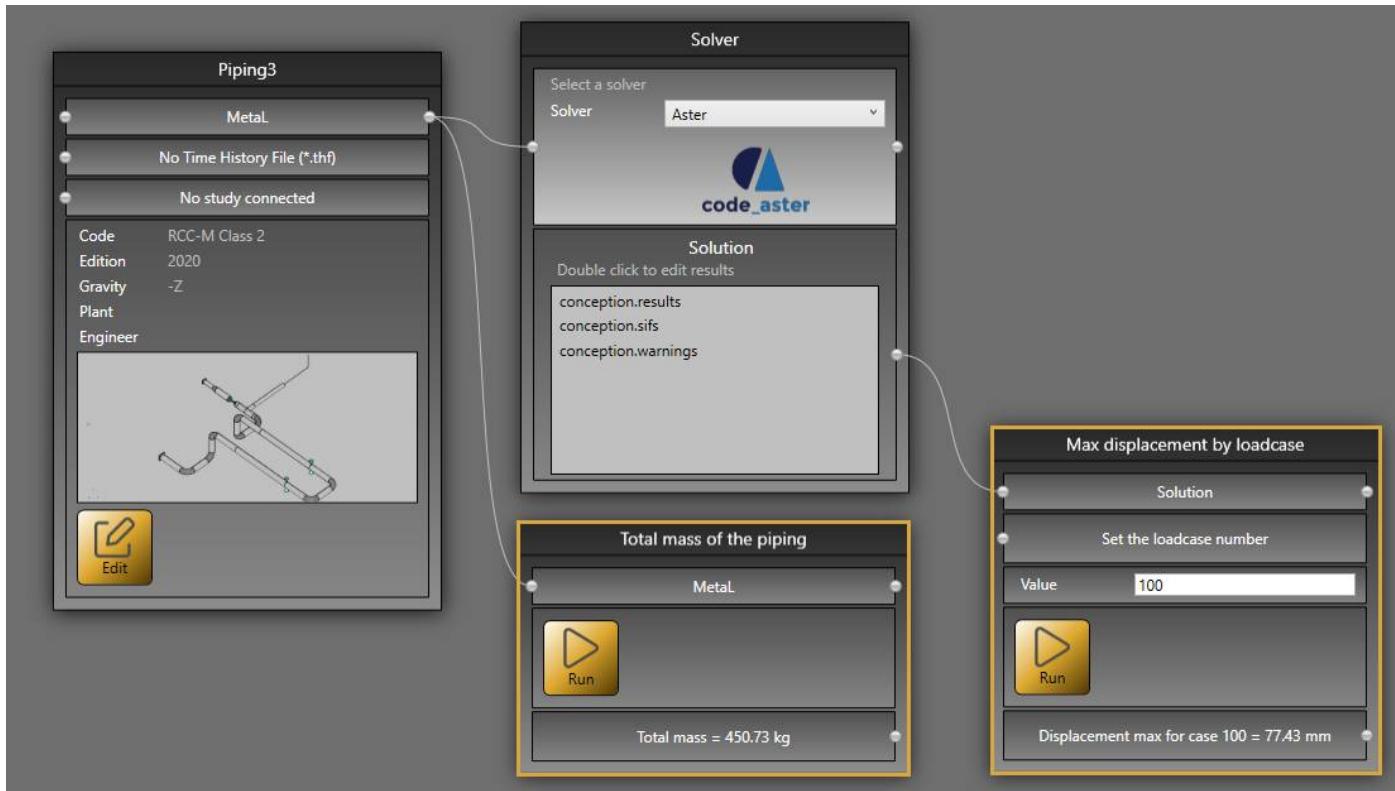
You can specify your favorite text editor in the settings.

The **plugin PIPESTRESS** also lets you configure all bricks by just importing a *.fre file. It checks if a *.thf file is needed, sets the Solver to PIPESTRESS and makes all connections automatically.



3.6 Python

You can create your own **brick** thanks to the Python scripts.



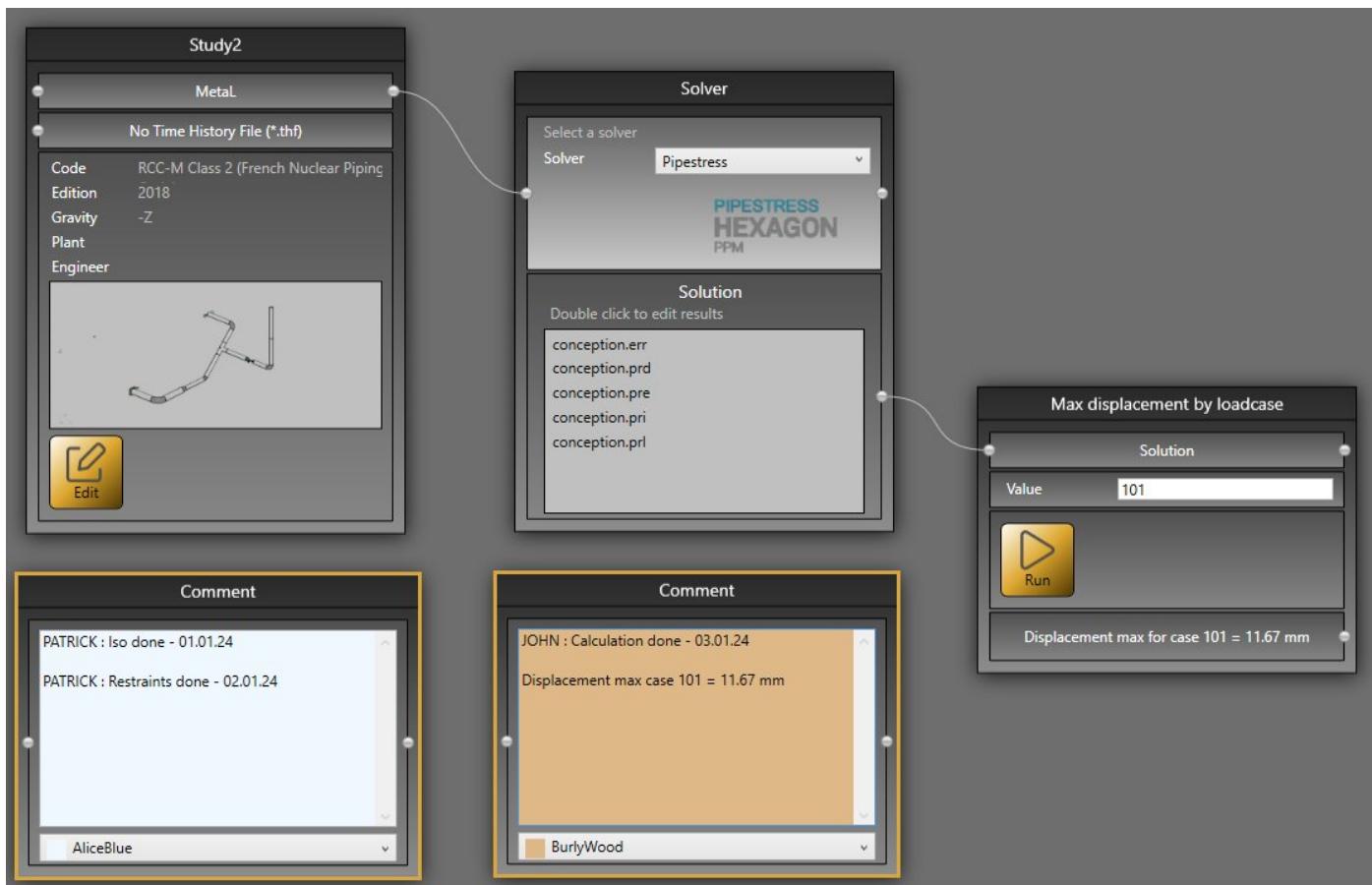
Example of a brick connected to the Metal and another one connected to the solution.

Click [here](#) to have more information about the script creation.

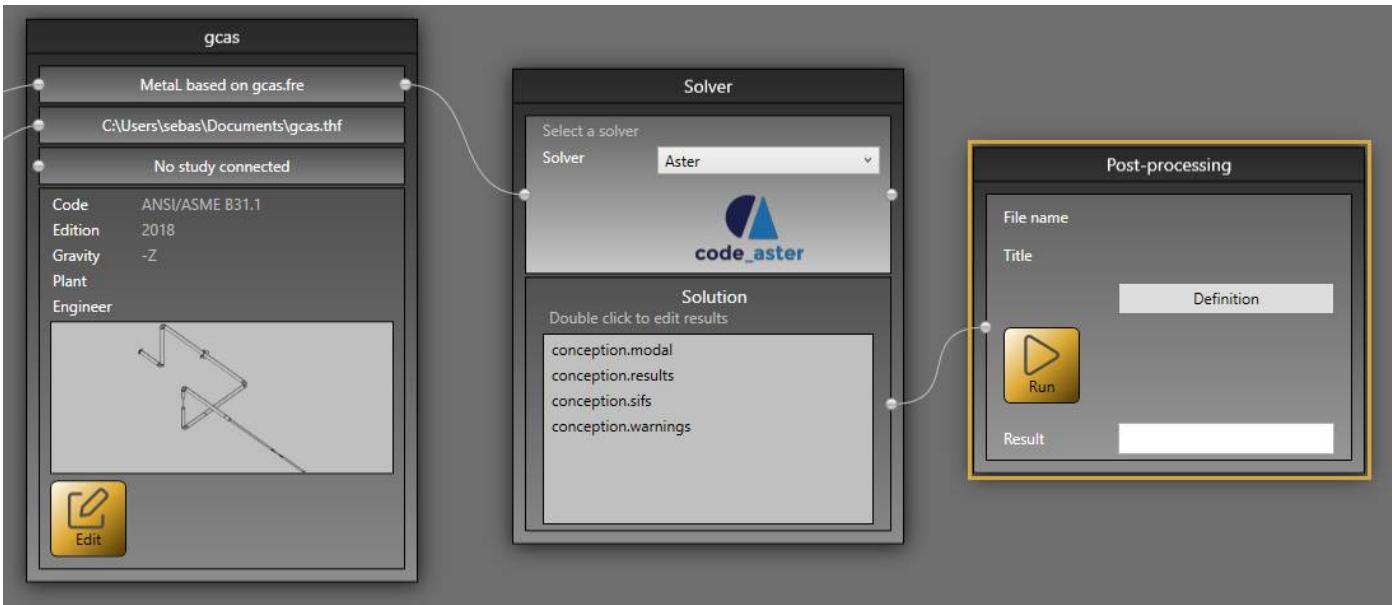
3.7 Comment

This creates a brick with text and color capabilities.

Useful to show the state of progress of the study to other users. A conventional color code can be established.

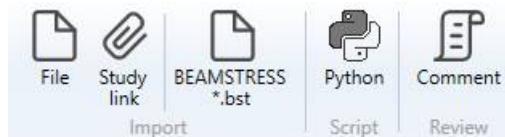


3.8 Post-processing : supports + flanges



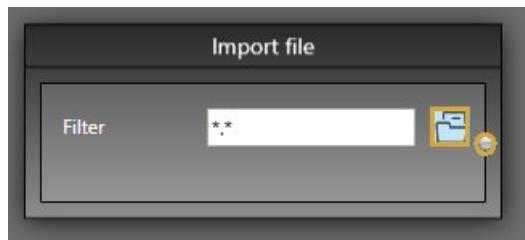
Documentation will come soon...

4. Structure bricks



4.1 File

This creates a brick that lets you import a file of any extension.



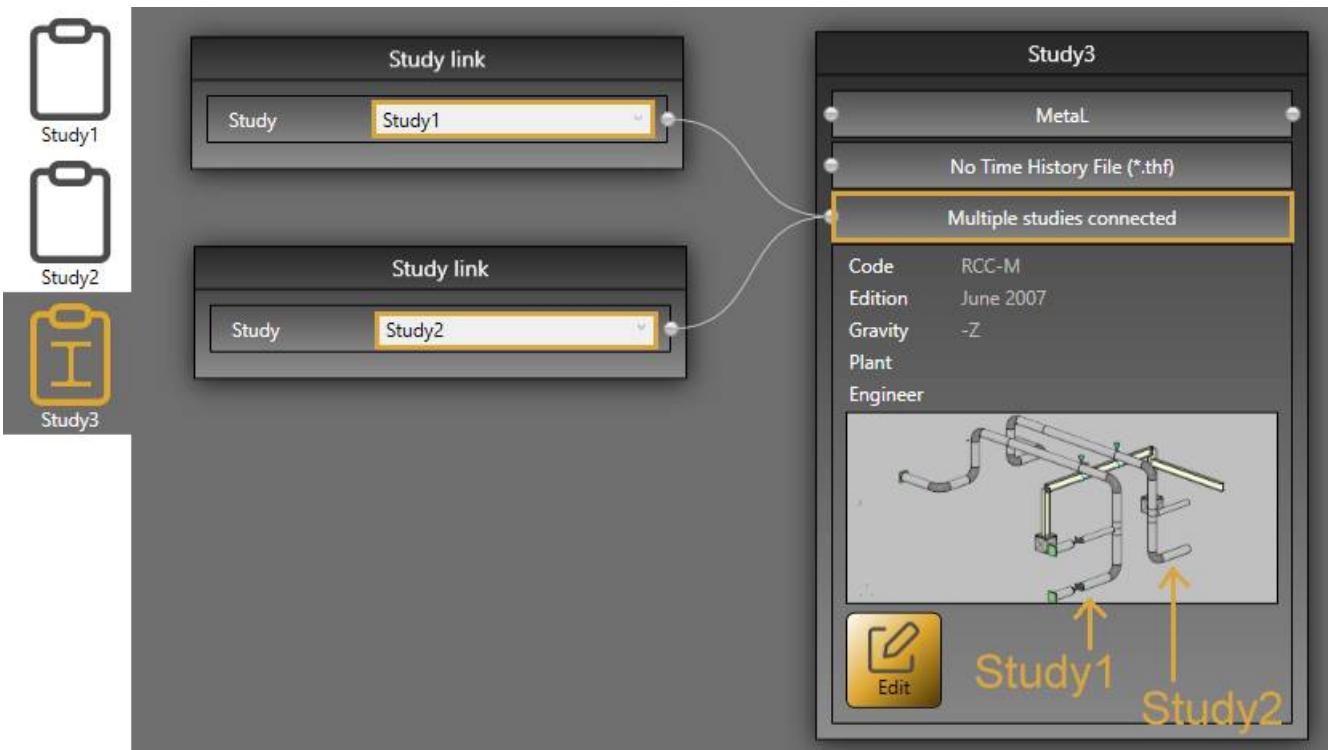
This is useful only for Python scripts that need a file as input (Ex : Excel file or Text file).

Click on the button to open a search file dialog.

Use the connector to send the file to another brick as an input.

4.2 Study link

This creates a brick that virtually replicates existing studies inside another one. This is useful to create new nodes based on those of linked studies.



WARNING

You can connect multiple studies. All must be defined in the same coordinate system XYZ.

WARNING

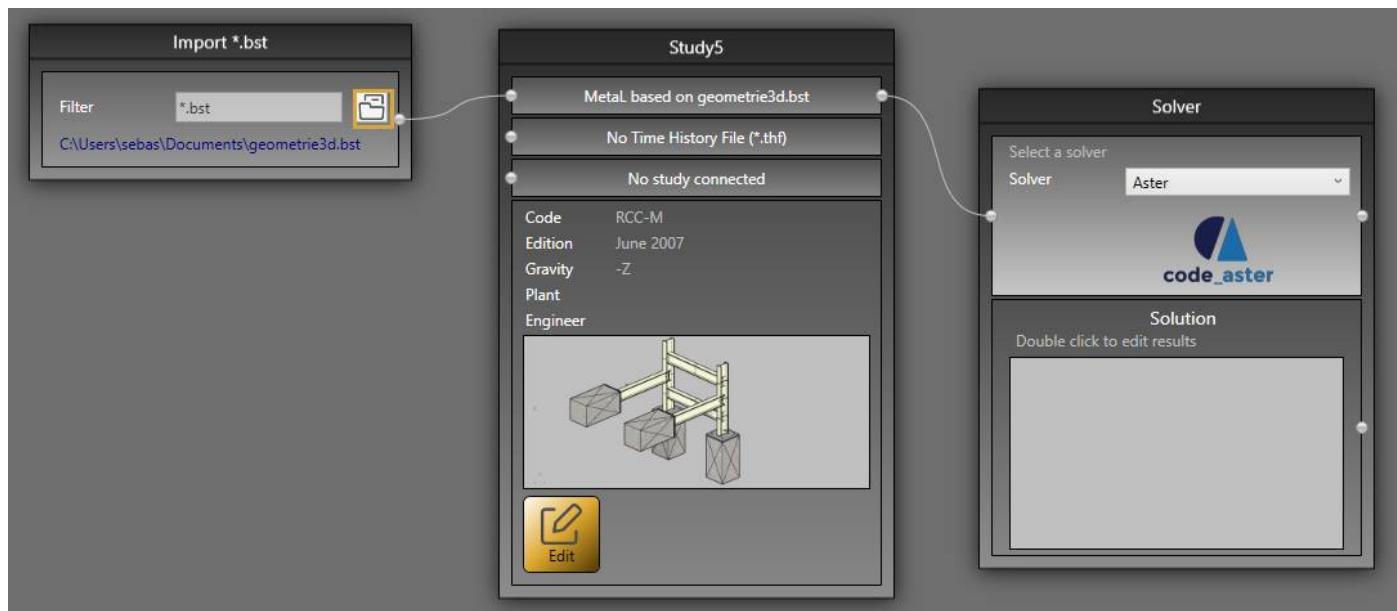
In design, you can hide the linked studies.

WARNING

In design, you cannot modify elements of linked studies.

4.3 BEAMSTRESS

This creates a brick that lets you import *.bst files that are converted to MetalL file.



WARNING

ATTENTION, only version 2.0.0 or higher *.bst files can be imported. If the file version is too old, we recommend to open it with a recent BEAMSTRESS and save the model again.

4.4 Python

You can create your own **brick** thanks to the Python scripts.

Click [here](#) to have more information about the script creation.

4.5 Comment

This creates a brick with text and color capabilities.

Useful to show the state of progress of the study to other users. A conventional color code can be established.

Python integration

MetaPiping offers to users the possibility to manipulate data produced by the software in order to operate custom treatment.

These manipulations can be done via **Python** scripts, a simple and well documented language, with great free mathematical, scientific and computer libraries, appreciated by engineers.

A script contains one or more *.py files and can use a **python virtual environment**.

Main.py is mandatory and represents the main access to the script.

WARNING

ATTENTION, the use of libraries takes place on the disk ! To be used with care.

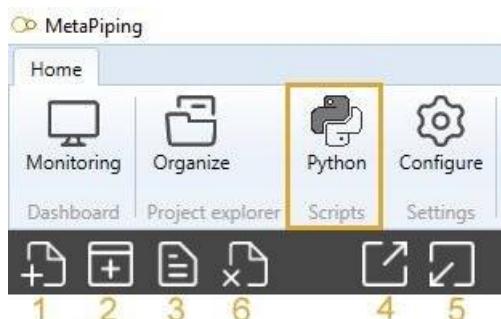
Script types

The integration can be done at 3 different levels :

- Explorer phase- [See Info page for more information](#)
- Project/study phase - [See Study page for more information](#)
- Design phase - [See Design page for more information](#)

Main menu

Select **Python** in the application ribbon :

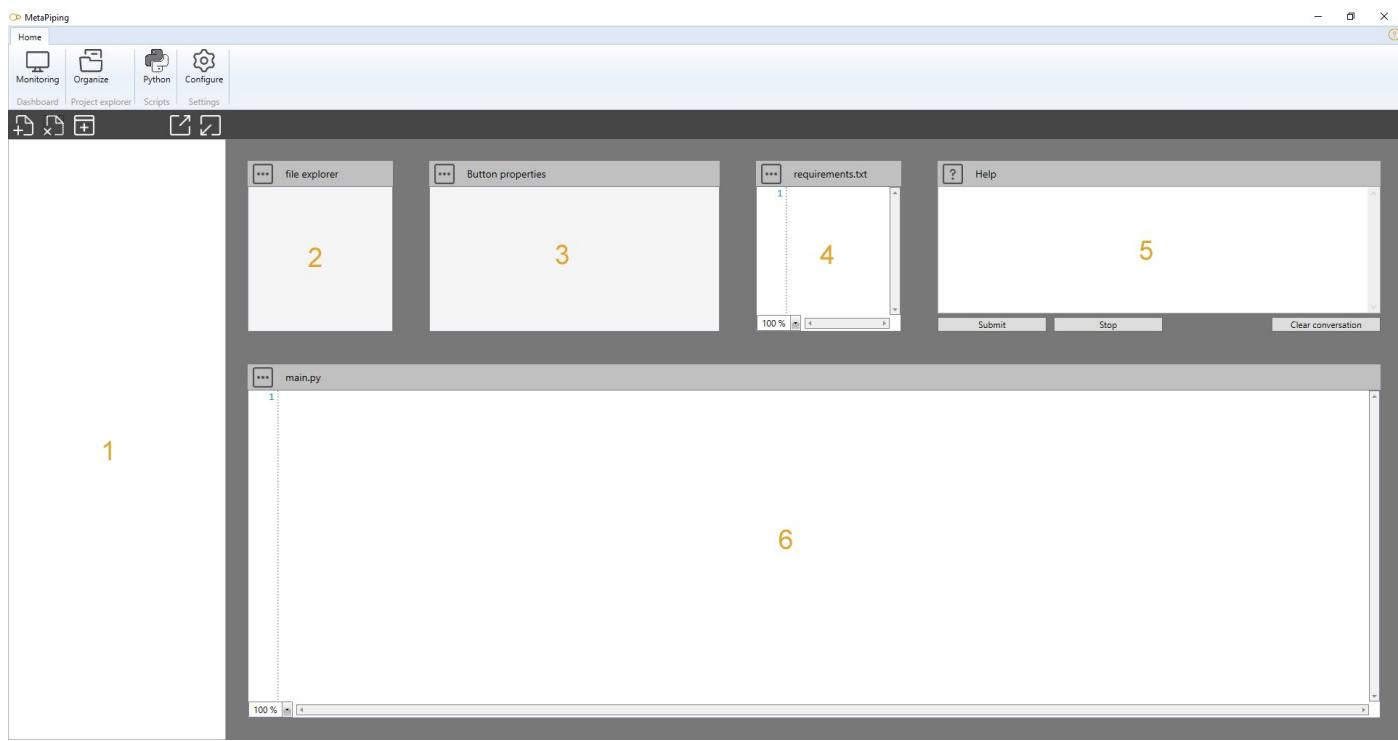


On the upper left menu, a list of buttons proposes several tools :

- 1 Create a script for **study** level - [See Study page for more information](#)
- 2 Create a script for **design** level - [See Design page for more information](#)
- 3 Create a script for **explorer** level - [See Info page for more information](#)
- 4 Export a script - [See Pyckage page for more information](#)
- 5 Import a script - [See Pyckage page for more information](#)
- 6 Remove the selected script

Screen

The screen is made up of different parts :



- 1 The list of all scripts (of all types)
- 2 The files (and directories) used in the script - selecting a file shows it in the editor (6)
- 3 The properties of the script (Type, Name, Input, Output...)
- 4 The **requirements.txt** of the eventual virtual environment of the script
- 5 The chatGPT help - [See chatGPT page for more information](#)
- 6 The Python **Editor** - or image viewer if you select an image in the file explorer (2)

WARNING

ATTENTION : The installation of libraries and the chatGPT help needs an internet connection !

TABLE OF CONTENTS

- [Configuration](#)
- [Study](#)
- [Design](#)
- [Info](#)
- [Classes](#)
- [Pyckage](#)
- [ChatGPT](#)
- [Samples](#)

Configuration

The **Python integration** in MetaPiping is automatically made during installation but you have the possibility to manually change some settings.

1. Python

MetaPiping 2024 is installed with Python version :

| Python 3.7

The directory of Python sources is :

| C:\ProgramData\Cwantic\MetaPiping\Settings\Pythons

This directory cannot be changed.

2. Scripts

The user's scripts are by default located in the directory :

| C:\ProgramData\Cwantic\MetaPiping\Datas\Scripts

Ce PC > Disque local (C:) > ProgramData > Cwantic > MetaPiping > Datas >		
Nom	Modifié le	Type
Databases	16-02-23 16:37	Dossier de fichiers
Scripts	16-02-23 16:03	Dossier de fichiers
Specifications	16-02-23 16:02	Dossier de fichiers

This **Datas** directory contains all user's data like :

- Databases
- Python Scripts
- Specifications

All user's data can so be shared between users.

This directory can be changed via the configuration settings :

The screenshot shows the MetaPiping application interface. At the top, there is a navigation bar with tabs: Home, Monitoring, Organize, Python, Configure (which is highlighted), and Settings. Below the navigation bar, there is a sidebar with icons for Sources, General, Databases, and Specifications. The main area is titled 'Sources' and contains a table with two sections: 'PROJECTS SOURCES' and 'DATA SOURCES'. Under 'PROJECTS SOURCES', there is a 'Directories' section with 'Directory 1' and a 'Current working directory' entry. Under 'DATA SOURCES', there is a 'Directories' section with 'Directory 1' and a 'Current data directory' entry. Both entries in the 'Current data directory' section are highlighted with orange boxes.

NAME	VALUE
PROJECTS SOURCES	
Directories	
Directory 1	
Current working directory	C:\ProgramData\Cwantic\MetaPiping\Projects C:\ProgramData\Cwantic\MetaPiping\Projects
DATA SOURCES	
Directories	
Directory 1	
Current data directory	C:\ProgramData\Cwantic\MetaPiping\Datas C:\ProgramData\Cwantic\MetaPiping\Datas

[See the settings page for more information](#)

WARNING

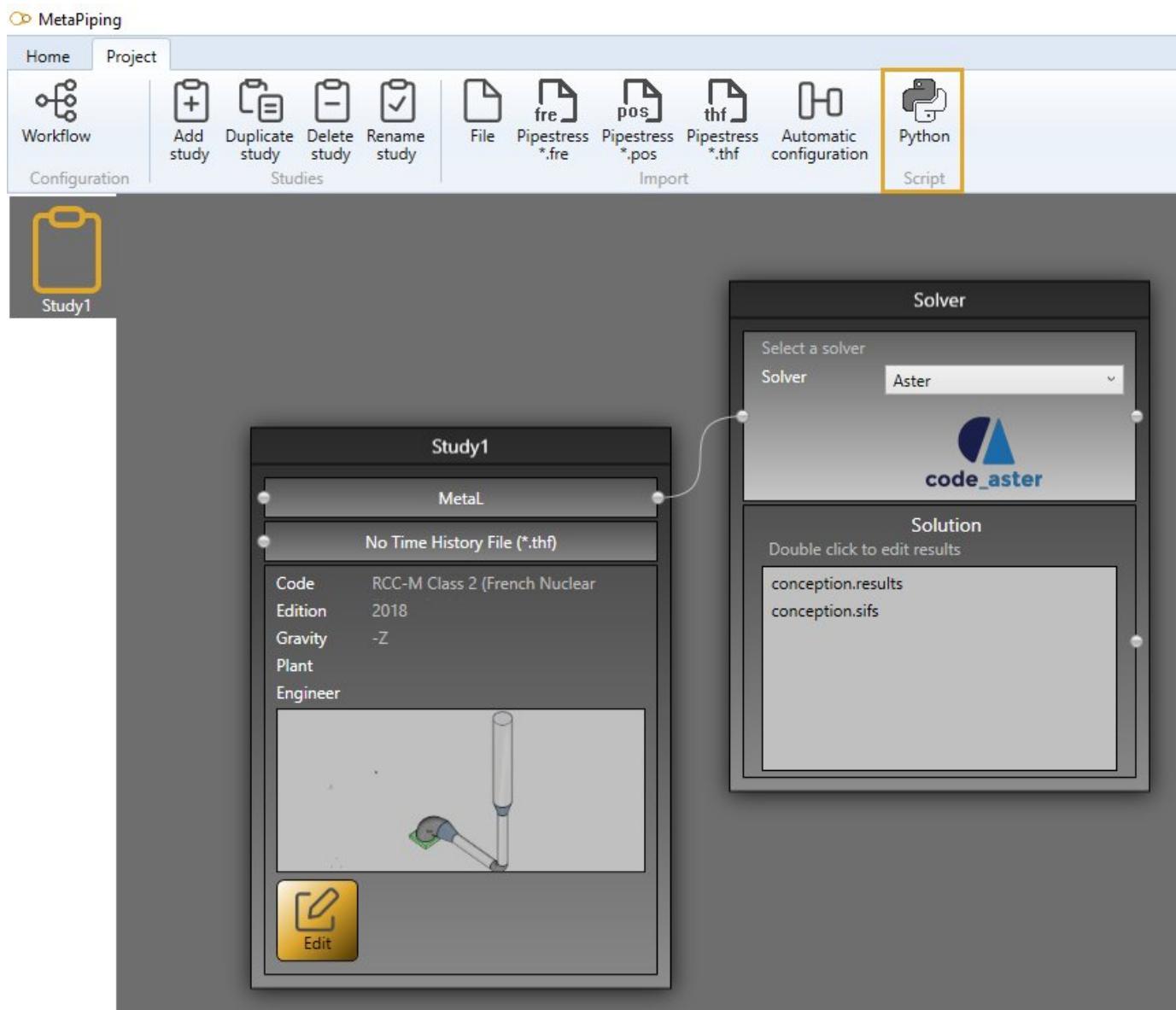
Attention ! Changing the data directory could break the link between Python scripts and projects.

Study

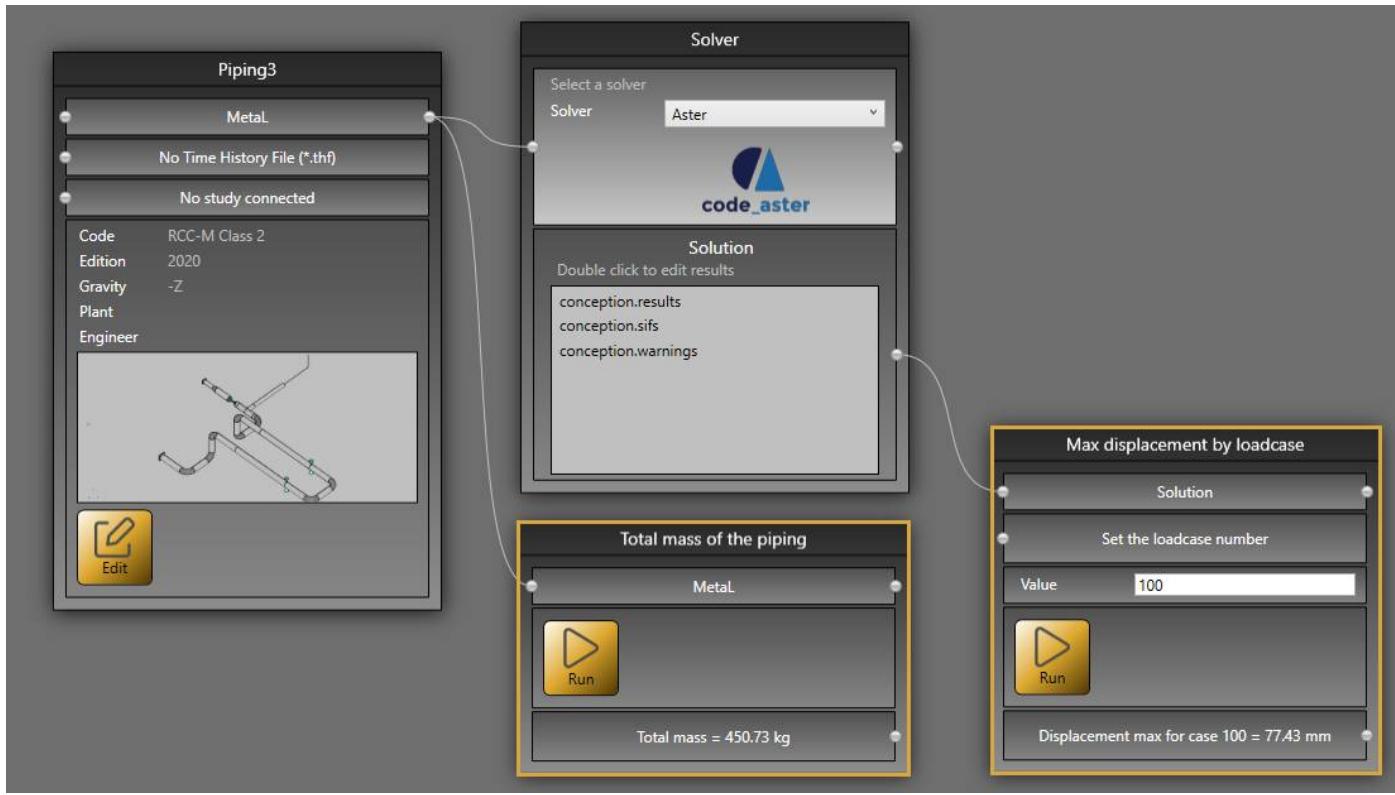
A study is a workflow of **bricks** and **connections** - [See Study page for more information](#)

User can create its own **bricks** via Python scripts.

Click on **Python** in application ribbon in the current study :



Select a script via the dropdown list :

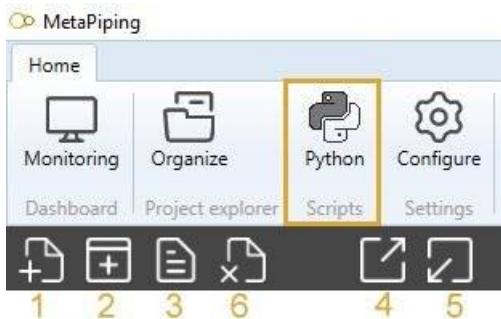


You can create scripts that are independent of the current study, you can **inject** data to the internal model, you can **extract** data from the internal model, or you can **extract** data from the solution.

Let's see how to create this kind of script.

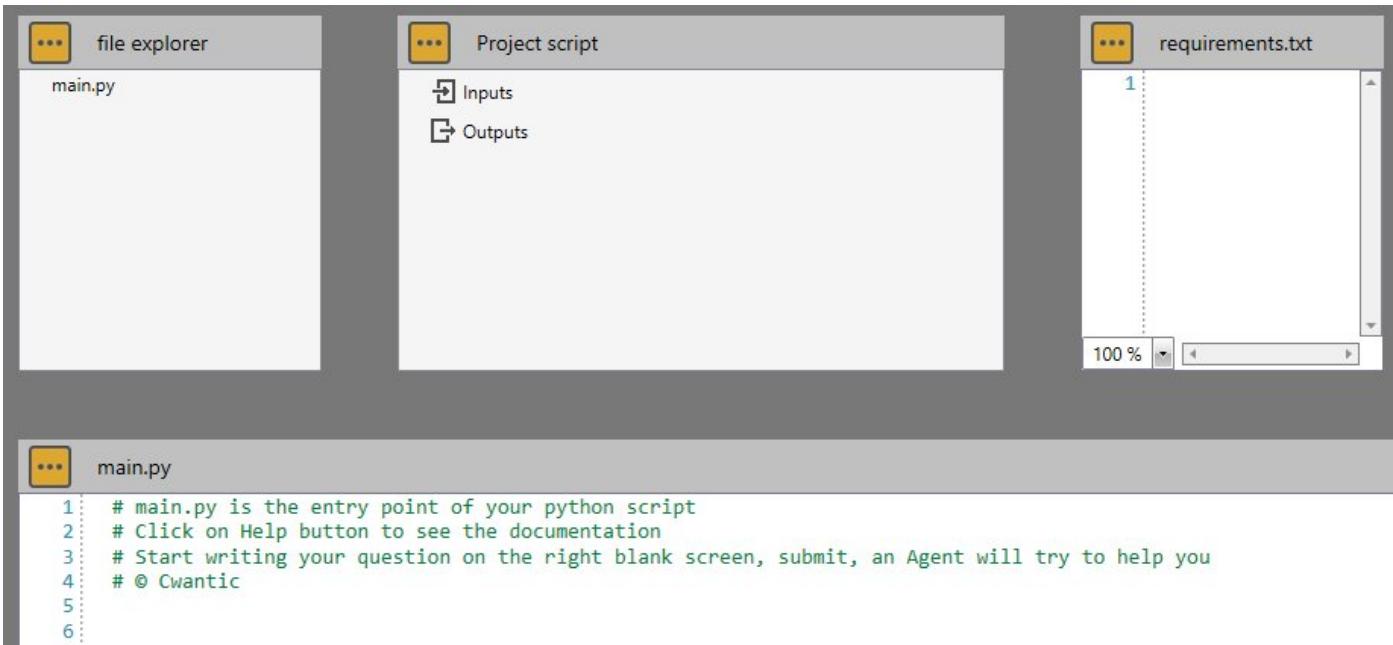
1. Script definition

From the Home/Python, click on button 1 (Add project script) :



Give it a name and answer **YES** if you need **Python Virtual Environment** for libraries. In this case, wait until the Virtual Environment is generating...

A new *main.py* will be created in the file explorer and *Inputs* and *Outputs* nodes in the Project script. Requirements.txt and the Editor are empty :



Requirements.txt is hidden if you answer **NO** to the question about **Python Virtual Environment**

Buttons with three points indicates the presence of a dropdown list :

In file explorer :

- Add a python file : Add an empty *.py file in the script directory
- Import image : Open a dialog to import *.jpg/*.png image
- Open folder in explorer : Open the current script folder
- Refresh : Refresh the file explorer to show the new imported files
- Remove selected file : Remove the selected file in the file explorer

In Project script :

- Add TEXT input : Add a text in the script brick
- Add EDIT input : Add an edit in the script brick where user can write values
- Add METAL input : Add an access to the internal structure metal
- Add SOLUTION input : Add an access to the solution
- Add TEXT output : Add an output text in the script brick
- Add FILENAME output : Add an output filename that can be edit by the user by double click
- Add CONSOLE output : Add an output text list in the script brick
- Save : Save the script definition
- Remove : Remove the selected item

The TEXT input can only receive an information by connection from a TEXT output of another script.

In requirements.txt :

- Save : Save the file
- Install : Install the libraries defined in the file

This file must contain the name and version of the Python libraries used in the script. MetaPiping will install all libraries by internally doing :

```
pip install -r requirements.txt
```

[See the specification of the file for more information](#)

In main.py

- Save : Save the current python file

A *study script* brick contains a **RUN** button that will execute the **main.py** file.



In the Editor, you have access to the object **study** that exposes some functions. A dropdown menu will appear after writing :

```
study.
```

```
1 # main.py is the entry point of your python script
2 # Click on Help button to see the documentation
3 # Start writing your question on the right blank s
4 # © Cwantic
5
6 study.
    Outputs[]
    getMetal()
    createMetal()
    getSolution()
    getDirectory()
    getMaterialLibraryModel()
    createMaterialLibraryModel()
    showInformation()
```

[See the description of the object study](#)

2. Independent script

This kind of script doesn't need to have access to the files of the current study/project/metaL/solution but only some interaction with the user.

It can for example ask some data from the user via INPUT **EDIT** and give answers after treatment via OUTPUTS **TEXT**.

[See an example of independent script](#) : Date, time and file

[See another example of independent script](#) : Send an email

3. MetaL injection

Based on the explained [MetaL](#) internal structure, user can create his own **CONVERTERS** from other file formats or from company's internal data.

Cwantic has created **PLUGINS**, with the same principle, that converts **PIPESTRESS**, **PIPSYS** and **PCF** file format to MetaL file format.

User can for example inject **LOADING TEMPLATES** in current project based on his own data.

[See an example of a CONVERTER from Excel file to Metal](#) : Converter

4. MetaL extraction

Based on the explained [MetaL](#) internal structure, user can create his own script to retrieve informations of the model.

[See an example of MetaL data extraction](#) : Calculate the total mass of a piping

[See another example of MetaL data extraction](#) : Get all static cases of a piping

5. Solution extraction

Based on the explained [Solution](#) internal structure, user can create his own script to retrieve informations of the solution.

[See an example of Solution data extraction](#) : Get the max displacement by static case number

Design

The **Design** mode is the part of the application dedicated to :

- Design : model the piping, configure the settings
- Loads : create several types of loads
- Review : review everything before launching the calculation
- Analysis : analyze the piping

MetaPiping 2024 gives an access to **Design** and **Review**.

The other parts will come soon...

1. Goal

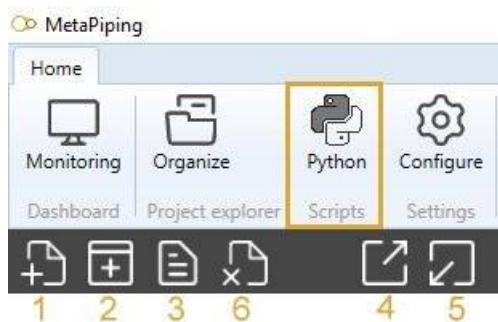
The goal of this kind of script is to create functions that doesn't exist in the application.

User can create his own **commands** to "interact" with the model and especially with the **selection** object.

He can add buttons on ribbons with a name, a description, an image and of course a **script** (main.py).

2. Script definition

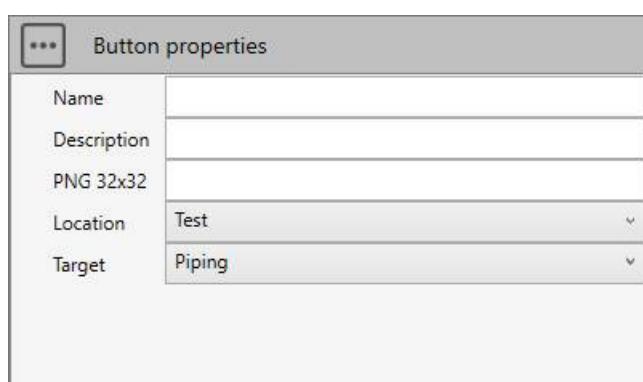
From Home/Python, click on button 3 (Add 3D script) :



Give it a name and answer **YES** if you need **Python Virtual Environment** for libraries. In this case, wait until the Virtual Environment is generating...

The whole screen and the dropdown lists are the same as the [study](#) scripts.

The only change is the **Button properties** :



This 32x32 button will appear on the possible locations :

- Test : nowhere, only for test purpose
- Design : In Design ribbon
- Loads : In Loads ribbon

- Review : In Review ribbon
- Analysis : In Analysis ribbon

The Target can be **Piping** or **Structure**.

A new *main.py* will be created in the file explorer and requirements.txt and the Editor will be empty.

In the Editor, you have access to the object **design** that exposes some functions. A dropdown menu will appear after writing :

```

design.

# main.py is the entry point of your python script
# Click on Help button to see the documentation
# Start writing your question on the right blank sc
# © Cwantic

design.

createCommand()
executeCommand()
getType()
isType()
result
selectedList
getVerticalVector()
getCurrentLayerID()

```

[See the description of the object design](#)

3. Design script

If a tool is missing, user can create his own **command** that can interact with the user and modify the selected elements or the whole model.

[See an example of a Design script](#) : Replace a selected pipe by a loop

4. Review script

If a tool is missing, user can create his own **tool** to inspect the objects in selection or the whole model.

[See an example of a Review script](#) : Measure the distance between 2 nodes by message

[See another example of a Review script](#) : Show the distance between 2 nodes in left frame

Info script

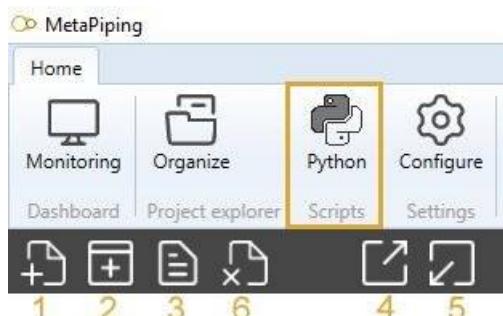
In the explorer of projects, user can show informations about the model and/or the results of a study via a special python script :



This script do not use **Python Virtual Environment** and is so very fast.

1. Script definition

From Home/Python, click on button 6 (Add simple script) :



Give it a name.

A new *main.py* will be created in the file explorer and *Inputs* and *Outputs* nodes in the Project script. Requirements.txt is hidden and the Editor is empty :

The screenshot shows the Cwantic IDE interface. At the top left is the "file explorer" panel with a single item "main.py". To its right is the "Project script" panel, which contains sections for "Inputs" and "Outputs". Below these panels is the main code editor window titled "main.py". The code in the editor is:

```
1 # main.py is the entry point of your python script
2 # Click on Help button to see the documentation
3 # Start writing your question on the right blank screen, submit, an Agent will try to help you
4 # © Cwantic
5
6
```

In the Editor, you have access to the object **study** that exposes some functions. A dropdown menu will appear after writing :

```
study.
```

The screenshot shows the code editor with the word "study." typed. A dropdown menu has appeared, listing several methods: "Outputs()", "getMetal()", "createMetal()", "getSolution()", "getDirectory()", "getMaterialLibraryModel()", "createMaterialLibraryModel()", and "showInformation()".

The main useful functions are **getMetal()**, **getSolution()** and **showInformation()**

[See the description of the object study](#)

2. Example

[See an example here](#) : Show informations about a study.



Classes

MetaPiping opens its classes in order to allow the user to **navigate** inside the structure of objects and create custom information processing.

The **study** object is the start point for "project script" to access the other objects.

The **design** object is the start point for "design script".

The internal format of the whole model in Design mode is the **metaL** object (aka **metaLanguage**).

After calculation, the **solution** object contains all the results.

The instructions that MetaPiping understands during model construction are the **command** objects.

Not all classes are explained in this documentation. The description of other classes can be given upon request...

TABLE OF CONTENTS

- [study](#)
- [design](#)
- [metaL](#)
- [solution](#)
- [command](#)
- [node](#)
- [element](#)
- [libraries](#)
- [restraint](#)
- [material](#)
- [section](#)

study

The **study** object gives access to several methods and can be used only in [Project script](#).

1. Properties

Name	Return	Description
Inputs	array of string	Access to an item [i] in Project script's Inputs
Outputs	array of string	Access to an item [i] in Project script's Outputs
getDirectory()	string	Get the current study directory
createMetal()	metal	Create an empty Metal object
getMetal()	metal	Access to current Metal object of the study
getSolution()	solution	Access to current Solution object of the study
getMaterialLibraryModel()	MaterialLibraryModel	Access to a material library by its name (without .materials)
createMaterialLibraryModel()	MaterialLibraryModel	Create a material library by a name (without .materials)
showInformation()	-	Add 2 strings into the description of a study (one for column 1 and one form column 2)
getScriptDirectory()	string	Return the current script directory
createVariableWindow()	createVariableWindow	Return an empty window

See [metal](#) for more information.

See [solution](#) for more information.

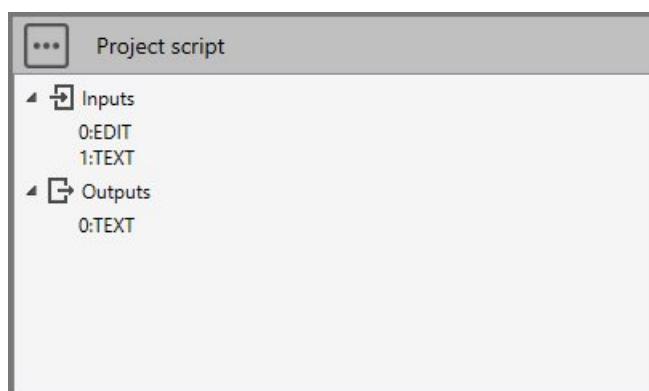
See [libraries](#) for more information about materialLibraryModel.

2. Examples

2.1 Inputs/Outputs

Imagine a **brick** (script) composed of an EDIT control (user can write something) and a TEXT control (text link from another brick).

We want that when clicking on **Run** button, the script returns as result the combination of the EDIT and the TEXT :



```
# Python script
study.Outputs[0] = study.Inputs[0] + " " + study.Inputs[1]
```

2.2 getDirectory()

Imagine user wants to save content in a text file in the current study directory :

```
# Python script
dir = study.getDirectory()
filename = os.path.join(dir, "info.txt")
with open(filename,'w') as f:
    f.write(content)
```

2.3 createMetal()

The **metal** object contains the whole definition of a piping : (geometry, loads, code and edition...).

Imagine user wants to create an empty model for the current study :

REM : a model contains default values but must have at least one layer

REM : the model of the study must be named "conception.metal"

```
# Python script
from Cwantic.MetaPiping.Core import Layer

metal = study.createMetal()

layer = Layer("0")
metal.Layers.Add(layer) # Attention ! metal.Layers is a C# List. Use Add instead of append

dir = study.getDirectory()
filename = os.path.join(dir, "conception.metal")

metal.SaveToFile(filename)
```

See [metal](#) for more information.

2.4 getMetal()

Imagine user wants to check if the 3D model exists (result as a text in Outputs[0]).

```
# Python script
metal = study.getMetal()
if metal != None:
    study.Outputs[0] = "The model exists"
else:
    study.Outputs[0] = "The model doesn't exist"
```

See [metal](#) for more information.

2.5 getSolution()

The **solution** object contains all results after calculation.

Imagine user wants to check if the solution exists (result as a text in Outputs[0]).

```
# Python script
solution = study.getSolution()
if solution != None:
    study.Outputs[0] = "The solution exists"
else:
    study.Outputs[0] = "The solution doesn't exist"
```

See [solution](#) for more information.

2.6 getMaterialLibraryModel()

Return a MaterialLibraryModel based on a name (without extension).

Return **None** if the library doesn't exist.

See [libraries](#) for more information about materialLibraryModel with an example.

2.7 createMaterialLibraryModel()

You can create your own library of material based on a name (without extension).

Return a new empty MaterialLibraryModel or **None** if already exists.

See [libraries](#) for more information about materialLibraryModel with an example.

2.8 showInformation()

This method is only accessible in scripts that show informations about the Metal and the results of the current study in **Explorer**.

```
metal = study.getMetal()
if metal != None:
    study.showInformation('Model exists !', '')
```

[See an example here](#) : Show informations about a study.

2.9 getScriptDirectory()

Returns the current script directory.

```
# Python script
scriptdirectory = design.getScriptDirectory()
```

2.10 createVariableWindow()

createVariableWindow() returns an empty window that will show user's variables.

```
# Python script
window = design.createVariableWindow()
```

Window components :

Method	Params	Description
AddComment	string	Add a comment (text)
AddValue	string, string, double	Add a numeric variable (variable name, text, default value)
AddImage	string	Add an image (local filename)
AddList	string, string, [string], int	Add a variable list of texts (variable name, text, array of texts, default index)
AddCheck	string, string, bool	Add a variable checkbox (variable name, text, default value)
ShowModal	bool	Show the window and return true if click on OK button
GetValue	string	Return a numerical value (variable name)

```
# Python script
directory = design.getScriptDirectory()

window = design.createVariableWindow()
window.AddComment("Fill the variables")
window.AddValue("L", "L =", 10)
window.AddList("CHOICE", "Choice =", ["Choice A", "Choice B", "Choice C"], 1)
window.AddCheck("ACTIVE", "Active ?", True)
window.AddImage(os.path.join(directory, "image.jpg"))
if window.ShowModal():
    val1 = window.GetValue("L")
    CHOICE_ID = window.GetValue("CHOICE")
    # CHOICE_ID = 0, 1 or 2
    val2 = 0
    if CHOICEID == 1:
        val2 = 0.5
    else:
        if CHOICE == 2:
            val2 = 3
    ACTIVE_ID = window.GetValue("ACTIVE")
    # ACTIVE_ID = 0 (False) or 1 (True)
    val3 = ACTIVE_ID == 1
```

In this example, we have 3 variables (L, CHOICE, ACTIVE), 1 comment and 1 image. We suppose image.jpg existing in the script directory next to main.py.

L will show the default value of 10

CHOICE will show the default value of "Choice B" (index 1)

ACTIVE will be checked by default

After window show :

val1 will receive the user value for L

val2 will receive the user value for CHOICE transformed to a real value (0, 0.5 or 3)

val3 will receive the user value for ACTIVE transformed to bool



design

The **design** object gives access to several methods and can be used only in [Design script](#) during the 3D modeling.

This object gives you access to the objects in selection, some current properties, the whole metal model. It gives you the opportunity to create your own [commands](#) and to show information on the left panel.

1. Properties

Name	Return	Description
selectedList	array of objects	Get the objects currently in selection (see § 2)
getType()	string	Get the type of an object in parameter (see § 3)
isType()	bool	Return True if param1 (object) is from type param2 (string) (see § 4)
createCommand()	CustomCommand	Return an empty command with a name (param1) (see § 5)
executeCommand()	-	Execute a CustomCommand passed in parameter (see § 6)
getVerticalVector()	array of 3 double	Return the current vertical vector (Y or Z)
getMetal()	metal	Access to current Metal object of the study
getMetalWithSelection()	metal	Access to the owner Metal of the selected element or node
addText()	-	Add a text on left panel
addSeparator()	-	Add a line separator on left panel
result	-	Set a message as output (empty = no message)
getCurrentLayer()	Layer	Return the current layer
getCurrentJointType()	JointType	Return the current joint type
getCurrentSpecValues()	CurrentPipingValues	Return the current piping/structure values (section, material, diameter, radius...)
getScriptDirectory()	string	Return the current script directory
createVariableWindow()	createVariableWindow	Return an empty window

2. selectedList

Retrieve the current selection list. This list can contain nodes, elements, restraints...

```
# Python script
n = len(design.selectedList)
if n==2:
    # we have 2 selected objects
    # you can access the object i : design.selectedList[i]
```

3. getType()

Retrieve the Assembly.Class type of an object.

```
# Python script
#
value = design.getType(design.selectedList[0])
result = value
```

This will show a message box with the **complete class type** of the first selected object

Example : "Cwantic.MetaPiping.Core.Node" for a node. **Cwantic.MetaPiping.Core** is the Assembly and **Node** is the class type.

4. isType()

Check if the object (param1) is from class type (param2)

```
# Python script
if design.isType(design.selectedList[0], "Node"):
    # ...
```

Unlike getType(), isType() does not refer to the assembly of the class type.

5. createCommand()

Return an empty [CustomCommand](#) with a given name (param1).

createCommand() must be used in conjunction with design.executeCommand()

```
# Python script
cmd = design.createCommand("MyCommand1")
# ...
# Fill cmd with sub commands
# ...
design.executeCommand(cmd)
```

See an example of [custom command](#).

6. executeCommand()

Execute the [CustomCommand](#) passed in parameter.

design.executeCommand() must be used in conjunction with createCommand()

```
# Python script
cmd = design.createCommand("MyCommand1")
# ...
# Fill cmd with sub commands
# ...
design.executeCommand(cmd)
```

See an example of [custom command](#).

7. getVerticalVector()

Return (0, 0, 1) if Z vertical or (0, 1, 0) if Y vertical

```
# Python script
verticalvec = design.getVerticalVector()
```

8. getMetal()

Imagine user wants to check if the 3D model exists (result as a text in Outputs[0]).

```
# Python script
metal = design.getMetal()
if metal != None:
    study.Outputs[0] = "The model exists"
else:
    study.Outputs[0] = "The model doesn't exist"
```

See [metal](#) for more information.

9. addText()

In review mode, it can be useful to shows some information about the model or about the selected objects.

These informations can be shown in a standard way using the same controls as MetaPiping.

You can show texts on the left panel using the **addText** method :

Param	Type	Description
Text	String	The text
IsTitle	Bool	True or False - if True, enlarged text
R	Byte	Red component of the text color
G	Byte	Green component of the text color
B	Byte	Blue component of the text color

```
# Python script
distance = 1.0

design.addText("Distances", True, 255, 255, 255)
design.addSeparator()
design.addText(" ")
design.addText("Distance = " + str(round(distance, 3)) + " m", False, 255, 255, 255)
```

10. addSeparator()

In the same way as the **addText** method, **addSeparator** just add a white horizontal line

```
# Python script
distance = 1.0

design.addText("Distances", True, 255, 255, 255)
design.addSeparator()
design.addText(" ")
design.addText("Distance = " + str(round(distance, 3)) + " m", False, 255, 255, 255)
```

11. result

If result is not an empty string, a message will appears at the end of execution of the script.

```
# Python script
res = "Error !"
if valid:
    res = ""
design.result = res
```

12. getScriptDirectory()

Returns the current script directory.

```
# Python script
scriptdirectory = design.getScriptDirectory()
```

13. createVariableWindow()

createVariableWindow() returns an empty window that will show user's variables.

```
# Python script
window = design.createVariableWindow()
```

13.1 Window components

Method	Params	Description
AddComment	string	Add a comment (text)
AddValue	string, string, double	Add a numeric variable (variable name, text, default value)
AddImage	string	Add an image (local filename)
AddList	string, string, [string], int	Add a variable list of texts (variable name, text, array of texts, default index)
AddCheck	string, string, bool	Add a variable checkbox (variable name, text, default value)
ShowModal	bool	Show the window and return true if click on OK button
GetValue	string	Return a numerical value (variable name)

```
# Python script
directory = design.getScriptDirectory()

window = design.createVariableWindow()
window.AddComment("Fill the variables")
window.AddValue("L", "L =", 10)
window.AddList("CHOICE", "Choice =", ["Choice A", "Choice B", "Choice C"], 1)
window.AddCheck("ACTIVE", "Active ?", True)
window.AddImage(os.path.join(directory, "image.jpg"))

if window.ShowModal():
    val1 = window.GetValue("L")
    CHOICE_ID = window.GetValue("CHOICE")
    # CHOICE_ID = 0, 1 or 2
    val2 = 0
    if CHOICEID == 1:
        val2 = 0.5
    else:
        if CHOICE == 2:
            val2 = 3
    ACTIVE_ID = window.GetValue("ACTIVE")
    # ACTIVE_ID = 0 (False) or 1 (True)
    val3 = ACTIVE_ID == 1
```

In this example, we have 3 variables (L, CHOICE, ACTIVE), 1 comment and 1 image. We suppose image.jpg existing in the script directory next to main.py.

L will show the default value of 10

CHOICE will show the default value of "Choice B" (index 1)

ACTIVE will be checked by default

After window show :

val1 will receive the user value for L

val2 will receive the user value for CHOICE transformed to a real value (0, 0.5 or 3)

val3 will receive the user value for ACTIVE transformed to bool

metal

The **metal** object gives access to the geometry and the loads of the whole piping.

1. Properties

Name	Return	Description
Plant	string	Plant
Engineer	string	Engineer
Title	string	Title
Units	Units	System units
ZVertical	bool	True if Z vertical, otherwise Yvertical
CalculationCode	PipingCode	Piping code
DefaultRoomTemperature	float	Default room temperature
DefaultOperatingDensity	float	Default operating density
DefaultTestDensity	float	Default test density
DefaultDesignCondition	OperatingCondition	Default design condition
ColdModulus	bool	True if cold modulus is used for all cases
HotAllowable	bool	True if allowable stress at operating temperature is used, otherwise design temperature is used
PressureStiffening	bool	True if pressure correction for bends
PressureElongation	bool	True if pressure elongation is considered for thermal expansion and test cases
SimplePressureStress	bool	True if simplified formula is used for pressure stress
ModalAnalysis	bool	True if modal extraction is performed
ModalRefCase	StaticCase	Young modulus is evaluated at the temperature of the reference case (Hot modulus option only)
CutOffFrequency	float	Cut-off frequency for modal extraction
MassGenerationFrequency	float	Frequency for automatic mass point generation
NbMaxModes	int	Maximum number of extracted mode shapes

The class **metal** also provides lists of the objects contained in the piping model:

Name	Description
Nodes	List of Node
Elements	List of Element
LumpedMasses	List of LumpedMass
Restraints	List of Restraint
Materials	List of Material
Sections	List of Section
DLCSS	List of DLCs (Defined Local Coordinate System)
Tees	Dictionary< Node , Tee >
UserSIFs	List of UserSIF (User-defined SIF)
RestrainedNodes	List of restrained nodes
Layers	List of Layer
Specifications	List of the specification names (string)

Other properties can be explained on demand...



solution

The **solution** object gives access to all analysis results and can be accessed through the [study](#) object.

1. Properties

Name	Return	Description
getMetal()	metal	Access to the analysis Metal object of the solution
getDisplacements()	list of (displacement, rotation)	Get the displacement/rotation results for a load case number on each node (see § 3)
getAccelerations()	list of accelerations	Get the acceleration results for a load case number on each node (see § 4)
getReactions()	list of (force, moment)	Get the force/moment results for a load case number on each restrained node (see § 5)
getForces()	list of (force, moment)	Get the force/moment results for a load case number on each element (see § 6)
getModeShape()	list of (displacement, rotation)	Get the displacement/rotation results for a mode number on each node (see § 7)
getStressIDList()	list of string	Get the list of all stress types for a load case number (see § 8)
getStresses()	list of double	Get the stress value for a load case number and a stress index on each element (see § 9)
getMaxStressRatio()	double	Get the max stress ratio for all elements and all load cases. You can specify if the result include the thermal (bool - false by default) (see § 10)

ATTENTION : the metal given by the solution (getMetal) is different from the metal given by the study !
The 'solution metal' has been transformed depending on the current piping code.

See [metal](#) for more information.

2. How to get the solution ?

The solution object can be reached via the [study](#) object. An analysis must have been done before.

```
# Python script
solution = study.getSolution()
if solution != None:
    # ...
```

3. getDisplacements()

Once the solution exists, you can retrieve the displacement and rotation on each [node](#) for a certain load case number (int).

getDisplacements() returns a list of (Item1, Item2). Item1 represents the displacement (Vector3D) and Item2 the rotation (Vector3D) of a node. This list is ordered like the metal node list.

```
# Python script
displacements = solution.getDisplacements(case)
if len(displacements) > 0:
    for item in displacements:
        displacement = item.Item1
        rotation = item.Item2
```

You can access the **X**, **Y** and **Z** properties like this :

```
# Python script
length = (displacement.X**2 + displacement.Y**2 + displacement.Z**2) ** 0.5
```

The displacement/rotation values are given in the [output units](#) specified in the model options.

4. getAccelerations()

Once the solution exists, you can retrieve the acceleration on each [node](#) for a certain load case number (int).

getAccelerations() returns a list of Vector3D. This list is ordered like the metal node list.

```
# Python script
accelerations = solution.getAccelerations(case)
if len(accelerations) > 0:
    for acceleration in accelerations:
        # you can access acceleration.X, acceleration.Y, acceleration.Z
```

The acceleration values are given in the [output units](#) specified in the model options.

5. getReactions()

Once the solution exists, you can retrieve the reaction on each [restrained node](#) for a certain load case number (int).

```
ATTENTION : the 'restrained nodes' are the nodes where a restraint has been defined.
It is a special list in the metal. The reactions are given in the same order.
```

getReactions() returns a list of (Item1, Item2). Item1 represents the force (Vector3D) and Item2 the moment (Vector3D) on a node.

```
# Python script
reactions = solution.getReactions(case)
if len(reactions) > 0:
    for reaction in reactions:
        force = reaction.Item1
        moment = reaction.Item2
        # you can access force.X, force.Y, force.Z
        # you can access moment.X, moment.Y, moment.Z
```

The force/moment values are given in the [output units](#) specified in the model options.

6. getForces()

Once the solution exists, you can retrieve the forces on each [element](#) for a certain load case number (int).

getForces() returns a list of (Item1, Item2). One couple for the extremity 1 and one couple for the extremity 2. Item1 represents the force (Vector3D) and Item2 the moment (Vector3D) on an extremity of an element. This list is ordered like the metal element list but with twice the size.

```
# Python script
forces = solution.getForces(case)
if len(forces) > 0:
    firstExtremity = True
    for item in forces:
        force = item.Item1
        moment = item.Item2
        # you can access force.X, force.Y, force.Z for an extremity of an element
        # you can access moment.X, moment.Y, moment.Z
        firstExtremity = not firstExtremity
```

The force/moment values are given in the [output units](#) specified in the model options.

7. getModeShape()

Once the solution exists, you can retrieve the displacement and rotation on each [node](#) for a certain mode number (int).

getModeShape() returns a list of (Item1, Item2). Item1 represents the displacement (Vector3D) and Item2 the rotation (Vector3D) of a node. This list is ordered like the metal node list.

```
# Python script
displacements = solution.getModeShape(mode)
if len(displacements) > 0:
    for item in displacements:
        displacement = item.Item1
        rotation = item.Item2
        # you can access displacement.X, displacement.Y, displacement.Z
        # you can access rotation.X, rotation.Y, rotation.Z
```

The displacement/rotation values are given in the [output units](#) specified in the model options.

8. getStressIDList()

Once the solution exists, you can retrieve the stress id list for a certain load case number/ or mode shape.

getStressIDList() returns a list of String depending on the load case (int) and internally the current piping code.

Example of result for RCCM class2, equation 2 : { "Ratio", "Eq. 6", "Sallow", "SPres", "SMomA" }

9. getStresses()

Once the solution exists, you can retrieve the stresses on each [element](#) for a certain load case number (int) and a certain stress index (int) from getStressIDList (see § 8).

getStresses() returns a list of Double. Two Double by element, one for the extremity 1 and one for the extremity 2. This list is ordered like the metal element list but with twice the size.

```
# Python script
stresses = solution.getStresses(case, stressIndex)
if len(stresses) > 0:
    firstExtremity = True
    for stress in stresses:
        # you have the stress value for an extremity of an element
        # ...
        firstExtremity = not firstExtremity
```

The stress value is given in the [output units](#) specified in the model options.

10. getMaxStressRatio()

Once the solution exists, you can retrieve the MAX stress ratio for the whole piping and for all load cases.

getMaxStressRatio() returns a Double. You can specify as parameter a Boolean to include Thermal expansion cases or not.

```
# Python script
maxStress = solution.getMaxStressRatio(True)
```

A value less than 1 indicates a valid piping model !

The max stress value is unitless.

CustomCommand

A custom command is simply a list of MetaPiping existing commands.

1. Creation

A **CustomCommand** object can only be created by **design** object via **createCommand** method :

```
# Python script
cmd = design.createCommand("MyCommand1")
# ...
```

The **CustomCommand** must be defined by a unique name (first param of the createCommand method).

[See the description of the object design](#)

2. Properties

Name	Return	Description
addSubCommand	bool	Add an existing command by a name and command params (array)

Example :

We want to remove the selected objects. First, create an empty array, add the selectedList from **design** object. Then add the name of the remove command and the params to the **CustomCommand**

```
# Python script
params = []
params.append(design.selectedList)

valid = cmd.addSubCommand("RemoveElementCommand", params)
```

Every command has its own param list !

Return False if the command name doesn't exists or the params are incorrect.

3. Execution

Finally, a **CustomCommand** (cmd in the example) can be executed by **design** object via **executeCommand** method:

```
# Python script
# ...
design.executeCommand(cmd)
```

4. Common objects

Two general objects control all properties of current tee, current material, current section, current bend radius...

- CurrentTeeValues
- CurrentPipingValues

You can set the wanted properties before creating a command and share these properties for all commands.

To use these objects, you need to import the classes in your Python script :

```
# Python script
from Cwantic.MetaPiping.Core import CurrentPipingValues, CurrentTeeValues
```

4.1 CurrentTeeValues

If you need to create a **tee**, you can create a **CurrentTeeValues** object and initialize its properties :

```
# Python script
from Cwantic.MetaPiping.Core import CurrentTeeValues

...
currentTeeValues = CurrentTeeValues()
currentTeeValues.Type = TeeType.BranchConnection
currentTeeValues.TN = 4.2
currentTeeValues.CollectorSize = 0.1429
currentTeeValues.BranchSize = 0.1429
```

If you don't need to create a **tee** but need to have a **CurrentTeeValues** for the command, just create an empty object :

```
# Python script
currentTeeValues = CurrentTeeValues()
```

According to the type of tee, only several properties must be set.

PROPERTIES :

Name	Type	Description	Unit	Default value
Type	TeeType	Type of tee	-	BranchConnection (see below)
TN	float	Branch pipe wall thickness	Diameter	0
RP	float	Branch Outer Radius	Diameter	0
PD	float	Pad Thickness for Reinforced Fabricated Tees	Diameter	0
R2	float	Branch-to-run Fillet Radius	Diameter	0
RX	float	Transition Radius	Diameter	0
Angle	float	Angle between run and branch	Degree	0
CollectorSize	float	Length of run	Length	0.3
BranchSize	float	Length of branch	Length	0.3

See [Units](#) for explanation of Length and Diameter units.

The type *TeeType* is an enumeration with the following values:

Value	Description
BranchConnection	Branch connection
WeldingTee	Butt welding tee
NonStandard	Junction of elements
Fabricated	Reinforced or un-reinforced fabricated tee
SweepletFlush	Bonney Forge Sweeplet® (flush welded)
SweepletAsWelded	Bonney Forge Sweeplet® (as welded)
Weldolet	Bonney Forge Weldolet®
ExtrudedOutlet	Extruded Outlet or Extruded welding tee
WeldedinContourInsert	Welded-in contour insert
BranchWeldedOnFitting	Branch welded-on fitting
Lateral	Piping lateral connection per WRC Bulletin 360
PartialPenetration	Branch connection with partial penetration welds or fillet welds

See an example of creation of a [CurrentTeeValues](#).

4.2 CurrentPipingValues

The **CurrentPipingValues** object contains all properties needed for piping element as layer, material, section, bend radius...

The easiest way to create a **CurrentPipingValues** object is to get the **current one** from design:

```
# Python script
from Cwantic.MetaPiping.Core import CurrentPipingValues

currentPipingValues = design.getCurrentPipingValues()
```

More properties can be explained on demand...

5. Commands

5.1 DrawPipingCommand

Param	Type	Description
Node1	Node	First extremity of the pipe - cannot be None
Node2	Node	Second extremity of the pipe - can be None
DX	Float	Distance along X axis of end point from Node1
DY	Float	Distance along Y axis of end point from Node1
DZ	Float	Distance along Z axis of end point from Node1
LocalX	Float	X component of local X axis
LocalY	Float	Y component of local X axis
LocalZ	Float	Z component of local X axis
PipingValues	CurrentPipingValues	Current piping properties (see §4.2)

Rem : LocalX, LocalY, LocalZ can be set to (0, 0, 0)

Imagine a customCommand cmd, a first node N1, a second node N2, a size of pipe (float), a direction (Vector3D), a currentPipingValues, here is how to create a pipe :

```
# Python script
params = []
params.append(N1)
params.append(N2)
params.append(size*dir[0])
params.append(size*dir[1])
params.append(size*dir[2])
params.append(0.0)
params.append(0.0)
params.append(0.0)
params.append(currentPipingValues)

valid = cmd.addSubCommand("DrawPipingCommand", params)
```

You can also write the command in this way :

```
# Python script
valid = cmd.addSubCommand("DrawPipingCommand", [N1, N2, size*dir[0], size*dir[1], size*dir[2], 0.0, 0.0, 0.0, currentPipingValues])
```

5.2 RemoveElementCommand

Param	Type	Description
Elements	Array	Array of element

See [Element](#) for more information.

Imagine a customCommand cmd and elements in selection, here is how to remove these elements :

```
# Python script
params = []
params.append(design.selectedList)

valid = cmd.addSubCommand("RemoveElementCommand", params)
```

You can also write the command in this way :

```
# Python script
valid = cmd.addSubCommand("RemoveElementCommand", [design.selectedList])
```

5.3 Other commands

Other commands can be explained on demand...

5.4 Example

Click [here](#) for a complete example using custom commands : Create a loop.

Node

1. Properties

Name	Return	Description
Name	String	Node name
Coor	Point3D	Coor.X, Coor.Y and Coor.Z are the X/Y/Z global coordinates
Elements	list of Element	List of the elements connected to the node
IsCoorInput	bool	True if the coordinates are set by the user
IsGenerated	bool	True if the node is generated during analysis
IsConstructionNode	bool	True if the node is a fictitious node at an elbow intersection
JointType	JointType	Type of joint - see description below
Mismatch	float	Mismatch for butt welds, as welded
FilletLength	float	Length of fillet weld
TMax	float	tmax for Class 1 transition within 1:3 slope envelope

2. Joint types

The type JointType is an enumeration. Its different values may be imported with the following instruction:

```
from Cwantic.MetaPiping.Core import JointType
```

and used like in this example :

```
if node.JointType == JointType.ButtWeldAsWelded:
```

2.1 Joint types for steel piping

Value	Description
None	No weld
ButtWeldFlush	Girth butt weld, flush
ButtWeldAsWelded	Girth butt weld, as welded
CappedEnd	Capped end, as welded
FilletWeld	Girth fillet weld
FullFilletWeld	Girth full fillet weld
TaperedFlush	Tapered transition joint, flush
TaperedAsWelded	Tapered transition joint, as welded
OneThirdSlopeFlush	Transition within 1:3 slope envelope, flush (Class 1 only)
OneThirdSlopeAsWelded	Transition within 1:3 slope envelope, as welded (Class 1 only)
Threaded	Threaded joint
Brazed	Brazed joint
LapFlange	Lap joint for flange
SingleWeldSlipOnFlange	Single-weld for slip-on flange (B31J only)
DoubleWeldSlipOnFlange	Double-weld for slip-on flange

2.2 Joint types for composite piping

Value	Description
None	None
AdhesiveBonded	Bell and spigot adhesive bonded joint
AdhesiveBondedWithOverlay	Bell and spigot adhesive bonded joint with laminated fiberglass overlay
GasketWithOverlay	Bell and spigot gasket joint with laminated fiberglass overlay
ButtAndStrap	Butt and strap joint

2.3 Joint types for HDPE piping

Value	Description
None	None
ConcentricFabricatedReducer	Concentric fabricated reducers
ThrustCollar	Thrust collar
ElectrofusionCoupling	Electrofusion coupling

1. Element

Element is the base class for all elements.

Properties

Name	Return	Description
Node1	Node	Start node
Node2	Node	End node
DL	Vector3D	DL.X, DL.Y and DL.Z are the coordinate changes from Node1 to Node2 in the global coordinate system
Material	Material	Material
RoomTemperature	float	Installation temperature. If null, same as the room temperature specified for the model
XDir	Vector3D	Local X' axis. If (0,0,0), the default local axes are used
Label	string	Label
LayerID	int	Layer number
Diameter	float	Element cross size (for graphics). Same as section outside diameter for Piping elements
Mass	float	Mass. The meaning depends on <i>MassModel</i>
MassModel	MassModel	Mass modeling option

The type *MassModel* is an enumeration with the following values:

Value	Description
AtEnd	<i>Mass</i> is a lumped mass located at the end node
AtStart	<i>Mass</i> is a lumped mass located at the start node
HalfAtBoth	<i>Mass</i> is a lumped mass divided between both nodes
LinearAtBoth	<i>Mass</i> is a linear mass. The total mass (<i>Mass</i> x <i>element length</i>) is lumped at both nodes
LinearAtEnd	<i>Mass</i> is a linear mass. The total mass (<i>Mass</i> x <i>element length</i>) is lumped at the end node
Linear	<i>Mass</i> is a linear mass
Density	<i>Mass</i> is a density (for Beam elements only)

2. Piping

Piping is an abstract class that inherits from **Element** (see § 1). Every **Piping** object has the properties of **Element**.

Properties

Name	Return	Description
Section	PipeSection	Pipe section
LongWeldType	LongWeldType	Longitudinal weld type
LongWeldMismatch	float	Longitudinal weld mismatch
DesignCondition	OperatingCondition	Design condition. If null, same as general design condition of the model
PipingCode	PipingCode	Piping code. If null, same as general piping code of the model
StresOff	bool	True if stress calculation is disabled
ThicknessFactor	float	Thickness factor
OutsideDiameter	float	Outside diameter
InsideDiameter	float	Inside diameter
Thickness	float	Thickness
ContentDiameter	float	Inside diameter for calculation of fluid weight

The type *LongWeldType* is an enumeration with the following values:

Value	Description
None	No weld
ButtWeldFlush	Butt weld, flush
ButtWeldAsWelded	Butt weld, as welded

3. Pipe

Pipe inherits from **Piping**. It represents straight pipes.

Properties

Name	Return	Description
NbDivisions	int	Number of subdivisions for analysis

4. Branch

Branch inherits from **Piping**. It is a fictitious element generated during the analysis that connects the tee center point to the run surface.

Properties

No additional properties.

5. Bend

Bend inherits from **Piping**. It represents elbows and miter bends.

Properties

Name	Return	Description
Miter	bool	True if miter bend
DL1	Vector3D	First tangent vector
DL2	Vector3D	Second tangent vector
Node3	Node	Intersection node
Radius	float	Geometrical radius
Angle	float	Angle (in radians)
NbDivisions	int	Number of subdivisions for analysis (elbow) or number of miter cuts (miter bend)
Spacing	float	Miter spacing at center line
NbFlanges	int	Number of attached flanges
UserFlex	float	User-defined flexibility factor

6. Reducer

Reducer inherits from **Piping**.

Properties

Name	Return	Description
Section2	PipeSection	Section at end node
Angle	float	Cone angle (in degrees)
L12	float	Minimum tangent L1, L2 (Class 1 and B31J)
R12	float	Minimum radius r1, r2 (Class 1 and B31J)

7. Valve

Valve inherits from **Piping**.

Properties

Name	Return	Description
Type	ValveType	Type of valve
PB	Node	Node at valve midpoint. Null if no midpoint
PC	Node	Node at end of valve stem. Null if no stem
BL	Vector3D	Midpoint to stem offset
IsStemEmpty	bool	True if the stem is empty

The type *ValveType* is an enumeration used for graphics. It has the following values:

Value	Description
Valve	Valve is represented depending on PB and PC
ToCenter	Valve end to center
FromCenter	Center to valve end
Stem	Center to stem

8. Flange

Flange inherits from **Piping**.

Properties

Name	Return	Description
FDiameter	float	Outside diameter
FThickness	float	Thickness

9. Socket

Socket inherits from **Piping**.

Properties

No additional properties.

10. Structural

Structural inherits from **Piping**.

Properties

No additional properties.

11. Structural

Structural inherits from **Piping**.

Properties

No additional properties.

12. Rigid

Rigid inherits from **Element**.

Properties

No additional properties.

13. Spring

Spring inherits from **Element**.

Properties

Name	Return	Description
ZDir	Vector3D	Local Z' direction. Only used if spring length is zero
Kx	float	Translation stiffness along X'
Ky	float	Translation stiffness along Y'
Kz	float	Translation stiffness along Z'
Krx	float	Rotation stiffness around X'
Kry	float	Rotation stiffness around Y'
Krz	float	Rotation stiffness around Z'

14. Matrix

Matrix inherits from [Element](#).

Properties

Name	Return	Description
ZDir	Vector3D	Local Z' direction
Stiffness	SquareMatrix	Stiffness matrix

SquareMatrix represents a 6x6 symmetrical matrix.

15. Bellow

Bellow inherits from [Piping](#).

Properties

Name	Return	Description
Type	BellowType	Type of bellow
SA	float	Axial stiffness
SL	float	Lateral stiffness
SB	float	Angular stiffness
ST	float	Torsional stiffness
A	float	Pressure thrust area

The type *BellowType* is an enumeration with the following values:

Value	Description
Axial	Axial
LateralSingle	Lateral in X'Z' plane
LateralAll	lateral in all planes
AngularSingle	Angular in X'Z' plane
AngularAll	Angular in all planes

16. Beam

Beam inherits from [Element](#).

Properties

Name	Return	Description
Section	BeamSection	Section
E	float	Young modulus
NbDivisions	int	Number of subdivisions

1. MaterialLibraryModel

A **MaterialLibraryModel** is a database containing a list of MaterialModel.

Name	Return	Description
GetMaterialByDescription()	MaterialModel	Access to a materialModel by its description (param1)
MaterialDescriptionExists()	bool	Return True if a material exists with this description (param1)
SetRefTemperature()	-	Set the reference temperature (param1) for all materials [°C in metric units]
AddMaterial()	MaterialModel	Add an empty materialModel to the library (without description)
DeleteMaterial()	-	Delete the materialModel (param1) from the library
Save()	-	Save the library

1.1 MaterialModel

A **MaterialModel** represents the definition of a material.

Name	Return	Description
Description		The description of the material
MaterialType		The type of material
MaxTemperature		Temperature max [°C in metric units]
RO		Density [kg/m³ in metric units]
SG		Poisson's ratio
Properties		List of MaterialProperties
Properties.Count	int	Number of MaterialProperties
Properties.[i]	MaterialProperties	i-th MaterialProperties
AddProperties()	MaterialProperties	Add a new line of properties for a temperature
DeleteProperties()	-	Delete a MaterialProperties (param1)

1.2 MaterialType

To access the **MaterialType** property of a **MaterialModel**, you need to import the object from *Cwantic.MetaPiping.Core* :

```
# Python script
from Cwantic.MetaPiping.Core import MaterialType
```

Values :

MaterialType.CarbonSteel
MaterialType.LowAlloySteel
MaterialType.MartensiticSteel
MaterialType.AusteniticSteel
MaterialType.NickelChromeSteel
MaterialType.NickelCopperSteel
MaterialType.Other
MaterialType.Composite
MaterialType.HDPE

Example :

```
# Python script
from Cwantic.MetaPiping.Core import MaterialType
...
if mat.MaterialType == MaterialType.CarbonSteel:
    ...
```

1.3 MaterialProperties

Property	Description	Unit Metric	Unit USA
TE	Temperature	°C	°F
EH	Modulus of Elasticity	kN/mm ²	10 ⁶ .psi
EX	Thermal Expansion	10 ⁻⁶ .mm/mm/°C	10 ⁻⁶ .in/in/°F
SH	Non-Class 1 Allowable Stress	N/mm ²	ksi
SY	Yield Stress	N/mm ²	ksi
SU	Ultimate Tensile Stress	N/mm ²	ksi
SM	Class 1 Allowable Stress	N/mm ²	ksi
CR	Creep	N/mm ²	ksi
GH	Shear Modulus	kN/mm ²	10 ⁶ .psi
CO	Class 1 Thermal Conductivity	kJ/hr/m/°C	btu/hr/ft/°F
DI	Class 1 Thermal Diffusivity	mm ² /s	ft ² /hr

1.4 Example

Imagine user wants to get the first temperature of a specific MaterialModel (study.Inputs[1]) from a specific library (study.Inputs[0]). Result as a text in Outputs[0].

```
# Python script
libname = study.Inputs[0]
matname = study.Inputs[1]

lib = study.getMaterialLibraryModel(libname)
if lib != None:
    mat = lib.GetMaterialByDescription(matname)
    if mat != None:
        study.Outputs[0] = str(mat.Properties[0].TE)
    else:
        study.Outputs[0] = "Material not found !"
else:
    study.Outputs[0] = "Library not found !"
```



1. Restraint

Restraint is the base class for all restraints.

Properties

Name	Return	Description
Node	Node	Restrained node
Level	string	Level (for seismic analysis)
AttachedElement	Element	Element attached to restraint (for local axes - see LCS)
LCS	CoordinateSystem	Coordinate system used for restraint directions
Label	string	Label
LayerID	int	Layer number

The type *CoordinateSystem* is an enumeration with the following values:

Value	Description
Global	Global coordinate system
LocalToConnectedElement	Local coordinate system of the adjacent element
LocalToPrecedingElement	Local coordinate system of the preceding element
LocalToFollowingElement	Local coordinate system of the following element
Local	Local coordinate system specified at restrained node

2. AxialRestraint

AxialRestraint is an abstract class that inherits from **Restraint**. Every **AxialRestraint** object has the properties of **Restraint**.

Properties

Name	Return	Description
Stiffness	float	Stiffness
Direction	Vector3D	Restrained direction

3. LinearRestraint

LinearRestraint inherits from **AxialRestraint**. It represents linear axial restraints.

Properties

Name	Return	Description
Type	LinearRestraintType	Type of restraint

The type *LinearRestraintType* is an enumeration with the following values:

Value	Description
Translational	Translational
Rotational	Rotational
Snubber	Snubber

4. SpringHanger

Branch inherits from **AxialRestraint**.

Properties

Name	Return	Description
Force	float	Type of restraint
PinnedCase	PinnedCase	Pinned case option

The type *PinnedCase* is an enumeration with the following values:

Value	Description
Design	Pinned for design weight analysis
DesignAndTest	Pinned for design weight and test weight analyses
DesignAndEmpty	Pinned for design weight and empty weight analyses

5. MultiRestraint

MultiRestraint inherits from **Restraint**.

Properties

Name	Return	Description
Kx	float	Stiffness along X: 0 if free, > 0 if rigid
Ky	float	Stiffness along Y: 0 if free, > 0 if rigid
Kz	float	Stiffness along Z: 0 if free, > 0 if rigid
Krx	float	Stiffness around X: 0 if free, > 0 if rigid
Kry	float	Stiffness around Y: 0 if free, > 0 if rigid
Krz	float	Stiffness around Z: 0 if free, > 0 if rigid

6. Anchor

Anchor inherits from **Restraint**.

Properties

Name	Return	Description
Kx	float	Stiffness along X
Ky	float	Stiffness along Y
Kz	float	Stiffness along Z
Krx	float	Stiffness around X
Kry	float	Stiffness around Y
Krz	float	Stiffness around Z

Material

1. Material

Material is the base class for all materials.

1.1 Properties

Name	Return	Description
Name	String	Material name
Type	MaterialType	Type of material
RefTemperature	float	Reference temperature
Density	float	Density
Poisson	float	Poisson coefficient
MaxTemperature	float	Max temperature
Description	string	Description

2. RegularMaterial

RegularMaterial inherits from **Material**. Every **RegularMaterial** object has the properties of **Material**.

RegularMaterial represents regular materials which properties are independent of time.

2.1 Properties

No additional properties.

Section

1. Section

Section is the base class for all sections.

Properties

Name	Return	Description
Name	String	Material name
A	float	Cross-sectional area
Ax	float	Shear area in local X' axis
Ay	float	Shear area in local Y axis
Ix	float	Inertia around local X' axis
Iy	float	Inertia around local Y axis
It	float	Torsional inertia
LinearMass	float	Linear mass
Description	string	Description

2. PipeSection

PipeSection inherits from **Section**. Every **PipeSection** object has the properties of **Section**.

PipeSection represents circular hollow pipe sections.

Properties

Name	Return	Description
Diameter	float	Diameter
Thickness	float	Thickness
InsulationThickness	float	Insulation thickness
Corrosion	float	Corrosion allowance
Erosion	float	Erosion allowance

3. BeamSection

BeamSection inherits from **Section**.

BeamSection represents beam sections.

Properties

Name	Return	Description
Type	BeamSectionType	Type of section
InputProperties	bool	True if section properties are input by the user, otherwise the properties are calculated from the section geometry
H	float	Height
B	float	Width
Tw	float	Web thickness
Tf	float	Flange thickness

The type *BeamSectionType* is an enumeration with the following values:

Value	Description
NonStandard	Non-standard section. The properties must be input
I	Double tee
Channel	Channel
Rect	Rectangular
Tee	Tee
EqualAngle	Equal angle. H is the side length, Tf is the thickness
UnequalAngle	Unequal angle. H and B are the side lengths, Tf is the thickness
Round	Round. H is the diameter, Tf is the thickness
Plate	Plate. H and B are used

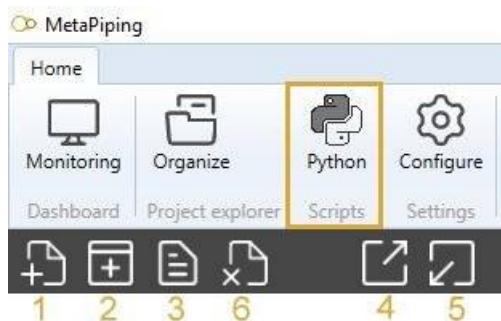
Pyckage

MetaPiping provides a simple tool that import/export a whole script (files - directories - definition and requirements.txt) from/to a single ***.pyckage** file.

1. Goal

The goal is to facilitate the **exchange** of complex scripts to several computers.

2. Use



2.1 Export

- 1 Select a script
- 2 Click on **button4**
- 3 Give it a name and save it on the computer

2.2 Import

- 1 Click on **button5**
- 2 Select a *.pyckage file that doesn't already exist on your script list
- 3 Open it

The **Python Virtual Environment** will automatically be installed.

ChatGPT

MetaPiping provides a privileged connection to **ChatGPT** through an official license to the API.

ChatGPT (Chat Generative Pre-trained Transformer) is a chatbot developed by OpenAI and launched in November 2022. It is built on top of OpenAI's GPT-3 family of large language models and has been fine-tuned (an approach to transfer learning) using both supervised and reinforcement learning techniques.



Since October 2023, MetaPiping uses the model **gpt-3.5-turbo** : Most capable GPT-3.5 model. It can do any task the other models can do, often with higher quality, longer output and better instruction-following. Also supports inserting completions within text.

WARNING

Attention ! Cwantic cannot be held responsible for the non-operation or non-availability of the service or for any errors generated by the AI.

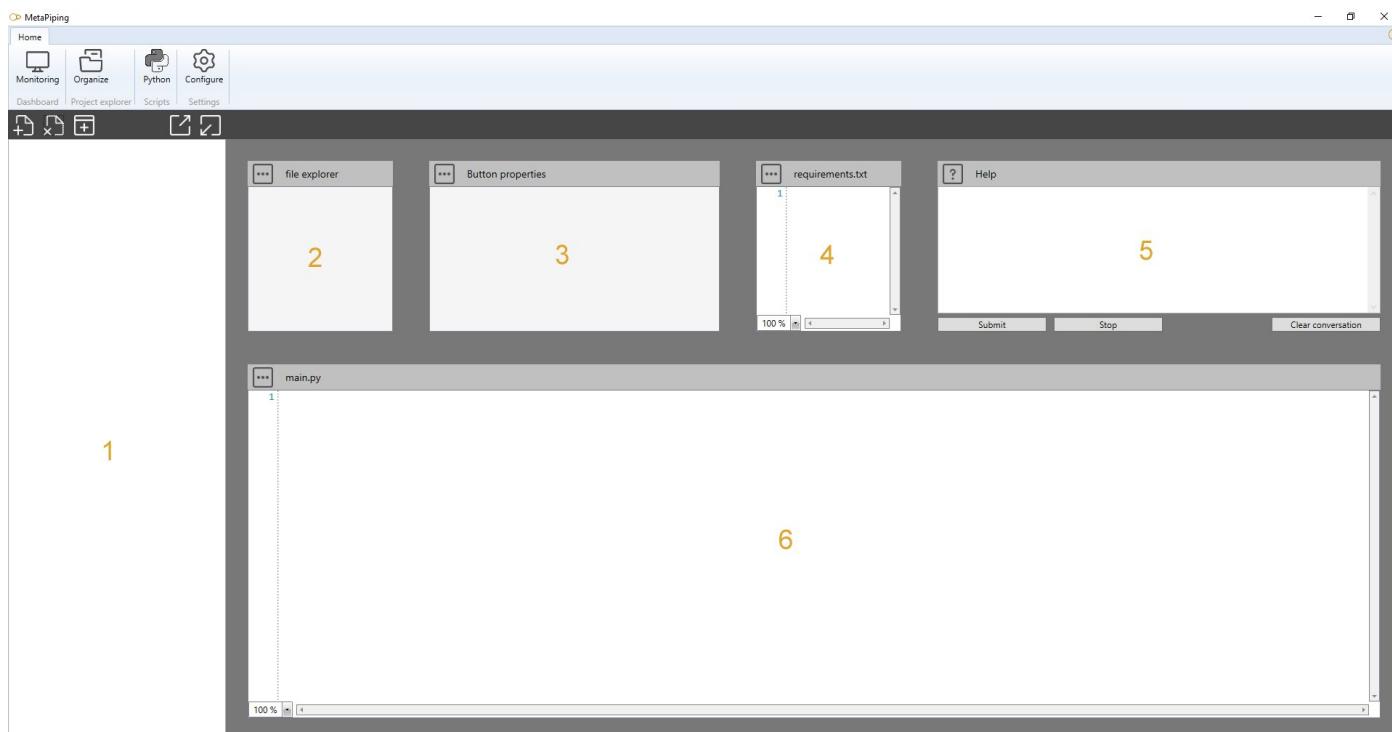
1. Codex

ChatGPT is able to help user in Python development thanks to its **Codex**.

OpenAI Codex is a descendant of GPT-3; its training data contains both natural language and billions of lines of source code from publicly available sources, including code in public GitHub repositories. OpenAI Codex is most capable in Python, ... It has a memory of 14KB for Python code, compared to GPT-3 which has only 4KB—so it can take into account over 3x as much contextual information while performing any task.

2. Screen

The prompt for a conversation with ChatGPT is on area 5 (in the Python screen)



3. Goal

One of the main feature of MetaPiping is the **Python integration**.

User can make his own scripts to interact with the application or manipulate the input and output data.

Cwantic chooses **Python** language to create these scripts. But Why **Python**? Let ChatGPT answers :

The screenshot shows a user interface for a conversational AI. At the top, there's a question bar with a help icon and a "Help" button. Below it, a blue-highlighted question asks, "what are the advantages of the python language compared to other languages ?". Three numbered steps are listed below the question:

1. Easy to read and understand: Python has a simple syntax and readable code, making it easier to read and understand compared to many other languages.
2. Cross platform support: Python is available on multiple platforms including Windows, Mac, Linux and many other systems. This means that you can develop applications that can run on any platform.
3. Extensive library support: Python comes with a wide range of libraries that can be used to develop applications quickly and efficiently.

At the bottom of the interface are three buttons: "Submit" (highlighted in yellow), "Stop", and "Clear conversation".

- 1 Write your question in English our in your native language
- 2 Submit
- 3 ChatGPT will answer

Here is its answer :

1. Easy to read and understand: Python has a simple syntax and readable code, making it easier to read and understand compared to many other languages.

2. Cross platform support: Python is available on multiple platforms including Windows, Mac, Linux and many other systems. This means that you can develop applications that can run on any platform.

3. Extensive library support: Python comes with a wide range of libraries that can be used to develop applications quickly and efficiently.

4. High-level language: Python can be used to develop high-level applications without having to write complex code.

5. Flexibility and dynamic typing: Python is flexible and does not enforce any particular style or structure on the code. In addition, it supports multiple programming paradigms.

4. Conversation

ChatGPT knows almost everything and retains the thread of the conversation.

Here is an example of a conversation. In blue the question asked by the user. Click on **Submit** button. Then in black the answer :

The screenshot shows a user interface for a conversational AI. At the top, there's a question bar with a help icon and a "Help" button. Below it, a blue-highlighted question asks, "what is the current standard for nuclear piping design in france ?". The AI response is:

The current standard for nuclear piping design in France is the French Nuclear Code (RCC-M). This code is based on the general framework of the French code, the Code of Nuclear Installations Safety and Protection (CISPR), which was introduced in 1987 and has been revised three times, in 1997, 2007 and most recently in 2018. It covers all aspects of nuclear safety and protection, including piping design and construction.
Done...

At the bottom of the interface are three buttons: "Submit" (highlighted in yellow), "Stop", and "Clear conversation".

The screenshot shows a user interface for a conversational AI. At the top, there's a question bar with a help icon and a "Help" button. Below it, a blue-highlighted question asks, "Wasn't there a revision in 2020 ?". The AI response is:

Yes, there was a revision in 2020. In particular, the 20th edition of the French Nuclear Code (RCC-M) was published in October 2020. This edition made revisions to the requirements for nuclear piping design and construction, as well as added new technical requirements for the prevention and mitigation of seismic events.
Done...

At the bottom of the interface are three buttons: "Submit" (highlighted in yellow), "Stop", and "Clear conversation".

? Help

construction, as well as added new technical requirements for the prevention and mitigation of seismic events.
Done...

What has changed between the 2018 and 2020 editions ?

The main changes between the 2018 and 2020 editions of the French Nuclear Code (RCC-M) include: improved seismic protection requirements, enhanced material and welding requirements, revised design requirements for general piping systems, revised requirements for fluctuating loading conditions, revised requirements for the evaluation of fatigue, revised requirements for thermal expansion, and updated requirements for fire protection systems.
Done...

Submit Stop Clear conversation

? Help

What is the young modulus of the carbon steel "P 265 GH" at 20°C ?

The young modulus of the carbon steel "P 265 GH" at 20°C is 200 GPa.
Done...

Can you be more precise ?

The young modulus of the carbon steel "P 265 GH" at 20°C is approximately 205 GPa.
Done...

Submit Stop Clear conversation

5. Python language

ChatGPT is also an **expert** in Python.

Examples :

? Help

Give me an example in python to get the current date and hour ?

```
import datetime

now = datetime.datetime.now()

print(now.strftime("%d-%m-%Y %H:%M:%S"))
Done...
```

Submit Stop Clear conversation

? Help

In python, how to combine a directory and a file ?

In Python, you can use the os.path.join() function to combine a directory and a file. For example:

```
import os
directory = "/home/user/Documents"
filename = "data.txt"
combined = os.path.join(directory, filename)
print(combined)
# Output: /home/user/Documents/data.txt
Done...
```

Submit Stop Clear conversation

Help

In python, how to save a list of string to file ?

1. Using the write() method:

```
f = open("filename.txt", "w")
for element in list:
    f.write(element + "\n")
f.close()
```

2. Using the writelines() method:

Submit Stop Clear conversation

These answers were used to create [Sample1](#).

6. Python libraries

ChatGPT is also an **expert** in all libraries of **pip**.

Examples used to create [Sample3](#) about **EXCEL** files :

Help

What are the libraries used in python to read Excel files and sheets ?

1. openpyxl
2. xlrd
3. xlwt
4. xlutils
5. pyexcel
6. pandas
Done...

Submit Stop Clear conversation

Help

6. pandas
Done...

Can you show an example to read XLSX on a particular sheet in one function with pandas ?

```
import pandas as pd

# read xlsx file
df = pd.read_excel("filename.xlsx", sheet_name="Sheet1")

Done...
```

Submit Stop Clear conversation

Help

```
df = pd.read_excel("filename.xlsx", sheet_name="Sheet1")
```

Done...

And how to iterate inside each row to get values of each column ?

```
# iterate over each row of dataframe
for index, row in df.iterrows():
    # access data using column names
    print(row['column name1'], row['column name2'], row['column name3'])
```

Submit Stop Clear conversation

7. Conclusion

ChatGPT is the perfect **companion** to help the user writing **scripts** by finding all the necessary information on any subject but more specifically on the use of the **python language** and its numerous mathematical, scientific and computer **libraries**.



Samples

You can find here some examples of script.

TABLE OF CONTENTS

- [Date Time and File](#)
- [Send an email](#)
- [Converter](#)
- [Total mass of piping](#)
- [Static loadcases](#)
- [Max displacement](#)
- [Add loop](#)
- [Add loop \(without virt. env.\)](#)
- [Distance between 2 nodes](#)
- [Distance between 2 nodes \(framed\)](#)
- [Study info](#)
- [Support](#)

Sample 1 : Date, time and file manipulation

1. Goal

Create a script that will (after pressing **RUN** button) save the current date and hour in a file and show it in the *brick*.

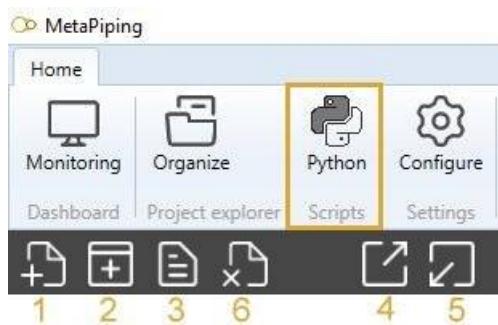
We suppose 4 predetermined dates :

- 1 Start date
- 2 Review date
- 3 Analysis date
- 4 Finish date

The user must clic on the button only when the phase is ready. The script must save the dates in the current study directory.

2. Script definition

From the Home/Python, click on button 1 (Add project script) :



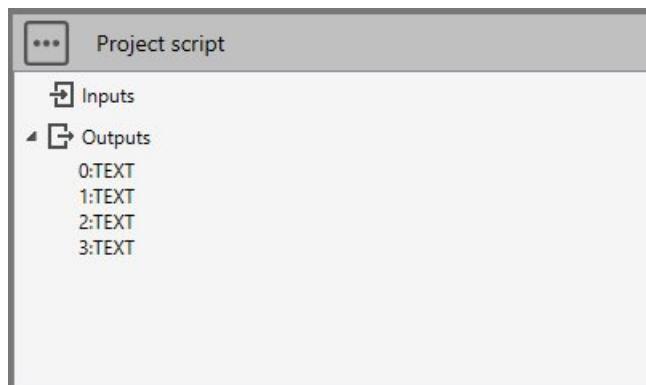
Give it the name "Click at each stage of the project"

Say **NO** to the question about creation of virtual environment. The Requirements.txt will be hidden.



3. Script properties

Add 4 TEXT output :



Don't forget to save !

4. Main.py

Copy/paste this code in the Editor :

```
import os
from datetime import datetime

# Initialize outputs
study.Outputs[0] = "Start date : "
study.Outputs[1] = "Review date : "
study.Outputs[2] = "Analysis date : "
study.Outputs[3] = "Finish date : "

dates = []

# Current date and hour
now = datetime.now()
now_string = now.strftime("%d/%m/%Y %H:%M:%S")

# info.txt is the file that contains all dates in the current study directory
dir = study.getDirectory()
filename = os.path.join(dir, "info.txt")

# Read the file if exists
if os.path.exists(filename):
    with open(filename) as f:
        dates = f.read().splitlines()

# Add current date to the list
if (len(dates) < 4):
    dates.append(now_string)

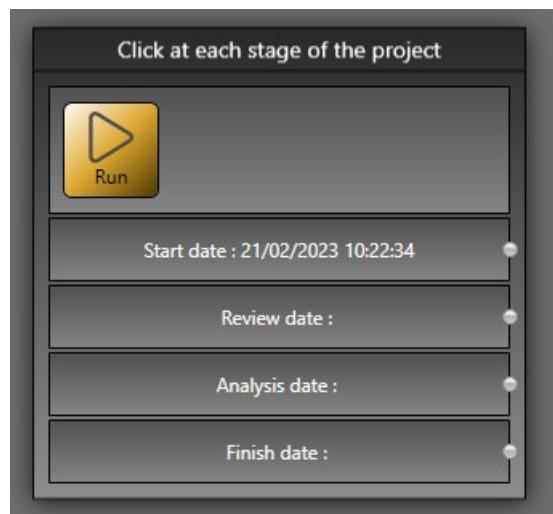
# Save the dates
with open(filename,'w') as f:
    f.write('\n'.join(dates))

# Show the dates
if (len(dates) > 0):
    study.Outputs[0] += dates[0]
if (len(dates) > 1):
    study.Outputs[1] += dates[1]
if (len(dates) > 2):
    study.Outputs[2] += dates[2]
if (len(dates) > 3):
    study.Outputs[3] += dates[3]
```

Save it !

5. Result

When you add this script in a current study, and after a first click on the **RUN** button :



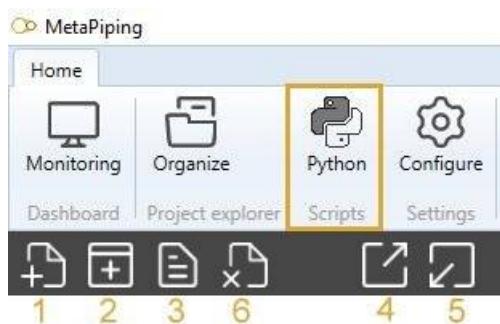
Sample 2 : Send an email

1. Goal

Create a script that will (after pressing **RUN** button) send an email to a fixed recipient with a SUBJECT defined by the user and a DATE coming from [Sample1](#).

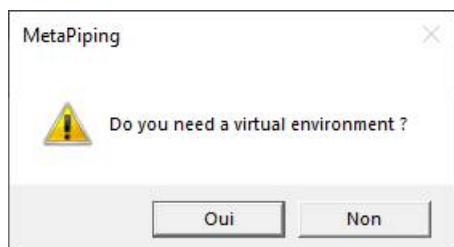
2. Script definition

From the Home/Python, click on button 1 (Add project script) :



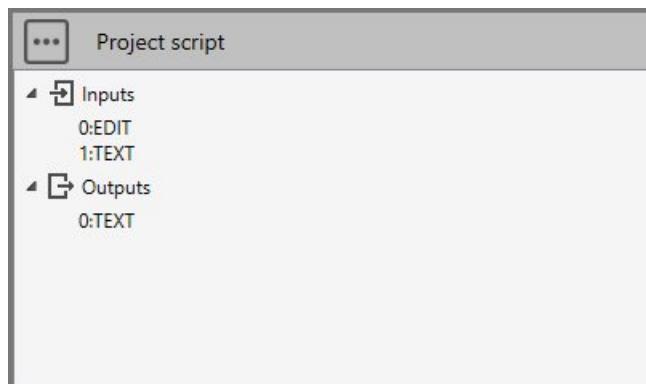
Give it the name "Send SUBJECT and DATE by email".

You need to create a virtual environment :



3. Script properties

Add 1 EDIT input, 1 TEXT input and 1 TEXT output :



Don't forget to save !

The EDIT is for the SUBJECT

The TEXT is for the date with a connection to script "Click at each stage of the project"

4. Main.py

Copy/paste this code in the Editor :

```
study.Outputs[0] = study.Inputs[0] + " - " + study.Inputs[1] + " must be sent by email"
```

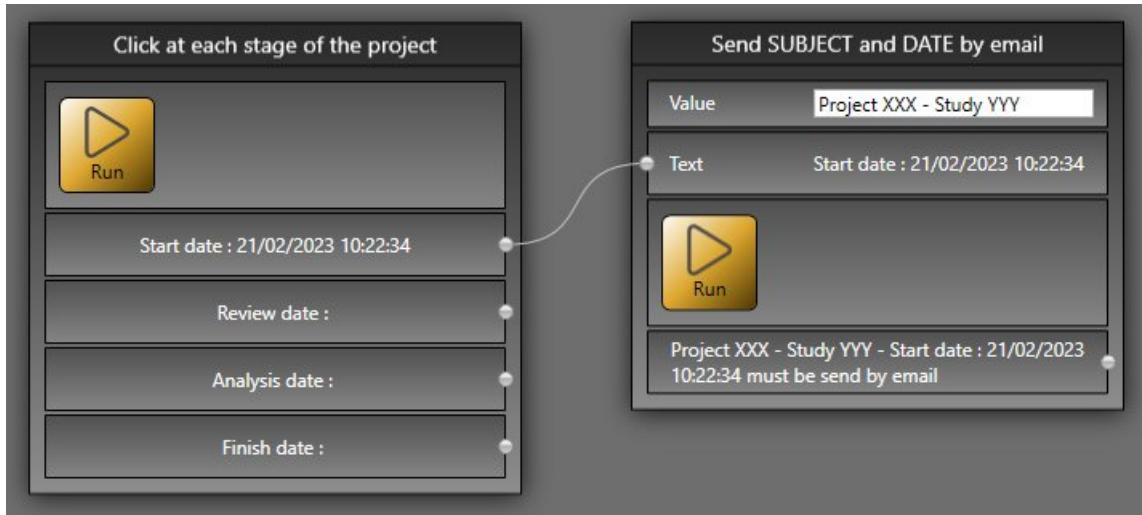
study.Inputs[0] take the text that user write in the EDIT.

study.Inputs[1] take the text coming from a connection.

Save it !

5. Result

When you add this script in a current study, write a SUBJECT, connect a DATE to the second input text and after a click on the **RUN** button :

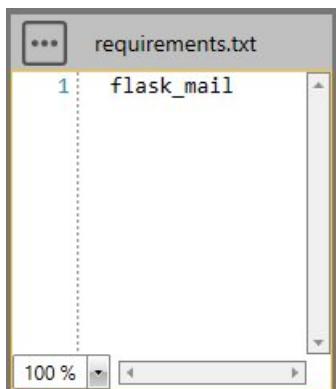


6. Add a new script for email

To send an email, we will use **flask_mail** library from pip.

6.1 requirements.txt

Add flask_mail in requirements.txt, **save** and **install** it from the three points button.



This may take a few minutes...

This window will appears :

```

Collecting flask_mail
  Using cached Flask-Mail-0.9.1.tar.gz (45 kB)
  Preparing metadata (setup.py): started
  Preparing metadata (setup.py): finished with status 'done'
Collecting Flask
  Downloading Flask-2.2.3-py3-none-any.whl (101 kB)
----- 101.8/101.8 kB 266.3 kB/s eta 0:00:00
Collecting blinker
  Using cached blinker-1.5-py2.py3-none-any.whl (12 kB)
Collecting Jinja2>=3.0
  Using cached Jinja2-3.1.2-py3-none-any.whl (133 kB)
Collecting itsdangerous>=2.0
  Using cached itsdangerous-2.1.2-py3-none-any.whl (15 kB)
Collecting importlib-metadata>=3.6.0
  Downloading importlib_metadata-6.0.0-py3-none-any.whl (21 kB)
Collecting Werkzeug>=2.2.2
  Downloading Werkzeug-2.2.3-py3-none-any.whl (233 kB)
----- 233.6/233.6 kB 1.2 MB/s eta 0:00:00
Collecting click>=8.0
  Using cached click-8.1.3-py3-none-any.whl (96 kB)
Collecting colorama
  Using cached colorama-0.4.6-py2.py3-none-any.whl (25 kB)
Collecting typing-extensions>=3.6.4
  Downloading typing_extensions-4.5.0-py3-none-any.whl (27 kB)
Collecting zipp>=0.5
  Downloading zipp-3.14.0-py3-none-any.whl (6.7 kB)
Collecting MarkupSafe>=2.0
  Downloading MarkupSafe-2.1.2-cp37-cp37m-win_amd64.whl (16 kB)
Installing collected packages: zipp, typing-extensions, MarkupSafe, itsdangerous, colorama, blinker, Werkzeug, Jinja2, importlib-metadata, click, Flask, flask_mail
  Running setup.py install for flask_mail: started
  Running setup.py install for flask_mail: finished with status 'done'
Successfully installed Flask-2.2.3 Jinja2-3.1.2 MarkupSafe-2.1.2 Werkzeug-2.2.3 blinker-1.5 click-8.1.3 colorama-0.4.6 flask_mail-0.9.1 importlib-metadata-6.0.0 itsdangerous-2.1.2 typing-extensions-4.5.0 zipp-3.14.0

```

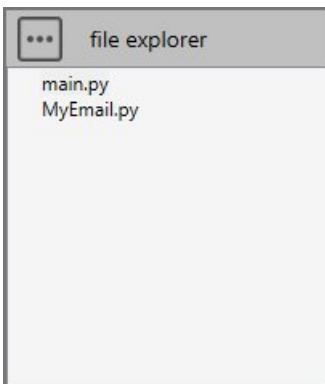
OK

WARNING

If you don't see this window, it means that the library has not been correctly installed. Close the application and reinstall requirements.txt.

6.2 file explorer

Add a new python file and name it **MyEmail.py** from the three points button.



WARNING

Use name that don't exists in Python. In this example, if you choose the name email.py, it would not work !

6.3 MyEmail.py

Select the file.

Copy/paste this code in the Editor :

```

import os
from flask import Flask
from flask_mail import Mail, Message
import mimetypes

MAIL_SERVER = ''          # To be completed by user
MAIL_PORT = 0              # To be completed by user
MAIL_USE_TLS = False       # To be completed by user
MAIL_USE_SSL = True        # To be completed by user
MAIL_USERNAME = ''          # To be completed by user
MAIL_PASSWORD = ''          # To be completed by user
MAIL SUBJECT_PREFIX = ''    # To be completed by user
MAIL_SENDER = ''            # To be completed by user

def send_email(to_email, subject, body, attachement = None):
    mail = Mail()

    app = Flask(__name__)
    app.config['MAIL_SERVER'] = MAIL_SERVER
    app.config['MAIL_PORT'] = MAIL_PORT
    app.config['MAIL_USE_TLS'] = MAIL_USE_TLS
    app.config['MAIL_USE_SSL'] = MAIL_USE_SSL
    app.config['MAIL_USERNAME'] = MAIL_USERNAME
    app.config['MAIL_PASSWORD'] = MAIL_PASSWORD

    mail.init_app(app)

    msg = Message(subject,
                  sender = MAIL_SENDER,
                  recipients = [to_email])
    msg.body = body
    if attachement != None:
        mime = mimetypes.guess_type(attachement)
        filename = os.path.basename(attachement)
        with open(attachement, 'rb') as f:
            msg.attach(filename=filename, content_type=mime[0], data=f.read(), disposition=None, headers=None)

    with app.app_context():
        mail.send(msg)

```

Complete it with your email server data and save it !

6.4 main.py

Select the file.

Copy/paste this code in the Editor and change the email `info@cwantic.com` to another destination :

```

from MyEmail import send_email

send_email(to_email='info@cwantic.com', subject=study.Inputs[0], body=study.Inputs[1])

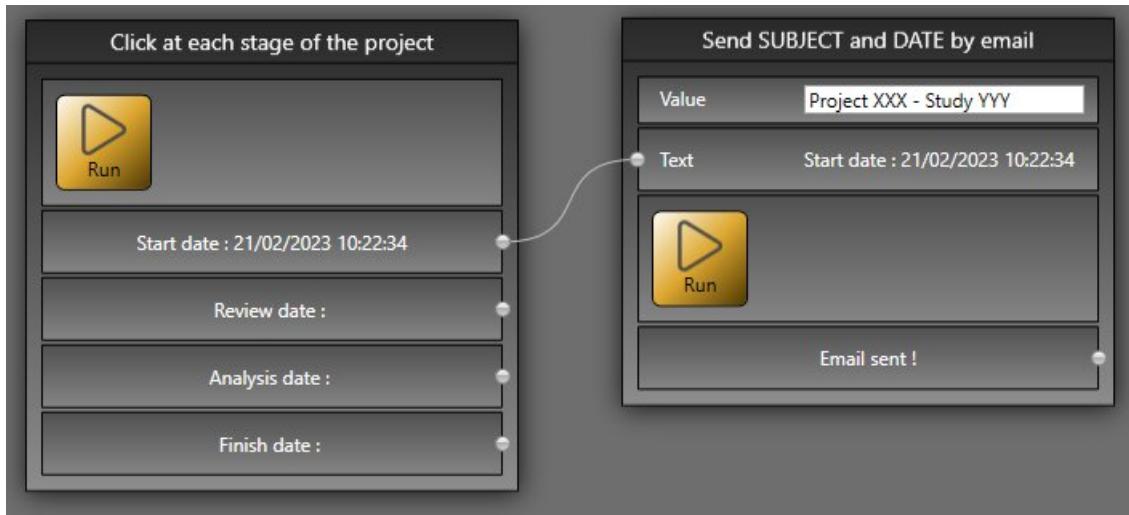
study.Outputs[0] = 'Email sent !'

```

Save it !

7. New result

After a click on the RUN button :



And in the email box :



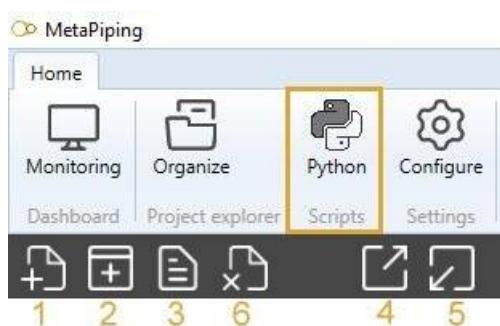
Sample 3 : Converter

1. Goal

Create a script that will (after pressing **RUN** button) convert user data in a EXCEL file to metaL file format.

2. Script definition

From the Home/Python, click on button 1 (Add project script) :



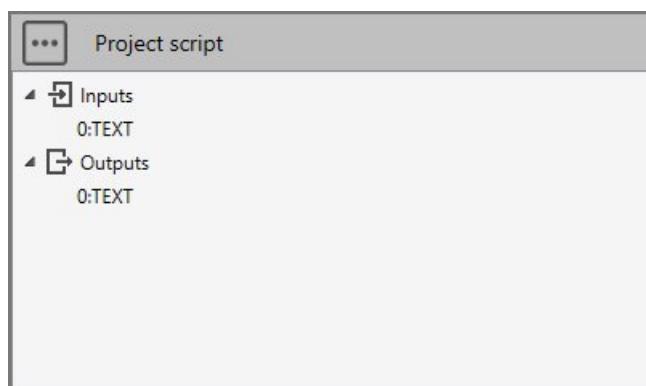
Give it the name "Convert XLSX to MetaL".

Say **NO** to the question about creation of virtual environment. The Requirements.txt will be hidden.



3. Script properties

Add 1 TEXT input and 1 TEXT output :



The input TEXT must receive a XLSX filename from a connection.

The output TEXT is the path to **conception.metaL** filename (the internal file format of MetaPiping). This output can be connected to the main **study** brick.

4. requirements.txt

To be able to read EXCEL file, we need some pip libraries (**pandas** & **openpyxl**):

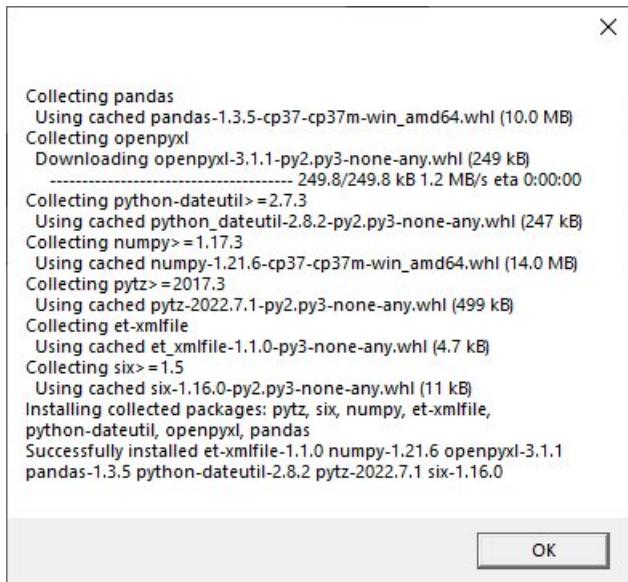
```

requirements.txt
1 pandas
2 openpyxl

```

Add this two libraries in requirements.txt, **save** and **install** it from the three points button.

This window will appears :



WARNING

If you don't see this window, it means that the library has not been correctly installed. Close the application and reinstall requirements.txt.

5. Data

Imagine that user has some data in EXCEL file and want to convert it into a 3D piping model.

The XLSX file has 4 sheets : MATERIALS, SECTIONS, NODES and ELEMENTS :

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	DI
1	ID	NAME	DESCRIPTION	TYPE	REFTEMPERATURE	MAXTEMPERATURE	DENSITY	POISSON	TEMP	EH	EX	SH	SY	SU	SM	CR	GH	CO		
2																				
3	0	100 Steel		2	21	370	7850	0.3		21	2.027E+11	1.15E-05	94500000	206900000		110300000				
4										370	1.758E+11	1.37E-05	86200000	148200000		99300000				
5																				
6																				

	A	B	C	D	E	F	G
1	ID	NAME	DESCRIPTION	DIAMETER	THICKNESS	LINEARMASS	
2				[m]	[m]	[kg/m]	
3	0	1	4"	0.1143	0.00602	16.1	
4							
5							
6							

	A	B	C	D	E	F	G
1	ID	NAME	X	Y	Z		
2			[m]	[m]	[m]		
3	0	10	0	0	0		
4	1	20	2	0	0		
5							
6							

◀ ▶ MATERIALS SECTIONS NODES ELEMENTS +

	A	B	C	D	E	F	G	H	I	J
1	ID	TYPE	ID NODE1	ID NODE2	ID MATERIAL	ID SECTION	XDIR_X	XDIR_Y	XDIR_Z	
2										
3	0	PIPE	0	1	0	0	0	0	0	
4										
5										
6										

◀ ▶ MATERIALS SECTIONS NODES ELEMENTS +

It represents a single pipe.

You can download this file at the bottom of this page.

6 main.py

Select the file.

Copy/paste this code in the Editor :

```
import os
import pandas as pd
from Cwantic.MetaPiping.Core import Layer, RegularMaterial, MaterialType, PipeSection, Node, Pipe
from System.Windows.Media.Media3D import Point3D, Vector3D

filename = study.Inputs[0]
study.Outputs[0] = "Invalid EXCEL file"

def readMaterials(df, metal):
    for index, row in df.iterrows():
        name = row["NAME"]
        temp = row["TEMP"]
        if not pd.isna(name):
            material = RegularMaterial()
            material.Name = str(int(name))
            material.Description = str(row["DESCRIPTION"])
            material.Type = MaterialType(int(row["TYPE"]))
            material.RefTemperature = row["REFTEMPERATURE"]
            material.MaxTemperature = row["MAXTEMPERATURE"]
            material.Density = row["DENSITY"]
            material.Poisson = row["POISSON"]
            metal.Materials.Add(material) # Attention ! metal.Materials is a C# list. Use Add instead of append
        elif not pd.isna(temp):
            if not pd.isna(row["EH"]):
                material SetProperty("EH", row["EH"], temp)
            if not pd.isna(row["EX"]):
                material SetProperty("EX", row["EX"], temp)
            if not pd.isna(row["SH"]):
                material SetProperty("SH", row["SH"], temp)
            if not pd.isna(row["SY"]):
                material SetProperty("SY", row["SY"], temp)
            if not pd.isna(row["SU"]):
                material SetProperty("SU", row["SU"], temp)
            if not pd.isna(row["SM"]):
                material SetProperty("SM", row["SM"], temp)
            if not pd.isna(row["CR"]):
                material SetProperty("CR", row["CR"], temp)
            if not pd.isna(row["GH"]):
                material SetProperty("GH", row["GH"], temp)
            if not pd.isna(row["CO"]):
                material SetProperty("CO", row["CO"], temp)
            if not pd.isna(row["DI"]):
                material SetProperty("DI", row["DI"], temp)
```

```

def readSections(df, metal):
    for index, row in df.iterrows():
        name = row["NAME"]
        if not pd.isna(name):
            section = PipeSection()
            section.Name = str(int(name))
            section.Description = str(row["DESCRIPTION"])
            section.Diameter = row["DIAMETER"]
            section.Thickness = row["THICKNESS"]
            section.LinearMass = row["LINEARMASS"]
            metal.Sections.Add(section) # Attention ! metal.Sections is a C# List. Use Add instead of append

def readNodes(df, metal):
    for index, row in df.iterrows():
        name = row["NAME"]
        if not pd.isna(name):
            node = Node()
            node.Name = str(int(name))
            node.Coor = Point3D(row["X"], row["Y"], row["Z"])
            metal.Nodes.Add(node) # Attention ! metal.Nodes is a C# List. Use Add instead of append

def readElements(df, metal):
    for index, row in df.iterrows():
        type = row["TYPE"]
        if type == "PIPE":
            node1 = metal.Nodes[int(row["ID_NODE1"])]
            node2 = metal.Nodes[int(row["ID_NODE2"])]
            material = metal.Materials[int(row["ID_MATERIAL"])]
            section = metal.Sections[int(row["ID_SECTION"])]
            dl = Vector3D(node2.Coord.X - node1.Coord.X, node2.Coord.Y - node1.Coord.Y, node2.Coord.Z - node1.Coord.Z)
            xdir = Vector3D(row["XDIR_X"], row["XDIR_Y"], row["XDIR_Z"])
            pipe = Pipe(0, node1, node2, dl, material, xdir, section)
            metal.Elements.Add(pipe) # Attention ! metal.Elements is a C# List. Use Add instead of append
        else:
            pass # TO DO : decode all possible types

if os.path.exists(filename):
    df_materials = pd.read_excel(filename, sheet_name="MATERIALS")
    df_sections = pd.read_excel(filename, sheet_name="SECTIONS")
    df_nodes = pd.read_excel(filename, sheet_name="NODES")
    df_elements = pd.read_excel(filename, sheet_name="ELEMENTS")

    if not df_materials.empty and not df_sections.empty and not df_nodes.empty and not df_elements.empty:
        # 1. output filename
        dir = study.getDirectory()
        filename = os.path.join(dir, "conception.metal")

        # 2. Create empty metal
        metal = study.createMetal()

        # 3. Layer
        layer = Layer("0")
        metal.Layers.Add(layer) # Attention ! metal.Layers is a C# List. Use Add instead of append

        # 4. Materials
        readMaterials(df_materials, metal)

        # 5. Sections
        readSections(df_sections, metal)

        # 6. Nodes
        readNodes(df_nodes, metal)

        # 7. Elements
        readElements(df_elements, metal)

        # 8. Save metal
        metal.SaveToFile(filename)

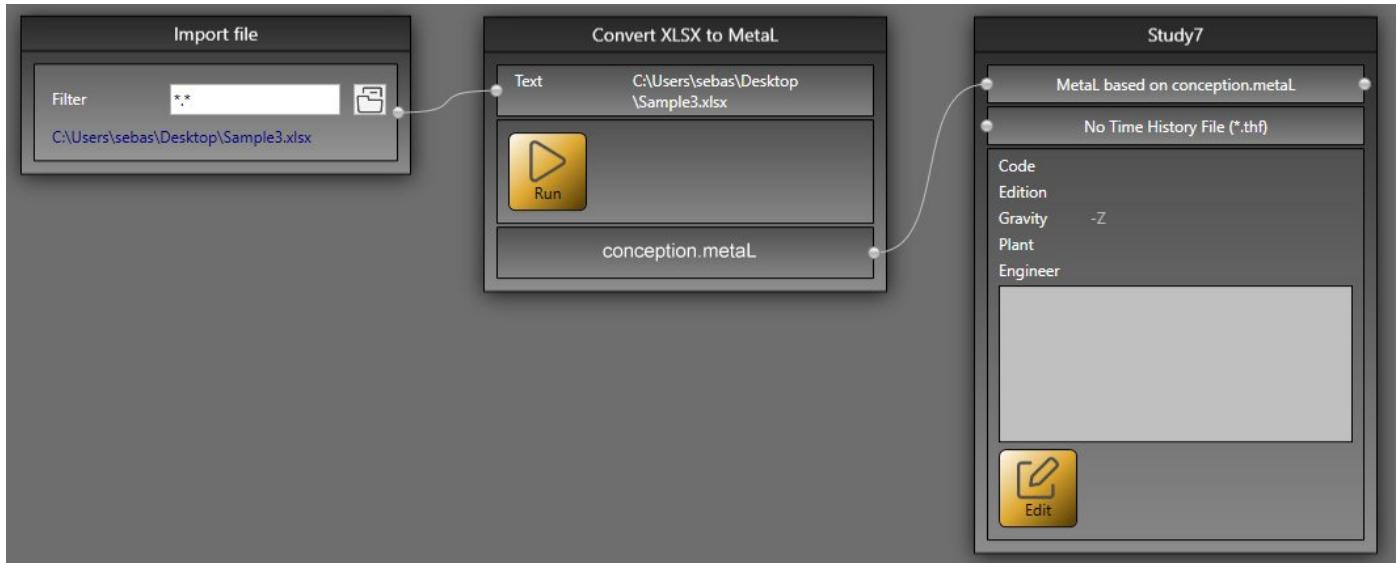
        # 9. Show filename for connection
        study.Outputs[0] = filename

```

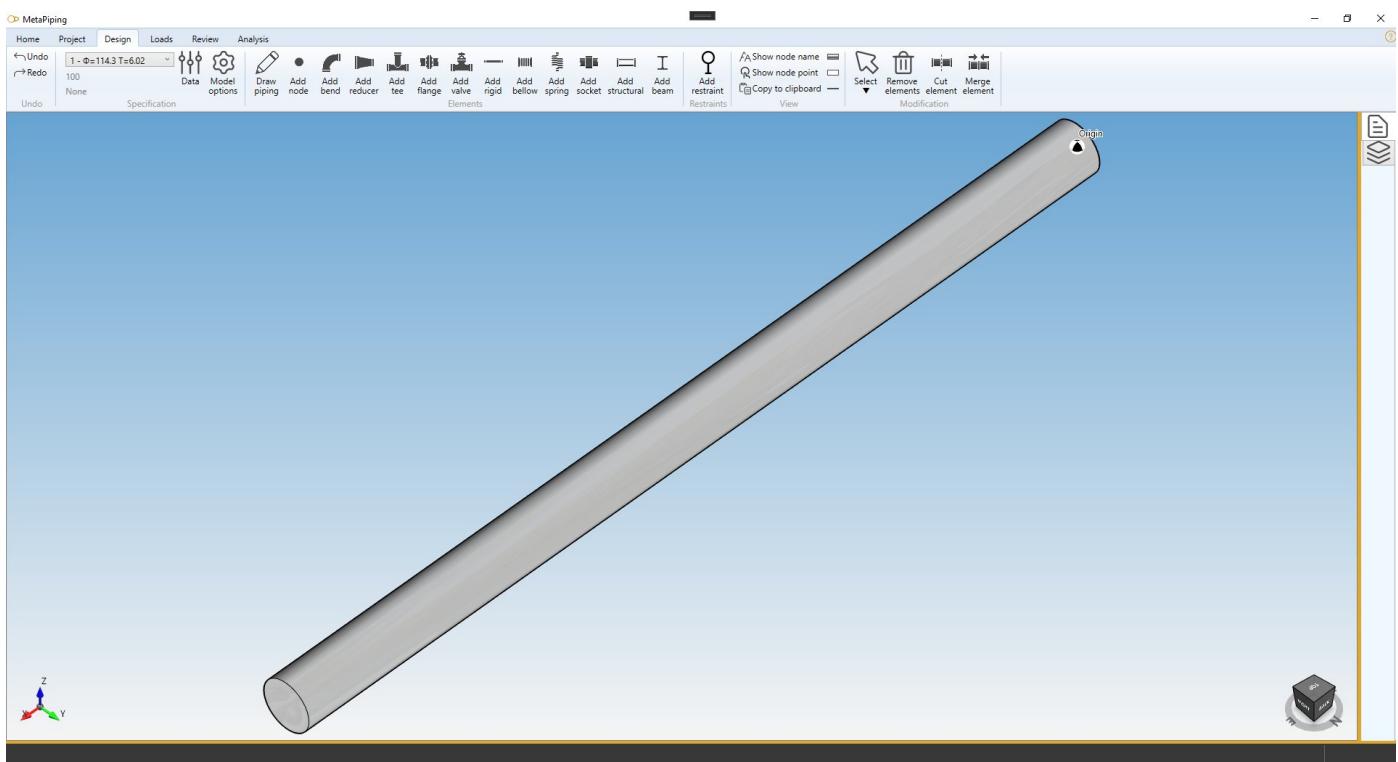
Save it !

7. Result

After IMPORT of the EXCEL file, the connection to the first TEXT and a click on the RUN button :



And in the Piping screen :



With the material and section correctly set.

Click [here](#) to download the EXCEL file used in this example.

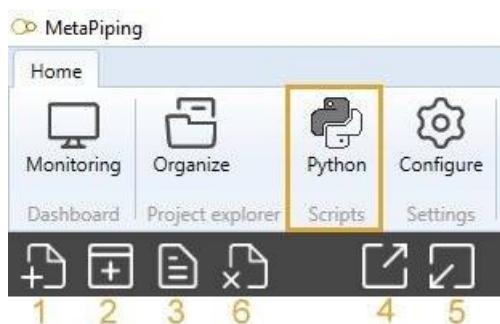
Sample 4 : Total mass of a piping

1. Goal

Create a script that will (after pressing **RUN** button) calculate the total mass of a piping based on the **MetaL** file.

2. Script definition

From the Home/Python, click on button 1 (Add project script) :



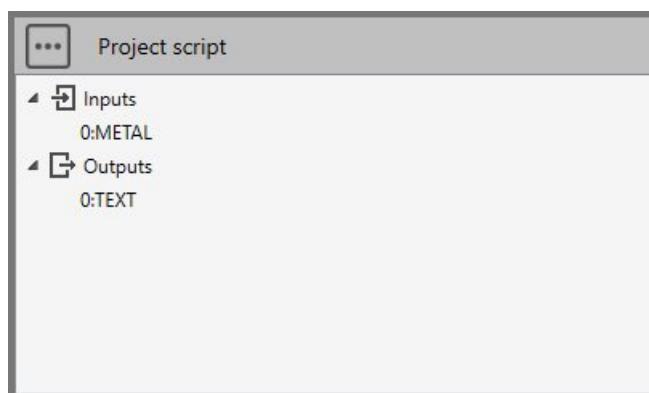
Give it the name "Total mass of the piping"

Say **NO** to the question about creation of virtual environment. The Requirements.txt will be hidden.



3. Script properties

Add 1 METAL input and 1 TEXT output :



The input METAL must receive the Metal file (the internal file format of MetaPiping) from a connection of the main **study** brick.

The output TEXT shows the result.

4. Main.py

Copy/paste this code in the Editor :

```

from Cwantic.MetaPiping.Core import Piping

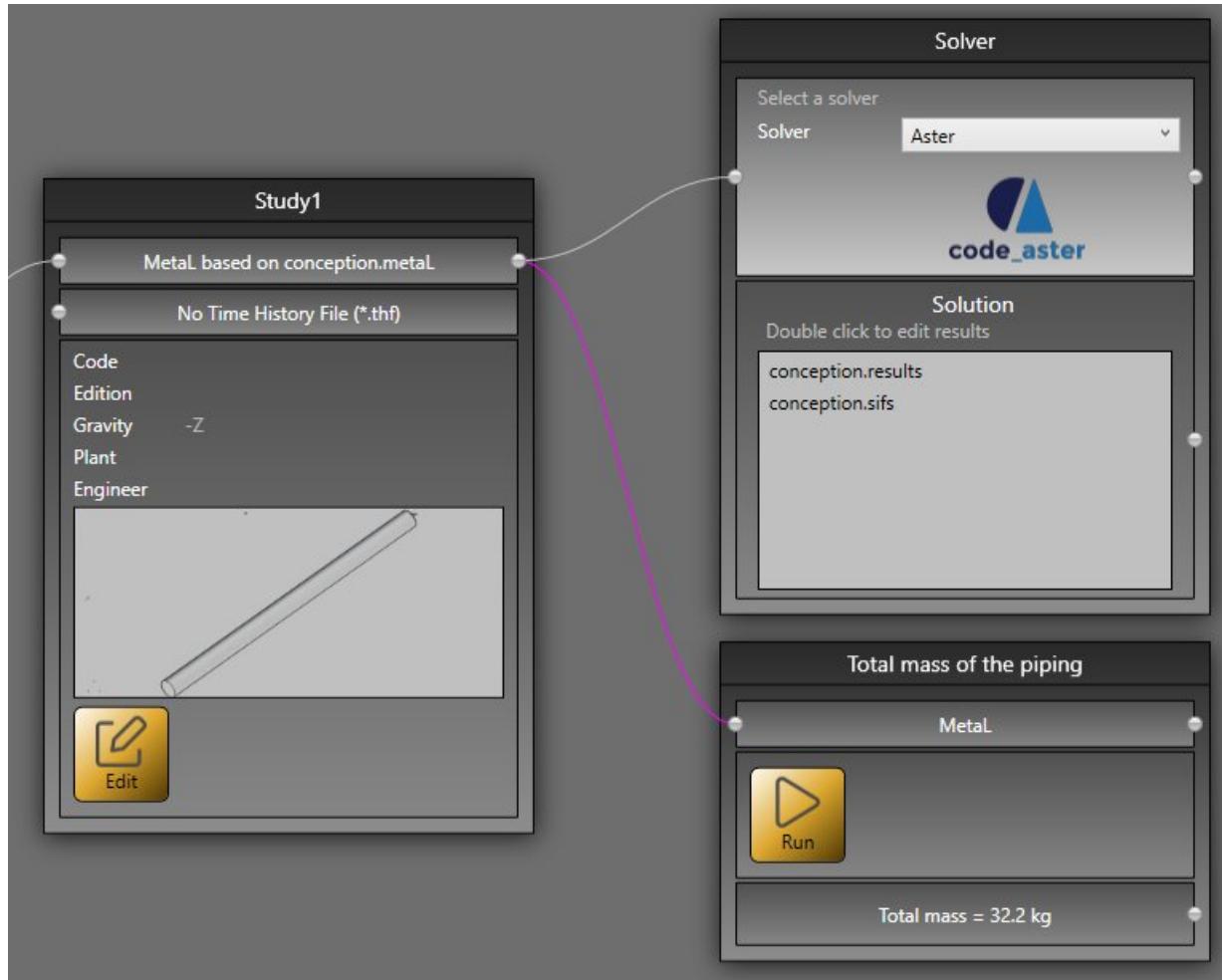
# Check if the connection has been made with the Metal connector
metal = study.getMetal()
if metal != None:
    total = 0
    for item in metal.Elements:
        if isinstance(item, Piping):
            # item is a Piping so we can access the Section property
            mass = item.Section.LinearMass * item.DL.Length
            total += mass
    study.Outputs[0] = "Total mass = {} kg".format(total)
else:
    study.Outputs[0] = "Please connect Metal !"

```

Save it !

5. Result

When you add this script in a current study, connect the Metal output of the **study brick** and after a click on the **RUN** button :



You have the total mass !

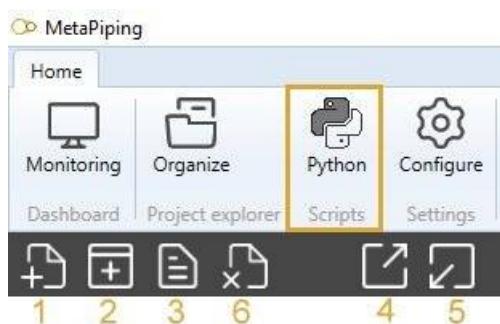
Sample 5 : Static loadcases of a piping

1. Goal

Create a script that will (after pressing **RUN** button) retrieve all static cases (Number + Title) of a piping based on the **Metal** file.

2. Script definition

From the Home/Python, click on button 1 (Add project script) :



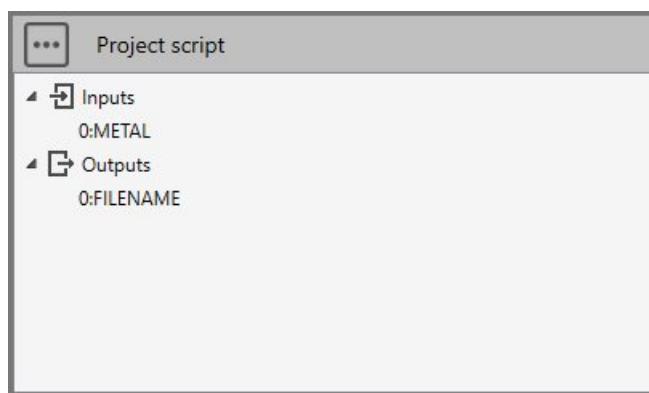
Give it the name "Static loadcases of the piping"

Say **NO** to the question about creation of virtual environment. The Requirements.txt will be hidden.



3. Script properties

Add 1 METAL input and 1 FILENAME output :



The input METAL must receive the Metal file (the internal file format of MetaPiping) from a connection of the main **study** brick.

The output FILENAME will expose a TXT file. You can double click to see the content.

4. Main.py

Copy/paste this code in the Editor :

```

import os

# Check if the connection has been made with the Metal connector
metal = study.getMetal()
if metal != None:
    dir = study.getDirectory()
    filename = os.path.join(dir, "StaticLoadcases.txt")

# Get all static cases
cases = []
for item in metal.StaticCases:
    st = "{} : {}".format(item.Number, item.Title)
    cases.append(st)

# Save the file
with open(filename,'w') as f:
    f.write('\n'.join(cases))

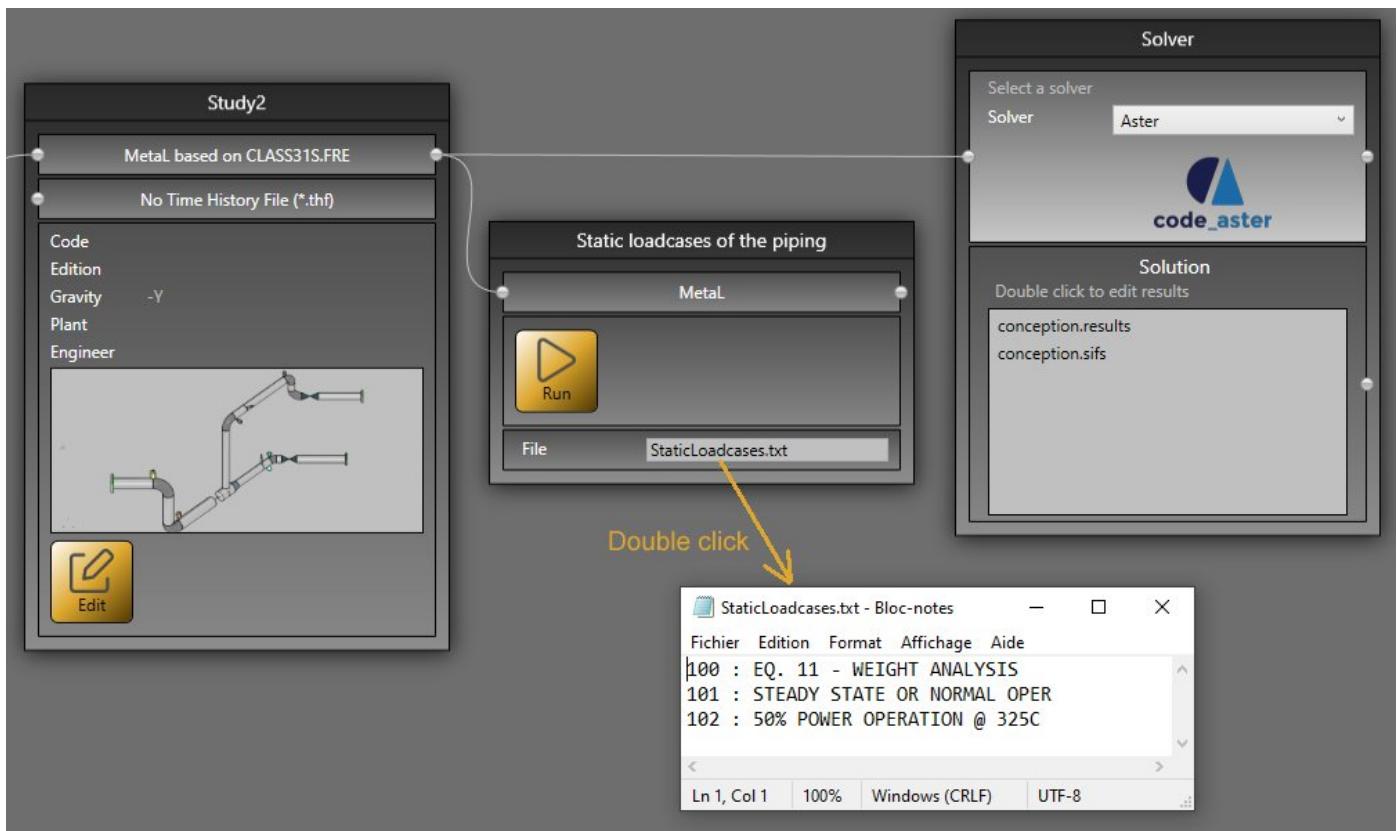
study.Outputs[0] = "StaticLoadcases.txt"
else:
    study.Outputs[0] = "Please connect Metal !"

```

Save it !

5. Result

When you add this script in a current study, connect the Metal output of the **study brick** and after a click on the **RUN** button :



You have the list of all static cases !



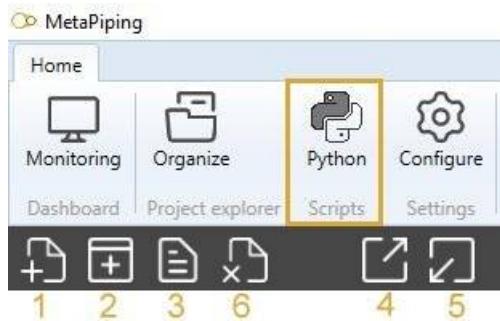
Sample 6 : Max displacement by static case number

1. Goal

Create a script that will (after pressing **RUN** button) retrieve the **displacement max** in mm of the whole piping for a particular loadcase number.

2. Script definition

From the Home/Python, click on button 1 (Add project script) :



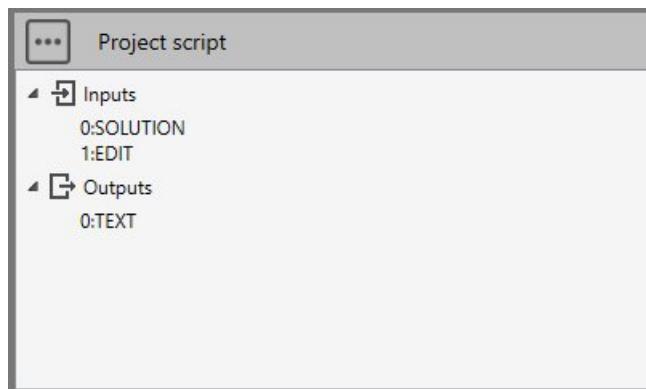
Give it the name "Max displacement by loadcase number"

Say **NO** to the question about creation of virtual environment. The Requirements.txt will be hidden.



3. Script properties

Add 1 SOLUTION input, 1 EDIT input and 1 TEXT output :



The input SOLUTION must receive a connection from the Solution connector of the **solver** brick.

The input EDIT must receive the static loadcase number.

The output TEXT will show the result.

4. Main.py

Copy/paste this code in the Editor :

```

import math

# Check if the connection has been made with the Solution connector
solution = study.getSolution()
if solution != None:
    # Check if Inputs[1] is an integer
    try:
        case = int(study.Inputs[1])
    except ValueError:
        case = 0

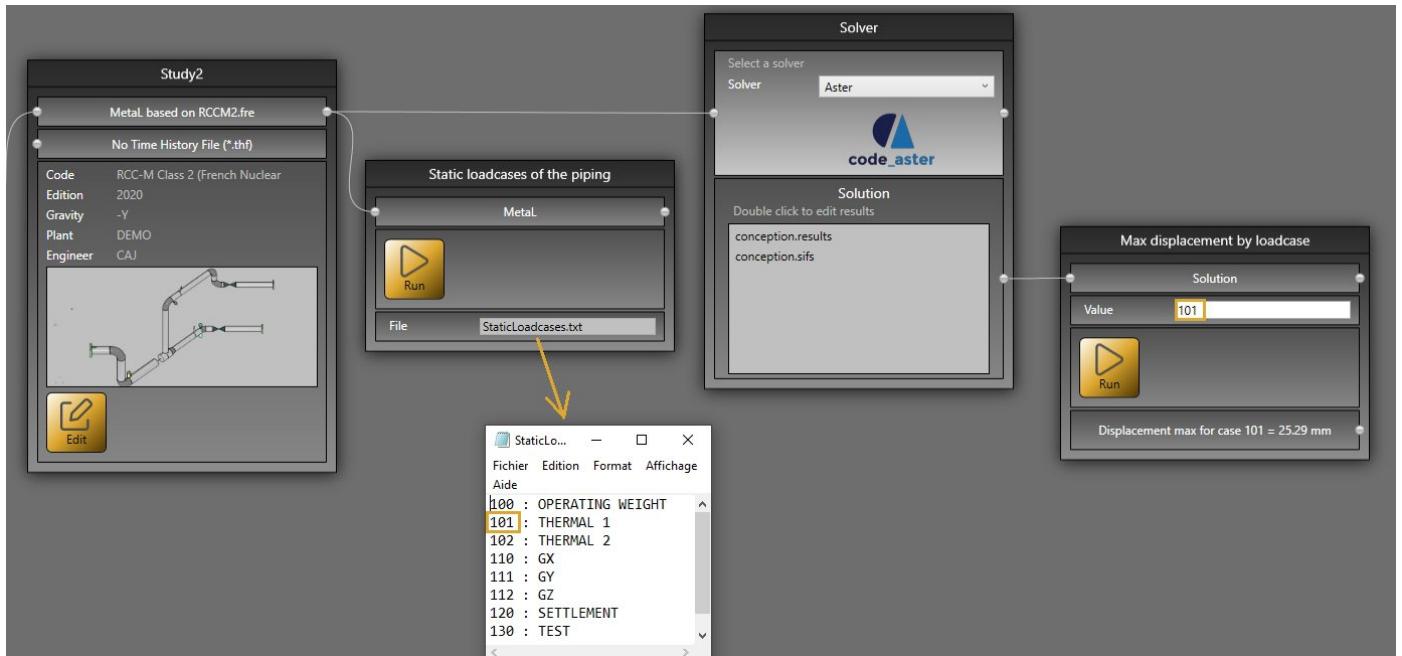
    if case>0:
        # Get displacements from solution with case number
        displacements = solution.getDisplacements(case)
        if len(displacements)>0:
            # Calculate the max
            max = -999999
            for displacement in displacements:
                d = displacement.Item1
                l = math.sqrt(d.X**2 + d.Y**2 + d.Z**2)
                if l > max:
                    max = l
            study.Outputs[0] = "Displacement max for case {} = {} mm".format(case, round(max, 2))
        else:
            study.Outputs[0] = "This loadcase number doesn't exists !"
    else:
        study.Outputs[0] = "Please enter an existing loadcase number !"
else:
    study.Outputs[0] = "Please connect Solution !"

```

Save it !

5. Result

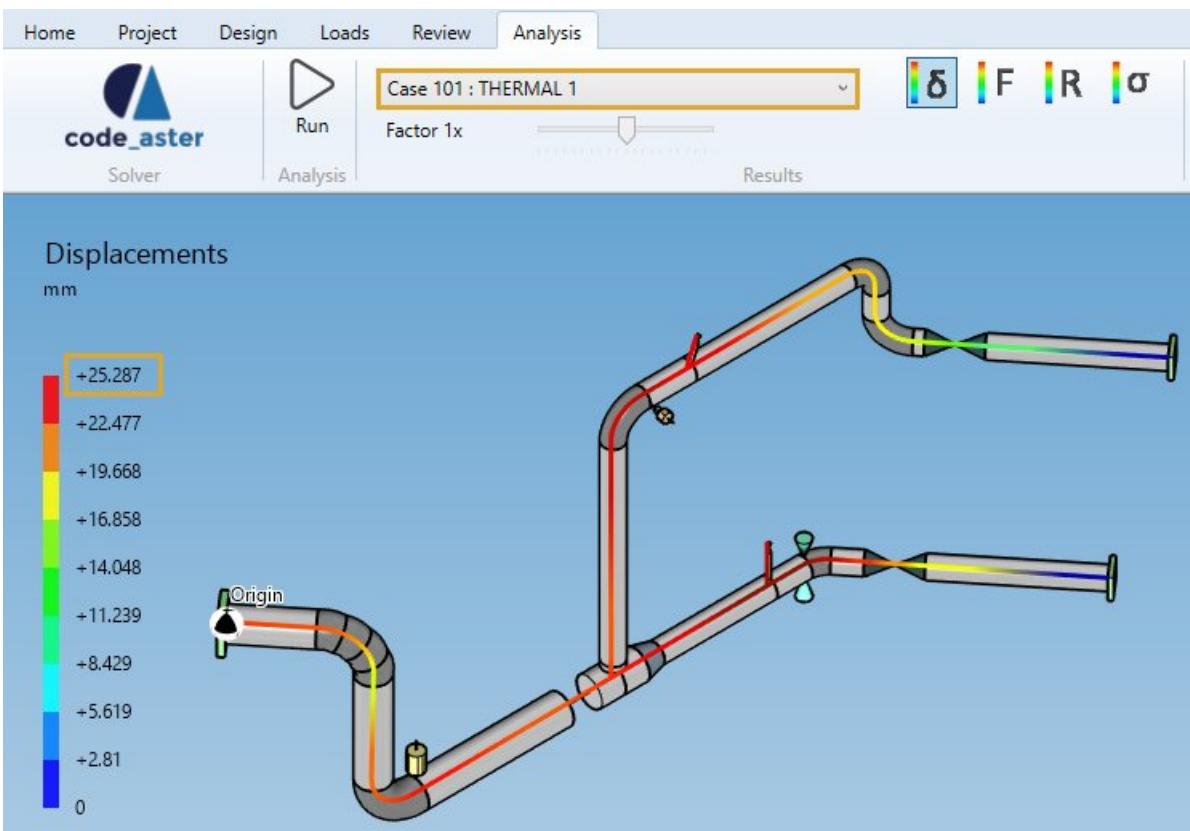
When you add this script in a current study, connect the Solution output of the **solver brick**, enter a static loadcase number and after a click on the **RUN** button :



You have the max displacement !

This sample use [sample5](#) to show all possible static loadcases.

And in **Design** mode :

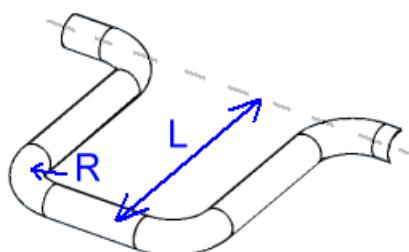


Sample 7 : Replace a selected pipe by an expansion loop (OBSOLETE)

See sample 8 instead !

1. Goal

Create a script that will (after pressing a **button** in the ribbon) replace a selected pipe by an expansion loop. A window will appear to define the size and the default bend radius.

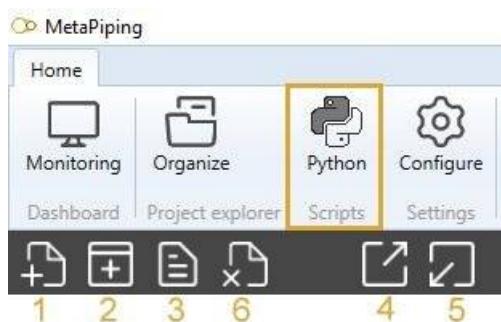


This feature does not exist in MetaPiping 2024 so it is useful that user can create his own COMMANDS based on the Metal internal format and the existing documented commands.

Since the user will have to create a COMMAND, the application will manage the undo/redo operations by itself !

2. Script definition

From the Home/Python, click on **button 2** (Add 3D script) :



Give it the name "Add loop"

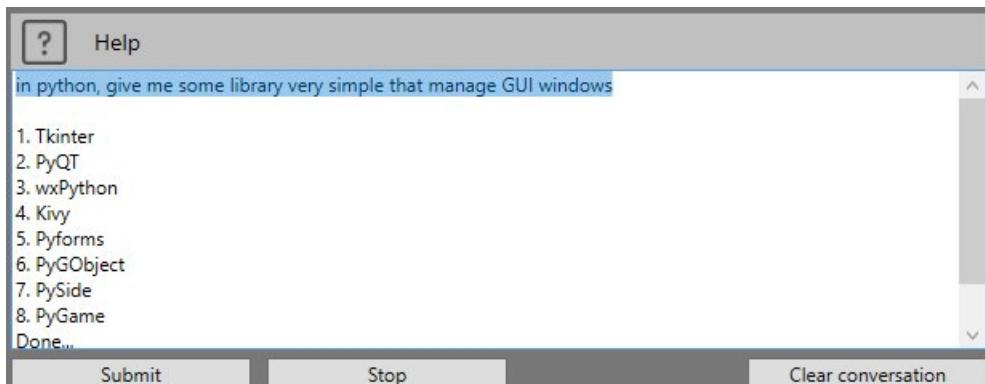
You need to create a virtual environment :



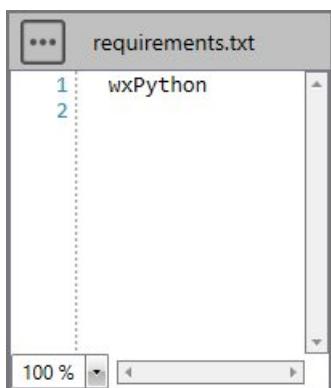
3. requirements.txt

We want to show a **window** so that the user can define the **L** and **R** values, respectively the size and the bend radius of the expansion loop.

We ask [ChatGPT](#) to recommend some python libraries :

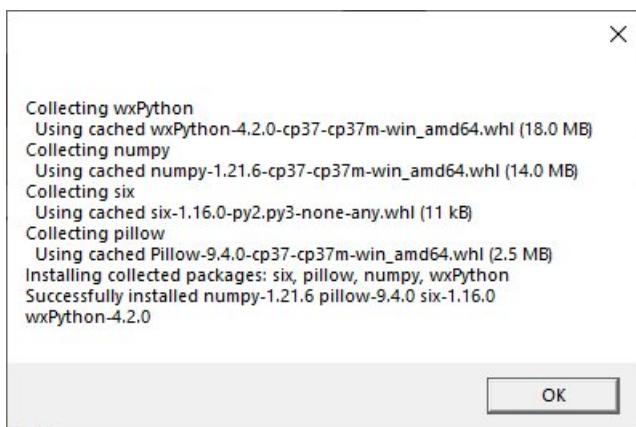


We will choose wxPython :



Add this library in requirements.txt, **save** and **install** it from the three points button.

This window will appears :

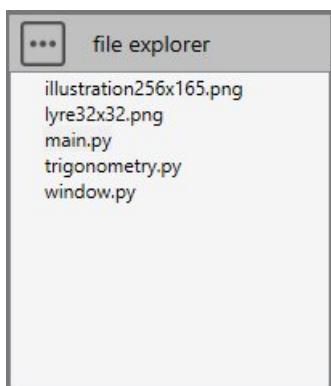


WARNING

If you don't see this window, it means that the library has not been correctly installed. Close the application and reinstall requirements.txt.

4. Script files

This kind of script needs some **images** in order to illustrate the **button** and the **window**.



Press the “three points” button and select “Import image”. You need to create a 32x32 transparent PNG for the icon of the button and an illustration of the expansion loop for the window.

Press the “three points” button and select “Add a python file”. We will create a file for some trigonometric functions and one for the definition of the window.

4.1 trigonometry.py

Select the file.

Copy/paste this code in the Editor :

```
def cross(vec1, vec2):
    return [vec1[1]*vec2[2] - vec1[2]*vec2[1], vec1[2]*vec2[0] - vec1[0]*vec2[2], vec1[0]*vec2[1] - vec1[1]*vec2[0]]

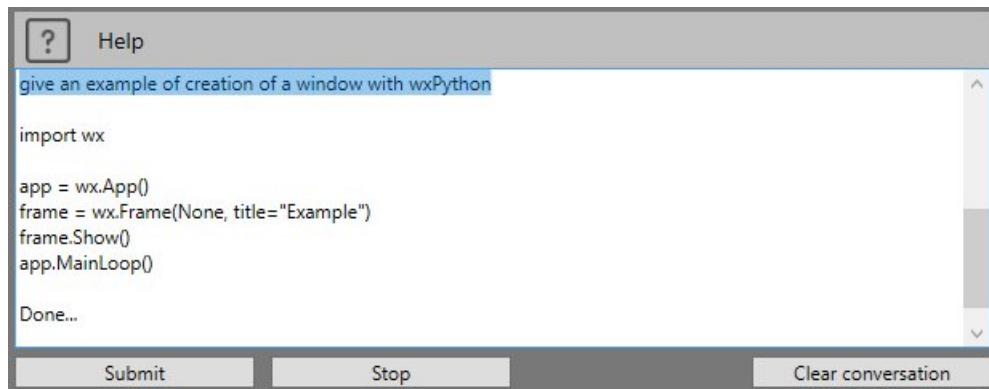
def length(vec):
    return (vec[0]**2 + vec[1]**2 + vec[2]**2)**0.5

def normalize(vec):
    l = length(vec)
    return [vec[0]/l, vec[1]/l, vec[2]/l]
```

Save it !

4.2 window.py

Ask help to [ChatGPT](#) :



The screenshot shows a ChatGPT interface. The user has asked for an example of creating a window with wxPython. The AI has provided the following Python code:

```
import wx

app = wx.App()
frame = wx.Frame(None, title="Example")
frame.Show()
app.MainLoop()
```

At the bottom of the interface, there are three buttons: "Submit", "Stop", and "Clear conversation".

After some research and multiple tests :

```

import wx
import os

class Frame(wx.Frame):
    def __init__(self, parent, id, title, app, defaultsize, defaultRadius):
        wx.Frame.__init__(self, parent, id, title, size=(300, 310))
        self.app = app

        panel = wx.Panel(self)
        panel.BackgroundColour = 'white'

        png = wx.Image(os.path.join(os.path.dirname(__file__), 'illustration256x165.png'), wx.BITMAP_TYPE_PNG).ConvertToBitmap()
        sb = wx.StaticBitmap(panel, -1, png, (0, 0), (png.GetWidth(), png.GetHeight()))

        text = wx.StaticText(panel, -1, "L =", pos = (10, 183))
        text.SetSize(text.GetBestSize())

        self.edit = wx.TextCtrl(panel, -1, str(defaultsize), size=(100, -1), pos = (50, 180))
        self.edit.SetInsertionPoint(0)

        text2 = wx.StaticText(panel, -1, "R =", pos = (10, 213))
        text2.SetSize(text2.GetBestSize())

        self.edit2 = wx.TextCtrl(panel, -1, str(defaultRadius), size=(100, -1), pos = (50, 210))
        self.edit2.SetInsertionPoint(0)

        btn = wx.Button(panel, -1, "Run", size=(100, 20), pos = (100, 240))
        self.Bind(wx.EVT_BUTTON, self.OnRun, btn)

        self.CenterOnScreen(wx.BOTH)

    def OnRun(self, evt):
        self.app.SetValue(self.edit.GetValue())
        self.app.SetRValue(self.edit2.GetValue())
        self.Close()

class App(wx.App):
    def __init__(self, defaultsize, defaultRadius):
        wx.App.__init__(self)
        self.Lvalue = ""
        self.Rvalue = ""
        self.frame = Frame(None, wx.ID_ANY, "Loop definition", self, defaultsize, defaultRadius)
        self.SetTopWindow(self.frame)
        self.frame.Show(True)

    def SetRValue(self, value):
        self.Rvalue = value

    def GetRValue(self):
        return self.Rvalue

    def SetLValue(self, value):
        self.Lvalue = value

    def GetLValue(self):
        return self.Lvalue

```

Save it !

This will create a window with an illustration and 2 edits, one for the length and one for the radius.

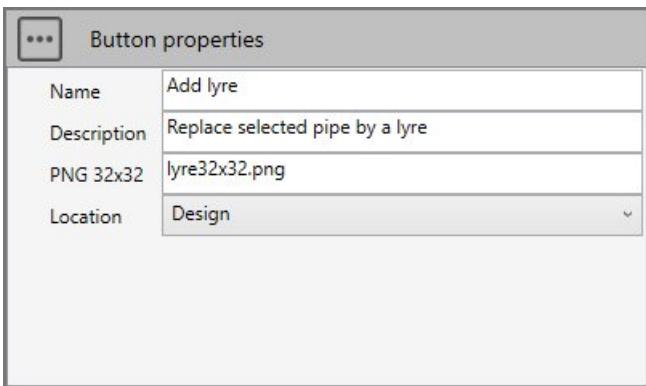
You can set the **default values** in the parameters of the App.

GetLValue() gives the user length.

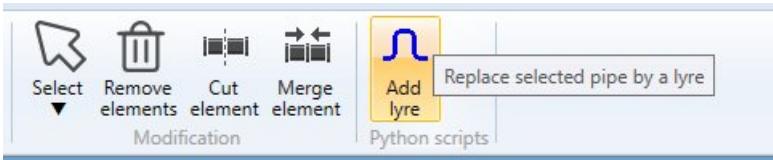
GetRValue() gives the user radius.

5. Script properties

In this kind of script, user has to define the **button** :



The location will be **Design** so that the button appears at the end of the ribbon with its name, icon and description :



6. main.py

Select the file.

Copy/paste this code in the Editor :

```
#####
# CWANTIC LOOP GUI EXAMPLE #
#####

from window import App
from System.Windows.Media.Media3D import Vector3D
from Cwantic.MetaPiping.Core import RemoveElementCommand, AddNodeCommand, DrawPipingCommand, InsertBendCommand

# Inspect selection
n = len(design.selectedList)
res = "Select a pipe !"

if n==1:
    # Check the type of selected element
    if design.isType(design.selectedList[0], "Pipe"):
        pipe = design.selectedList[0]
        p1 = pipe.Node1.Coor
        p2 = pipe.Node2.Coor

        # Get the current model
        model = design.getMetal()

        # Get the scene vertical vector (+Z or +Y)
        verticalvec = design.getVerticalVector()

        # Get the current piping values (section, material, radius,...)
        currentValues = design.getCurrentSpecValues()

        # Memorize current radius
        currentRadius = currentValues.MKS_BendRadius

        # Get the pipe direction vector
        vec1 = Vector3D(pipe.DL.X, pipe.DL.Y, pipe.DL.Z)

        # Compute the cross product to determine the loop direction
        dir = Vector3D.CrossProduct(vec1, verticalvec)
        dir.Normalize()

        # Create a new USER command : cmd
        cmd = design.createCommand("AddLoop")

        # 1 : Remove the selected pipe

        # 1.1 : Create params for command "RemoveElementCommand" (see Help)
        params = []
        params.append(design.selectedList)
```

```

# 1.2 : Add sub command to user command cmd
valid = cmd.addSubCommand("RemoveElementCommand", params)

if valid:
    # 2.0 : Launch window to get Loop's size and bend radius. Default size = 1.0, default radius = currentRadius
    app = App(1.0, currentRadius)
    app.MainLoop()

    # retrieve the size and radius from the app
    size = float(app.GetLValue())
    # Set the new Radius for next commands
    currentValues.MKS_BendRadius = float(app.GetRValue())

    # TIP : Create 2 new nodes (N3 and N4) with "AddNodeCommand"
    node1Cmd = AddNodeCommand(model, p1.X + size*dir.X, p1.Y + size*dir.Y, p1.Z + size*dir.Z, "", False, currentValues)
    N3 = node1Cmd.Node

    node2Cmd = AddNodeCommand(model, p2.X + size*dir.X, p2.Y + size*dir.Y, p2.Z + size*dir.Z, "", False, currentValues)
    N4 = node2Cmd.Node

    # Create new pipe perpendicular to selected pipe from node1

    # 2.1 : Create params for command "DrawPipingCommand" (see Help)
    params = []
    params.append(pipe.Node1)
    params.append(N3)
    params.append(size*dir.X)
    params.append(size*dir.Y)
    params.append(size*dir.Z)
    params.append(0.0)
    params.append(0.0)
    params.append(0.0)
    params.append(currentValues)

    # 2.2 : Add sub command
    valid = cmd.addSubCommand("DrawPipingCommand", params)

if valid:
    # 3 : Create new pipe parallel to selected pipe

    # 3.1 : Create params for command "DrawPipingCommand" (see Help)
    params = []
    params.append(N3)
    params.append(N4)
    params.append(vec1.X)
    params.append(vec1.Y)
    params.append(vec1.Z)
    params.append(0.0)
    params.append(0.0)
    params.append(0.0)
    params.append(currentValues)

    # 3.2 : Add sub command
    valid = cmd.addSubCommand("DrawPipingCommand", params)

if valid:
    # 4 : Create new pipe to close the Loop

    # 4.1 : Create params for command "DrawPipingCommand" (see Help)
    params = []
    params.append(N4)
    params.append(pipe.Node2)
    params.append(-size*dir.X)
    params.append(-size*dir.Y)
    params.append(-size*dir.Z)
    params.append(0.0)
    params.append(0.0)
    params.append(0.0)
    params.append(currentValues)

    # 4.2 : Add sub command
    valid = cmd.addSubCommand("DrawPipingCommand", params)

if valid:
    # 5 : Insert a bend at the last node

    # 5.1 Create params for command "InsertBendCommand"

```

```

# 5.1 : Create parameters for command under command
params = []
params.append(pipe.Node2)
params.append(currentValues)

# 5.2 : Add sub command
valid = cmd.addSubCommand("InsertBendCommand", params)

# Execute command
if valid:
    design.executeCommand(cmd)
    res = ""
else:
    res = "Incorrect params"
# Restore the radius
currentValues.MKS_BendRadius = currentRadius
else:
    res = "The selected element is not a pipe"

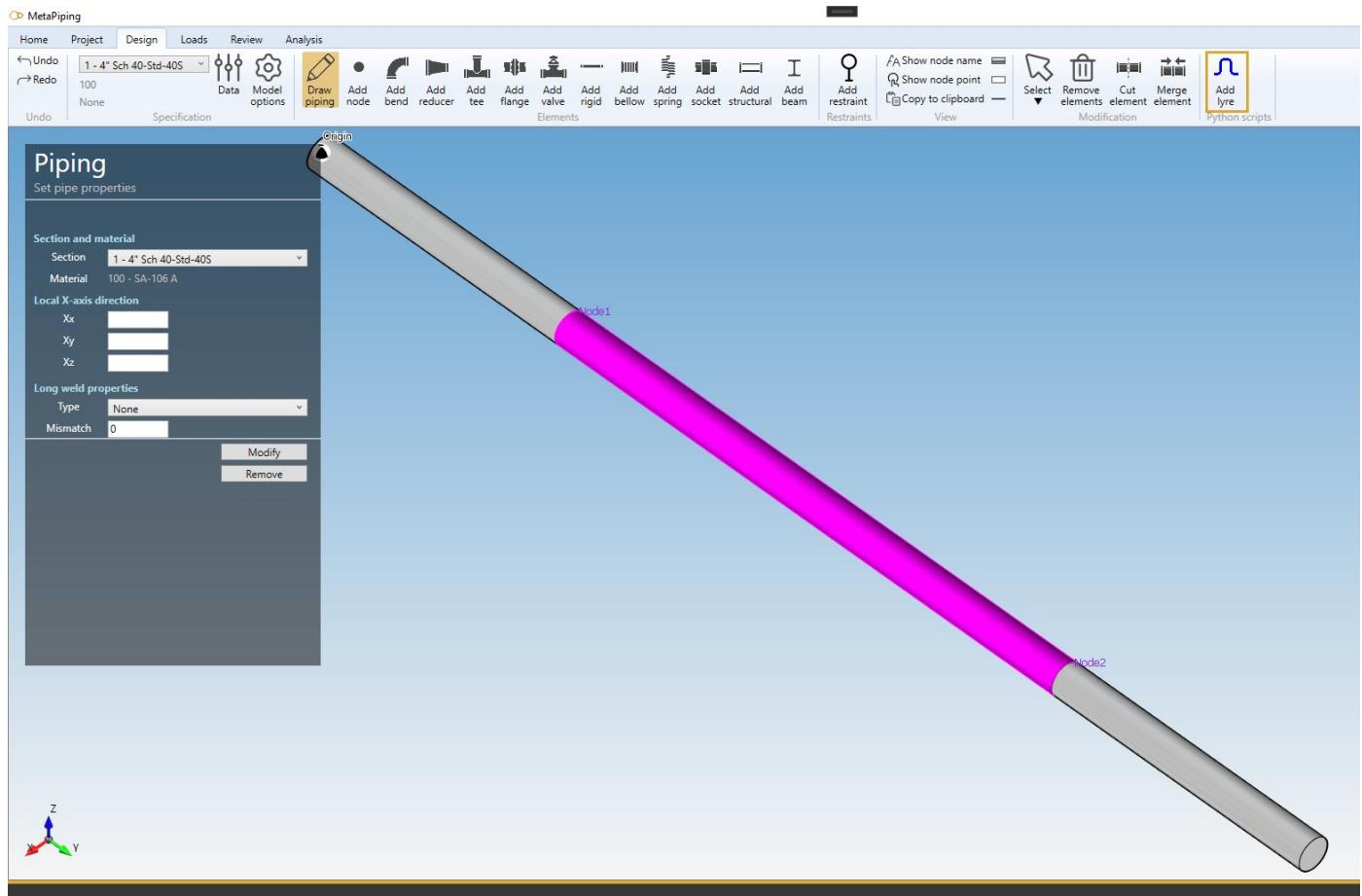
# Show a message (if res != "")
design.result = res

```

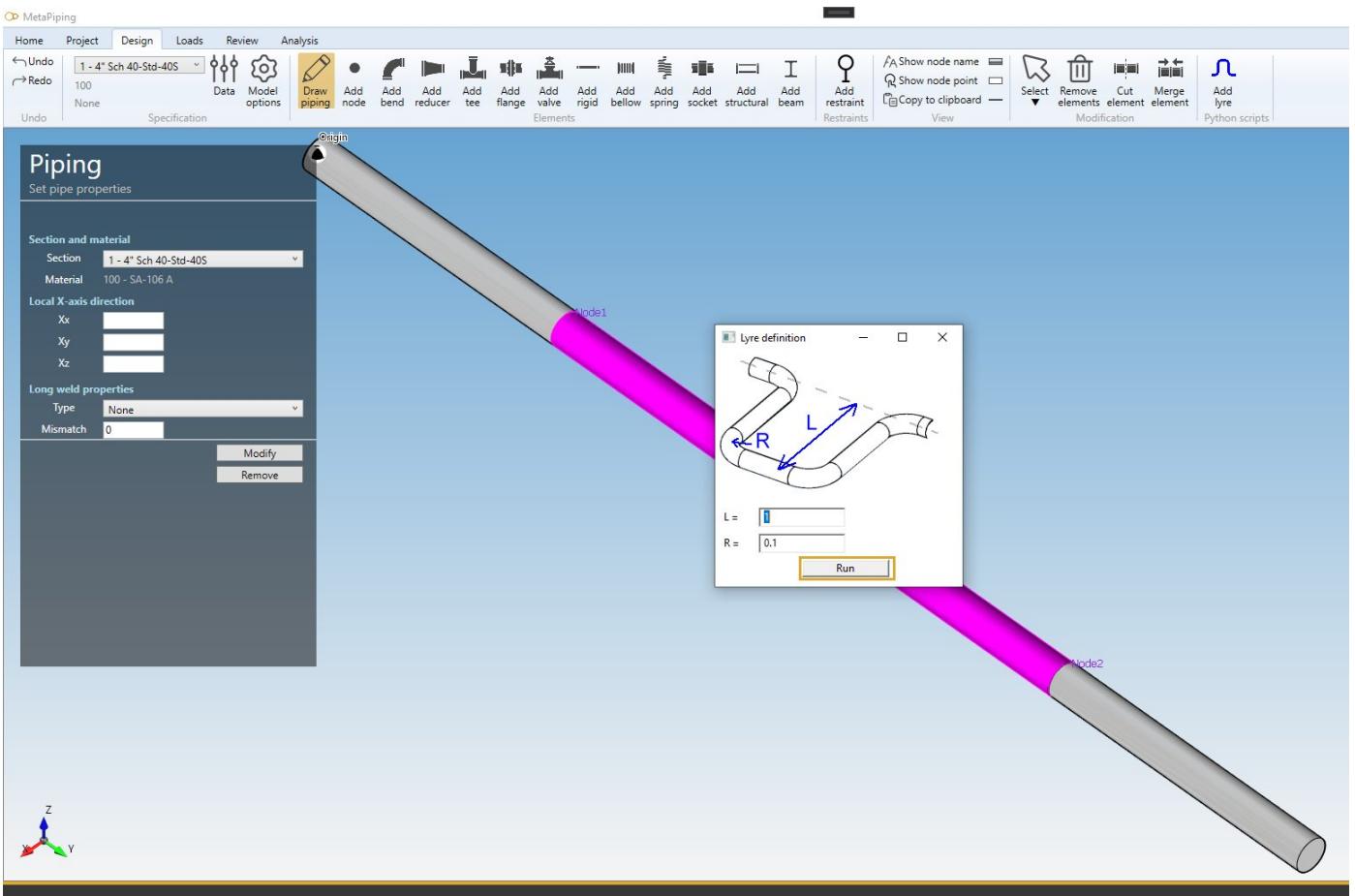
Save it !

7. Result

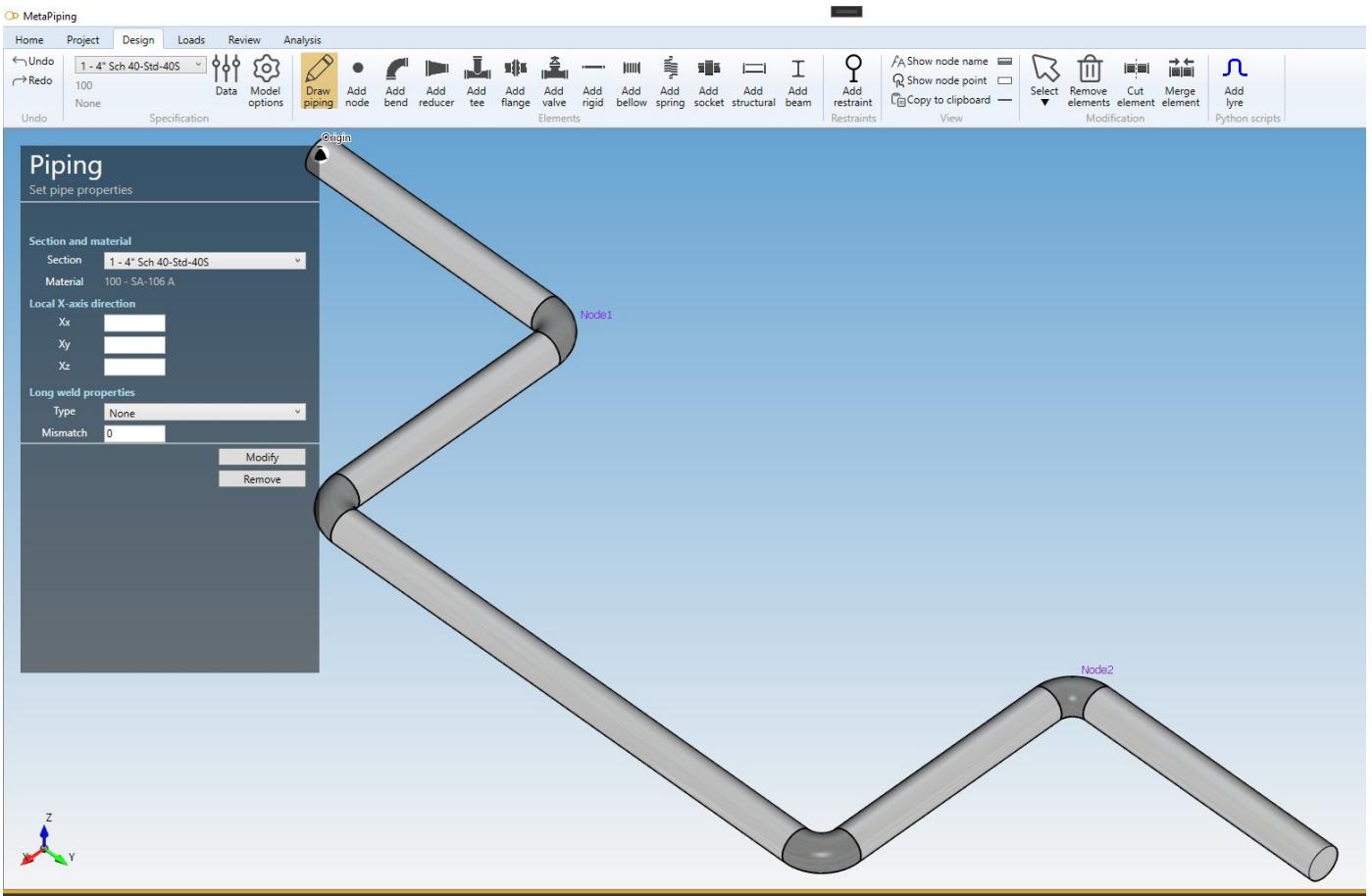
In **Design mode**, select a pipe that you want to replace by an expansion loop :



Click on the new button **Add loop** :



Complete L, R and press Run :



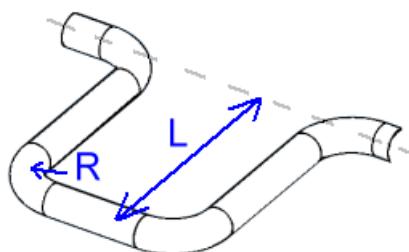
The expansion loop appears !

You can undo/redo your COMMAND !

Sample 8 : Replace a selected pipe by an expansion loop (without virtual env.)

1. Goal

Create a script that will (after pressing a **button** in the ribbon) replace a selected pipe by an expansion loop. A window will appear to define the size and the default bend radius.

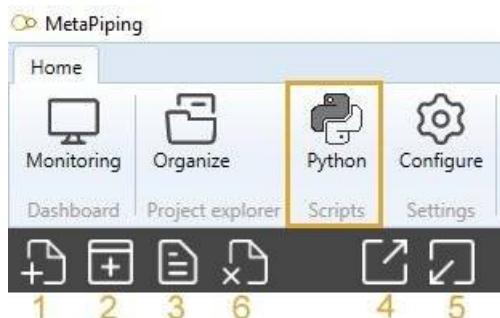


This feature does not exists in MetaPiping 2024 so it is useful that user can create his own COMMANDS based on the Metal internal format and the existing documented commands.

Since the user will have to create a COMMAND, the application will manage the undo/redo operations by itself !

2. Script definition

From the Home/Python, click on **button 2** (Add 3D script) :



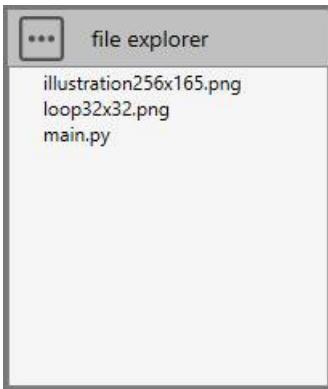
Give it the name "Add loop"

Say **NO** to the question about creation of virtual environment. The Requirements.txt will be hidden.



3. Script files

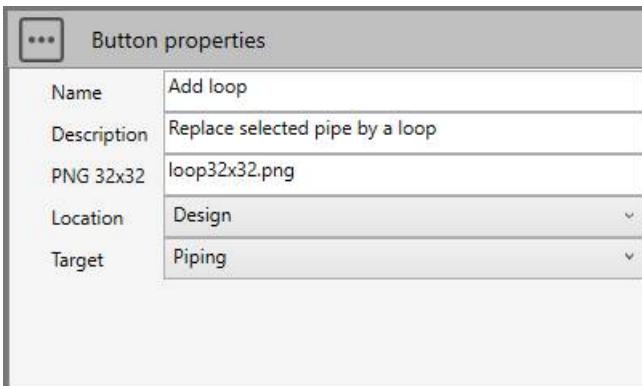
This kind of script needs some **images** in order to illustrate the **button** and the **window**.



Press the “three points” button and select “Import image”. You need to create a 32x32 transparent PNG for the icon of the button and an illustration of the expansion loop for the window.

4. Script properties

In this kind of script, user has to define the **button** :



The location will be **Design** (Piping ribbon) so that the button appears at the end of the ribbon with its name, icon and description :



The target is for piping system.

5. main.py

Select the file.

Copy/paste this code in the Editor :

```
#####
# CWANTIC LOOP GUI EXAMPLE #
#####

import os
from System.Windows.Media.Media3D import Vector3D
from Cwantic.MetaPiping.Core import RemoveElementCommand, AddNodeCommand, DrawPipingCommand, InsertBendCommand

# Inspect selection
n = len(design.selectedList)
res = "Select a pipe !"

if n==1:
    # Check the type of selected element
    if design.isType(design.selectedList[0], "Pipe"):
        pipe = design.selectedList[0]
        p1 = pipe.Node1.Coor
        p2 = pipe.Node2.Coor

    # Get the current model
    model = design.getMetal()
```

```

# Get the scene vertical vector (+Z or +Y)
verticalvec = design.getVerticalVector()

# Get the current piping values (section, material, radius,...)
currentValues = design.getCurrentSpecValues()

# Memorize current radius
currentRadius = currentValues.MKS_BendRadius

# Get the pipe direction vector
vec1 = Vector3D(pipe.DL.X, pipe.DL.Y, pipe.DL.Z)

# Get the script directory
directory = design.getScriptDirectory()

# Compute the cross product to determine the Loop direction
dir = Vector3D.CrossProduct(vec1, verticalvec)
dir.Normalize()

# Create a new USER command : cmd
cmd = design.createCommand("AddLoop")

# 1 : Remove the selected pipe

# 1.1 : Create params for command "RemoveElementCommand" (see Help)
params = []
params.append(design.selectedList)

# 1.2 : Add sub command to user command cmd
valid = cmd.addSubCommand("RemoveElementCommand", params)

if valid:
    # Get variable values - create a window
    window = design.createVariableWindow()
    window.AddComment("Fill the variables and click on OK. The loop will be perpendicular to the pipe. L can be positive or negative. [")
    window.AddValue("L", "L =", 1)
    window.AddValue("R", "R =", currentRadius)
    window.AddImage(os.path.join(directory, "illustration256x165.png"))
    if window.ShowDialog():
        # retrieve the size and radius from the window
        size = window.GetValue("L")
        currentValues.MKS_BendRadius = window.GetValue("R")

    # TIP : Create 2 new nodes (N3 and N4) with "AddNodeCommand"
    node1Cmd = AddNodeCommand(model, p1.X + size*dir.X, p1.Y + size*dir.Y, p1.Z + size*dir.Z, "", False, currentValues)
    N3 = node1Cmd.Node

    node2Cmd = AddNodeCommand(model, p2.X + size*dir.X, p2.Y + size*dir.Y, p2.Z + size*dir.Z, "", False, currentValues)
    N4 = node2Cmd.Node

    # Create node sub commands
    cmd.addSubCommand("AddNodeCommand", [N3])
    cmd.addSubCommand("AddNodeCommand", [N4])

    # Create new pipe perpendicular to selected pipe from node1

    # 2.1 : Create params for command "DrawPipingCommand" (see Help)
    params = []
    params.append(pipe.Node1)
    params.append(N3)
    params.append(size*dir.X)
    params.append(size*dir.Y)
    params.append(size*dir.Z)
    params.append(0.0)
    params.append(0.0)
    params.append(0.0)
    params.append(currentValues)

    # 2.2 : Add sub command
    valid = cmd.addSubCommand("DrawPipingCommand", params)

if valid:
    # 3 : Create new pipe parallel to selected pipe

    # 3.1 : Create params for command "DrawPipingCommand" (see Help)
    params = []

```

```

    params.append(N3)
    params.append(N4)
    params.append(vec1.X)
    params.append(vec1.Y)
    params.append(vec1.Z)
    params.append(0.0)
    params.append(0.0)
    params.append(0.0)
    params.append(currentValues)

    # 3.2 : Add sub command
    valid = cmd.addSubCommand("DrawPipingCommand", params)

    if valid:
        # 4 : Create new pipe to close the Loop

        # 4.1 : Create params for command "DrawPipingCommand" (see Help)
        params = []
        params.append(N4)
        params.append(pipe.Node2)
        params.append(-size*dir.X)
        params.append(-size*dir.Y)
        params.append(-size*dir.Z)
        params.append(0.0)
        params.append(0.0)
        params.append(0.0)
        params.append(currentValues)

        # 4.2 : Add sub command
        valid = cmd.addSubCommand("DrawPipingCommand", params)

        if valid:
            # 5 : Insert a bend at the last node

            # 5.1 Create params for command "InsertBendCommand"
            params = []
            params.append(pipe.Node2)
            params.append(currentValues)

            # 5.2 : Add sub command
            valid = cmd.addSubCommand("InsertBendCommand", params)

    # Execute command
    if valid:
        design.executeCommand(cmd)
        res = ""
    else:
        res = "Incorrect params"
    # Restore the radius
    currentValues.MKS_BendRadius = currentRadius
else:
    res = "The selected element is not a pipe"

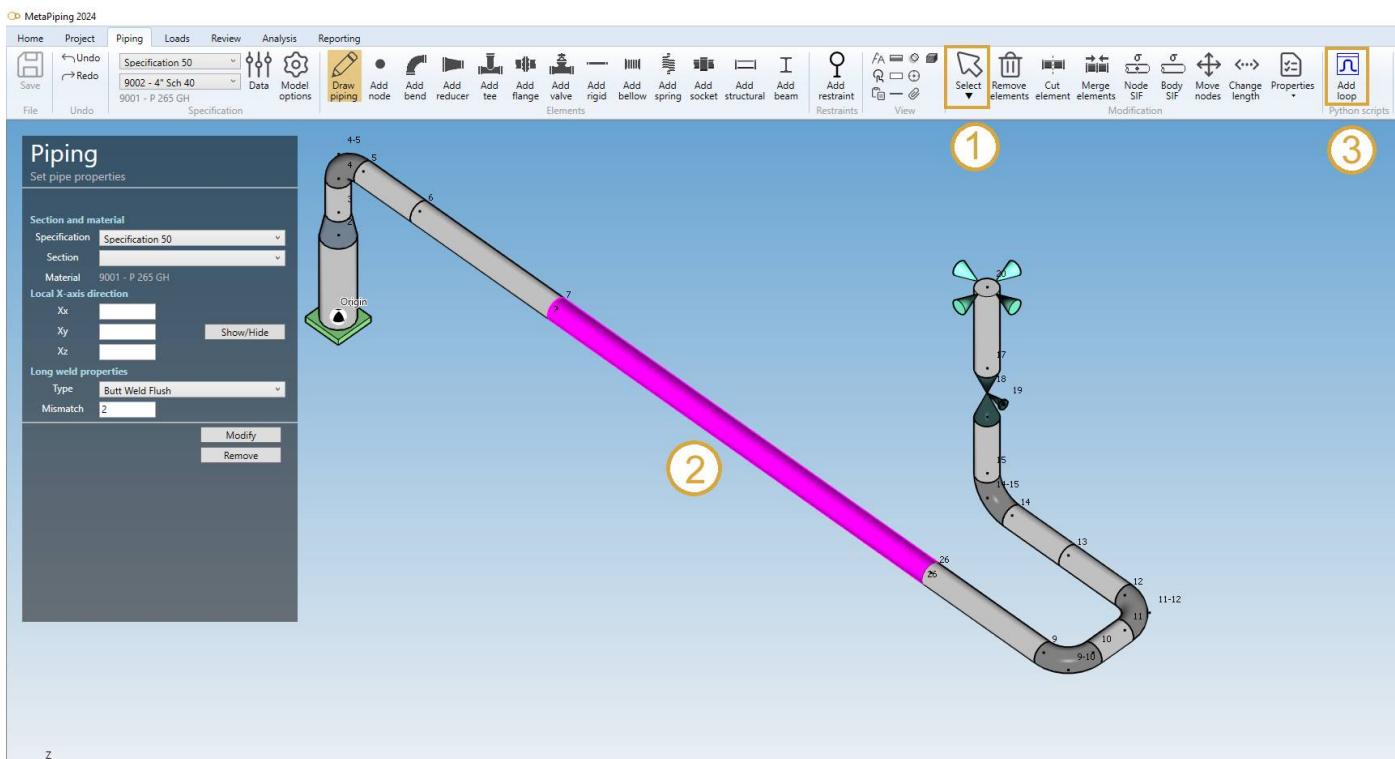
# Show a message (if res != "")
design.result = res

```

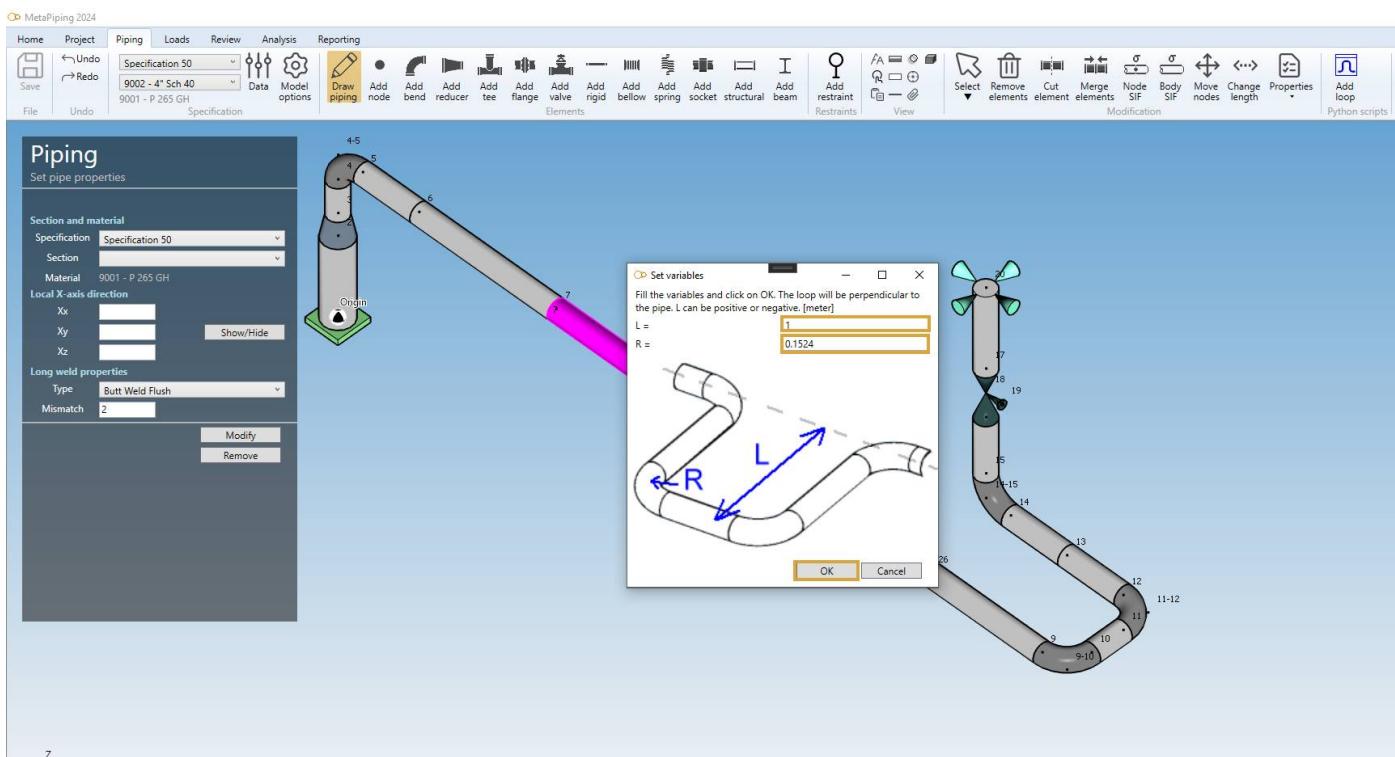
Save it !

6. Result

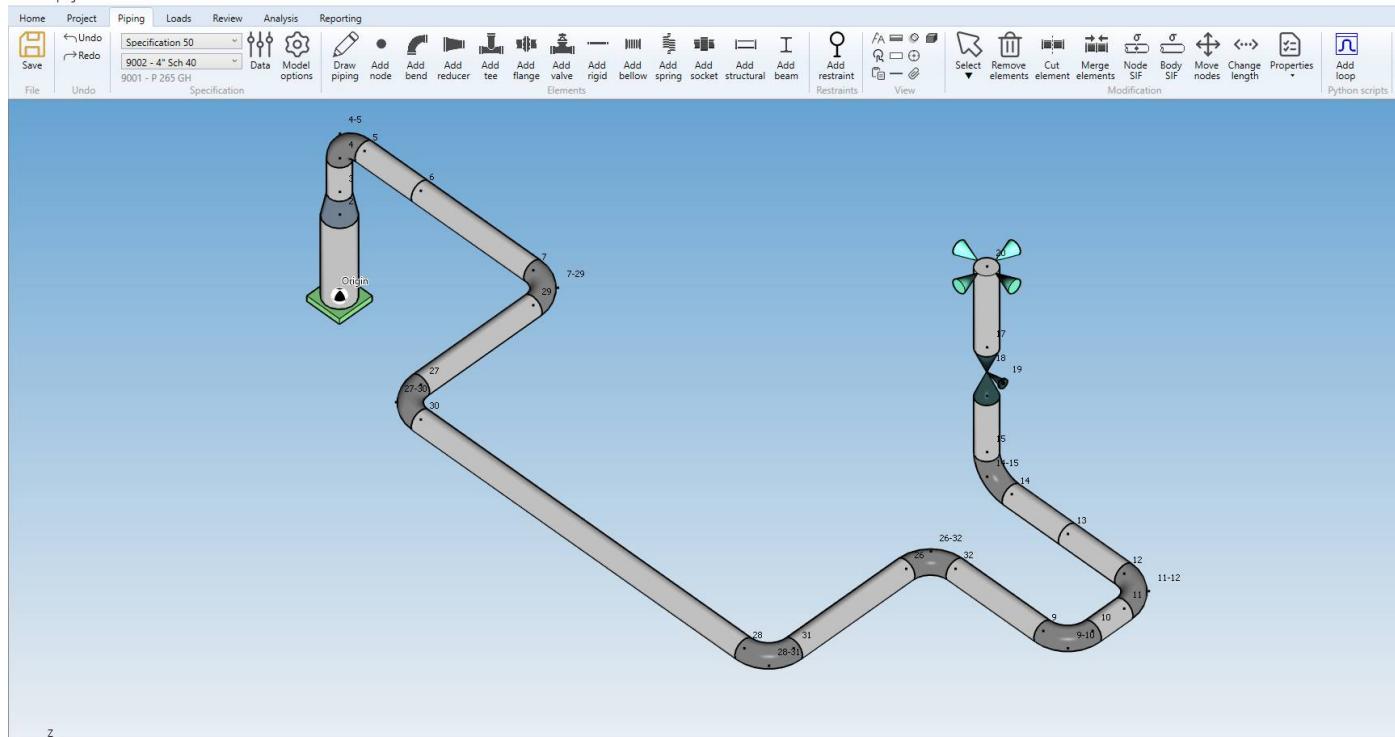
In **Piping screen**, select a pipe (1 & 2) that you want to replace by an expansion loop :



Click on the new button **Add loop** (3).



Complete **L**, **R** and press **OK**.



The expansion loop appears !

You can undo/redo your COMMAND !

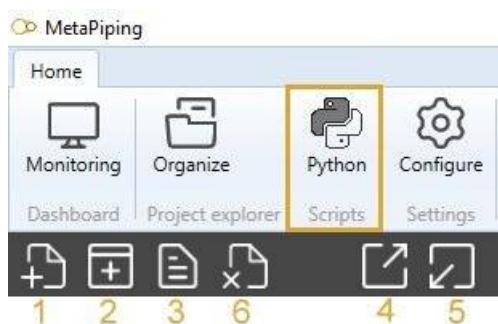
Sample 9 : Measure the distance between 2 nodes by message

1. Goal

Create a script that will (after pressing a **button** in the ribbon) show by message the distance between 2 selected nodes.

2. Script definition

From the Home/Python, click on **button 2** (Add 3D script) :



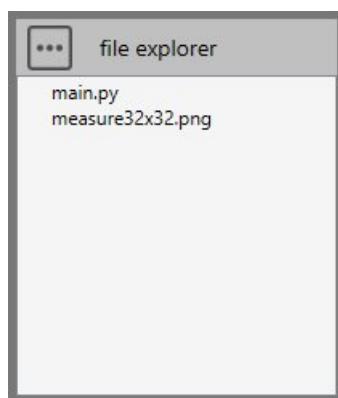
Give it the name "Distance between 2 nodes by message"

Say **NO** to the question about creation of virtual environment. The Requirements.txt will be hidden.



3. Script files

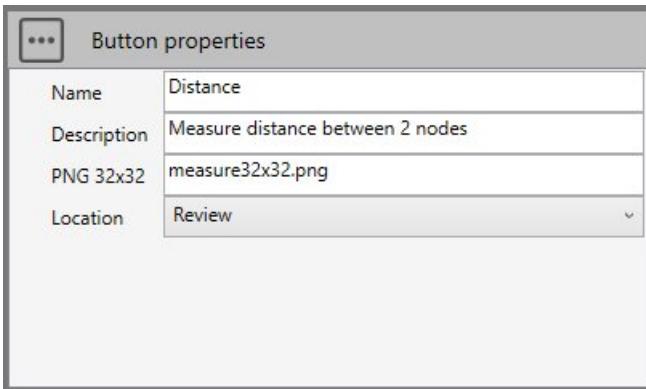
This kind of script needs an **image** in order to illustrate the **button**.



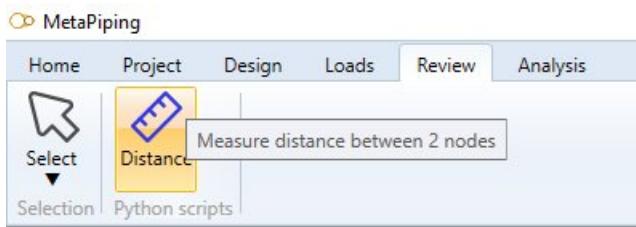
Press the "three points" button and select "Import image". You need to create a 32x32 transparent PNG for the icon of the button.

4. Script properties

In this kind of script, user has to define the **button** :



The location will be **Review** so that the button appears at the end of the ribbon with its name, icon and description :



5. main.py

Select the file.

Copy/paste this code in the Editor :

```
from math import *

# Inspect selection
n = len(design.selectedList)
res = "Select 2 nodes !"

if n==2:
    # Check the type of selected
    if design.isType(design.selectedList[0], "Node") and design.isType(design.selectedList[1], "Node"):
        node1 = design.selectedList[0]
        node2 = design.selectedList[1]
        dx = node2.Coor.X - node1.Coor.X
        dy = node2.Coor.Y - node1.Coor.Y
        dz = node2.Coor.Z - node1.Coor.Z
        d = sqrt(dx*dx + dy*dy + dz*dz)

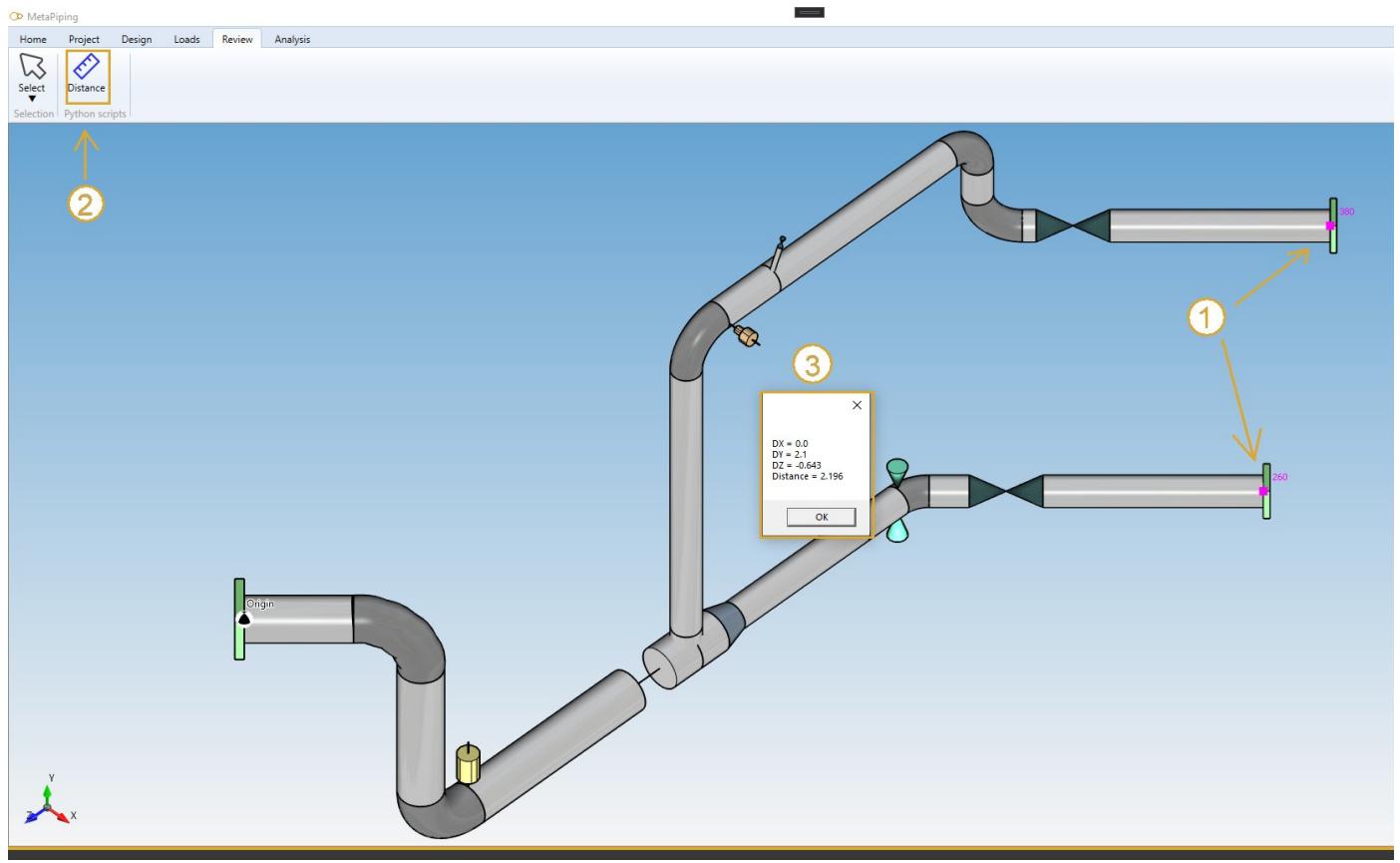
        res = "DX = " + str(round(dx, 3)) + "\n" + "DY = " + str(round(dy, 3)) + "\n" + "DZ = " + str(round(dz, 3)) + "\n" + "Distance = " + str(round(d, 3))

# Show a message (if res != "")
design.result = res
```

Save it !

6. Result

In **Review mode**, select 2 nodes and press the **button** :



A message shows the distance !

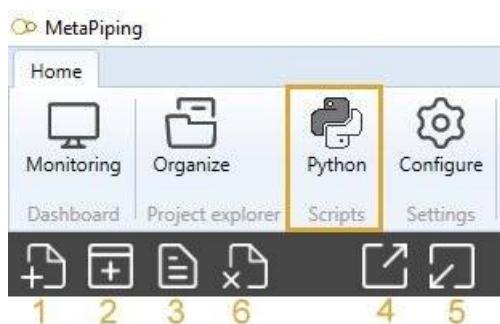
Sample 10 : Show the distance between 2 nodes in a frame

1. Goal

Create a script that will (after pressing a **button** in the ribbon) show in left frame the distance between 2 selected nodes.

2. Script definition

From the Home/Python, click on **button 2** (Add 3D script) :



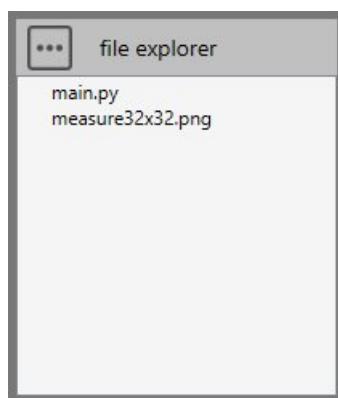
Give it the name "Show distance between 2 nodes"

Say **NO** to the question about creation of virtual environment. The Requirements.txt will be hidden.



3. Script files

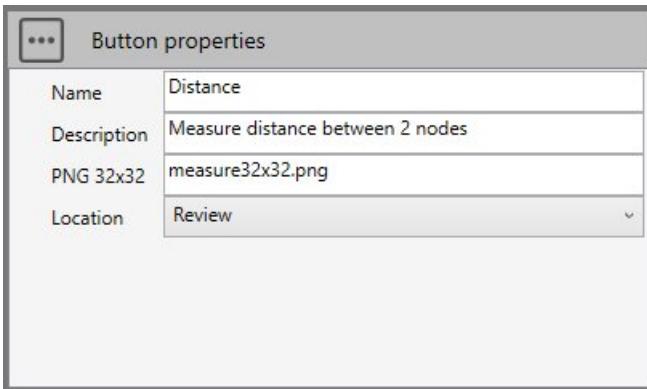
This kind of script needs an **image** in order to illustrate the **button**.



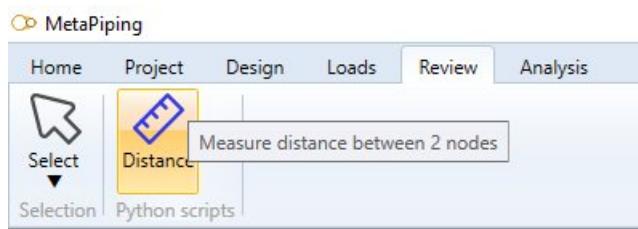
Press the "three points" button and select "Import image". You need to create a 32x32 transparent PNG for the icon of the button.

4. Script properties

In this kind of script, user has to define the **button** :



The location will be **Review** so that the button appears at the end of the ribbon with its name, icon and description :



5. main.py

Select the file.

Copy/paste this code in the Editor :

```
from math import *

# Inspect selection
n = len(design.selectedList)
res = "Select 2 nodes !"

if n==2:
    # Check the type of selected
    if design.isType(design.selectedList[0], "Node") and design.isType(design.selectedList[1], "Node"):
        node1 = design.selectedList[0]
        node2 = design.selectedList[1]
        dx = node2.Coor.X - node1.Coor.X
        dy = node2.Coor.Y - node1.Coor.Y
        dz = node2.Coor.Z - node1.Coor.Z
        d = sqrt(dx*dx + dy*dy + dz*dz)

        design.addText("Distances", True, 255, 255, 255)
        design.addSeparator()
        design.addText(" ")
        design.addText("DX = " + str(round(dx, 3)), False, 255, 100, 100)
        design.addText("DY = " + str(round(dy, 3)), False, 100, 255, 100)
        design.addText("DZ = " + str(round(dz, 3)), False, 100, 100, 255)
        design.addText(" ")
        design.addText("Distance = " + str(round(d, 3)) + " m", False, 255, 255, 255)

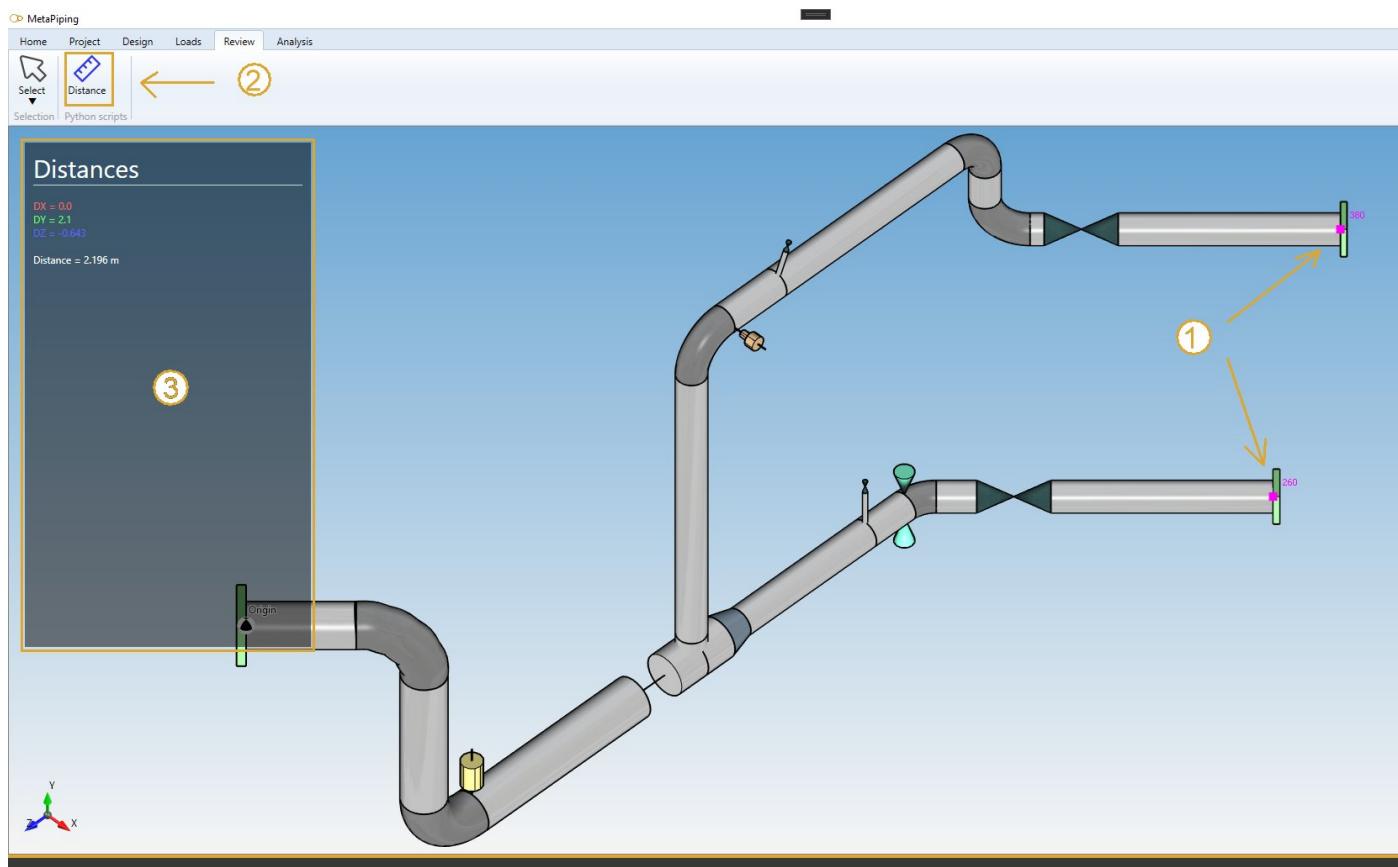
        res = ""

# Show a message (if res != "")
design.result = res
```

Save it !

6. Result

In **Review mode**, select 2 nodes and press the **button** :



A left frame shows the distance !

Sample 11 : Show informations about a study

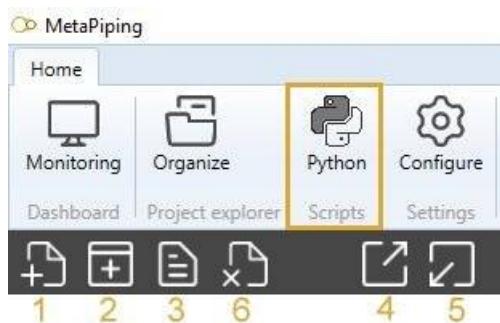
1. Goal

Create a script that will show informations about the Metal and the results of the current selected study in **Explorer**.

In this example, we will just access the **Metal** file and retrieve the number of nodes then access the **solution** and retrieve the **maxStressRatio** value.

2. Script definition

From the Home/Python, click on **button 3** (Add simple script) :



Give it the name "Show study infos"

3. main.py

Select the file.

Copy/paste this code in the Editor :

```
metal = study.getMetal()
if metal != None:
    study.showInformation('Model', '')
    study.showInformation('====', '')
    nb = metal.Nodes.Count
    study.showInformation('Nb noeuds', str(nb))

study.showInformation('', '')

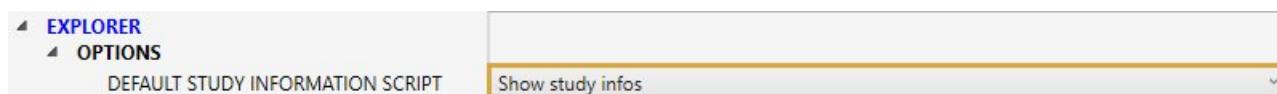
solution = study.getSolution()
if solution != None:
    study.showInformation('Solution', '')
    study.showInformation('=====', '')
    maxStressRatio = solution.getMaxStressRatio()
    study.showInformation('Max stress ratio', str(maxStressRatio))
```

Save it !

Click [here](#) to have more information about the object **study**.

4. Settings

Restart the application and go to the **settings**. In the tab **General**, change the "Default study information script" to "Show study infos" (our current script) :

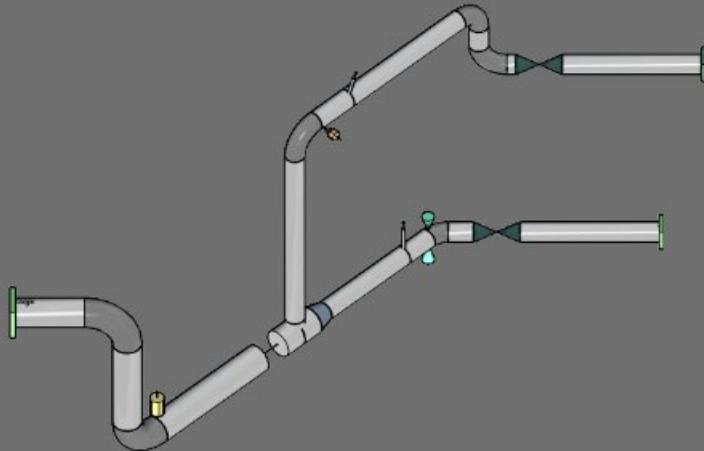


5. Result

In **Explorer of project**, select a study and visualize the informations :

Study

Name	RCCM2
Solver	Pipetress
Gravity	-Y
Units	Metric
Code	RCC-M Class 2 (French Nuclear Piping Code)
Edition	2018
Plant	ITER
Engineer	Tata

**Model**

====

Nb noeuds

41

Solution

=====

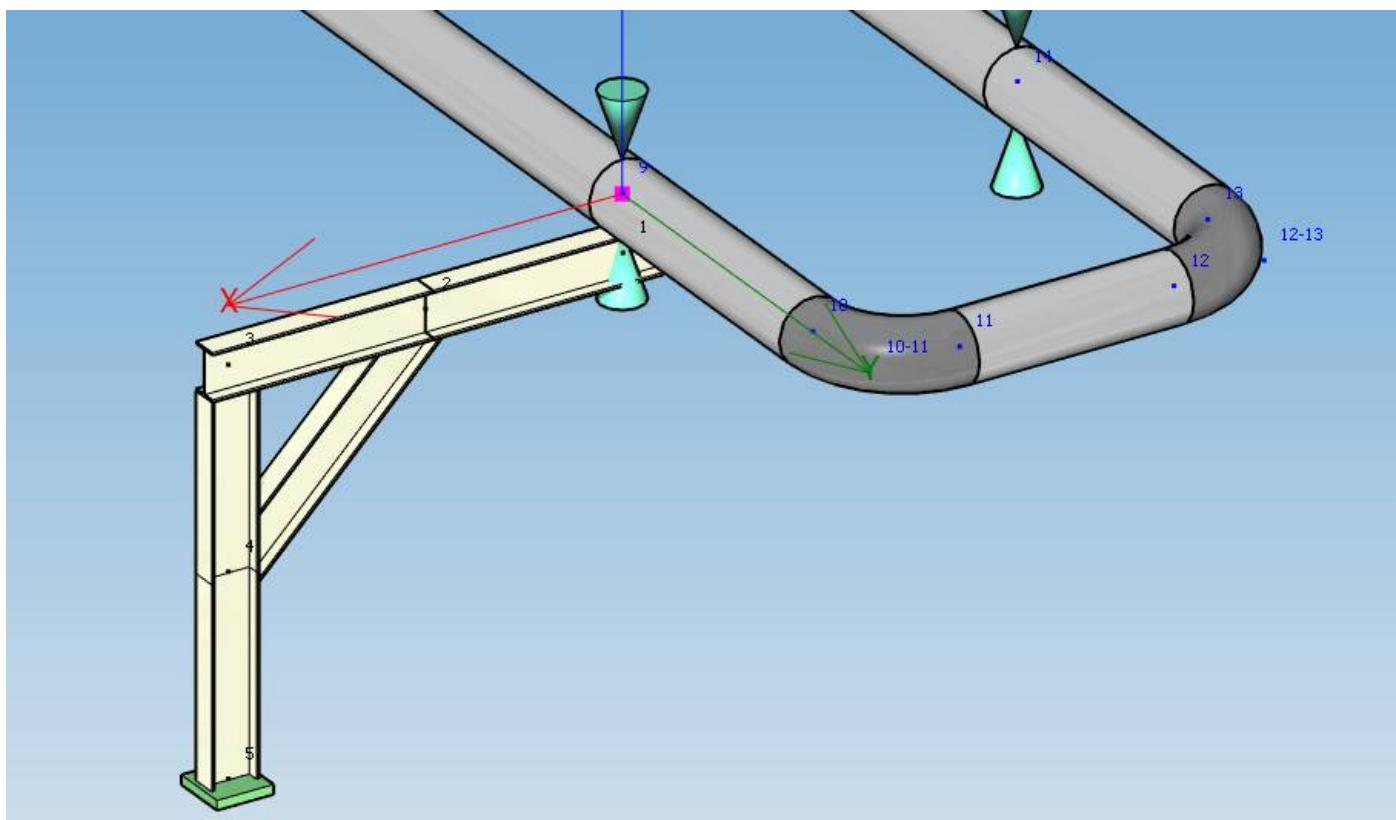
Max stress ratio

1.928

Sample 12 : Create a beam support of a piping on a node

1. Goal

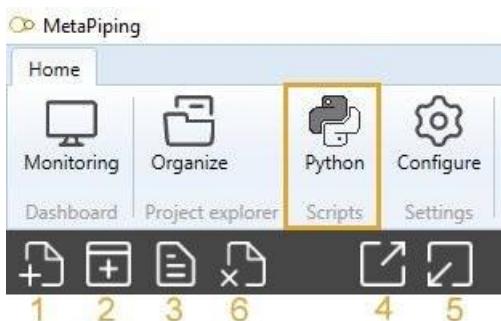
Create a script that will (after selecting a piping node with a restraint and pressing a **button** in the ribbon) show a window with variables and build several beams like in this picture :



Since the user will have to create a COMMAND, the application will manage the undo/redo operations by itself !

2. Script definition

From the Home/Python, click on **button 2** (Add 3D script) :

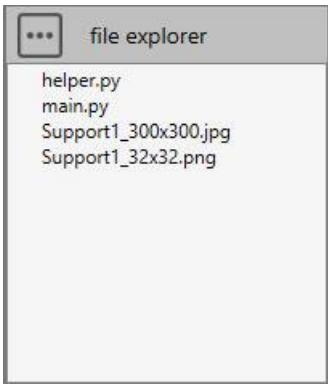


Give it the name "Support1"

Say **NO** to the question about creation of virtual environment. The Requirements.txt will be hidden.

3. Script files

This kind of script needs some **images** in order to illustrate the **button** and the **window**.



Press the "three points" button and select "Import image". You need to create a 32x32 transparent PNG for the icon of the button and an illustration of the support for the window.

Press the "three points" button and select "Add a python file". We will create a file for some helper functions.

3.1 helper.py

Select the file.

Copy/paste this code in the Editor :

```
#####
# Model helper functions #
#####

from Cwantic.MetaPiping.Core import BeamExtremity, LTBModel

# Method that check if a node contains a restraint
def getRestraint(node, model):
    for restraint in model.Restraints:
        if restraint.Node == node:
            return restraint
    return None

# Method that return the first pipe on node
def getFirstPipe(design, node):
    for element in node.Elements:
        if design.isType(element, "Pipe"):
            return element
    return None

# Initialize a new CurrentValues for beam
def initializeCurrentValues(currentValues):
    other = currentValues.Clone()
    other.BeamIndexExtremity1 = 0
    other.BeamIndexExtremity2 = 0
    other.BeamOffset1 = 0
    other.BeamOffset2 = 0
    other.BeamJoint1 = None
    other.BeamJoint2 = None
    other.BeamLTBModel = LTBModel.BothHingedUniform
    other.BeamExtremity1 = BeamExtremity(0)    # Tip : cannot set BeamExtremity.None so use 0 instead
    other.BeamExtremity2 = BeamExtremity(0)    # Tip : cannot set BeamExtremity.None so use 0 instead
    other.CheckModel = False
    return other
```

Save it !

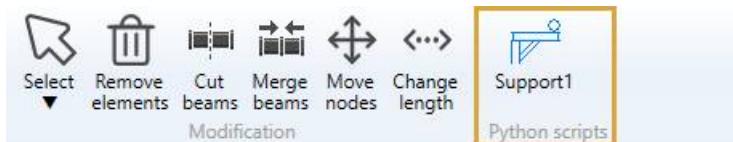
4. Script properties

In this kind of script, user has to define the **button** :

Button properties	
Name	Support1
Description	Create a support under a restrained node
PNG 32x32	Support1_32x32.png
Location	Design
Target	Structure

The **Target** is Structure!

The location will be **Design** so that the button appears at the end of the ribbon with its name, icon and description :



5. main.py

Select the file.

Copy/paste this code in the Editor :

```
#####
# Create support on selected node #
#####

import os
from System.Windows.Media.Media3D import Vector3D
from System.Collections.Generic import List
from helper import getRestraint, getFirstPipe, initializeCurrentValue
from Cwantic.MetaPiping.Core import Node, Element, AddNodeCommand, AddBeamCommand, MassModel, BeamExtremity, CoordinateSystem

# Default error message
message = "Select a node with restraint !"

# Inspect selection
n = len(design.selectedList)
if n==1:
    # Check the type of selected element
    if design.isType(design.selectedList[0], "Node"):
        node = design.selectedList[0]

    # Get the current metal
    model = design.getMetal()

    # Get the metal of the node (can be a LINKED metal so do not use getMetal)
    selectedModel = design.getMetalWithSelection(node)

    # Get the script directory
    directory = design.getScriptDirectory()

    # Check if a restraint exists on this node
    restraint = getRestraint(node, selectedModel)
    if restraint != None:
        # Get the first pipe on node
        pipe = getFirstPipe(design, node)
        if pipe != None:
            # Get the pipe ray
            ray = pipe.Section.Diameter / 2
            # Get the pipe direction
            pipevec = Vector3D(pipe.DL.X, pipe.DL.Y, pipe.DL.Z)
            pipevec.Normalize()
            # Get the model vertical vector (+Z or +Y)
            verticalvec = design.getVerticalVector()
            # Get the horizontal vector perpendicular to pipe
            horizontalvec = Vector3D.CrossProduct(pipevec, verticalvec)
            horizontalvec.Normalize()
```

```

horizontalvec.normalize()

# Get the current beam default values (section, material, extremities,...)
currentValues = design.getCurrentSpecValues()

# Add the half distance of the current beam section to ray
dist = ray + currentValues.MKS_SpecBeamSection.H / 2

# Get node altitude
if verticalvec.Y == 1:
    altitude = node.Coor.Y
else:
    altitude = node.Coor.Z

# Initialize offset and anchor
offset = 0
anchor = True

# Get variable values - create a window
window = design.createVariableWindow()
window.AddComment("Fill the variables and click on OK. The support will be perpendicular to the pipe and tangent. L can be positive")
window.AddValue("L", "L =", 2)
window.AddValue("H", "H =", altitude)
window.AddValue("DL", "L1 =", 1)
window.AddValue("DH", "H1 =", 0.5)
window.AddList("OFFSET", "Offset =", ["No offset", "Pipe ray", "0.5 m"], 1)
window.AddCheck("ANCHOR", "Anchor =", anchor)
window.AddImage(os.path.join(directory, "Support1_300x300.jpg"))
if window.ShowDialog():
    L = window.GetValue("L")
    H = window.GetValue("H")
    DL = window.GetValue("DL") * L / abs(L)
    DH = window.GetValue("DH")
    OFFSET_ID = window.GetValue("OFFSET")
    if OFFSET_ID == 1:
        offset = ray
    else:
        if OFFSET_ID == 2:
            offset = 0.5
    ANCHOR_ID = window.GetValue("ANCHOR")
    anchor = ANCHOR_ID == 1

    # Create a new USER command : cmd
    cmd = design.createCommand("AddSupport1")

    # Create nodes - we need temporary commands to get new nodes (1 to 5) to pass to node, beam and anchor commands
    node1Cmd = AddNodeCommand(model, node.Coor.X - dist*verticalvec.X, node.Coor.Y - dist*verticalvec.Y, node.Coor.Z - dist*verticalvec.Z)
    node1 = node1Cmd.Node

    node2Cmd = AddNodeCommand(model, node1.Coor.X + (L-DL)*horizontalvec.X, node1.Coor.Y + (L-DL)*horizontalvec.Y, node1.Coor.Z)
    node2 = node2Cmd.Node

    node3Cmd = AddNodeCommand(model, node1.Coor.X + L*horizontalvec.X, node1.Coor.Y + L*horizontalvec.Y, node1.Coor.Z + L*horizontalvec.Z)
    node3 = node3Cmd.Node

    node4Cmd = AddNodeCommand(model, node3.Coor.X - DH*verticalvec.X, node3.Coor.Y - DH*verticalvec.Y, node3.Coor.Z - DH*verticalvec.Z)
    node4 = node4Cmd.Node

    node5Cmd = AddNodeCommand(model, node3.Coor.X - H*verticalvec.X, node3.Coor.Y - H*verticalvec.Y, node3.Coor.Z - H*verticalvec.Z)
    node5 = node5Cmd.Node

    # Create node sub commands
    cmd.addSubCommand("AddNodeCommand", [node1])
    cmd.addSubCommand("AddNodeCommand", [node2])
    cmd.addSubCommand("AddNodeCommand", [node3])
    cmd.addSubCommand("AddNodeCommand", [node4])
    cmd.addSubCommand("AddNodeCommand", [node5])

    # Create beam sub commands
    # Tip : each command must have its own currentValues, initializeCurrentValues create a clone and initialize values
    currentValues = initializeCurrentValues(currentValues)
    currentValues.BeamOffset1 = offset
    cmd.addSubCommand("AddBeamCommand", [node1, node2, node2.Coor.X-node1.Coor.X, node2.Coor.Y-node1.Coor.Y, node2.Coor.Z-node1.Coor.Z])

    currentValues = initializeCurrentValues(currentValues)
    beamCmd = AddBeamCommand(model, node2, node3, node3.Coor.X-node2.Coor.X, node3.Coor.Y-node2.Coor.Y, node3.Coor.Z-node2.Coor.Z)
    beam1 = beamCmd.Beam

```

```

cmd.addSubCommand("AddBeamCommand", [beam1])

currentValues = initializeCurrentValues(currentValues)
beamCmd2 = AddBeamCommand(model, node3, node4, node4.Coor.X-node3.Coor.X, node4.Coor.Y-node3.Coor.Y, node4.Coor.Z-node3.Coo
beam2 = beamCmd2.Beam
cmd.addSubCommand("AddBeamCommand", [beam2])

# Set connectivity between Beam1 and Beam2
beam1.Extremity2 = BeamExtremity.Back
beam1.Extremity2Beam = beam2
beam2.Extremity1 = BeamExtremity.Front
beam2.Extremity1Beam = beam1

currentValues = initializeCurrentValues(currentValues)
beamCmd3 = AddBeamCommand(model, node4, node5, node5.Coor.X-node4.Coor.X, node5.Coor.Y-node4.Coor.Y, node5.Coor.Z-node4.Coo
beam3 = beamCmd3.Beam
cmd.addSubCommand("AddBeamCommand", [beam3])

currentValues = initializeCurrentValues(currentValues)
currentValues.BeamExtremity1 = BeamExtremity.Front
currentValues.BeamIndexExtremity1 = 1
currentValues.BeamExtremity2 = BeamExtremity.Front
currentValues.BeamIndexExtremity2 = 1
cmd.addSubCommand("AddBeamCommand", [node4, node2, node2.Coor.X-node4.Coor.X, node2.Coor.Y-node4.Coor.Y, node2.Coor.Z-node4

# Create anchor command - need a List of node as first parameter
if anchor:
    Nodes = List[Node]()
    Nodes.Add(node5)

    Attached = List[Element]()
    Attached.Add(beam3)

    TSTIFF = model.ANCH_TSTIFF
    RSTIFF = model.ANCH_RSTIFF
    cmd.addSubCommand("CreateAnchorCommand", [Nodes, CoordinateSystem.LocalToConnectedElement, Attached, "", TSTIFF, TSTIFF

design.executeCommand(cmd)
message = ""
else:
    message = "Cancel by user !"
else:
    message = "No pipe on this node !"
else:
    message = "No restraint on this node !"

design.result = message

```

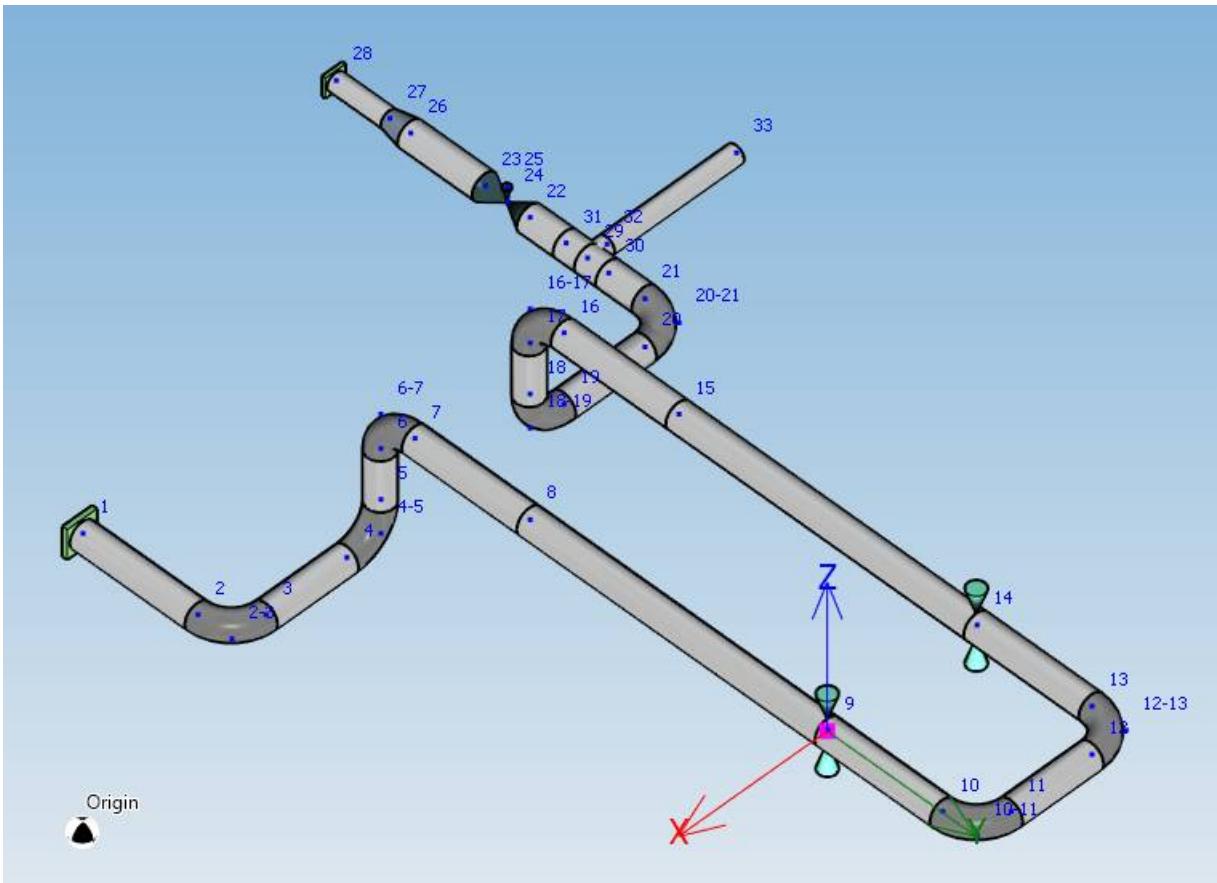
Save it !

6. Result

Create a **piping study** with some restraints on node.

Create a **structure study** with a link to the previous study.

Select a restrained node in the link model and press the **Support1** button.



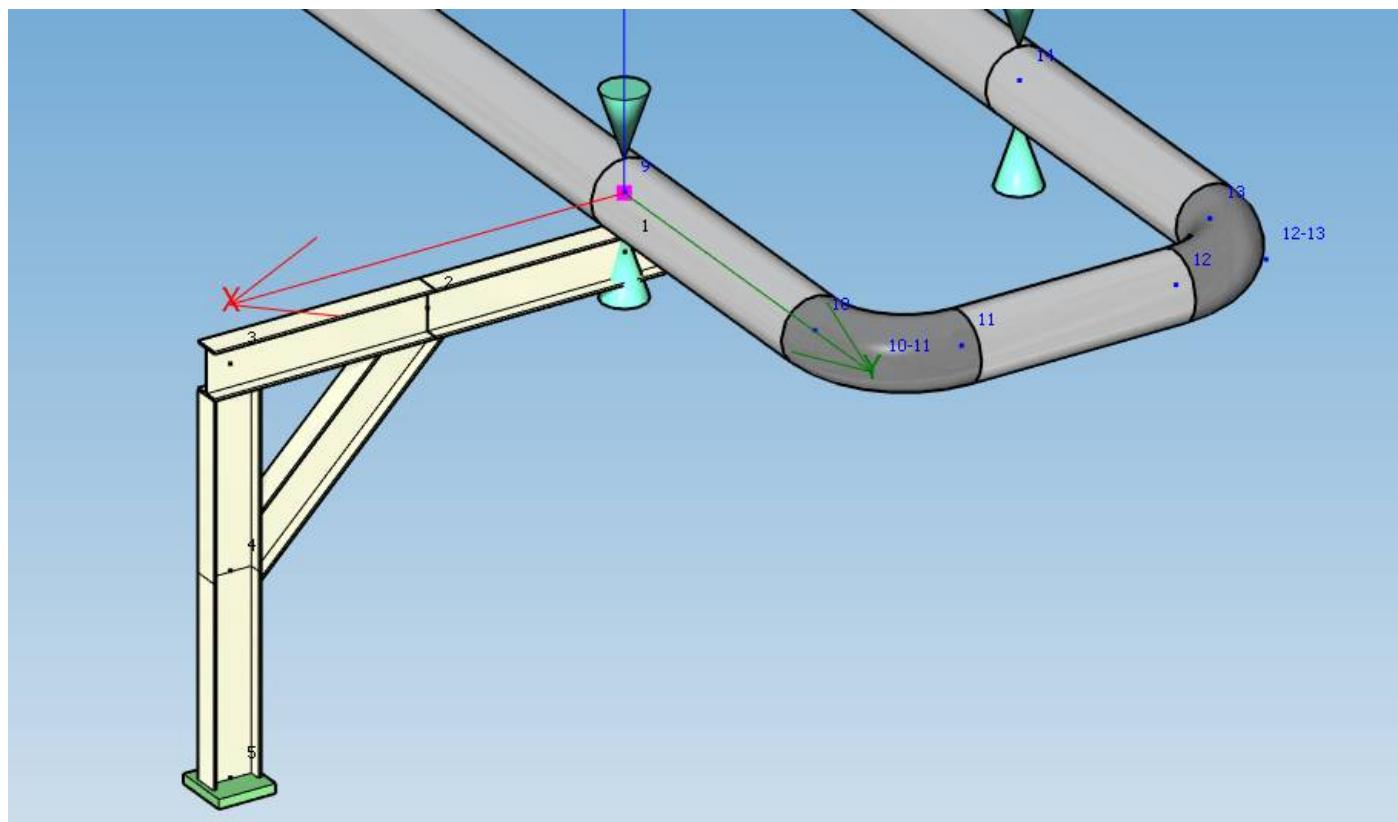
This window will appear :

Set variables X

Fill the variables and click on OK. The support will be perpendicular to the pipe and tangent. L can be positive or negative. [meter]

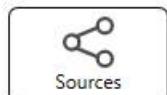
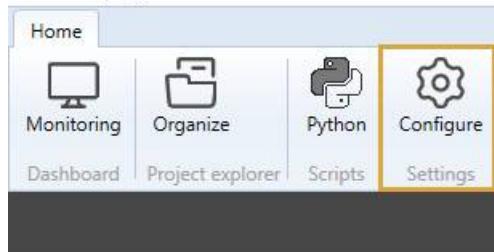
L =	<input type="text" value="2"/>
H =	<input type="text" value="2.8"/>
L1 =	<input type="text" value="1"/>
H1 =	<input type="text" value="0.5"/>
Offset =	<input type="button" value="Pipe ray"/>
Anchor =	<input checked="" type="checkbox"/>

Change values and click OK :



Settings

MetaPiping



Sources



General



Databases



Specifications



Reporting

Click on **Configure** button to access the settings panel.

TABLE OF CONTENTS

- [Sources](#)
- [General](#)
- [Databases](#)
- [Specifications](#)
- [Reporting](#)



Sources

NAME	VALUE
PROJECTS SOURCES	
Directories	
Directory 1	C:\ProgramData\Cwantic\MetaPiping\Projects
Current working directory	
DATAS SOURCES	
Directories	
Directory 1	C:\ProgramData\Cwantic\MetaPiping\Datas
Current data directory	

1. Projects sources

Define the *local directory* or *server directory* where all projects will be saved. [See Project page for more information.](#)

After installation, the default directory will be:

```
C:\ProgramData\Cwantic\MetaPiping\Projects
```

You can add supplement directories by clicking the + button and remove others by clicking - button.

There must be at least one working directory.

This way you can organize yourself by splitting the projects into main categories and/or for sharing project between users.

2. Data sources

Define the *local directory* or *server directory* where all data will be saved.

The data directory is the directory where the **databases**, the python **scripts** and the **specifications** are saved.

```
(C:) > ProgramData > Cwantic > MetaPiping > Datas
```

Nom	Modifié le
Databases	26-06-23 13:22
Scripts	27-06-23 11:16
Specifications	22-05-23 14:16

[See Database page for more information.](#)

[See Python page for more information.](#)

[See Specification page for more information.](#)

After installation, the default directory will be:

```
C:\ProgramData\Cwantic\MetaPiping\Datas
```

You can add supplement directories by clicking the + button and remove others by clicking - button. You may not remove the first row (the default directory after installation).

There must be at least one data directory.

This way you can easily share your own data.

General

General

NAME	VALUE
ASTER	
PATH	C:\ProgramData\Cwantic\MetaPiping\Settings\Aster\v2021\bin\as_run.bat
WINDOWS SOLVER PATH	
LINUX SOLVER PATH	
OPTION	Windows
SOLVER	
EXPLORER	
OPTIONS	Show study infos
DEFAULT STUDY INFORMATION SCRIPT	
METAL	
NODE NAME	
NODE NAME START	1
NODE NAME STEP	1
METAPIPING	
OPTIONS	
USE IMPERIAL NOTATION	<input type="checkbox"/>
PIPESTRESS	
PATH	C:\Program Files (x86)\PepS 7.2\PS.420\PIPESTRESS64W.exe
SOLVER PATH	C:\Program Files (x86)\PepS 7.2\VersionFiles\PS.420\PIPESTRESS.chm
HELP PATH	
OPTIONS	
MAX FIELD LENGTH	15
ROTATION UNITS	Radians
GENERATE RESTRAINTS INLINE	<input type="checkbox"/>
GENERATE DLCS INLINE	<input type="checkbox"/>
GENERATE INDI INLINE	<input type="checkbox"/>
PROJECT	
OPTIONS	
TEXT EDITOR PATH	
REVIEW	
END CONDITIONS	
NODE SYMBOL SIZE	10
STRUCTURE	
OPTIONS	
MAX DISTANCE TO PIPE NODE [m]	0.5

Define all options proposed by all **PLUGINS**.

1. Code Aster

Code Aster is the main **SOLVER** of MetaPiping. It is developed and maintained by EDF (<https://code-aster.org>).

The Windows version is installed by MetaPiping but you can install the Linux version manually and specify the location on your computer.

In this case, you have to select **LINUX** in the option combobox.

2. Explorer

The project explorer lets you **calculate** a python script that explores the project/results and show information.

All projects share the same script that you can specify in this option.

[See Info page for more information.](#)

3. Metal

Metal is the name of the 3D model inside MetaPiping and also the file extension of the saved file.

You can specify here 2 values (integers):

- The name of the very first node

- The increment for the next node name

In the example, the nodes will start with name "10" then "20", "30", "40"...

4. MetaPiping

The application lets the user work with English units. In this case, it is also possible to specify to work with **imperial** notation.

Example: 4"1/8 for a diameter

5. PIPESTRESS

For the users of PIPESTRESS that have installed the plugin, they can specify the path of the SOLVER (*PIPESTRESSW.exe* or *PIPESTRESS64W.exe*) and the HELP file (*PIPESTRESS.chm*).

You can also specify:

- Max field length
- Rotation units
- Inline of the restraints, DLCS and INDI

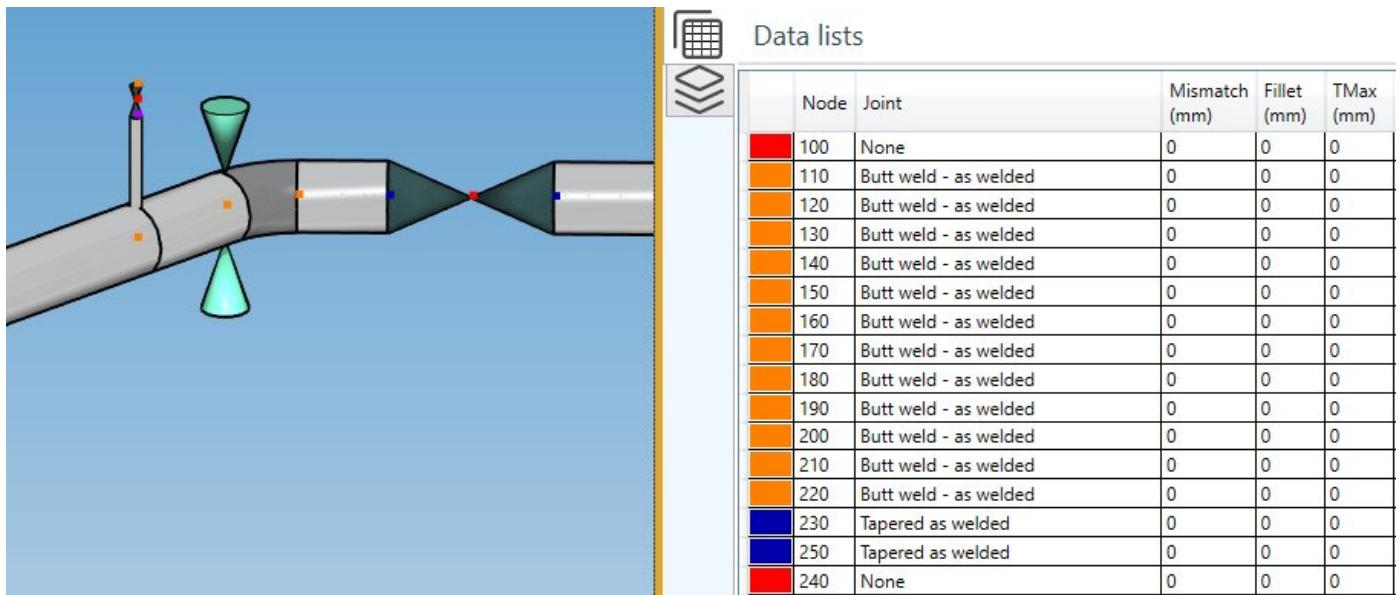
6. Project

You can specify the path of your favorite text editor for text report editing.

If blank, MetaPiping uses **Windows NotePad**.

7. Review

You can define the size in pixel of the point for the **End conditions** screen (default = 10 pixels) :



8. Structure

You can specify the maximum distance (in meter) between a pipe node and a structure node during the **Node link** operation. MetaStructure will search structure nodes that are distant less than this value from the selected piping node.



Databases

The screenshot shows the 'Databases' tab of the MetaPiping application. On the left, there's a sidebar with icons for Sources, General, Databases (which is highlighted in orange), and Specifications. The main area has tabs for Materials, Piping, Components, Supports, Spectra, Beam sections, Fasteners, and Bolts. The 'Materials' tab is active, showing a tree view with 'Materials' expanded, revealing 'Bolting', 'Piping' (with sub-options for ASME II Part D 2008, MAT SEB TEST, and RCC-M ZI 2013), 'Structure', and 'Welding' (with sub-option BS_Welding). There are also icons for creating new databases and deleting existing ones.

MetaPiping lets you create databases of common data as materials, components, spectra, ratings, beam sections, fasteners or bolts...

The databases are **JSON** files stored in the directory *Databases* in the **Data** source.

[See Sources page for more information](#)

TABLE OF CONTENTS

- [Materials](#)
- [Piping](#)
- [Components](#)
- [Supports](#)
- [Spectra](#)
- [Sections](#)
- [Fasteners](#)
- [Bolts](#)
- [Ratings](#)

Materials

In this section, libraries of materials can be created.

A library is a **JSON** file with a name, a target, units, a reference temperature and a list of materials.

Target can be :

- Piping
- Structure
- Bolting
- Welding

Click the **+ Folder** button to add a new folder, click the **+ File** to add a new library or click the **-** button to remove the selected folder or an **unused** library (not defined in a specification).

You can also import materials from BEAMSTRESS.

You can also drag a file/folder on another folder.

The screenshot shows a software interface for managing material libraries. At the top, there are tabs for Materials, Piping, Components, Supports, Spectra, Beam sections, Fasteners, and Bolts. Below the tabs is a toolbar with icons for adding (+), deleting (-), saving (floppy disk), and other operations. A sidebar on the left contains a tree view of categories: Materials, Bolting, Piping, Structure, and Welding. Under Piping, 'ASME II Part D 2008' is selected and highlighted with a yellow bar. This selection is reflected in the configuration panel on the right, which includes fields for Standard (set to ASME II Part D 2008), Target (set to Piping), Units (set to Metric), Ref. temperature (set to 21.1), and a Materials list. The Materials list shows 'ASME II Part D 2008' and has a small '+' icon next to it.

Example : "ASME II Part D 2008"

To know the UNIT of a property, just let the mouse over the property name.
For example : °C for the Ref. temperature

The screenshot shows a detailed view of a material's properties. On the left, a tree view shows 'Materials' expanded, with 'Name' selected. Under 'Name', there are fields for 'Description' (Carbon Steel), 'Type' (Mean coefficient calculated by linear interpolation of submitted values), 'Thermal expansion option' (Max. temperature), 'Max. temperature' (371), 'Density' (7850), and 'Poisson' (0.3). Below this, 'Properties' is expanded, showing a table with columns for Temperature (EH, EX, SH, SY, SU, SM (C1), CR, GH (EDF), CO (C1), DI (C1)) and rows for specific temperatures (21, 65, 93, 149, 204, 260, 316, 343, 371). Each row has a '+' icon to add more data and a '-' icon to remove it.

Click the **+** button (next to Materials) to add a new material or click the **-** button (next to Name) to remove one.

Define the description, the type, the thermal expansion option, the temperature max, the density and the Poisson's ratio.

Click the **+** button (next to Properties) to add a new row of properties for a **specific temperature** or click the **-** button (end of row) to remove one.

Property	Description	Unit Metric	Unit USA
Max. temperature	Temperature max	°C	°F
Density	Density	kg/m ³	lb/ft ³
Poisson	Poisson's ratio	-	-
EH	Modulus of Elasticity	kN/mm ²	10 ⁶ .psi
EX	Thermal Expansion	10 ⁻⁶ .mm/mm/°C	10 ⁻⁶ .in/in/°F
SH	Non-Class 1 Allowable Stress	N/mm ²	ksi
SY	Yield Stress	N/mm ²	ksi
SU	Ultimate Tensile Stress	N/mm ²	ksi
SM	Class 1 Allowable Stress	N/mm ²	ksi
CR	Creep	N/mm ²	ksi
GH	Shear Modulus	kN/mm ²	10 ⁶ .psi
CO	Class 1 Thermal Conductivity	kJ/hr/m/°C	btu/hr/ft/°F
DI	Class 1 Thermal Diffusivity	mm ² /s	ft ² /hr
E2	Modulus of Elasticity 2	kN/mm ²	10 ⁶ .psi

See [Units](#) for more informations.

WARNING

ATTENTION, temperatures must be entered in ascending order!

HDPE MATERIALS :

Name	HDPE EDF1
Description	HDPE EDF1
Type	HDPE
Max. temperature	
Density	
Poisson	0.45
Thermal expansion coef.	
Spec	EDF1 : D305914006648 [E] EDF guide

The HDPE materials have simpler properties :

Property	Description	Unit Metric	Unit USA
Max. temperature	Temperature max	°C	°F
Density	Density	kg/m ³	lb/ft ³
Poisson	Poisson's ratio	-	-
Thermal expansion coef.	Thermal Expansion	10 ⁻⁶ .mm/mm/°C	10 ⁻⁶ .in/in/°F
Spec	Spec value (see below)	-	-

Spec values :

- ISO : EN 15494:2015 with coefficient = 1.25
- ASTM : ASTM with factor = 0.63
- ASME: ASTM with factor = 0.5
- EDF1 : D305914006648 [E] EDF guide
- EDF2 : D305921021240 [A] EDF guide

All other properties for the calculation are automatically set.

Piping

In this section, piping libraries can be created.

A library is a **JSON** file with a name, units and a list of pipe sizes, bend, reducer, welding tee and flange definitions.

Click the **+ Folder** button to add a new folder, click the **+ File** to add a new library or click the **-** button to remove a folder or an **unused** library (not defined in a specification).

You can also drag a file/folder on another folder.

The screenshot shows the Piping library configuration interface. At the top, there are tabs for Materials, Piping, Components, Supports, Spectra, Beam sections, Fasteners, and Bolts. Below the tabs is a toolbar with icons for adding (+), deleting (-), and saving (disk).

The main area is divided into two columns. The left column contains a tree view of piping components:

- Pipings** (selected)
- ANSI B16.9
- My Piping Library** (highlighted in yellow)
- Group**
 - PIPE SEB TEST
 - test

The right column contains configuration tables for various piping components:

- Standard Units**: Set to My Piping Library, USA.
- Pipe sizes**:
 - 4"** (selected)

Schedule	Outside diameter	Thickness	Linear mass
Sch 40	100	10	0
 - 2"**

Schedule	Outside diameter	Thickness	Linear mass
Sch 40	80	10	0
- Bends**:
 - Pipe size 4"**

Long radius	Small radius	Mass
0	0	0
- Reducers**:
 - 4"**

Reduce to 2"	Length	Mass
	0	0
- Welding tees**:
 - 4"**

Length	Branch size	Mass
0	4"	0
- Flanges**:
 - 4"**

Length	Mass
0	0

Example : "My Piping Library"

1. Pipe sizes

To add new pipe sizes, click on **+** button and give it a name (Ex : 4") :

The screenshot shows a table for defining pipe sizes. The first row is a header with columns: Schedule, Outside diameter, Thickness, and Linear mass. The 'Outside diameter' column is highlighted in yellow. The table lists several schedule options for 4-inch pipe:

Schedule	Outside diameter	Thickness	Linear mass
Sch 5S	114.3	2.11	5.84
Sch 10-10S	114.3	3.05	8.37
Sch 40-Std-40S	114.3	6.02	16.1
Sch 80-XS-80S	114.3	8.56	22.3
Sch 120	114.3	11.12	28.3
Sch 160	114.3	13.49	33.5
Sch XXS	114.3	17.12	41

Each size exists in different schedules. Click on **+** button to add schedules :

The screenshot shows a table for defining unit conversions for piping properties. The columns are: Property, Unit Metric, and Unit USA.

Property	Unit Metric	Unit USA
Schedule	-	-
Outside diameter	mm	in
Thickness	mm	in
Linear mass	kg/m	lb/ft

To know the UNIT of a property, just let the mouse over the property name.
For example : mm for the Outside diameter

WARNING

ATTENTION, the name of the pipe size is very important (Ex : 4"). It will be used as a **reference** for all other piping elements.

WARNING

ATTENTION, the sizes must be defined in an **ascending** order.

2. Bends

To add **bend** properties for each pipe size, click on + button and select a pipe size :

Bends			
Pipe size	Long radius	Small radius	Mass
1/2"	0.0191	0.0127	0
3/4"	0.0286	m	0
1"	0.0381	0.0254	0
1 1/4	0.0476	0.0318	0
1 1/2	0.0572	0.0381	0
2"	0.0762	0.0508	0
2 1/2	0.0953	0.0635	0
3"	0.1143	0.0762	0
3 1/2	0.1334	0.0889	0
4"	0.1524	0.1016	0

Standard radius and mass can be entered :

Property	Unit Metric	Unit USA
Long radius	m	ft
Small radius	m	ft
Mass	ton	kips

WARNING

ATTENTION, for a better readability, we recommend to enter the sizes in **ascending** order.

3. Reducers

To add **reducer** properties for each pipe size, click on + button and select a pipe size :

Reducers			
3/4"	Length	Mass	+
3/4"	0.0508	0	-
1"	0.0508	0	ton
1 1/4	0.0508	0	-

Click on + button (next to Mass) to add all possible reduction from the current size to a **smaller** one.

Length and mass can be entered :

Property	Unit Metric	Unit USA
Length	m	ft
Mass	ton	kips

WARNING

ATTENTION, for a better readability, we recommend to enter the sizes in **ascending** order.

4. Welding tees

Welding tee is a special fitting with a header and a branch.

The section of the branch must be equal or smaller than the section of the header pipe.

To add **welding tee** properties for each pipe size, click on + button and select a pipe size :

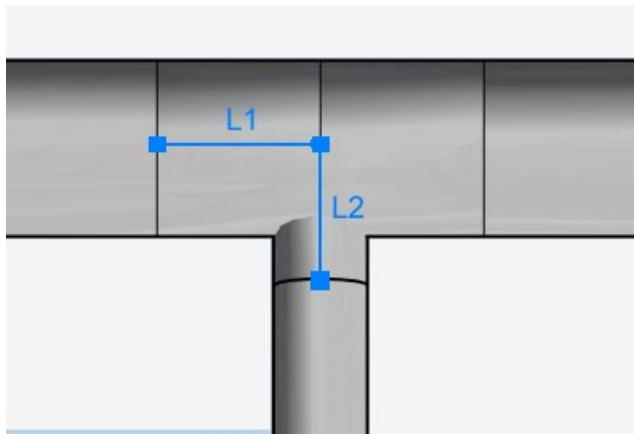
Welding tees	
1/2"	<input type="button" value="+"/>
3/4"	<input type="button" value="-"/>
Length	0.0286 L1
Branch size	Length Mass
3/8"	0.0286 L2 0 <input type="button" value="+"/>
1/2"	0.0286 m 0 <input type="button" value="-"/>
3/4"	0.0286 0 <input type="button" value="-"/>

WARNING

ATTENTION, for a better readability, we recommend to enter the sizes in **ascending** order.

Property	Unit Metric	Unit USA
Length	m	ft

The length corresponds to half the length of the header (L1 in the picture) :



Click on + button (next to Mass) to add all possible branch sizes from the current size to a **smaller** one.

Length and mass can be entered :

Property	Unit Metric	Unit USA
Length of the branch L2	m	ft
Mass	ton	kips

5. Flanges

To add **flange** properties for each pipe sizes, click on + button and select a pipe size :

Flanges	
1/8"	<input type="button" value="+"/>
6"	<input type="button" value="-"/>
Nominal pressures	<input type="button" value="+"/>
Ratings	<input type="button" value="-"/>
50	<input type="button" value="-"/>
Type	Length Mass
Welding neck	0.1 0.003 <input type="button" value="+"/>
Socket welding	0.15 0.025 <input type="button" value="-"/>

WARNING

ATTENTION, for a better readability, we recommend to enter the sizes in **ascending** order.

For this size, create several **Ratings** (max service pressure). It is a text and is just informative (Ex : 50).

Click the + button (next to Mass) to add a new flange of a certain type :

- Welding neck
- Slip-on
- Lap joint
- Threaded
- Socket welding
- Blind

Length and mass can be entered :

Property	Unit Metric	Unit USA
Length of the flange	m	ft
Mass	ton	kips

Components

In this section, libraries of components can be created.

A library is a **JSON** file with a name, a reference to a piping library and a list of components.

Components are :

- Valves
- Flanges
- Bellows
- Sockets

The screenshot shows the 'Components' tab selected in a top navigation bar. The left panel displays a tree view of component categories: 'Components' (expanded), 'Athena Engineering' (expanded), 'My Component Library' (selected and highlighted in yellow), 'Test1', 'Test2', and 'DFT' (expanded). Under 'DFT', there is a folder named 'Vannes'. The right panel contains configuration fields for the selected library:

Name	My Component Library
Pipe library	ANSI B16_9
Units	Metric
Type	All
Valves	[+]
Flanges	[+]
Bellows	[+]
Sockets	[+]

1. Organisation

On the left panel, you can create folders, sub-folders and libraries.

The - button is used both for removing folders and libraries.

A folder can be a type of component or a specific provider.

You can also drag a file/folder on another folder.

2. Library

A library can be managed as you want.

It can correspond to a provider of several component (Ex : flanges, bellows and sockets) or be specific to a product (Ex : only valves).

A library is referenced to an existing [Piping library](#) for the size definitions. Units are then imposed.

You can choose the type :

- All
- Valves
- Flanges
- Bellows
- Sockets

2.1 Valves

Click the + button to add valves for a specific pipe size. Select a size (Ex : 4") :

The screenshot shows the 'Valves' category selected in the left navigation pane. Under '4"', the 'Nominal pressures' section is expanded, showing a table with a row for '50'. The right panel displays detailed configuration for this valve:

Type	Length	Mass	Thickness factor	Filled stem	Connection	Description
With exentricity	0.3	0.01	1	<input type="checkbox"/>	Butt weld - as welded	4" DFT® Ultra-Trol® Valve

For this size, create several **Ratings** (max service pressure). It is a text and is just informative (Ex : 50).

Click the + button to add a new valve of a certain type :

- Simple
- With middle point
- With eccentricity

Property	Unit Metric	Unit USA
Length	m	ft
Mass	ton	kips
Thickness factor	-	-

The wall thickness is increased by the thickness factor by equally increasing the outer diameter and decreasing the inner diameter, keeping the

The connection indicates how the valve is connected to the pipes :

- None
- Butt weld - flush
- Butt weld - as welded
- Fillet weld
- Full fillet weld
- Threaded
- Brazed

2.2 Flanges

Click the + button to add flanges for a specific pipe size. Select a size (Ex : 4") :

Flanges	+ []	[]
4"	+ []	[]
Nominal pressures	+ []	[]
Ratings	50 []	[]
	Type	Length
	Welding neck	0.08
	Mass	Mass
	0.005	3
	Thickness factor	Mass at end
	3	[]
	Diameter	Diameter
	130	20
	Thickness	Thickness
	[]	[]
	Description	[]

For this size, create several **Ratings** (max service pressure). It is a text and is just informative (Ex : 50).

Click the + button to add a new flange of a certain type :

- Welding neck
- Slip-on
- Lap joint
- Threaded
- Socket welding
- Blind

Property	Unit Metric	Unit USA
Length	m	ft
Mass	ton	kips
Thickness factor	-	-
Diameter of the flange	mm	in
Thickness of the flange	mm	in

The wall thickness is increased by the thickness factor by equally increasing the outer diameter and decreasing the inner diameter, keeping the



2.3 Bellows

Click the + button to add bellows for a specific pipe size. Select a size (Ex : 4") :

Type	Length	Distributed mass	Pressure area	Axial stiffness	Lateral stiffness	Angular stiffness	Torsional stiffness	Description
Axial								<input type="button" value="+"/>

For this size, create several **Ratings** (max service pressure). It is a text and is just informative (Ex : 50).

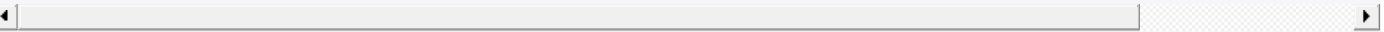
Click the + button to add a new bellow of a certain type :

- Axial : *Axial and lateral displacements and rotations are permitted*
- Lateral in a single plane : *Lateral displacements are only permitted in the plane defined by the local axes Z' and X'. No axial displacements nor rotations are permitted*
- Lateral in all planes : *Lateral displacements are permitted in all directions perpendicular to the element axis Z'. No axial displacements nor rotations are permitted*
- Angular in a single plane : *Rotations are only permitted about the local axis Y'. The movements are thus restricted to the plane defined by the local axes Z' and X'*
- Angular in all planes : *Rotations are permitted about all axes perpendicular to Z'*

Property	Unit Metric	Unit USA
Length	m	ft
Distributed mass	ton	kips
Pressure area	mm ²	in ²
Axial stiffness	kN/mm	kips/in
Lateral stiffness	kN/mm	kips/in
Angular stiffness	kN.m/rad	kips.ft/rad
Torsional stiffness	kN.m/rad	kips.ft/rad

If Torsional stiffness is zero or blank, the joint is modeled as rigid in torsion and the user should verify that the moments about the element

One half of the mass will be concentrated at each end of the bellow.



2.4 Sockets

Click the + button to add sockets for a specific pipe size. Select a size (Ex : 4") :

Length	Mass	Thickness factor	Description
			<input type="button" value="+"/>

For this size, create several **Ratings** (max service pressure). It is a text and is just informative (Ex : 50).

Click the + button to add a new socket.

Property	Unit Metric	Unit USA
Length	m	ft
Mass	ton	kips
Thickness factor	-	-

The wall thickness is increased by the thickness factor by equally increasing the outer diameter and decreasing the inner diameter, keeping the



Supports

In this section, libraries of components can be created.

A library is a **JSON** file with a name, a unit and a list of supports.

Predefined support are :

- Rotational restraints
- Translational restraints
- Snubbers
- Variable spring hangers
- Constant spring hangers

Name	Units	
Rotational restraints	Metric	[+]
Translational restraints		[+]
Snubbers		[+]
Variable spring hanger series		[+]
Constant spring hanger series		[+]

1. Organisation

On the left panel, you can create folders, sub-folders and libraries.

The - button is used both for removing folders and libraries.

A folder can be a type of component or a specific provider.

You can also drag a file/folder on another folder.

2. Library

A library can be managed as you want. One library for all type of support or one provider of a specific type.

For all type, the main property is the **stiffness**. All other properties are essentially informative.

2.1 Rotational restraints

Click the + button to add a new support :

Property	Unit Metric	Unit USA	Note
Stiffness	kN.m/rad	kips.ft/rad	
Max moment	kN.m	kips.ft	informative
Size	mm	in	informative
Mass	ton	kips	informative

2.2 Translational restraints

Click the + button to add a new support :

Translational restraints		+	Stiffness	Max force	Size	Mass	Description	-
Property	Unit Metric		Unit USA		Note			
Stiffness	kN/mm		kips/in					
Max force	kN		kips		informative			
Size	mm		in		informative			
Mass	ton		kips		informative			

2.3 Snubbers

Click the + button to add a new support :

Snubbers		+	Stiffness	Max force	Size	Mass	Description	-
Property	Unit Metric		Unit USA		Note			
Stiffness	kN/mm		kips/in					
Max force	kN		kips		informative			
Size	mm		in		informative			
Mass	ton		kips		informative			

2.4 Variable spring hangers

Click the + button to add a new range of forces :

Variable spring hanger series		→	+	Min Force	Max force	-	Stiffness	Hanger size	Can size	Mass	Description	+	-
				Unit USA			Note						
Min force	kN			kips			informative						
Max force	kN			kips			informative						

Property	Unit Metric		Unit USA		Note							
Min force	kN		kips		informative							
Max force	kN		kips		informative							

Click the + button (next to the description) to add a new support :

Property	Unit Metric		Unit USA		Note			
Stiffness	kN/mm		kips/in					
Hanger size	mm		in		informative			
Can size	mm		in		informative			
Mass	ton		kips		informative			

2.5 Constant spring hangers

This support is fully informative inside the library ! There is no stiffness but a max displacement.

Click the + button to add a new range of forces :

Constant spring hanger series

	Min Force	Max force					
→							
	Max displacement	Hanger size	Can size	Mass	Description		

Property	Unit Metric	Unit USA	Note
Min force	kN	kips	informative
Max force	kN	kips	informative

Click the + button (next to the description) to add a new support :

Property	Unit Metric	Unit USA	Note
Max displacement	mm	in	informative
Hanger size	mm	in	informative
Can size	mm	in	informative
Mass	ton	kips	informative

Spectra

In this section, spectra can be created.

A spectrum is a **JSON** file with a name, a displacement max, a directional type and a list of dampings.

1. Organisation

On the left panel, you can create folders, sub-folders and spectra.

The - button is used both for removing folders and spectra.

A folder can be a site, a phase, a location or a floor for several spectrum files.

You can also drag a file/folder on another folder.

2. Spectrum

A spectrum defines the **curves** frequency/acceleration for different directions. And that for several dampings.

An *informative* dispacement max can be entered :

Properties	Unit Metric	Unit USA
Dx, Dy, Dz	mm	in

The **type** that defines the curves can be :

Property	Nb curves	Note
X + Y + Z	3	One curve by direction
X + YZ	2	One curve for X and one curve for both Y and Z
Y + XZ	2	One curve for Y and one curve for both X and Z
Z + XY	2	One curve for Z and one curve for both X and Y
XYZ	1	One curve for all directions

Based on this type, 1, 2 or 3 tables must be filled.

Click the + button (next to Dampings) to add a damping value (%) and a label :

Name	Floor 1		
Displacement max	Dx	Dy	Dz
Type	X + YZ		
Dampings	<input style="border: 2px solid orange; border-radius: 50%; width: 15px; height: 15px; vertical-align: middle;" type="button" value="+"/> 5		
Value %	5-10 %		
Label	X Frequency Acceleration <input style="border: 2px solid orange; border-radius: 50%; width: 15px; height: 15px; vertical-align: middle;" type="button" value="+"/> Frequency Acceleration <input style="border: 2px solid orange; border-radius: 50%; width: 15px; height: 15px; vertical-align: middle;" type="button" value="+"/> 0.1 0.01 0.1 0.01		
	1 0.1 1 0.09		
	10 0.5 10 0.45		
	20 0.5 20 0.45		
	30 0.3 30 0.3		
Value %	10		
Label	10-15 %		
	YZ Frequency Acceleration <input style="border: 2px solid orange; border-radius: 50%; width: 15px; height: 15px; vertical-align: middle;" type="button" value="+"/> Frequency Acceleration <input style="border: 2px solid orange; border-radius: 50%; width: 15px; height: 15px; vertical-align: middle;" type="button" value="+"/> 0.1 0.009 0.1 0.009		
	1 0.09 1 0.09		
	10 0.45 10 0.45		
	20 0.45 20 0.45		
	30 0.27 30 0.27		

Fill the tables by adding rows for the pair frequency/acceleration.

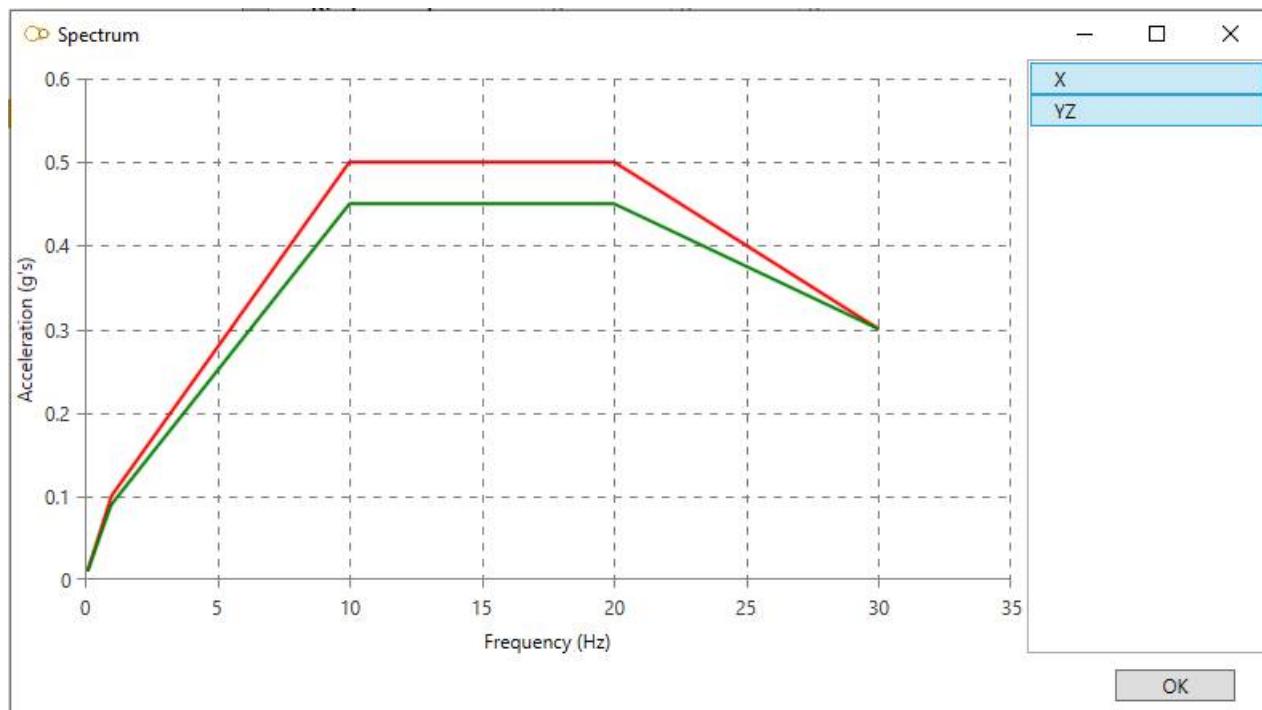
Property	Unit
Frequency	Hz
Acceleration	G's

WARNING

ATTENTION, the frequencies must be filled in ascending order

Each table can have its own number of rows

You can also verify the tables of a damping by clicking on the last button next to Value % :



Beam sections

In this section, beam section can be created.

A section is a **JSON** file with a name, a type, units and a list of geometrical properties.

The screenshot shows the 'Beam sections' tab in a software application. On the left, there's a sidebar with categories like 'Beams', 'EuropeanSections', 'EqualAngle', 'HEA', 'HEB', 'IPE', 'IPN', 'UAP', 'UPE', and 'UPN'. Under 'EqualAngle', 'L 050 x 050 x 5' is selected and highlighted with a yellow bar. The main panel shows a diagram of an L-shaped section with dimensions: height (h), thickness (t), and width (h). It also shows coordinate axes X and Y. Below the diagram, under 'Fabrication', it says 'Hot rolled'. Under 'Dimensions', it lists height (h) as 50, thickness (t) as 5, and area A as 4.80258 cm². Under 'First properties', it lists reduced area Ax as 2.5, reduced area Ay as 2.5, torsional inertia Ix as 0.39583, bending inertia Ix as 10.96, and bending inertia ly as 10.96. Under 'Second properties', it lists shear resistance Wx as 0.826, torsion module Wt as 3.05, elastic module WEly as 3.05, plastic module WPly as 3.05, plastic module WPLx as 3.05, and warping inertia Iw as 0.

1. Organisation

On the left panel, you can create folders, sub-folders and sections.

The - button is used both for removing folders and sections.

You can also import sections from BEAMSTRESS.

You can drag a file/folder on another folder.

To know the UNIT of a property, just let the mouse over the property name.
For example : cm² for the Area A

2. Section

The type can be :

Non standard

- I
- Channel
- Rect
- Tee
- Equal angle
- Unequal angle
- Round
- Plate

The preview image of the section and the properties vary with the type.

Properties :

Property	Description	Unit Metric	Unit USA
Fabrication	Hot rolled, cold rolled or welded	-	-
h	Height	mm	in
b	Basis	mm	in
tw	Web thickness	mm	in
tf	Flange thickness	mm	in
t	Thickness	mm	in
y0	Shear center Y	mm	in
x0	Shear center X	mm	in
A	Area	cm ²	in ²
Ax	Reduced area X	cm ²	in ²
Ay	Reduced area Y	cm ²	in ²
It	Torsional inertia	cm ⁴	in ⁴
Ix	Bending inertia X	cm ⁴	in ⁴
ly	Bending inertia Y	cm ⁴	in ⁴
Wy	Shear resistance Y	cm ²	in ²
Wx	Shear resistance X	cm ²	in ²
Wt	Torsion module	cm ³	in ³
WEly	Elastic module Y	cm ³	in ³
WElx	Elastic module X	cm ³	in ³
WPly	Plastic module Y	cm ³	in ³
WPLx	Plastic module X	cm ³	in ³
Iw	Warping inertia	cm ⁶	in ⁶

Press the  button for an automatic calculation of the ares and inertias.

Fasteners

In this section, fasteners can be created.

A fastener is a **JSON** file with a name, a type, units and a list of properties.

Name	SPIT TRIGA Z M12
ETA reference	Réf. xxx valide jusqu'au XX/XX/XX
Type	Mechanical
Units	Metric
Dimensions	Diameter d External diam. dnom Anchor depth hef Bearing area Ah Instal. factor yinst
Steel	Tensile strength fuk Yield strength fyk
Concrete cone	
Pull-out	
Tension splitting failure	NRk,sp,0 Normal ymsp Accidental ymsp
Shear pry-out failure	k Shear concrete edge Lf
Displacements	
Seismic	agap reduction factor NRk,s,eq NRk,p,eq τRk,eq VRk,s,eq

1. Organization

On the left panel, you can create folders, sub-folders and fasteners.

The - button is used both for removing folders and fasteners.

You can also import fasteners from BEAMSTRESS.

You can drag a file/folder on another folder.

2. Fastener

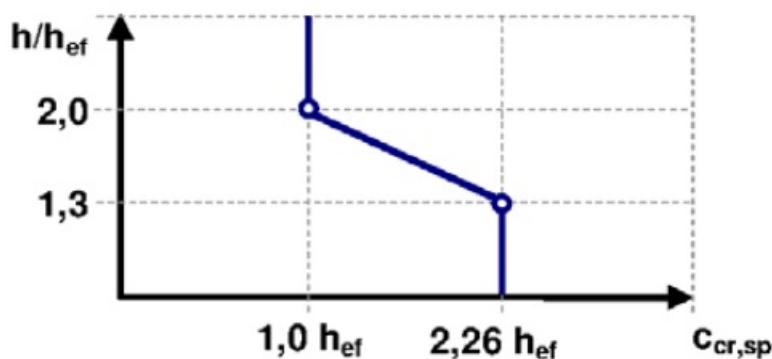
The fastener characteristics are defined by means of properties normally given in the ETA's (European Technical Assessments) provided by the manufacturer. The notations are those of code EN 1992-4 edition 2018 :

Name	SPIT TRIGA Z M12							
ETA reference	Réf. xxx valide jusqu'au XX/XX/XX							
Type	Mechanical							
Units	Metric							
Dimensions	Diameter d	External diam. dnom	Anchor depth hef	Bearing area Ah	Instal. factor yinst			
	12	17.5	80		1			
Cracked concrete	Min thickness hmin	Min spacing smin	for dist to edge c >=	Min dist to edge cmir	for spacing s >=			
	160	80	160	80	200			
					<input checked="" type="checkbox"/> Same uncracked values			
Steel	Tensile strength fuk	Yield strength fyk						
Tension failure	NRk,s	Normal yms	Accidental yms					
	67	1.5	1.5					
Shear failure	VRk,s	Normal yms	Accidental yms					
	72.7	1.25	1.25					
Mounting	Without spacing k7	With spacing MRk,s,0						
	0.8	0.105						
Concrete cone	Ccr,n	Cracked k1	Uncracked k1	Normal ymc	Accidental ymc			
	120	7.2	10.1	1.5				
Pull-out	Tension failure	Nrk,p cracked C20/25	Nrk,p uncracked C20/	Normal ymp	Accidental ymp			
				1.5				
Class factor ψ_c	C25/30	C30/37	C35/45	C40/50	C45/55	C50/60	C55/67	C60/75
Cracked concrete		1.22		1.41		1.55		
Uncracked concrete		1.22		1.41		1.55		
Chemical anchors	$\psi_{us,0}$	τ_{Rk} cracked C20/25	τ_{Rk} uncracked C20/25					
Tension splitting failure	NRk,sp,0	Normal ymsp	Accidental ymsp					
		1.5						
	Ccr,sp							<input type="checkbox"/> Ccr,sp depends on concrete thickness h
Shear pry-out failure	k	Shear concrete edge Lf failure						
	2	80						
Displacements								
Static tension	δN_0 cracked	δN_0 uncracked	δN_{∞} cracked	δN_{∞} uncracked				
Static shear	δV_0 cracked	δV_0 uncracked	δV_{∞} cracked	δV_{∞} uncracked				
Seismic	gap reduction factor	NRk,s,eq	NRk,p,eq	$\tau_{Rk,eq}$	VRk,s,eq	δN_{eq} (DLS)	δV_{eq} (DLS)	k15
		67	9.4		28.4	2.64	5.83	1

The ultimate tensile strength fuk and yield strength fyk allow to calculate the partial safety factors for the steel. If not available, the user must enter these safety factors directly.

The resistance to pull-out Nrk,p must be defined for non-bonded fasteners. For bonded fasteners, the strength is calculated based on the parameters provided for chemical anchors.

For some fasteners, the characteristic distance Ccr,sp (splitting failure) depends on the concrete thickness :



In this case, check the box **Ccr,sp depends on concrete thickness** and enter the ratio Ccr,sp/hef for two values of the ratio h/hef (beginning with the smallest value of h/hef), where h is the concrete thickness.

The parameters δN_0 , δN_{∞} , δV_0 and δV_{∞} are flexibilities (in mm/kN or in/kips). MetaStructure multiplies these values by the tensile and shear forces on the fasteners to obtain the resulting displacements. Two sets of values are given by the manufacturers: short-term loadings (index 0) and long-term loadings (index ∞). MetaStructure will calculate the displacements for these two durations.

To know the UNIT of a property, just let the mouse over the property name.

3. Plate anchors

The fastener library can be called during anchor plate definition.

Bolts

In this section, bolt library can be created.

A bolt library is a **JSON** file with a name, units and a list of bolts.

Name	Diameter Db	Pitch Pc	Resistant area	Pitch diameter Dp	Bolt nut diameter	Thread half angle Be
M10	10	1.5	75 mm ²	0	0	0
M12	12	1.75	84.3 mm ²	0	0	0
M14	14	2	115	0	0	0
M16	16	2	157	0	0	0
M18	18	2.5	192	0	0	0
M20	20	2.5	245	0	0	0
M22	22	2.5	303	0	0	0
M24	24	3	353	0	0	0
M27	27	3	459	0	0	0
M30	30	3.5	561	0	0	0

1. Organisation

On the left panel, you can create folders, sub-folders and bolt libraries.

The - button is used both for removing folders and libraries.

You can drag a file/folder on another folder.

To know the UNIT of a property, just let the mouse over the property name.

For example : mm² for the Resistant area

2. Bolt

Click the + button (next to Bolts) to add a new bolt or click the - button (at the end of a row) to remove one.

Properties :

Property	Description	Unit Metric	Unit USA
Name	The name of the bolt	-	-
Diameter Db	Outside diameter	mm	in
Pitch Pc	Distance between threads	mm	in
Resistant area	Tensile stress area	mm ²	in ²
Pitch diameter Dp	Middle diameter	mm	in
Bolt nut diameter	Nut diameter	mm	in
Thread half angle Be	Half angle	°	°

3. Bolting plate

The Bolt library can be called during bolting plate definition.

Ratings

In this section, rating library can be created.

A rating library is a **JSON** file with a name, units and a list of temperature/pressure curves.

The screenshot shows the Ratings interface. At the top, there is a navigation bar with tabs: Materials, Piping, Components, Supports, Spectra, and Ratings. Below the navigation bar is a toolbar with icons for creating a folder, creating a file, and opening a file. On the left, there is a sidebar with a tree view. The tree starts with 'Ratings', then 'P50', and finally 'RatingP50 A' which is highlighted with a yellow background. To the right of the sidebar is a table with columns for Name, Title, Units, and Curve. The 'Curve' column contains a table of temperature (T) and pressure (P) pairs:

T (°C)	P (N/mm²)	Add (+)	Remove (-)
100	5	<input type="checkbox"/>	<input type="checkbox"/>
120	5	<input type="checkbox"/>	<input type="checkbox"/>
140	6	<input type="checkbox"/>	<input type="checkbox"/>
160	6	<input type="checkbox"/>	<input type="checkbox"/>
180	7	<input type="checkbox"/>	<input type="checkbox"/>

1. Organisation

On the left panel, you can create folders, sub-folders and rating libraries.

The - button is used both for removing folders and libraries.

You can drag a file/folder on another folder.

2. Rating

Click the + button (next to Curve) to add a new temperature/pressure pair or click the - button (at the end of a row) to remove one.

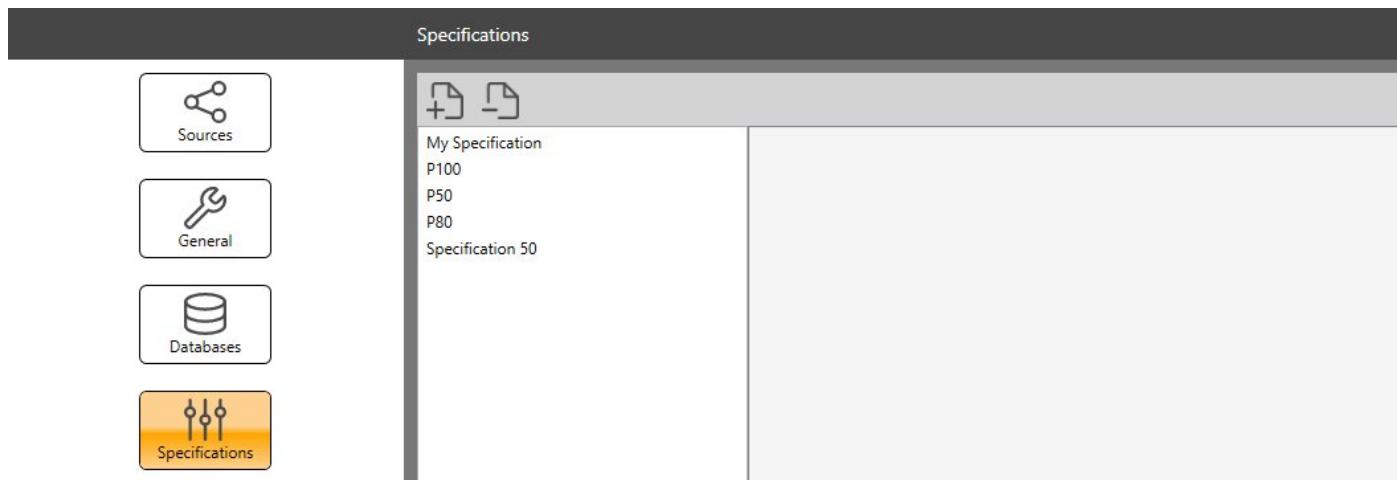
3. Supports/Flanges post-processing

These libraries are only used during **Supports/Flanges post-processing**.

Click [here](#) for more informations.

Specifications

The specification screen is only available if MetaPiping is active on your license.



MetaPiping lets you create **specifications** = predefined choice of piping elements and properties dedicated to a particular project.

It brings together a variety of information :

- Calculation code and edition
- Library of materials
- Library of piping elements

Click the + button to add a new specification or click the - button to remove an **unused** one.

WARNING

ATTENTION, do not remove specification used in one or more projects otherwise the projects may not open properly.

1. Header

Description	My Specification
Units	Metric
Code	RCC-M Class 2 (French Nuclear Piping Code)
Edition	All
Max design temperature	0
Max design pressure	0
Standards	
Materials	Undefined
Pipes	Undefined
Pipes	Size Schedule Material Node connection Mismatch / fillet length Long weld type Long weld mismatch Linear mass + insulation Insulation thickness External corrosion Internal erosion Bend thickness Description
Bends	Size Long radius Small radius Mass
Reducers	
Tees	
Flanges	Size Nominal pressure Type Length Mass

Properties :

- Units - Metric or USA
- Code - See [Codes](#) for more information
- Edition - See [Codes](#) for more information
- Max design temperature - °C or °F
- Max design pressure - N/mm² or psi
- Material library - See [Materials](#) for more information
- Piping library - See [Piping](#) for more information

WARNING

ATTENTION, a specification has UNITS, so only libraries of materials and piping elements with the same UNITS can be connected.

WARNING

ATTENTION, when starting a new project, be sure to select a specification that will be compatible with the Units, Code and Edition of the project and do not change these 3 properties during design.

2. Elements

To create a **specification**, just click to + button and select several pipe sizes :

Pipes												
Size	Schedule	Material	Node connection	Mismatch / fillet length	Long weld type	Long weld mismatch	Linear mass + insulation	Insulation thickness	External corrugation	Internal erosion	Bend thickness	Description
4"	Sch 5S	SA-106 A	None	0	None	0	5.84	0	0 mm	0	2.11	
2"	Sch 5S	SA-106 A	None	0	None	0	2.39	0	0	0	1.65	
Bends												
Size	Long radius	Small radius	Mass									
4"	0.1524	0.1016	0									
2"	0.0762	0.0508	0									
Reducers												
4" 4"	Reduce to	Length	Mass	Angle								
2"		0.1016	0	0								
Tees												
4" 4"	Branch size	Branch type	L1	R2	TN or Y	Thickness type	RE	RX	PD	Collector length	Branch length	Mass
4"	Branch connect	v	0	0	0	Constant	v					
2"	Branch connect	v	0	0	0	Constant	v					
4" 2"	Branch size	Branch type	L1	R2	TN or Y	Thickness type	RE	RX	PD	Collector length	Branch length	Mass
2"	Branch connect	v	0	0	0	Constant	v					
Flanges												
Size	Nominal pressure	Type	Length	Mass								
4"	50	Welding neck	0.08	0.003								
2"	50	Welding neck	0.07	0.0025								

MetaPiping fills automatically the tables and let you make some choices.

2.1 Pipes

For each size, define the **schedule** and the **material**. A **pipe** knows now its section and its material :

Pipes												
Size	Schedule	Material	Node connection	Mismatch / fillet length	Long weld type	Long weld mismatch	Linear mass + insulation	Insulation thickness	External corrugation	Internal erosion	Bend thickness	Description
4"	Sch 5S	SA-106 A	None	0	None	0	5.84	0	0 mm	0	2.11	
2"	Sch 5S	SA-106 A	None	0	None	0	2.39	0	0	0	1.65	

Select then the **node connections** of the pipe between :

- None
- Butt weld - flush
- Butt weld - as welded
- Fillet weld
- Full fillet weld
- Threaded
- Brazed

Based on this property, define the **mismatch** or the **fillet length** [mm or in].

Select then the **Long weld type** of the pipe between :

- None
- Butt weld flush
- Butt weld as welded

Based on this property, define the **Long weld mismatch** [mm or in].

Other properties :

Property	Unit Metric	Unit USA	Default
Linear mass + insulation	kg/m	lb/ft	Linear mass of the pipe
Insulation thickness	mm	in	0
External corrosion	mm	in	0
Internal erosion	mm	in	0
Bend thickness	mm	in	Thickness of the pipe

2.2 Bends

MetaPiping automatically fills the table with the corresponding information :

Bends			
Size	Long radius	Small radius	Mass
4"	0.1524	0.1016	0
2"	0.0762	0.0508	0

Property	Unit Metric	Unit USA
Long radius	m	ft
Small radius	m	ft
Mass	ton	kips

To know the UNIT of a property, just let the mouse over the property name.

2.3 Reducers

MetaPiping automatically fills the table with the corresponding information :

Reducers			
4"	Length	Mass	Angle
Reduce to 2"	0.1016	0	0

Property	Unit Metric	Unit USA	Note
Length	m	ft	
Mass	ton	kips	
Angle	°	°	Class 1 only

To know the UNIT of a property, just let the mouse over the property name.

2.4 Tees

MetaPiping automatically fills the table with the corresponding informations :

Tees												
4"	Branch size	Branch type	L1	R2	TN or Y	Thickness type	RE	RX	PD	Collector length	Branch length	Mass
4"	WeldingTee	v	0	0	0	Constant	v	0	PD	0.1048	0.1048	0
2"	Branch connect	v	0	0	0	Constant	v					
2"	Branch size	Branch type	L1	R2	TN or Y	Thickness type	RE	RX	PD	Collector length	Branch length	Mass
2"	Branch connect	v	0	0	0	Constant	v					

For each combination, you can define the type of tee :

- Branch connection
- WeldingTee
- Non Standard

- Fabricated
- Sweeplet flush weld
- Sweeplet as welded
- Weldolet
- Extruded weldingTee
- Welded-in contour insert
- Branch welded-on fitting
- Lateral connection
- Partial penetration branch connection

Based on the type, some other properties must be filled :

Property	Description	Unit Metric	Unit USA
L1	Length of reinforcement	mm	in
R2	Branch-to-run fillet radius	mm	in
TN	Branch connections and lateral connections CONSTANT thickness	mm	in
Y	Branch connections and lateral connections VARIABLE thickness	mm	in
Thickness type	Variable or constant	-	-
RE	Pad or saddle outer radius for reinforced fabricated tees	mm	in
RX	Transition radius	mm	in
PD	Pad thickness for reinforced fabricated tees	mm	in
Collector length	Half length of the header of welding tees	m	ft
Branch length	Branch length of welding tees	m	ft
Mass	Mass of the tee	ton	kips

2.5 Flanges

MetaPiping automatically fills the table with the corresponding information :

Flanges		Nominal pressure	Type	Length	Mass
Size	4"	50	Welding neck	0.08	0.003
	2"	50	Welding neck	0.07	0.0025

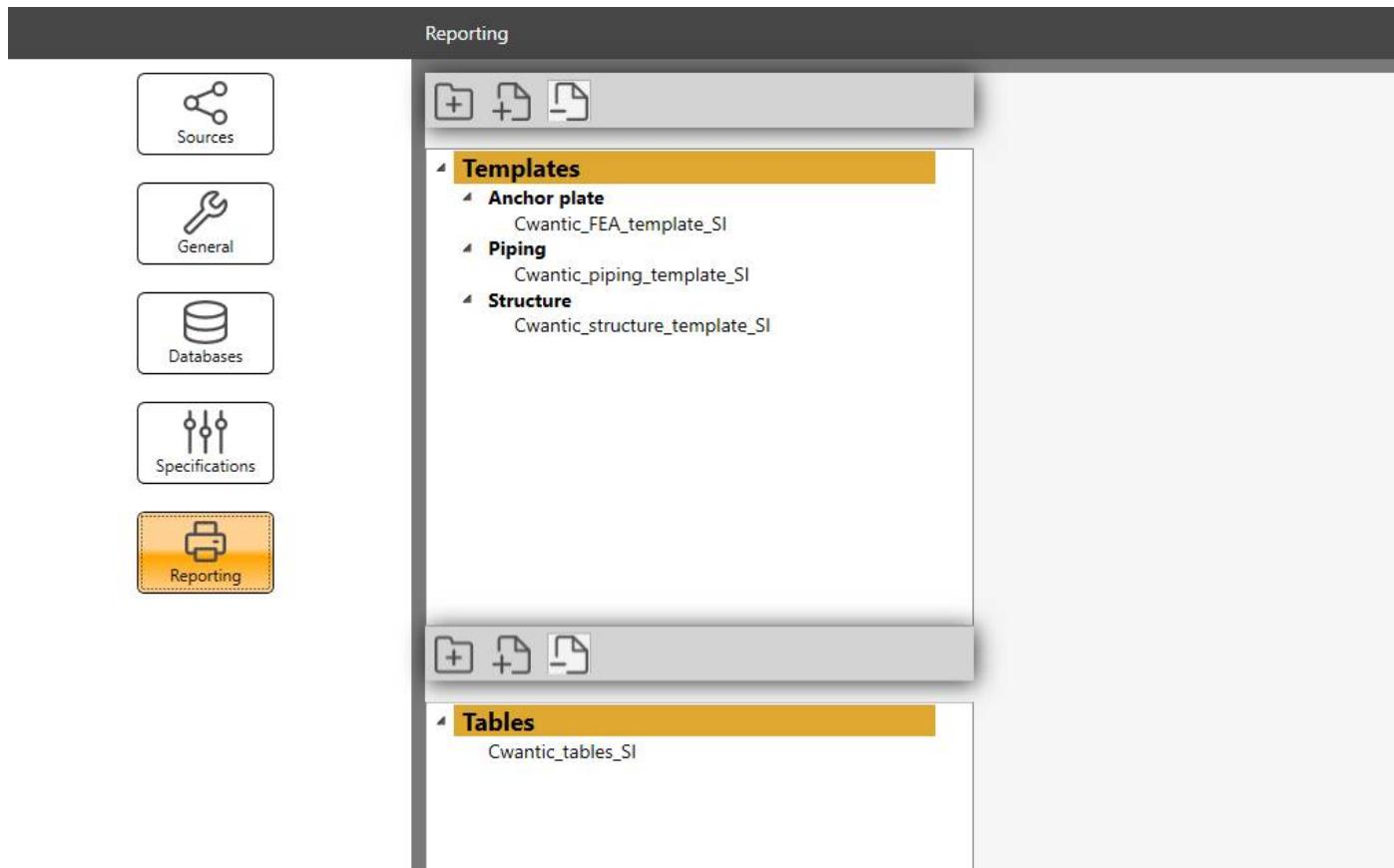
For each size, select the **Nominal pressure** (Rating) and select the predefined **type** of flange :

- Welding neck
- Slip-on
- Lap joint
- Threaded
- Socket welding
- Blind

The predefined properties will be shown.

Property	Unit Metric	Unit USA
Length	m	ft
Mass	ton	kips

Reporting



MetaPiping and MetaStructure can generate **reports** with all the details of the model input, some review pictures and all results of an analysis.

The report is exclusively based on *.docx file.

WARNING

ATTENTION, it is mandatory that Microsoft Word must be installed on user computer !

A **report** is based on an empty **template** (*.docx) that contains **keywords**.

A **keyword** is associated to a **section** described in a **table** document (*.docx).

User can prepare **template** document and **table** document and then import them in the software (the first one in the template section, the second in the table section) as shown in the upper figure.

WARNING

ATTENTION, the documents are copied in the settings. The original word files are not moved nor modified.

1. Keyword mechanism

The keyword is useful to make a correspondence between the **template** and the **table** document but with specific decorators :

```
 $$keyword$$ for the template
 [keyword] for the table
```

Example :

Keyword	Template keyword	Table keyword
MATERIALS	\$\$MATERIALS\$\$	[MATERIALS]

A digit is authorized after a keyword [0..9] to enlarge the possibilities.

Example :

Keyword	Template keyword	Table keyword
MATERIALS	\$\$MATERIALS0\$\$ -> \$\$MATERIALS9\$\$	[MATERIALS0] -> [MATERIALS9]

If the software encounters the keyword \$\$MATERIALS3\$\$ in the template, it will know that it corresponds to the internal keyword MATERIALS and will try to find the section in the table document with the keyword [MATERIAL3]. If succeed, it will **insert** the content of the table at the position of the template keyword and fill all materials.

WARNING

ATTENTION, the keyword must be written in CAPITAL letters.

2. Template

A **template** is a Word *.docx file with user defined layout and content.

MetaPiping/MetaStructure let you **insert** datas from the current study inside the template at a certain position in the document via **keywords**.

2.2 Keywords

Keyword	Module(s)	Description	Other possible keyword or remark
STUDY NAME	MetaPiping/MetaStructure/Finite element analysis	The name of the current study	No table
PICTURE	MetaPiping/MetaStructure/Finite element analysis	Take a picture (\$2.3)	No table
SECTIONS	MetaPiping/MetaStructure	The list of all sections	CROSS SECTION
MATERIALS	MetaPiping/MetaStructure	The list of all materials	
NODES	MetaPiping/MetaStructure	The list of all nodes	NODES INPUT
ELEMENTS	MetaPiping/MetaStructure	The list of all elements	ELEMENTS INPUT
ELEMENT COORDINATE SYSTEMS	MetaPiping/MetaStructure	The list of all elements with non default CS	
NODE COORDINATE SYSTEMS	MetaPiping/MetaStructure	The list of all DLCS	
LUMPED MASSES	MetaPiping/MetaStructure	The list of all lumped masses	
SUPPORTS	MetaPiping/MetaStructure	The list of all restraints	SUPPORT
STATIC CASES	MetaPiping/MetaStructure	The list of all static cases	
EXTERNAL CASES	MetaStructure	The list of all external cases	from linked studies
FORCES	MetaPiping/MetaStructure	The list of all nodal load forces	also the external forces (linked study)
MOVEMENTS	MetaPiping/MetaStructure	The list of all restraint movements	DISPLACEMENTS
OPERATING CONDITIONS	MetaPiping/MetaStructure	The list of all operating conditions	TEMPERATURE & PRESSURE
DISTRIBUTED FORCES	MetaPiping/MetaStructure	The list of all distributed forces	
WIND	MetaPiping/MetaStructure	The list of all wind forces	
SNOW	MetaPiping/MetaStructure	The list of all snow forces	

ACCELERATIONS Keyword	MetaPiping/MetaStructure Module(s)	The list of all static accelerations Description	Other possible keyword or remark
DENSITIES	MetaPiping	The list of all content densities	
STRATIFICATIONS	MetaPiping	The list of all stratifications	
COLD SPRINGS	MetaPiping	The list of all cold springs	
TRANSIENTS	MetaPiping	The list of all transients	
LOAD SETS	MetaPiping	The list of all load sets	
SPECTRA	MetaPiping/MetaStructure	The list of all spectra	
DYNAMIC EVENTS	MetaPiping/MetaStructure	The list of all dynamic node loads	
DYNAMIC CASES	MetaPiping/MetaStructure	The list of all dynamic cases	
PRIMARY RESPONSE	MetaPiping/MetaStructure	The list of all primary cases	
SECONDARY RESPONSE	MetaPiping/MetaStructure	The list of all secondary cases	
COMBINATION CASES	MetaPiping/MetaStructure	The list of all combination cases	
COMBINED STRESS CASES	MetaPiping	The list of all combined stress cases	
RDISPLACEMENTS	MetaPiping/MetaStructure	The list of all displacement results	Possible JSON parameters (see §3)
RFORCES	MetaPiping/MetaStructure	The list of all force and moment results	Possible JSON parameters (see §3)
REACTIONS	MetaPiping/MetaStructure	The list of all reaction results	Possible JSON parameters (see §3)
STRESSES	MetaPiping/MetaStructure	The list of all stress results	Possible JSON parameters (see §3)
ANCHOR PLATES	MetaStructure	The list of all anchor plate results	
JOINTS	MetaStructure	The list of all joint results	
ANCHORS RESULTS	Finite element analysis	The list of all fastener results	
MESH RESULTS	Finite element analysis	The list of all mesh results	
ANALYSIS NAME	Finite element analysis	The name of the current analysis	No table
CONFIGURATION NAME	Finite element analysis	The name of the current configuration	No table
HEIGHT	Finite element analysis	The height of the current assembly	No table
MESH SIZE	Finite element analysis	The meshing size	No table
IDNODE	Finite element analysis	The name of the current node	No table
LOADCASE	Finite element analysis	The current load case	No table
STATIC EQUILIBRIUM	Finite element analysis	The static equilibrium value	No table
DISPLACEMENT MAX	Finite element analysis	The displacement max	No table
STRESS MAX	Finite element analysis	The stress max	No table
STRAIN MAX	Finite element analysis	The strain max	No table
FASTENER RATIO MAX	Finite element analysis	The max ratio on all fasteners	No table

2.3 Pictures

It is possible to include pictures in the report with use of the keyword **PICTURE**.

When the software encounters this keyword, it simply makes a screenshot of the 3D engine.

It is possible to change the kind of visualization via a **JSON structure** just after the keyword separated by a semicolon character :

```
$$PICTURE;{...}$$
```

2.3.1 Piping & structure parameters

JSON parameters :

Parameter	Description	Default value
Layers	An array of visible layer name	Empty list = all layers will be visible
View	Orientation of the camera	39 (= FrontFaceTopLeft - see below)
Mode	Display mode	0 (= Shaded - see below)
Type	Ribbon type	0 (= Conception - see below)
ReviewType	Review category	0 (= Sections - see below)
ResultType	Result category	0 (= Displacements - see below)
LoadCase	Index of mode or load case in the list	-1
Factor	Amplification factor of the displacements	1
StressIndex	Index in possible stress list	0
StressMax	1 = show the max stress colorisation	0
ShowNodes	1 = show node point and name	0
ShowOrigin	1 = show origin	0
ShowAxis	1 = show axis	0

No parameter is mandatory. If a parameter is not specified, its default value will be used.

View values :

Front	= 0
Right	= 1
Rear	= 2
Left	= 3
Top	= 4
Bottom	= 5
Isometric	= 6
Dimetric	= 7
FrontFaceLeft	= 12
RightFaceBottom	= 13
RightFaceRight	= 14
RightFaceTop	= 15
RightFaceLeft	= 16
BackFaceBottom	= 17
BackFaceRight	= 18
BackFaceTop	= 19
BackFaceLeft	= 20
LeftFaceBottom	= 21
LeftFaceRight	= 22
LeftFaceTop	= 23
LeftFaceLeft	= 24
BottomFaceBottom	= 25
BottomFaceRight	= 26
BottomFaceTop	= 27
BottomFaceLeft	= 28
TopFaceBottom	= 29
TopFaceRight	= 30
TopFaceTop	= 31
TopFaceLeft	= 32
FrontFaceBottomLeft	= 33
FrontFaceBottomRight	= 34
FrontFaceTopLeft	= 35
FrontFaceTopRight	= 36
BackFaceBottomLeft	= 37
BackFaceBottomRight	= 38
BackFaceTopLeft	= 39
BackFaceTopRight	= 40

Mode values :

```
Shaded = 0  
Hiddenlines = 1  
Iso = 2
```

Type values :

```
Conception = 0  
Loads = 10  
Review = 20  
Analysis = 30
```

Review type values :

```
Sections = 0  
Materials = 10  
Specifications = 20  
StressOff = 30  
Soils = 40  
Codes = 50  
RoomTemperatures = 60  
DesignConditions = 70
```

Result type values :

```
None = -1  
Displacements = 0  
Reactions = 1  
Forces = 2  
RestraintForces = 3  
Stresses = 4  
Anchors = 5  
Joints = 6
```

Examples:

```
$$PICTURE$$
```

```
$$PICTURE;{"View":35}$$
```

```
$$PICTURE;{"Type":20,"ReviewType":10,"View":35}$$
```

```
$$PICTURE;{"Type":30,"ResultType":0,"Layers":["0"],"LoadCase":100,"View":35,"Factor":8}$$
```

2.3.2 Finite element analysis parameters

JSON parameters :

Parameter	Description	Default value
Groups	An array of visible group name	Empty list = all groups will be visible
View	Orientation of the camera	35 (= FrontFaceTopLeft - see above)
ResultType	Result category	0 (= Group - see below)
Factor	Amplification factor of the displacements	1
Dim	1 = show dimensions	1

Result type values :

```

Group = 0
Displacement = 1
Stress = 2
Strain = 3
Compression = 4
IsoDisplacement = 5
IsoStress = 6
IsoStrain = 7
IsoCompression = 8

```

Examples:

```
$$PICTURE$$
```

```
$$PICTURE;{"ResultType":2,"Groups":["PLATE"],"View":4,"Dim":0}$$
```

3. Results params

For **results** keywords, it is possible to filter the output via a **JSON structure** just after the keyword separated by a semicolon character :

```

$$RDISPLACEMENTS;{...}$$
$$RFORCES;{...}$$
$$REACTIONS;{...}$$
$$STRESSES;{...}$$

```

JSON parameters :

Parameter	Description	Default value
Layers	An array of visible layer name	Empty list = all layers will be visible
Loadcase	A loadcase number	-1 (all loadcases)
StressIndex	A stress index	0 (for stress results only)

Examples:

```
$$RDISPLACEMENTS$$
```

This will show a table with all displacements (all layers) for all loadcases.

```
$$RDISPLACEMENTS;{"Layers":["0","1"],"LoadCase":100}$$
```

This will show a table with the displacements of nodes from layer "0" and "1" and only for loadcase 100.

You can combine in your report a **picture** and a result **table** :

```

FORCES & MOMENTS:
-----
1) Loadcase 100 - layer 0

$$PICTURE;{"Type":30,"ResultType":2,"Layers":["0"],"LoadCase":100,"View":35}$$
$$RFORCES;{"Layers":["0"],"LoadCase":100}$$

```

This will first show a **PICTURE** (see §2.3.1) :

Parameter	Description
"Type":30	Analysis screen
"ResultType":2	Forces
"Layers":["0"]	Only elements of layer "0"
"LoadCase":100	Only for loadcase number 100
"View":35	FrontFaceTopLeft view

...followed by a **table** :

Parameter	Description
"Layers":["0"]	Only results of elements of layer "0"
"LoadCase":100	Only for loadcase number 100

4. Table

A **table** is a Word *.docx file with user defined layout and content based on predefined keyword and structure.

A **table** can define several keyword content.

The layout of a keyword content is :

```
[KEYWORD]
[HEADER]
[CONTENT]
[FOOTER]
[END]
```

The table's **[KEYWORD]** corresponds to the template's **\$\$KEYWORD\$\$**.

Example for **\$\$NODES\$\$** :

```
[NODES]
[HEADER]
```

NODES

Name	Global coordinates [m]			Joint type	Mismatch [mm]	Fillet length [mm]	T _{max} [mm]
	X	Y	Z				
[CONTENT]							
\PT\	\X\	\Y\	\Z\	\JT\	\MM\	\CX\	\TM\
[FOOTER]							

Joint type:

All	Steel	Composite	HDPE
0 None	1 Butt weld flush	101 Adhesive bonded	201 Concentric fabricated reducer
	2 Butt weld as welded	102 Adhesive bonded with overlay	202 Thrust collar
	3 Fillet weld	103 Gasket with overlay	203 Electrofusion coupling
	4 Capped end	104 Butt and strap	
	6 Full fillet weld		
	11 Tapered flush		
	12 Tapered as welded		
	13 Threaded		
	14 Brazed		
	15 One third slope flush		
	16 One third slope as welded		
	18 Lap flange		
	19 Double weld slip on flange		
	20 Single weld slip on flange		

[END]

Between the [HEADER] section and the [CONTENT] section, user can define an introduction of the table, a title and table headers and units.

Between the [CONTENT] section and the [FOOTER] section, user can define a one line table (aligned with the header). On each cell, user can define **properties**. The software will repeat this row with every item of this type and fill the properties.

Between the [FOOTER] section and the [END] section, user can define a conclusion of the table or some remarks.

4.1 Properties

Depending of the type of the keyword, properties are recoverable.

A property is defined between 2 backslashes \XX\

4.1.1 Pipe section

Property	Description	Unit Metric	Unit USA
\CD\	Name	-	-
\AL\	Description	-	-
\OD\	Diameter	mm	in
\ID\	Inside diameter	mm	in
\WT\	Thickness	mm	in
\IN\	Insulation thickness	mm	in
\SO\	Operating density	-	-
\ST\	Test density	-	-
\MA\	Linear mass	kg/m	lb/ft
\CO\	Corrosion	mm	in
\CI\	Erosion	mm	in
\OV\	Ovalization ratio	-	-
\LN\	Liner thickness	mm	in
\TC\	TopCoat thickness	mm	in
\EM\	Special thickness	mm	in
\BT	Bend thickness	mm	in

4.1.2 Beam section

Property	Description	Unit Metric	Unit USA
\CD\	Name	-	-
\AL\	Description	-	-
\H\	Height	mm	in
\B\	Basis	mm	in
\TW\	Web thickness	mm	in
\TF\	Flange thickness	mm	in
\IX\	Bending inertia X	cm^4	in^4
\IY\	Bending inertia Y	cm^4	in^4
\IT\	Torsional inertia	cm^4	in^4
\A\	Area	cm^2	in^2
\AX\	Reduced area X	cm^2	in^2
\AY\	Reduced area Y	cm^2	in^2

4.1.3 Regular material

Property	Description	Unit Metric	Unit USA
\CD\	Name	-	-
\AL\	Description	-	-
\SG\	Poisson's ratio	-	-
\TX\	Temperature max	°C	°F
\TA\	Temperature ref	°C	°F
\RO\	Density	kg/m³	lb/ft³
\TE\	Temperature i	°C	°F
\EX\	Thermal Expansion	10^-6.mm/mm/°C	10^-6.in/in/°F
\EH\	Modulus of Elasticity	kN/mm²	10^6.psi
\SH\	Non-Class 1 Allowable Stress	N/mm²	ksi
\SY\	Yield Stress	N/mm²	ksi
\SU\	Ultimate Tensile Stress	N/mm²	ksi
\SM\	Class 1 Allowable Stress	N/mm²	ksi
\CR\	Creep	N/mm²	ksi
\GH\	Shear Modulus	kN/mm²	10^6.psi
\CO\	Class 1 Thermal Conductivity	kJ/hr/m/°C	btu/hr/ft/°F
\DI\	Class 1 Thermal Diffusivity	mm²/s	ft²/hr
\E2\	Modulus of Elasticity 2	kN/mm²	10^6.psi

4.1.4 Node

Property	Description	Unit Metric	Unit USA
\PT\	Name	-	-
\X\	X global coordinate	m	ft
\Y\	Y global coordinate	m	ft
\Z\	Z global coordinate	m	ft
\JT\	Joint type (see below)	-	-
\MM\	Mismatch for welded joints	mm	in
\CX\	Length of fillet weld Cx	mm	in
\TM\	for Class 1 transition within 1:3 slope envelope	mm	in

Joint type values :

```

None = 0
// For steel
ButtWeldFlush = 1
ButtWeldAsWelded = 2
FilletWeld = 3
CappedEnd = 4
FullFilletWeld = 6
TaperedFlush = 11
TaperedAsWelded = 12
Threaded = 13
Brazed = 14
OneThirdSlopeFlush = 15
OneThirdSlopeAsWelded = 16
LapFlange = 18
DoubleWeldSlipOnFlange = 19
SingleWeldSlipOnFlange = 20
// For composite
AdhesiveBonded = 101
AdhesiveBondedWithOverlay = 102
GasketWithOverlay = 103
ButtAndStrap = 104
// For HDPE
ConcentricFabricatedReducer = 201
ThrustCollar = 202
ElectrofusionCoupling = 203

```

4.1.5 Element

Property	Description	Unit Metric	Unit USA
\TY\	Type	-	-
\PT1\	Node1 name	-	-
\PT2\	Node2 name	-	-
\LE\	Length	m	ft
\RA\	Bend radius (only for bend)	m	ft
\MA\	Material description	-	-
\AL\	Label	-	-
\XX\	X-axis direction on X	-	-
\XY\	X-axis direction on Y	-	-
\XZ\	X-axis direction on Z	-	-
\CS\	Section description	-	-
\CL\	Piping code (empty for beam)	-	-

Special properties for beam :

Property	Description	Unit Metric	Unit USA
\J1\	Joint type on Node 1 (see below)	-	-
\J2\	Joint type on Node 2 (see below)	-	-
\BX\	Buckling on X	-	-
\BY\	Buckling on Y	-	-
\BZ\	Buckling on Z	-	-
\LTB\	Lateral Torsional Buckling (see below)	-	-

Joint type values :

```
None = 0
Joint = 1
BoltedJoint = 2
WeldedJoint = 3
```

Lateral Torsional Buckling values :

```
None = 0
BothHingedUniform = 1
BothHingedNodal = 2
BothFixedUniform = 3
BothFixedNodal = 4
Moments = 5
CantileverUniform = 6
CantileverNodal = 7
```

4.1.6 DLCS

Property	Description	Unit Metric	Unit USA
\PT\	Node name	-	-
\XX\	X-axis direction on X	-	-
\XY\	X-axis direction on Y	-	-
\XZ\	X-axis direction on Z	-	-
\ZX\	Z-axis direction on X	-	-
\ZY\	Z-axis direction on Y	-	-
\ZZ\	Z-axis direction on Z	-	-

4.1.7 Lumped mass

Property	Description	Unit Metric	Unit USA
\PT\	Node name	-	-
\X\	X global coordinate	m	ft
\Y\	Y global coordinate	m	ft
\Z\	Z global coordinate	m	ft
\MA\	Mass	ton	kips

4.1.8 Restraint

Property	Description	Unit Metric	Unit USA
\PT\	Node name	-	-
\AL\	Restraint type	-	-
\LO\	Coordinate system (see below)	-	-
\LV\	Level	-	-
\DESCR\	Label	-	-
\SX\	Translation stiffness on X	kN/mm	Kips/in
\SY\	Translation stiffness on Y	kN/mm	Kips/in
\SZ\	Translation stiffness on Z	kN/mm	Kips/in
\SPX\	Rotation stiffness on X	kN.m/rad	Kips.ft/rad
\SPY\	Rotation stiffness on Y	kN.m/rad	Kips.ft/rad
\SPZ\	Translation stiffness on Z	kN.m/rad	Kips.ft/rad
\FO\	Pre-compression/pre-tension force	kN	Kips

Coordinate system values :

```

G = Global
L = Local to attached element
N = Local to node's DLCS

```

4.1.9 Static case : Forces & moments on node

Property	Description	Unit Metric	Unit USA
\PT\	Node name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\FX\	Force on X	kN	kips
\FY\	Force on Y	kN	kips
\FZ\	Force on Z	kN	kips
\MX\	Moment on X	kN.m	kips.ft
\MY\	Moment on Y	kN.m	kips.ft
\MZ\	Moment on Z	kN.m	kips.ft
\LO\	Local	-	-

4.1.10 Static case : Restraints movement

Property	Description	Unit Metric	Unit USA
\PT\	Node name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\DX\	Displacement on X	mm	in
\DY\	Displacement on Y	mm	in
\DZ\	Displacement on Z	mm	in
\RX\	Rotation on X	rad	rad
\RY\	Rotation on Y	rad	rad
\RZ\	Rotation on Z	rad	rad

4.1.11 Static case : Distributed loads

Property	Description	Unit Metric	Unit USA
\TY\	Type	-	-
\PT1\	Node1 name	-	-
\PT2\	Node2 name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\X\	Load on X	N/m	lb/ft
\Y\	Load on Y	N/m	lb/ft
\Z\	Load on Z	N/m	lb/ft

4.1.12 Static case : Wind loads

Property	Description	Unit Metric	Unit USA
\TY\	Type	-	-
\PT1\	Node1 name	-	-
\PT2\	Node2 name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\PR\	Pressure	N/m ²	lb/ft ²
\X\	Wind direction on X	-	-
\Y\	Wind direction on Y	-	-
\Z\	Wind direction on Z	-	-

4.1.13 Static case : Snow loads

Property	Description	Unit Metric	Unit USA
\TY\	Type	-	-
\PT1\	Node1 name	-	-
\PT2\	Node2 name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\PR\	Pressure	N/m ²	lb/ft ²

4.1.14 Static case : Operating conditions

Property	Description	Unit Metric	Unit USA
\TY\	Type	-	-
\PT1\	Node1 name	-	-
\PT2\	Node2 name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\TE\	Temperature	°C	°F
\PR\	Pressure	N/m ²	lb/ft ²

4.1.15 Static case : Densities

Property	Description	Unit Metric	Unit USA
\TY\	Type	-	-
\PT1\	Node1 name	-	-
\PT2\	Node2 name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\RO\	Density	-	-

4.1.16 Static case : Stratifications

Property	Description	Unit Metric	Unit USA
\TY\	Type	-	-
\PT1\	Node1 name	-	-
\PT2\	Node2 name	-	-
\CA\	Load number	-	-
\ME\	Stratification method (see below)	-	-
\AL\	Mean coef. of thermal expansion of pipe	10^-6mm/mm/°C	10^-6in/in/°F
\T1\	Temperature at bottom of the pipe or temp. gradient Gu	°C or °C/mm	°F or °F/in
\T2\	Temperature at top of the pipe or temp. gradient Gv	°C or °C/mm	°F or °F/in
\V1\	Vertical coordinate of the discontinuity or laminar flow down	mm	in
\V2\	Vertical coordinate of the laminar flow up	mm	in
\X\	X component of the local V vector	-	-
\Y\	Y component of the local V vector	-	-
\Z\	Z component of the local V vector	-	-

Stratification method values :

```

HorizontalLinear = 0
HorizontalStep = 1
HorizontalLaminar = 2
HorizontalGu = 3
Skewed = 4
General = 5

```

4.1.17 Static case : Cold springs

Property	Description	Unit Metric	Unit USA
\TY\	Type	-	-
\PT1\	Node1 name	-	-
\PT2\	Node2 name	-	-
\CA\	Load number	-	-
\LE\	Length change	mm	in

4.1.18 Static case : Accelerations

Property	Description	Unit Metric	Unit USA
\TY\	Type	-	-
\PT1\	Node1 name	-	-
\PT2\	Node2 name	-	-
\CA\	Load number	-	-
\T\	Load title	-	-
\GX\	Fraction of G on X	-	-
\GY\	Fraction of G on Y	-	-
\GZ\	Fraction of G on Z	-	-

4.1.19 Static case

Property	Description	Unit Metric	Unit USA
\CA\	Load number	-	-
\TI\	Load title	-	-
\TY\	Load category (see below)	-	-
\LV\	Level	-	-
\EQ\	Equation (see below)	-	-
\CF\	Check fasteners (structure only)	-	-

Category values :

```

OperatingWeight = 0
TestWeight = 1
EmptyWeight = 2
DesignWeight = 3
Wind = 4
Snow = 5
Distributed = 6
Acceleration = 7
SAM = 8
Thermal = 9
Settlement = 10
ColdSpring = 11
Dummy = 12
UserDefined = 13

```

Equation values :

```

None (no stress calculation) = 1
Equation 6 (sustained) = 2
Equation 10 B (occasional) = 3
Equation 10 C (occasional) = 4
Equation 10 D (occasional) = 5
Equation 10 T (test) = 6
Equation 7 (thermal expansion) = 7
Equation 8 (thermal expansion + sustained) = 8
Equation 9 (settlement) = s

```

4.1.20 Time history event : Dynamic forces & moments on node

Property	Description	Unit Metric	Unit USA
\PT\	Node name	-	-
\TI\	Event title	-	-
\FX\	Force on X	kN	kips
\FY\	Force on Y	kN	kips
\FZ\	Force on Z	kN	kips
\MX\	Moment on X	kN.m	kips.ft
\MY\	Moment on Y	kN.m	kips.ft
\MZ\	Moment on Z	kN.m	kips.ft

4.1.21 Time history cases

Property	Description	Unit Metric	Unit USA
\CA\	Load number	-	-
\TI\	Load title	-	-
\LV\	Level	-	-
\EQ\	Equation (see above)	-	-
\EV\	Event number	-	-
\DA\	Damping value	-	-
\RC\	Rigid correction	-	-
\RM\	Resultant moment	-	-
\ES\	Explicit scheme	-	-
\TS\	Time step	-	-
\AR\	Archive rate	-	-
\CF\	Check fasteners (structure only)	-	-

4.1.22 Class 1 thermal case : Transients

Property	Description	Unit Metric	Unit USA
\TY\	Type	-	-
\PT1\	Node1 name	-	-
\PT2\	Node2 name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\TR\	Transient name	-	-

4.1.23 Class 1 case : Load sets

Property	Description	Unit Metric	Unit USA
\TI\	Load title	-	-
\RF\	Reference case number	-	-
\SI\	Situation number	-	-
\TR\	Thermal case number	-	-
\CY\	Number of cycles	-	-
\PR\	Pressure case number	-	-
\MO\	Moment case number	-	-
\DY\	Dynamic load flag	-	-

4.1.24 Spectrum

Property	Description	Unit Metric	Unit USA
\EV\	Event number	-	-
\TI\	Load title	-	-
\NA\	Spectrum name	-	-
\XA\	Period or frequency	-	-
\ME\	Interpolation method (see below)	-	-
\SH\	Shift value	%	%
\DA\	Damping value	-	-
\DU\	Duration value	s	s

Interpolation method values :

```

Linear / Linear acceleration = 0
Linear frequency / Linear acceleration = 1
Linear period / Linear acceleration = 2
Log frequency / Linear acceleration = 3
Log frequency / Log acceleration = 4

```

4.1.25 Primary floor response cases

Property	Description	Unit Metric	Unit USA
\CA\	Load number	-	-
\TI\	Load title	-	-
\LV\	Level	-	-
\EV\	Event number	-	-
\EQ\	Equation (see above)	-	-
\IM\	Inter modal combination (see below)	-	-
\IL\	Inter level combination (see below)	-	-
\OR\	Order (see below)	-	-
\RC\	Rigid correction (see below)	-	-
\X\	X factor	-	-
\Y\	Y factor	-	-
\Z\	Z factor	-	-
\CF\	Check fasteners (structure only)	-	-

Inter modal combination values :

```

Grouping = 1
Ten percent = 2
Double sum = 3
SRSS = 4
All coupling = 5
Rosenblueth = 6
Der Kiureghian = 7

```

Inter level combination values :

```

Absolute without phase = 0
SRSS without phase = 1
Algebraic = 2
Absolute with phase = 3
SRSS with phase = 4
Envelope = 5
SRSS with counterphase = 6

```

Order values :

```

Interlevel / Intermodal / Interspatial = 0
Interlevel / Interspatial / Intermodal = 1
Intermodal / Interlevel / Interspatial = 2

```

Rigid correction values :

```

SRSS = 0
None = 1
Absolute = 2
SRSS with modal = 3
Gupta = 4
Lindley-Yow = 5

```

4.1.26 Secondary floor response cases

Property	Description	Unit Metric	Unit USA
\CA\	Load number	-	-
\TI\	Load title	-	-
\LV\	Level	-	-
\EV\	Event number	-	-
\EQ\	Equation (see above)	-	-
\ME\	Method of combination (see below)	-	-
\TP\	Treat as primary	-	-
\X\	X factor	-	-
\Y\	Y factor	-	-
\Z\	Z factor	-	-
\CF\	Check fasteners (structure only)	-	-

Combination method values :

```

Absolute = 0
SRSS = 1

```

4.1.27 Combination cases

Property	Description	Unit Metric	Unit USA
\CA\	Load number	-	-
\TI\	Load title	-	-
\RF\	Reference case number	-	-
\LV\	Level	-	-
\EQ\	Equation (see above)	-	-
\ME\	Method of combination (see below)	-	-
\DY\	Dynamic load flag	-	-
\CY\	Number of cycles	-	-
\TY\	Analysis type	-	-
\C1\	factor(i) x case(i)	-	-
\CF\	Check fasteners (structure only)	-	-

Combination method values :

```

Algebraic = 0
Absolute = 1
SRSS = 2
Seismic = 3
MaxAbsolute = 4
MaxResultant = 5
MaxAlgebraic = 6
MinAlgebraic = 7
Range = 8
MomentRange = 9
StressRange = 10

```

4.1.28 Combination stress cases

Property	Description	Unit Metric	Unit USA
\CA\	Load number	-	-
\TI\	Load title	-	-
\RF\	Reference case number	-	-
\LV\	Level	-	-
\EQ\	Equation (see above)	-	-
\ME\	Method of combination (see above)	-	-
\MU\	Allowable multiplier	-	-
\MA\	factor x sustained load case	-	-
\MB\	factor x occasional load case	-	-
\MC\	factor x thermal load case	-	-

4.1.29 External cases

Property	Description
\CA\	Load number
\TI\	Load title
\SN\	External study name
\LC\	External study load case number

4.1.30 Results : Displacements

Property	Description	Unit Metric	Unit USA
\PT\	Node name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\DX\	X global displacement	mm	in
\DY\	Y global displacement	mm	in
\DZ\	Z global displacement	mm	in
\RX\	X global rotation	Rad	Rad
\RY\	Y global rotation	Rad	Rad
\RZ\	Z global rotation	Rad	Rad

4.1.31 Results : Forces & moments on element's nodes

Property	Description	Unit Metric	Unit USA
\TY\	Type	-	-
\PT\	Node extremity name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\FX\	Force on X	N	lb
\FY\	Force on Y	N	lb
\FZ\	Force on Z	N	lb
\MX\	Moment on X	N.m	lb.ft
\MY\	Moment on Y	N.m	lb.ft
\MZ\	Moment on Z	N.m	lb.ft

4.1.32 Results : Reactions on restraint's node

Property	Description	Unit Metric	Unit USA
\PT\	Node name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\FX\	Force on X	N	lb
\FY\	Force on Y	N	lb
\FZ\	Force on Z	N	lb
\MX\	Moment on X	N.m	lb.ft
\MY\	Moment on Y	N.m	lb.ft
\MZ\	Moment on Z	N.m	lb.ft

4.1.33 Results : Stresses on elements

Property	Description	Unit Metric	Unit USA
\TY\	Type	-	-
\PT1\	Node 1 name	-	-
\PT2\	Node 2 name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\ST\	Stress max	N/mm ²	ksi

4.1.34 Results : Anchor plate max ratios

Property	Description	Unit Metric	Unit USA
\PT\	Node name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\RA\	Ratio max	-	-

4.1.35 Results : Beam joint ratios

Property	Description	Unit Metric	Unit USA
\TY\	Type	-	-
\PT\	Node name	-	-
\CA\	Load number	-	-
\TI\	Load title	-	-
\RA\	Ratio max	-	-

4.1.36 Results : Finite element analysis fastener report

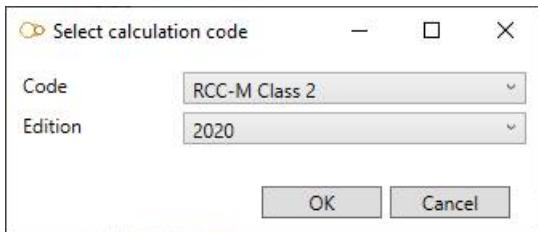
Property	Description	Unit Metric	Unit USA
\LINE\	Report line(i)	-	-

4.1.37 Results : Finite element analysis node

Property	Description	Unit Metric	Unit USA
\ID\	Node id	-	-
\X\	X coordinate	mm	in
\Y\	Y coordinate	mm	in
\Z\	Z coordinate	mm	in
\UX\	X displacement	mm	in
\UY\	Y displacement	mm	in
\UZ\	Z displacement	mm	in
\STRESS\	Stress	N/mm ²	ksi
\STRAIN\	Strain	%	%
\CO\	Compression	N/mm ²	ksi

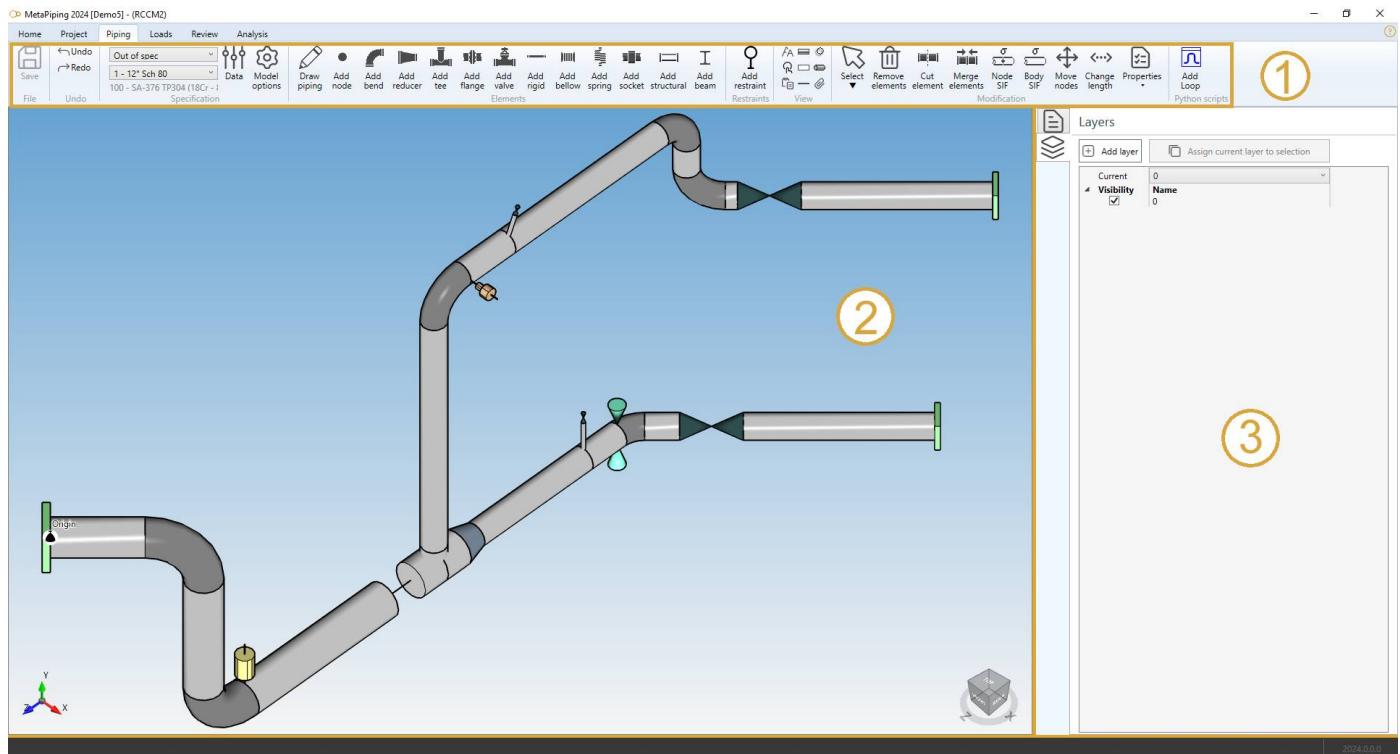
Piping

After creating a new **piping study** and pressing the **Edit** button, a window appears to define the calculation code :



Set the **Code** and **Edition** and press **OK** : new tabs appear in the ribbon menu on top of the application :

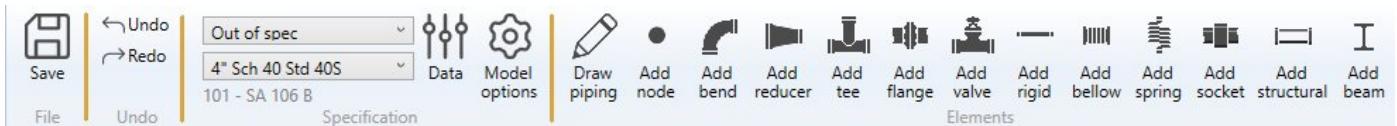
- Piping
- Load - click [here](#) to have more information about loading
- Review - click [here](#) to have more information about review tools
- Analysis - click [here](#) to have more information about analysis



The **Piping** screen consists of a ribbon menu (1), a 3D visualization engine (2) and a data panel on the right (3).

The **Data Panel** can be opened/closed with the shortcut **F2**

1. Ribbon menu



1.1 File

During the design of the model, every command is automatically saved in a temporary file (conception.~metaL).

You can decide to save the last modifications or cancel it and return to the study screen.

A floppy disk on the left side of the ribbon indicates the state of the file on disk :



Save means that everything has been saved.



Save means that something has changed and the MetaL needs to be saved.

WARNING

If the application crashes, MetaPiping will prompt to reopen the last modifications

1.2 Undo/Redo



Every command in MetaPiping is stored in a **command list**.

You can *navigate* through this list by pressing the **Undo/Redo** buttons.

1.3 Specification



This group of controls lets you define the current **Section/Material** for the next elements.

You have an access to a window that defines the piping sections and materials for the current model.

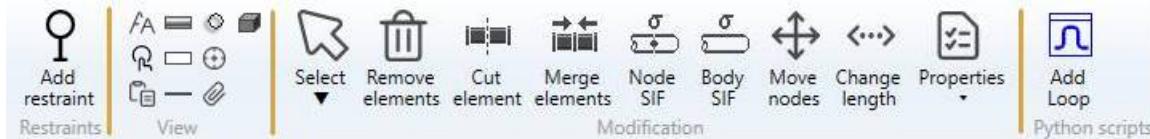
You have an access to the model options (code, edition, calculation options...).

Click [here](#) to have more information about the **Data** button and the **Model options** button.

1.4 Elements



Click [here](#) to have more information about piping element creation.



1.5 Restraints

Click [here](#) to have more information about restraints.

1.6 View options

show/hide the node name and valve/support label

show/hide the node point

copy the 3D view to the clipboard

show the piping in shaded mode

show the piping in hidden lines mode

- show the piping in wireframe mode
- show/hide the shadows
- show/hide the origin symbol
- show/hide the linked study models
- show/hide the concrete (for MetaStructure only)

1.7 Modification tools

Click [here](#) to have more information about all modification tools.

1.8 Python scripts

MetaPiping lets you create your own **tools**.

If a tool is missing, user can create his own **command** that can interact with the user and modify the selected elements or the whole model.

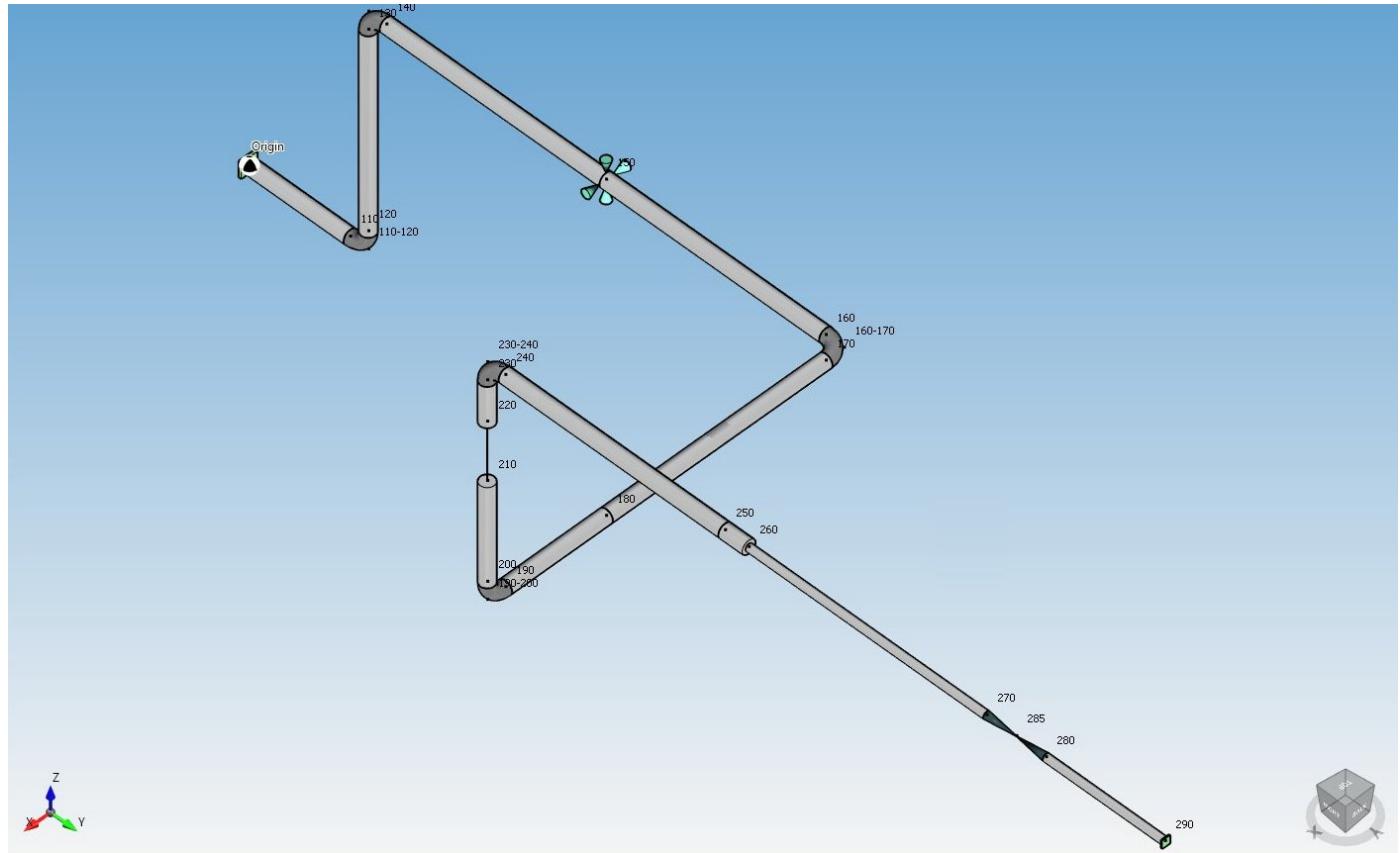
MetaPiping will take care of the Undo/Redo mechanism.

Click [here](#) to have more information about python script creation.

[See an example of a Design script](#) : Replace a selected pipe by a loop

2. 3D Visualization

MetaPiping is equipped with a powerful 3D engine :



2.1 Navigation

You navigate inside the view with the **middle button** of the mouse pressed :



hold down the **CTRL key** during navigation to translate the view.

You zoom in by scrolling the mouse wheel up :



You zoom out by scrolling the mouse wheel down :

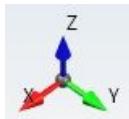


You zoom all by **double click** the mouse wheel button :



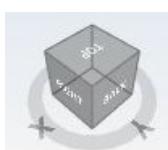
2.2 Axis

The **global axis** are shown in the bottom left corner of the screen :



2.3 Predefined views

An interactive **cube** with predefined views is shown in the bottom right corner of the screen :



- Click on top corners of the cube to change to predefined axonometric view.
- Click twice on the faces of the cube to change to 2D views.

3. Data panel

The **Data Panel** can be opened/closed with the shortcut **F2**

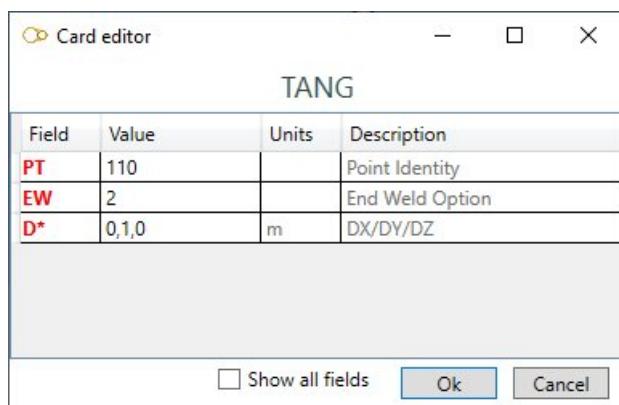
3.1 PIPESTRESS editor

The **Plugin PIPESTRESS** converts automatically every command of MetaPiping into HIGH FIDELITY **FRE** file format.

```
Refresh  Hide line numbers

57 MATD TE=21 EH=203.4 EX=10.08 SH=103.4 SM=137.9 SY=241.3
58 MATD TE=93 EH=198.6 EX=10.96 SH=103.4 SM=137.9 SY=219.9
59 MATD TE=149 EH=195.1 EX=11.57 SH=103.4 SM=137.9 SY=213.7
60 MATD TE=204 EH=191.0 EX=12.13 SH=103.4 SM=137.9 SY=206.8
61 MATD TE=260 EH=188.2 EX=12.71 SH=103.4 SM=130.3 SY=195.1
62 MATD TE=316 EH=184.1 EX=13.10 SH=103.4 SM=119.3 SY=178.6
63 MATD TE=371 EH=175.8 EX=13.52 SH=98.6 SM=115.8 SY=173.7
64
65 *INITIAL OPERATING CONDITIONS
66 ****
67 DESN TE=200 PR=1
68 OPER CA=100 PR=1 TE=200
69 OPER CA=101 TE=204 PR=1
70 OPER CA=102 TE=150 PR=1
71
72 *INITIAL MATERIAL
73 ****
74 MATL CD=101
75
76 *INITIAL CROSS SECTION
77 ****
78 CROS CD=1 OD=114.30 WT=6.02 SO=1 ST=0 MA=16.08 AL=/4" Sch 40 Std 40S/
79
80 *GEOMETRY
81 ****
82 *
83 *
84 ANCH PT=100
85 AMVT PT=100 CA=100 DX=2. DZ=1.5
86 TANG PT=110 EW=2 DY=1
87 BRAD PT=120 EW=2 RA=0.1524 *SD=15
88 TANG PT=130 EW=2 DZ=2
89 BRAD PT=140 EW=2 RA=0.1524 *SD=15
```

- Double click on a **card** shows a window with all fields :



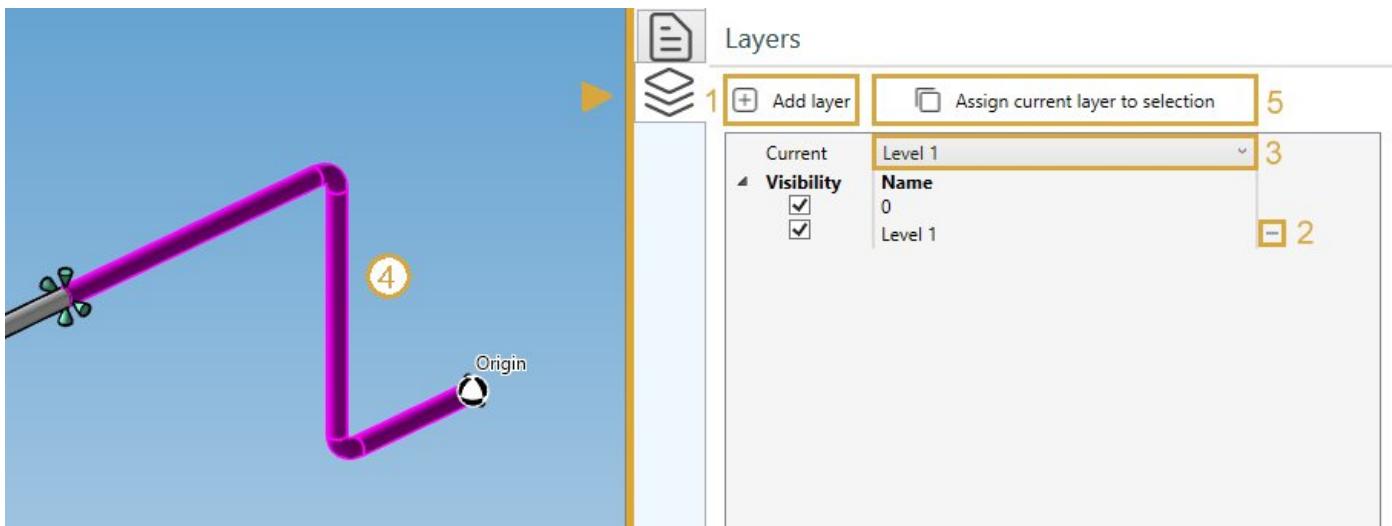
- Select a line and click **F9** on keyboard will select the object in the 3D view.
 - Select a line and click **F1** on keyboard will open PIPESTRESS help (if defined in the settings).
- Click [here](#) to have more information about the settings.
- You can also write/change the file by yourself in this editor. Press the **Refresh** button (or F6) on top to update the model with the content of the editor.

WARNING

Attention, this action is irreversible

3.2 Layers

MetaPiping let you simplify your visualization by using the **Layer** concept :



By default, all piping elements are assigned to **Layer 0**.

To create a new layer, click on **Add layer** button (1).

Give it a name, for example "Level 1".

The layer "Level 1" will appear in the list of layers and will become the current layer (3).

To modify the layer of objects, select elements and/or restraints (4) and click on **Assign current layer to selection** button (5).

If you decide to remove a layer, click on - button (2) next to the layer name. All elements and restraints of this layer will be assigned to the default **Layer 0**.

You can now show/hide elements and restraints by activating/deactivating the visibility buttons.

All layer manipulations support Undo/Redo mechanism.

TABLE OF CONTENTS

- [Units](#)
- [Specification](#)
- [Elements](#)
- [Restraints](#)
- [Modification](#)
- [Selection](#)

Units

The main units are given here below.

1. Input units

Property	Unit Metric	Unit USA
Element length	m	ft
Thickness, diameter, etc.	mm	in
Temperature	°C	°F
Pressure	MPa	psi
Mass	Ton	Kips
Force	kN	Kips
Moment	kN.m	Kips.ft
Applied displacement	mm	in
Applied rotation	rad	rad
Translational stiffness	kN/mm	Kips/in
Rotational stiffness	kN.m/rad	Kips.ft/rad
Acceleration	g's	g's
Stress	MPa	ksi
Young modulus	10^3 MPa	10^3 ksi
Thermal expansion coefficient	10^-6 mm/mm/°C	10^-6 in/in/°F

2. Output units

Property	Unit Metric	Unit USA
Displacement	mm	in
Rotation	mrad	mrad
Force	N	lbf
Moment	N.m	lbf.ft
Stress	MPa	ksi

Specification



The specification tool lets you define :

- the current "specification"
- the current section

By default there is no predefined specification. Everything you create will be "Out of spec".

You can define one or more **specification** by clicking the **Data** button.

The goal of a specification is to supervise the engineer's work by predefining all piping elements, sections, materials, bend radius, reducer lengths, tees...

Click [here](#) for more information about creation of specification.

The possible sections must be defined in the **Data** window. You can there create all materials and all piping sections or, selecting predefined **specifications**.

TABLE OF CONTENTS

- [Data](#)
- [Model options](#)

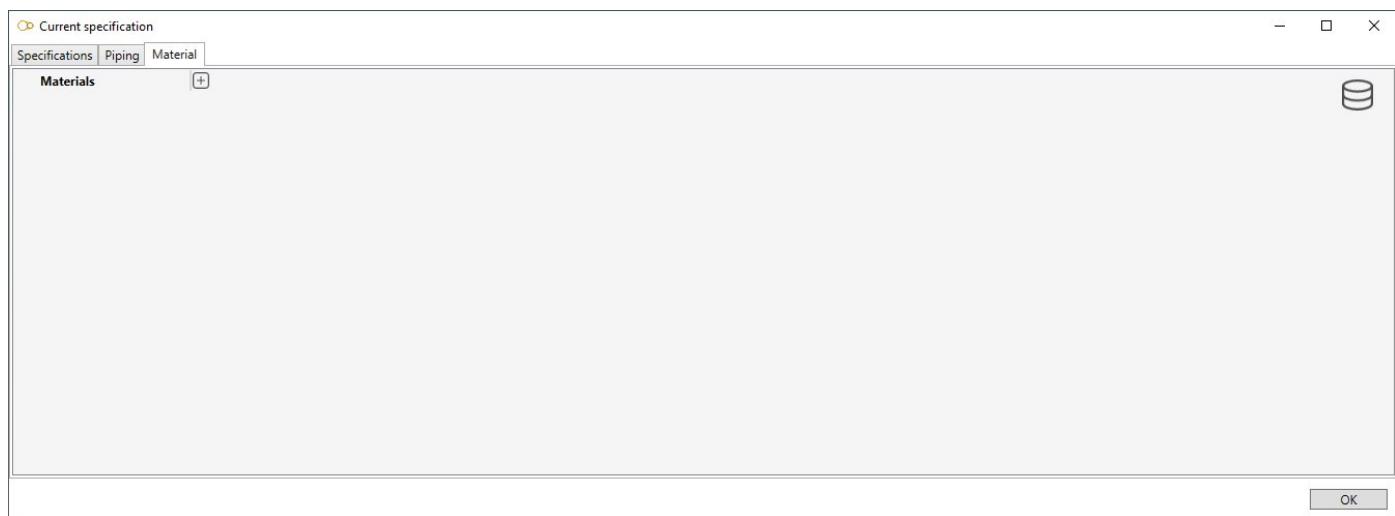
Data

The **Data** window lets you create all materials and all piping sections of the model.

It also lets you select one or more predefined **specifications**.

1. New project

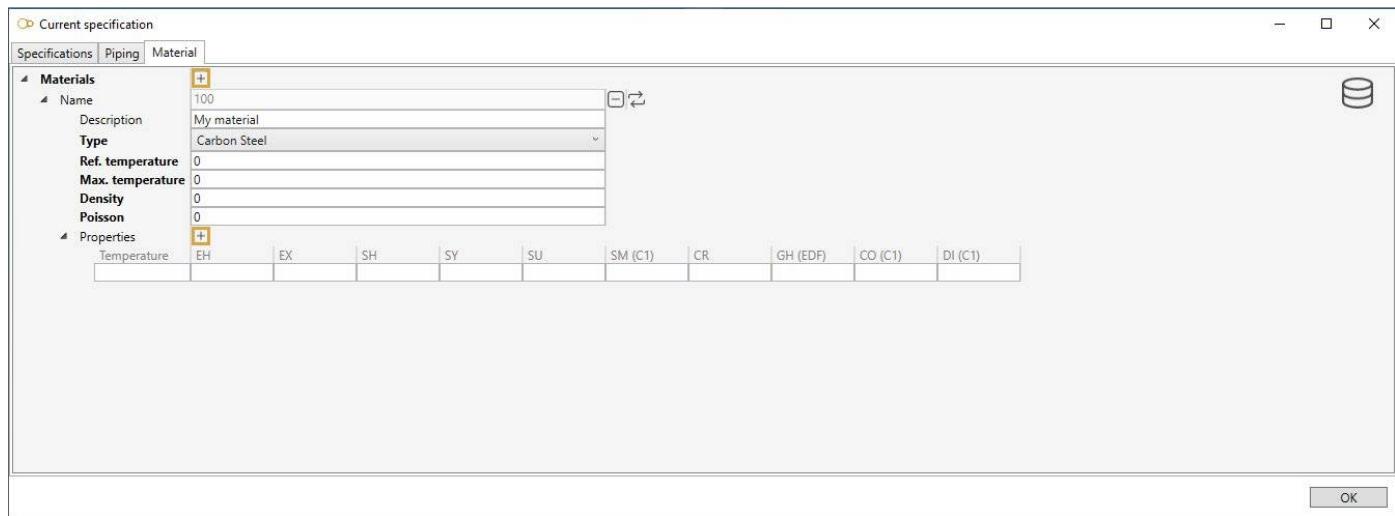
When starting a new model, the **Data** window opens :



1.1 Material definition

Let's define a new material.

Click on the + button and enter the description of the new material (Ex : My material).



A new name has been created for this material (Ex : 100).

Click the - button (next to Name) to remove it.

Define the description, the type, the temperature max, the density and the Poisson's ratio.

Click the + button (next to Properties) to add a new row of properties for a **specific temperature** or click the - button (end of row) to remove one.

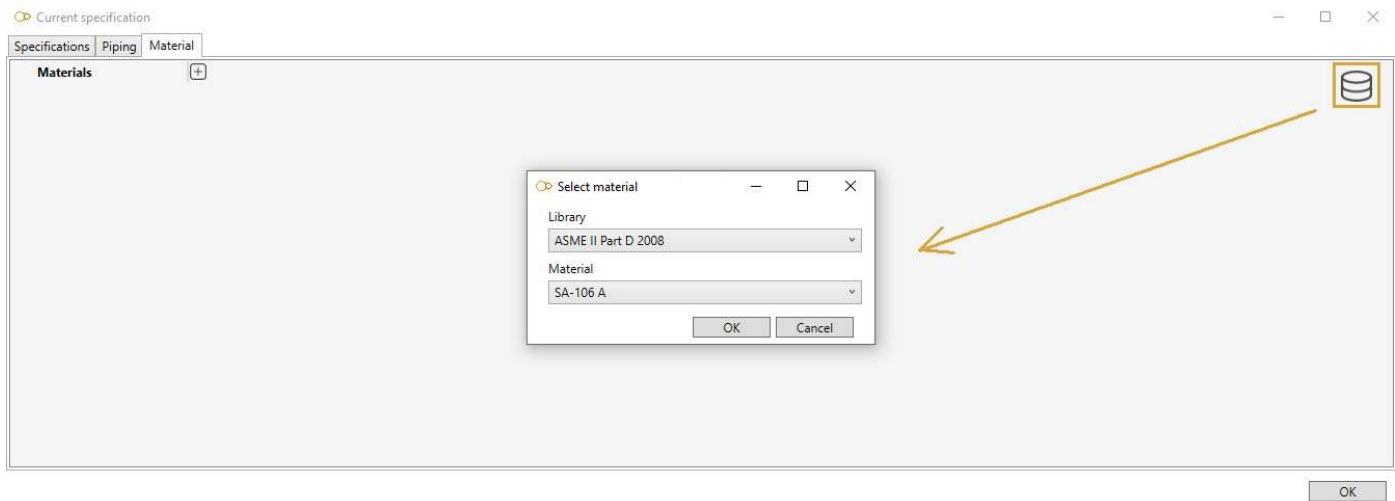
Property	Description	Unit Metric	Unit USA
Max. temperature	Temperature max	°C	°F
Density	Density	kg/m ³	lb/ft ³
Poisson	Poisson's ratio	-	-
EH	Modulus of Elasticity	kN/mm ²	10 ⁶ .psi
EX	Thermal Expansion	10 ⁻⁶ .mm/mm/°C	10 ⁻⁶ .in/in/°F
SH	Non-Class 1 Allowable Stress	N/mm ²	ksi
SY	Yield Stress	N/mm ²	ksi
SU	Ultimate Tensile Stress	N/mm ²	ksi
SM	Class 1 Allowable Stress	N/mm ²	ksi
CR	Creep	N/mm ²	ksi
GH	Shear Modulus	kN/mm ²	10 ⁶ .psi
CO	Class 1 Thermal Conductivity	kJ/hr/m/°C	btu/hr/ft/°F
DI	Class 1 Thermal Diffusivity	mm ² /s	ft ² /hr
E2	Modulus of Elasticity 2	kN/mm ²	10 ⁶ .psi

See [Units](#) for more information.

WARNING

ATTENTION, temperatures must be entered in ascending order!

To save time, you can also directly select a material from **database** :



Click on the **Database** button, select a **Library** and a **Material**. The tables will be filled automatically :

Current specification

Specifications Piping Material

Materials

Name	100
Description	SA-106 A
Type	Carbon Steel
Ref. temperature	21.1
Max. temperature	371
Density	7850
Poisson	0.3

Properties

Temperature	EH	EX	SH	SY	SU	SM (C1)	CR	GH (EDF)	CO (C1)	DI (C1)
21	202.7	11.52	94.5	206.9		110.3				
65	200	11.88	94.5	194.4		110.3				
93	198.6	12.06	94.5	189.6		110.3				
149	195.1	12.42	94.5	182.7		110.3				
204	192.4	12.78	94.5	176.5		110.3				
260	188.2	13.14	94.5	168.2		110.3				
316	182.7	13.32	94.5	158.6		105.5				
343	179.3	13.5	94.5	153.1		100.7				
371	175.8	13.68	86.2	148.2		99.3				

OK

Only libraries with **Target = Piping** will be shown !

Click [here](#) for more information about creation of library of materials.

1.2 Piping definition

Current specification

Specifications Piping Material

Pipe sizes

Bends	+ 1
Pipe size	+ Long radius Small radius Mass
Reducers	+ []

2

OK

To add new pipe sizes, click on + button (1) and give it a name (Ex : 4").

Each size exists in different schedules. click on + button to add schedules :

Property	Unit Metric	Unit USA
Schedule	-	-
Outside diameter	mm	in
Thickness	mm	in
Linear mass	kg/m	lb/ft

To know the UNIT of a property, just let the mouse over the property name.

For example : mm for the Outside diameter

4"

Schedule	Outside diameter	Thickness	Linear mass	[+]
Sch 5S	114.3	2.11	5.84	[]
Sch 10-10S	114.3	3.05	8.37	[]
Sch 40-Std-40S	114.3	6.02	16.1	[]
Sch 80-XS-80S	114.3	8.56	22.3	[]
Sch 120	114.3	11.12	28.3	[]
Sch 160	114.3	13.49	33.5	[]
Sch XXS	114.3	17.12	41	[]

WARNING

ATTENTION, the name of the pipe size is very important (Ex : 4"). It will be used as a **reference** for all other piping elements.

WARNING

ATTENTION, the sizes must be defined in an **ascending** order.

To add **bend** properties for each pipe size, click on + button and select a pipe size :

Bends		+	Long radius	Small radius	Mass	-
↳ Pipe size						
1/2"	0.0191	m	0.0127	0		
3/4"	0.0286		0.0191	0		
1"	0.0381		0.0254	0		
1 1/4	0.0476		0.0318	0		
1 1/2	0.0572		0.0381	0		
2"	0.0762		0.0508	0		
2 1/2	0.0953		0.0635	0		
3"	0.1143		0.0762	0		
3 1/2	0.1334		0.0889	0		
4"	0.1524		0.1016	0		

Standard radius and mass can be entered :

Property	Unit Metric	Unit USA
Long radius	m	ft
Small radius	m	ft
Mass	ton	kips

WARNING

ATTENTION, for a better readability, we recommend to enter the sizes in **ascending** order.

To add **reducer** properties for each pipe size, click on + button and select a pipe size :

Reducers		+
↳ 3/4"		-
↳ 1"		-
↳ 1 1/4		-
Reduce to		
1/2"	0.0508	Mass
3/4"	0.0508	+
1"	0.0508	ton

Click on + button (next to Mass) to add all possible reduction from the current size to a **smaller** one.

Length and mass can be entered :

Property	Unit Metric	Unit USA
Length	m	ft
Mass	ton	kips

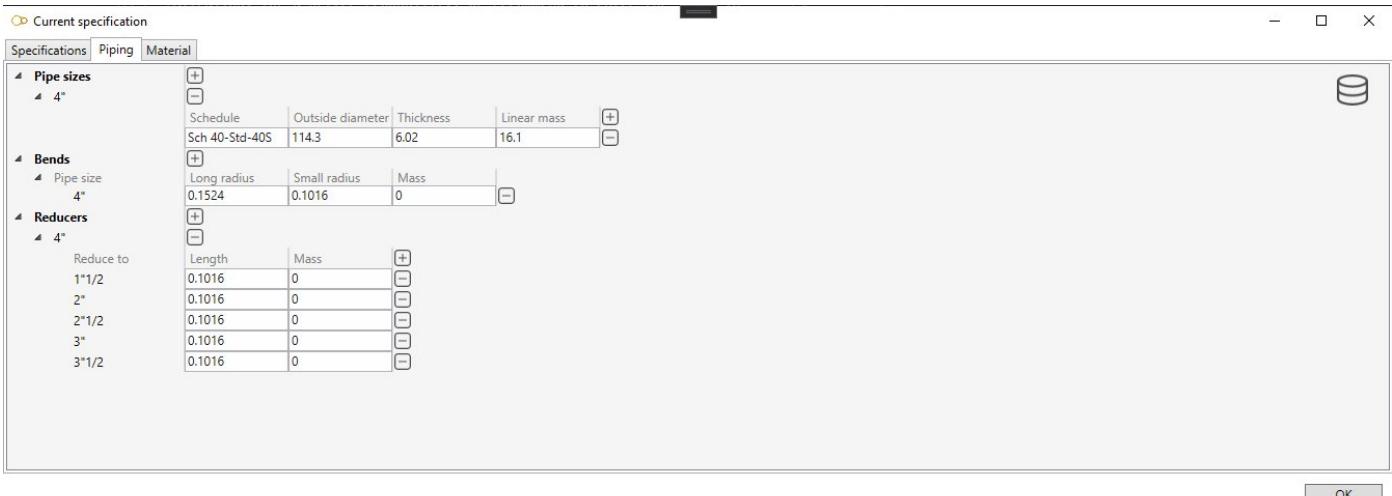
WARNING

ATTENTION, for a better readability, we recommend to enter the sizes in **ascending** order.

To save time, you can also directly select a pipe size from **database** (2) :



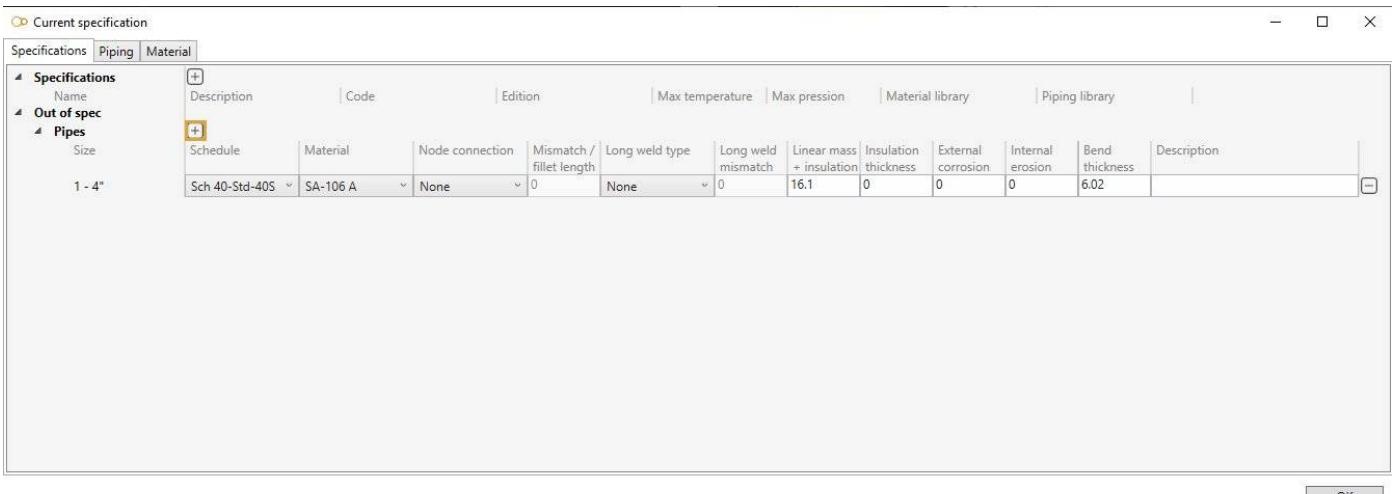
Select a **Library**, a **Pipe size** and a **Schedule**. The tables will be filled automatically :



Click [here](#) for more information about creation of library of piping elements.

Finally, you can define the **sections** of the model by defining pairs of Piping size/Material + special properties of the project.

Click to + button and select pipe sizes (Ex : 4") :



For each size, define the **schedule** and the **material**. A **pipe** knows now its section and its material.

Select then the **node connections** of the pipe between :

- None

- Butt weld - flush
- Butt weld - as welded
- Fillet weld
- Full fillet weld
- Threaded
- Brazed

Based on this property, define the **mismatch** or the **fillet length** [mm or in].

Select then the **Long weld type** of the pipe between :

- None
- Butt weld flush
- Butt weld as welded

Based on this property, define the **Long weld mismatch** [mm or in].

Other properties :

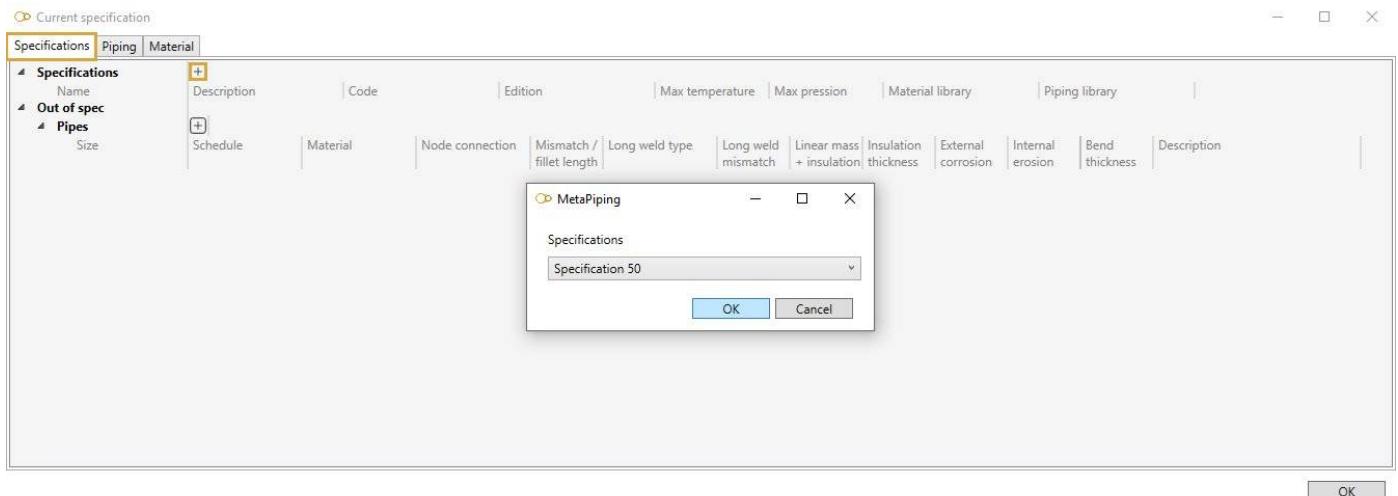
Property	Unit Metric	Unit USA	Default
Linear mass + insulation	kg/m	lb/ft	Linear mass of the pipe
Insulation thickness	mm	in	0
External corrosion	mm	in	0
Internal erosion	mm	in	0
Bend thickness	mm	in	Thickness of the pipe

1.3 Spec driven mode

Another way to define **Materials** and **Piping elements** is to use a predefined **Specification**.

The goal of a specification is to supervise the engineer's work by predefining all piping elements, sections, materials, bend radius, reducer lengths, tees...

Click to + button and select a specification :



Current specification

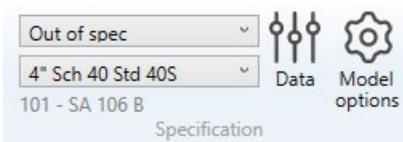
Specifications Piping Material

Specifications		Description	Code	Edition	Max temperature	Max pressure	Material library	Piping library				
Name	Specification 50	Spec 50	RCC-M Class 2 (French Nuc)	All	200	5	RCC-M ZI 2013	ANSI B16_9				
Out of spec												
Pipes		Schedule	Material	Node connection	Mismatch / fillet length	Long weld type mismatch	Linear mass + insulation	Insulation thickness	External corrosion	Internal erosion	Bend thickness	Description

OK

Click [here](#) for more information about creation of specification.

Now, you can use these 2 dropdown lists to select the current **section** :





Model options

1. General

Model options

General Analysis Modal extraction

Calc. code : RCC-M Class 2 (French Nuclear P)

Calc. edition : 2020

Gravity : -Y -Z

Input units : Metric USA USA2

Output units : Metric USA USA2

Plant :

Engineer :

OK Cancel

This tab lets you define the **calculation code** and its **Edition**, the direction of the **gravity**, the input and output **Units**, the **Plant** name and the **Engineer** name.

WARNING

ATTENTION, when starting a new project, be sure to select a specification that will be compatible with the Units, Code and Edition of the project and do not change these three properties during design.

2. Analysis

Model options

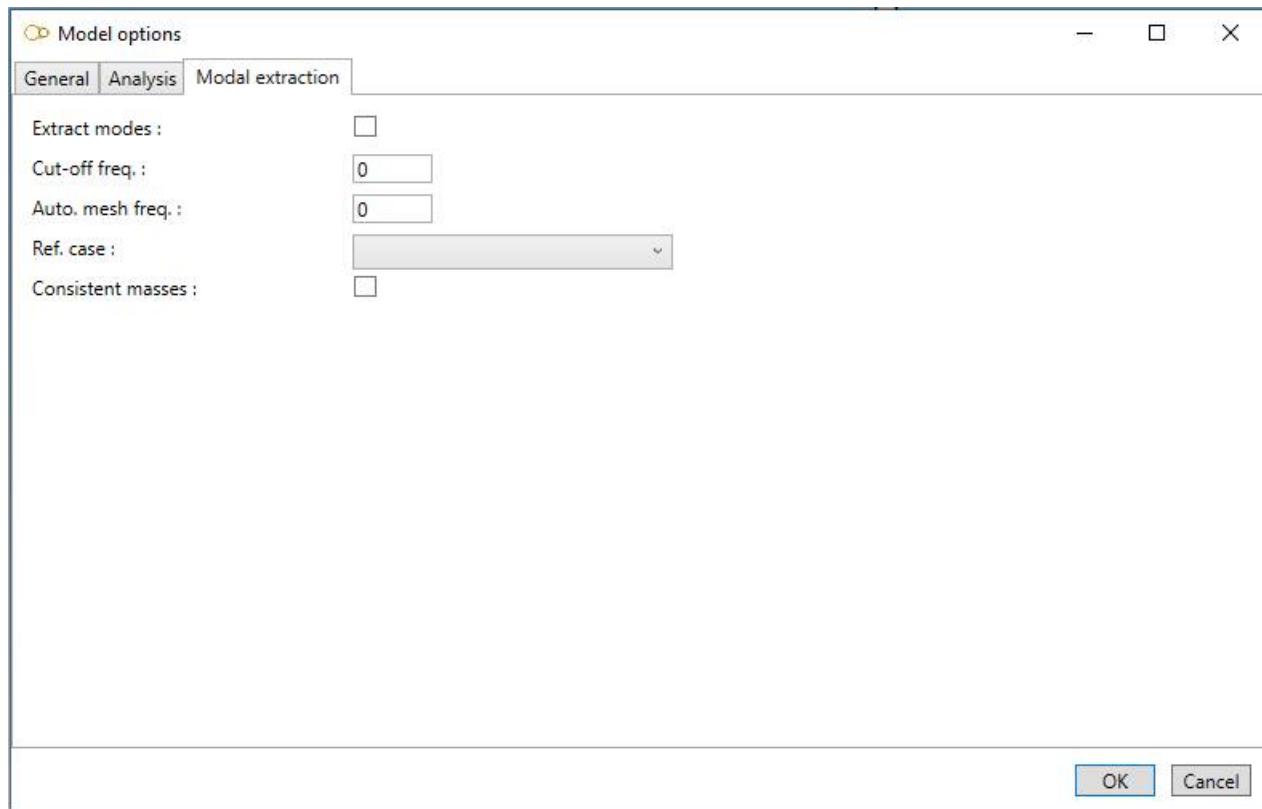
General Analysis Modal extraction

Def. room T° :	21.1
Def. oper. density :	<input type="text"/>
Def. test density :	<input type="text"/>
Def. design T° :	21.1
Def. design P :	5
Mass modeling :	Unif. stat. / Lumped dyn.
Hot modulus :	<input type="checkbox"/>
Hot allowable :	<input type="checkbox"/>
Pres. stiffening :	<input type="checkbox"/>
Pres. elongation :	<input type="checkbox"/>
PD/t pres. stress :	<input type="checkbox"/>
Branch forces at surface	<input type="checkbox"/>
3.Sm average value (Class 1)	<input type="checkbox"/>
Alternative Ke factor (Class 1)	<input type="checkbox"/>
Optimize residual moments (Class 1)	<input type="checkbox"/>
True transient range (Class 1)	<input type="checkbox"/>

OK **Cancel**

- **Def. room T°** : default room temperature (used to calculate thermal expansion)
- **Def. oper. density** and **Def. test density** : default operating and test density. These values can be superseded in **Data** for some cross sections or in the load case definition
- **Def. design T°** and **Def. design P** : default design temperature and pressure. For non-Class 1 piping codes, the allowable stress Sh is evaluated at the design temperature if the **Hot allowable** option is not checked
- **Mass modeling** :
 - Lumped stat. and dyn. : lumped mass for static analysis, lumped mass for dynamic analysis
 - Unif. stat. / Lumped dyn. : uniform mass for static analysis, lumped mass for dynamic analysis
 - Lumped stat. / Lumped dyn. + rot. : lumped mass for static analysis, lumped mass and rotational inertia for dynamic analysis
 - Unif. stat. / Lumped dyn. + rot. : uniform mass for static analysis, lumped mass and rotational inertia for dynamic analysis,
- **Hot modulus** : for most non-Class 1 piping codes, the Young modulus Ec at room temperature is used by default for all load cases. If **Hot modulus** is checked, the Young modulus Eh at the operating temperature is used instead. For Class 1 piping codes, the Young modulus Eh at the operating temperature is always used
- **Hot allowable** : for non-Class 1 piping codes, the allowable stress at design temperature is used by default for all load cases. If **Hot allowable** is checked, the allowable at the operating temperature is used instead
- **Pres. stiffening** : check this option to take into account the pressure stiffening effect on elbows and miter bends (if permitted by the piping code)
- **Pres. elongation** : if PIPESTRESS solver is used and this option is checked, the elongation due to internal pressure (also called "Bourdon effect") will be taken into account for all thermal expansion and test weight cases. If Code_Aster solver is used, this option has no effect, the option **Include Bourdon effect** in every load case definition is used instead
- **PD/t pres. stress** : the pressure stress is calculated with the formula $Pd^2/(D0^2 - d^2)$ by default. The simpler formula PD/t is used if this option is checked
- **Branch forces at surface** : check this option to use the forces and moments at run surface instead of intersection point for tees (if permitted by the piping code)
- **3.Sm average value (Class 1)** : check this option to use the average value of 3Sm between two load sets instead of the minimum value
- **Alternative Ke factor (Class 1)** : if checked, the penalty factor Ke.therm is used for austenitic stainless steels and Ni-Cr-Fe alloys (RCC-M code only)
- **Optimize residual moments (Class 1)** : contact CWANTIC for further explanations and theoretical background
- **True transient range (Class 1)** : by default, the range of the thermal gradient stress for a load set pair (LS1, LS2) is calculated assuming that the sign of the thermal stress is positive for heat-up transients and negative for cool-down transients. If this option is checked, the true range of the thermal gradient stress is calculated

3. Modal extraction



- **Extract modes** : check this option to extract the mode shapes
- **Cut-off freq.** : only the mode shapes below to the cut-off frequency are extracted
- **Auto. mesh freq.** : if different from 0, additional mass points will be generated to ensure that the mode shapes up to this frequency are accurate. This frequency should be equal or larger than the cut-off frequency
- **Ref. case** : if the **Hot modulus** option is checked, the Young modulus Eh used for the modal extraction is evaluated at the temperature of the reference case
- **Consistent masses** : if checked, a consistent mass matrix is used



Elements

Click on the links below :

TABLE OF CONTENTS

- [Node](#)
- [Pipe](#)
- [Bend](#)
- [Reducer](#)
- [Tee](#)
- [Flange](#)
- [Valve](#)
- [Rigid](#)
- [Bellow](#)
- [Spring](#)
- [Socket](#)
- [Structural](#)
- [Beam](#)
- [Orientation](#)

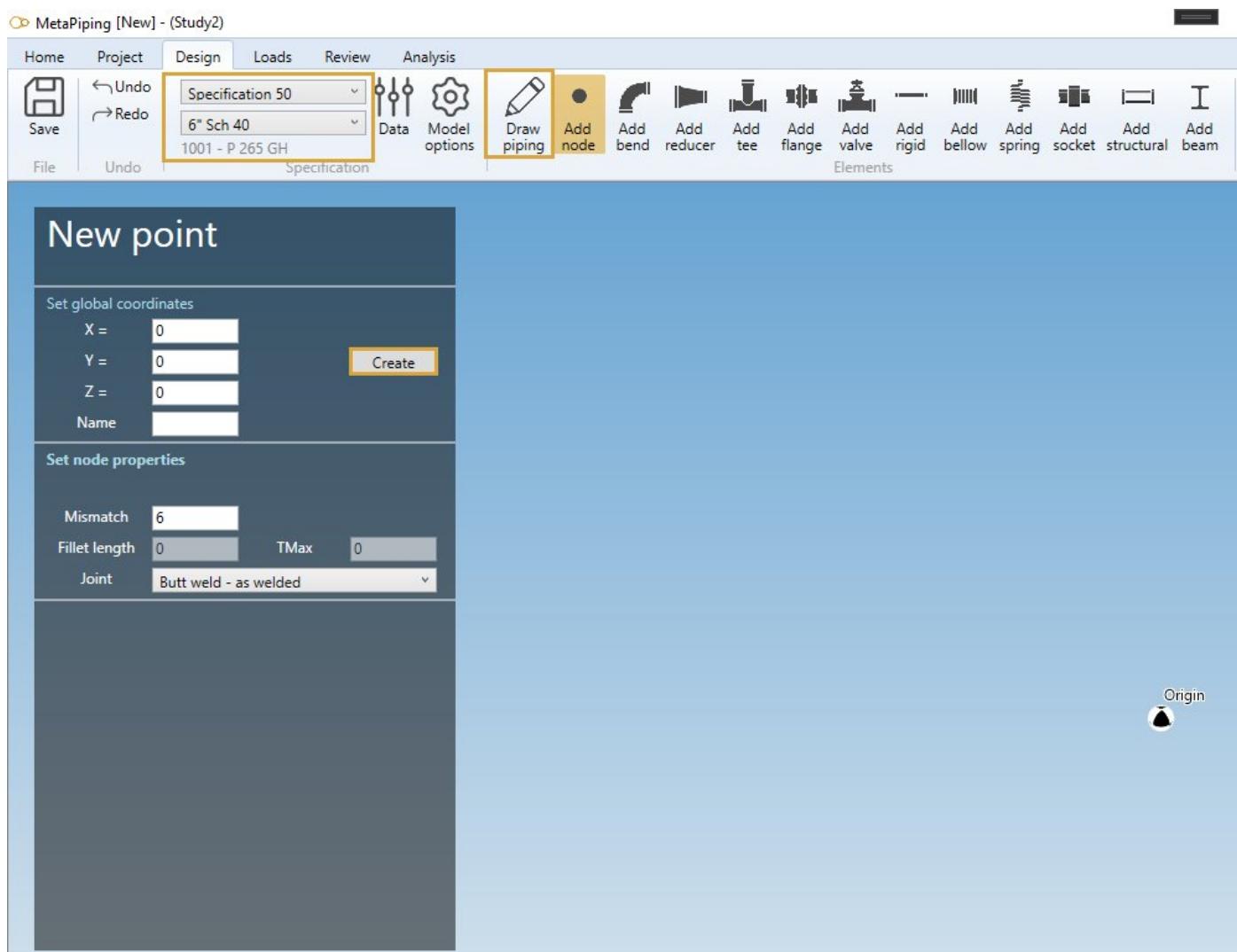
Create nodes

There are 3 possibilities of creation :

- Creation of the **FIRST** node
- Creation of a **GLOBAL** node
- Creation of a **RELATIVE** node

1. First node

The first node panel is called when a first **element** needs a starting point. Example for a pipe :



Set the **GLOBAL** X, Y, Z coordinates and a name.

See **Global node** just below :

2. Global node

If the selection is **EMPTY**, click on the **Add node** button to create a new global node :



New point

Set global coordinates

X =	0
Y =	0
Z =	0
Name	

Create

1

Set node properties

Mismatch	6
Fillet length	0
TMax	0
Joint	Butt weld - as welded

2

WARNING

ATTENTION, the current specification is important (Ex : 6" Sch 40) for the default node connection.

Fill the **GLOBAL** X, Y, Z coordinates and a name (1).

Property	Description	Unit Metric	Unit USA
X	X global coordinate	m	ft
Y	Y global coordinate	m	ft
Z	Z global coordinate	m	ft
Name	Text or number	-	-

To know the UNIT of the value, just let the mouse over the cell.

If **Name** is blank, the new node will receive an automatic name based on the **Node settings**.

Click [here](#) for more information about node naming.

All new node follow the **current specification** node connection properties (2) :

Property	Description	Unit Metric	Unit USA
Mismatch	Mismatch for welded joints	mm	in
Fillet length	Length of fillet weld Cx	mm	in
TMax	for Class 1 transition within 1:3 slope envelope	mm	in

The **Joint** can be :

- None
- Butt weld - flush
- Butt weld - as welded
- Fillet weld
- Full fillet weld
- Threaded
- Brazed

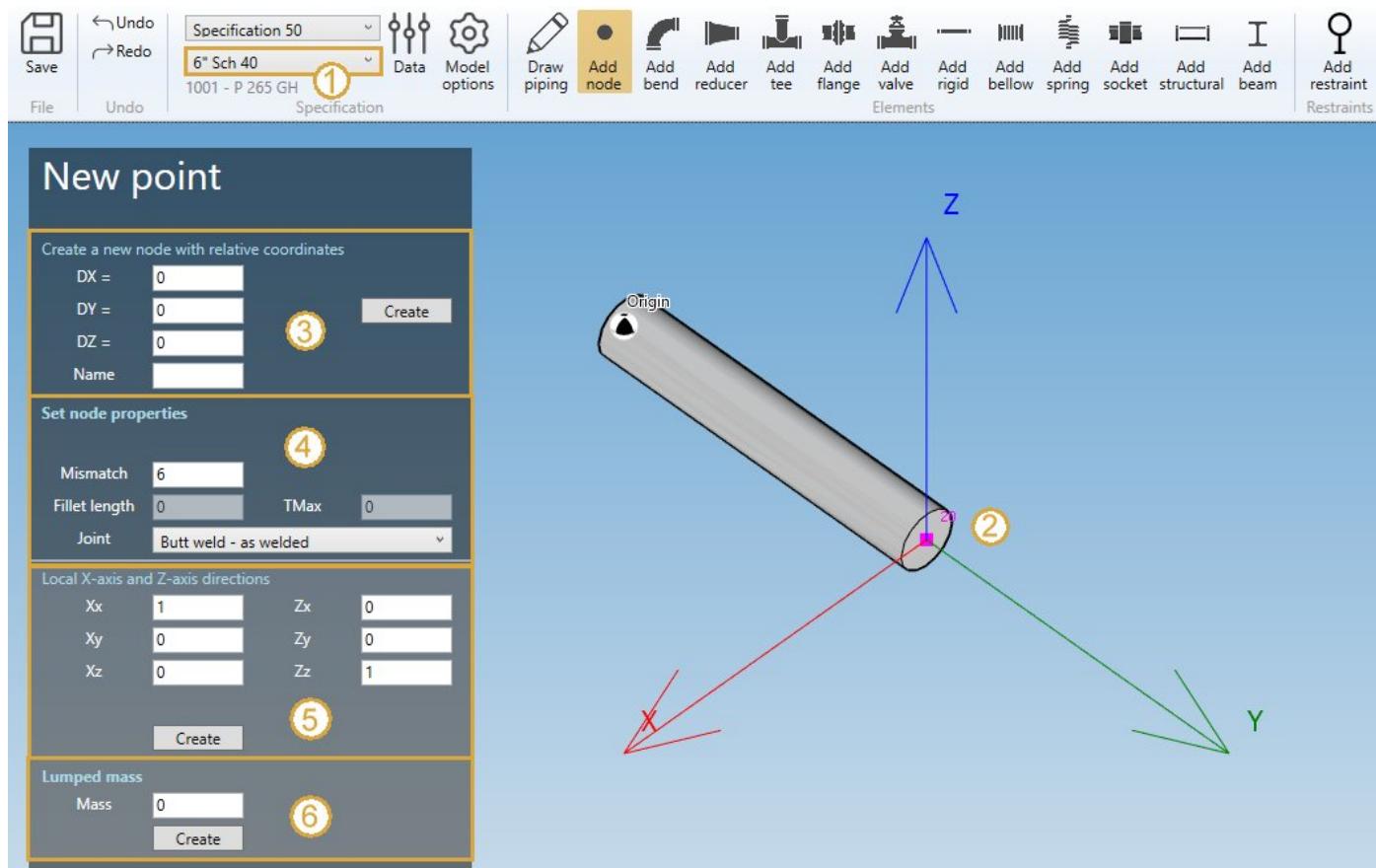
In this example, the values correspond to **current specification** preset :

Pipes	Size	Schedule	Material	Node connection	Mismatch / fillet length	Long weld type	Long weld mismatch	Linear mass + insulation	Insulation thickness	External corrosion	Internal erosion	Bend thickness	Description
	6"	Sch 40-Std-40S	P 265 GH	Butt weld - as w	6	Butt Weld Flush	6.6	11.3	0	0	0	7.11	6" Sch 40
	4"	Sch 40-Std-40S	P 265 GH	Butt weld - as w	4	Butt Weld Flush	4.4	5.84	0	0	0	6.02	4" Sch 40
	2"	Sch 40-Std-40S	P 280 GH	Butt weld - as w	2	Butt Weld Flush	2.2	2.39	0	0	0	3.91	2" Sch 40

Click the **Create** button to create the first node of the model.

3. Relative node

If a node is selected, you can create a new node relative to it in **Global directions** :



- 1 Select the current **section/material** in the specification box.
- 2 Select a **node**.
- 3 Define the DX, DY, DZ, name of the new node.
- 4 Define the node properties

Click [here](#) for more information about the selection tool.

Property	Description	Unit Metric	Unit USA
DX	Relative distance from the selected node in X global direction	m	ft
DY	Relative distance from the selected node in Y global direction	m	ft
DZ	Relative distance from the selected node in Z global direction	m	ft
Name	Text or number	-	-

If **Name** is blank, the new node will receive an automatic name based on the **Node settings**.

Click [here](#) for more information about node naming.

All new node follow the **current specification** node connection properties (2) :

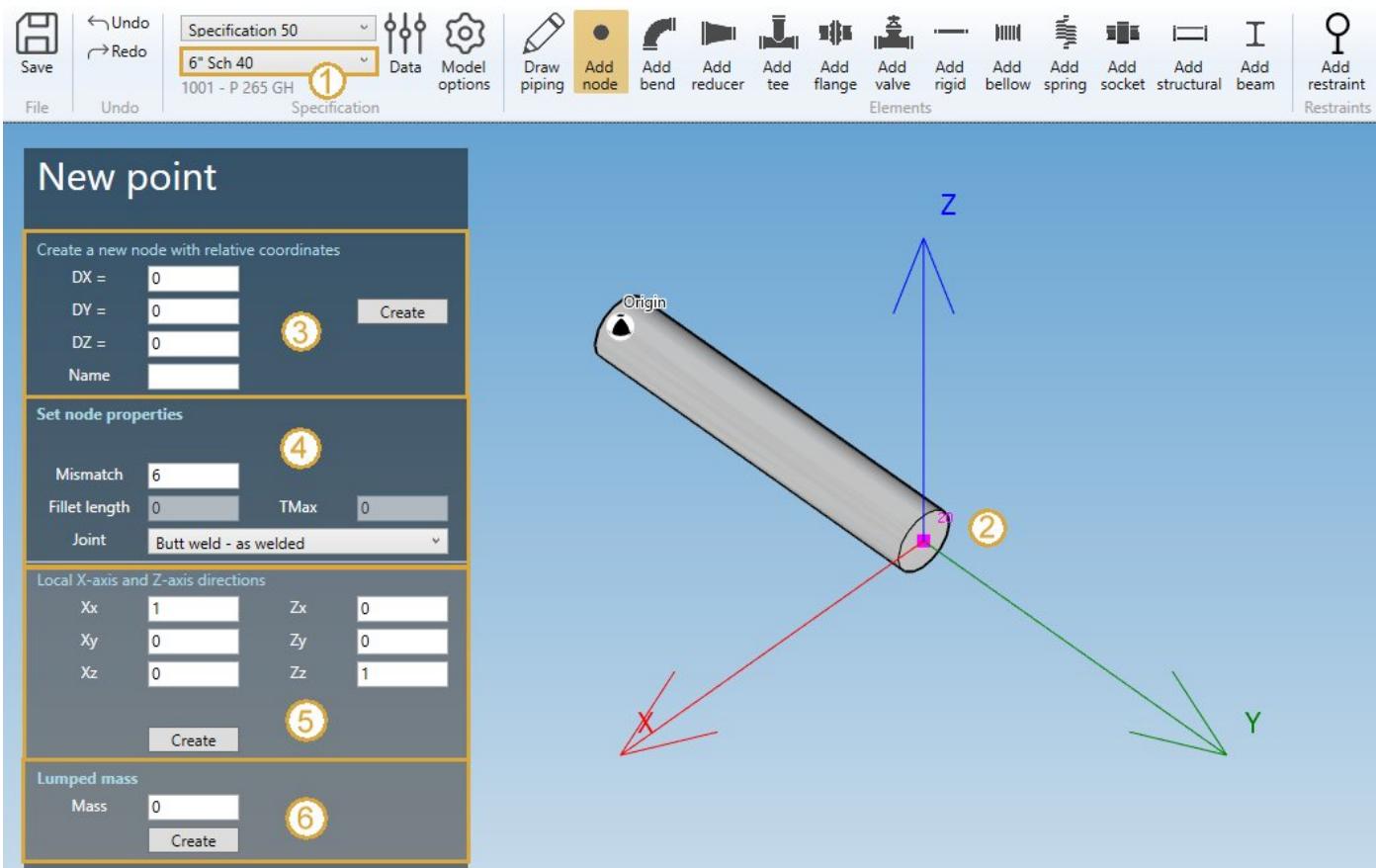
Property	Description	Unit Metric	Unit USA
Mismatch	Mismatch for welded joints	mm	in
Fillet length	Length of fillet weld Cx	mm	in
TMax	for Class 1 transition within 1:3 slope envelope	mm	in

The **Joint** can be :

- None
- Butt weld - flush
- Butt weld - as welded
- Fillet weld
- Full fillet weld
- Threaded
- Brazed

4. Creation of a local coordinate system

You can define a new **local coordinate system** on selected node (5) :



Just define the new **X** axis vector (X_x, X_y, X_z) in global coordinates and the new **Z** axis vector (Z_x, Z_y, Z_z). The **Y** axis will be automatically defined by the right-hand rule.

Example :

New point

Create a new node with relative coordinates

DX =	0
DY =	0
DZ =	0
Name	<input type="text"/>

Create

Set node properties

Mismatch	6
Fillet length	0
TMax	0
Joint	Butt weld - as welded

Local X-axis and Z-axis directions

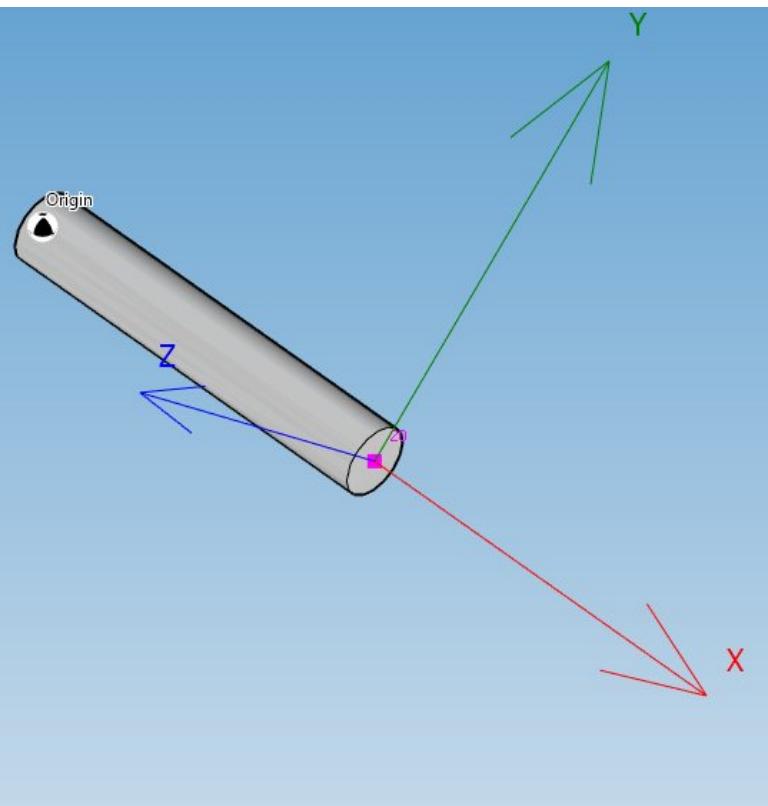
Xx	0	Zx	0.707
Xy	1	Zy	0
Xz	0	Zz	0.707

Create

Lumped mass

Mass	0
------	---

Create

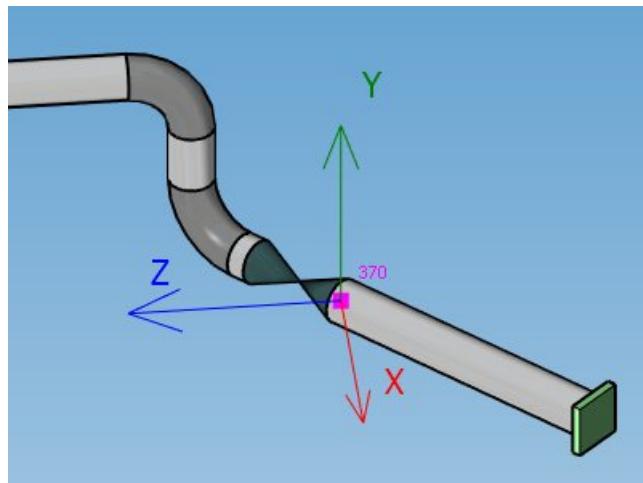


Click on the **Create** button to add a new local coordinate system to the model.

You can later modify or remove this object by selecting this node and click on **Modify/Remove** buttons.

You can **Undo** this command.

Another example with an off-axis piping :



The default global axis are :

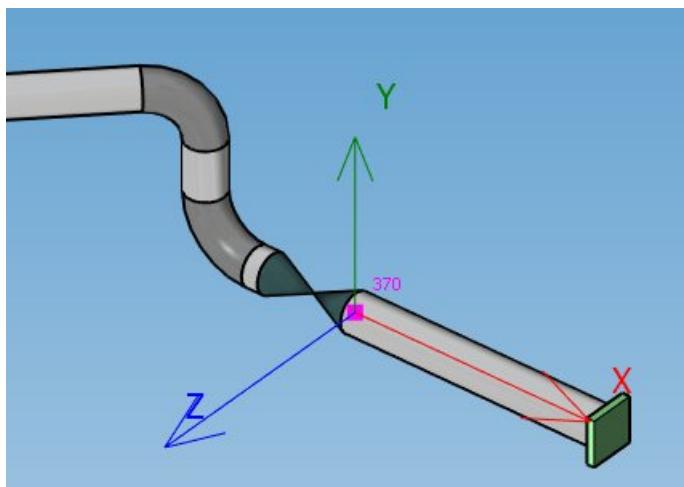
Local X-axis and Z-axis directions

Xx	1	Zx	0
Xy	0	Zy	0
Xz	0	Zz	1

Create

To define the **Local X-axis** tangent to the pipe, change the vectors like this :

Local X-axis and Z-axis directions	
Xx	1
Xy	0
Xz	-1
Zx	1
Zy	0
Zz	1



5. Creation of a lumped mass

You can define a **lumped mass** on selected node (6) :

Screenshot of the software interface showing the creation of a lumped mass:

- Specification:** 6" Sch 40 1001 - P 265 GH (1)
- Add node:** (2)
- New point dialog:**
 - Create a new node with relative coordinates:** DX = 0, DY = 0, DZ = 0 (3)
 - Set node properties:** Mismatch = 6 (4)
 - Local X-axis and Z-axis directions:** Xx = 1, Xy = 0, Xz = 0; Zx = 0, Zy = 0, Zz = 1 (5)
 - Lumped mass:** Mass = 0 (6)
- 3D View:** Shows the pipe with a local coordinate system (X, Y, Z) and a lumped mass node at the end of the pipe.

Property	Unit Metric	Unit USA
Mass	ton	kips

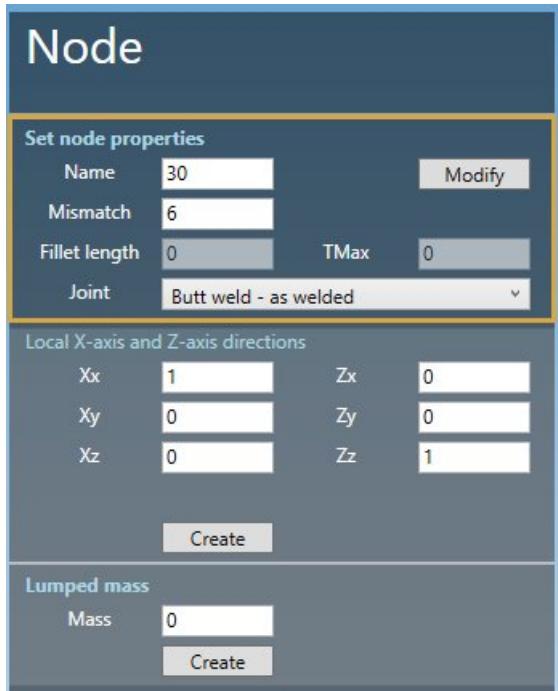
Click on the **Create** button to add a new lumped mass to the node.

You can later modify or remove this object by selecting this node and click on **Modify/Remove** buttons.

You can **Undo** this command.

6. Modify a node

Change the **Selection mode** to POINT and select a node :



Click [here](#) for more information about the selection tool.

You can change the node properties.

Click on the **Modify** button to change the selected node.

You can **Undo** this command.

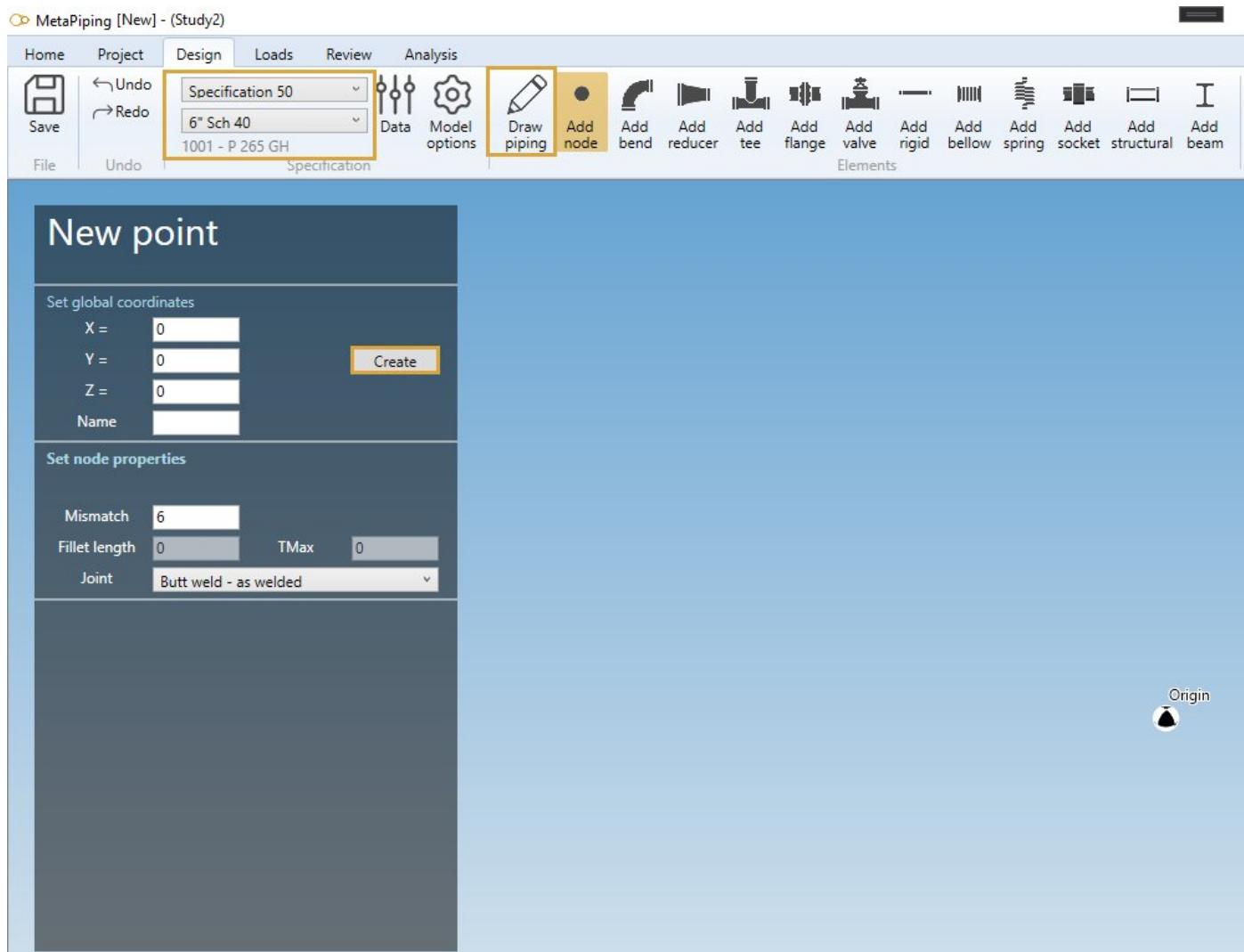
Create pipes

1. New project

Select a **specification** (default = Out of spec) and a current **section**.

Click on the **Draw piping** button.

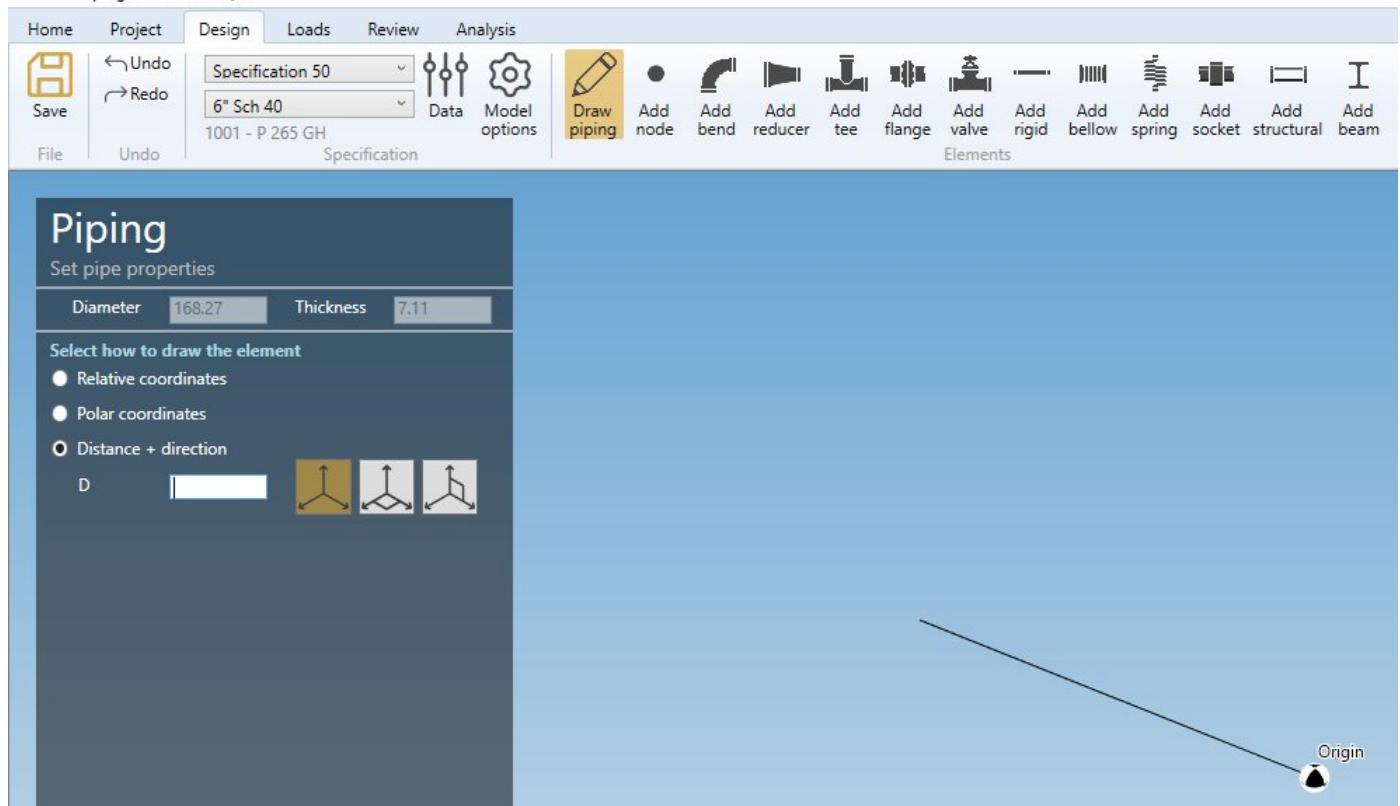
As we start the modeling, we need to create a **First node** :



Click [here](#) for more information about the **First node** creation.

1.2. Draw the first pipe

After the first node has been created, you can draw your first pipe :



The Diameter and Thickness properties come from the **current specification**.

To know the UNIT of the value, just let the mouse over the cell.

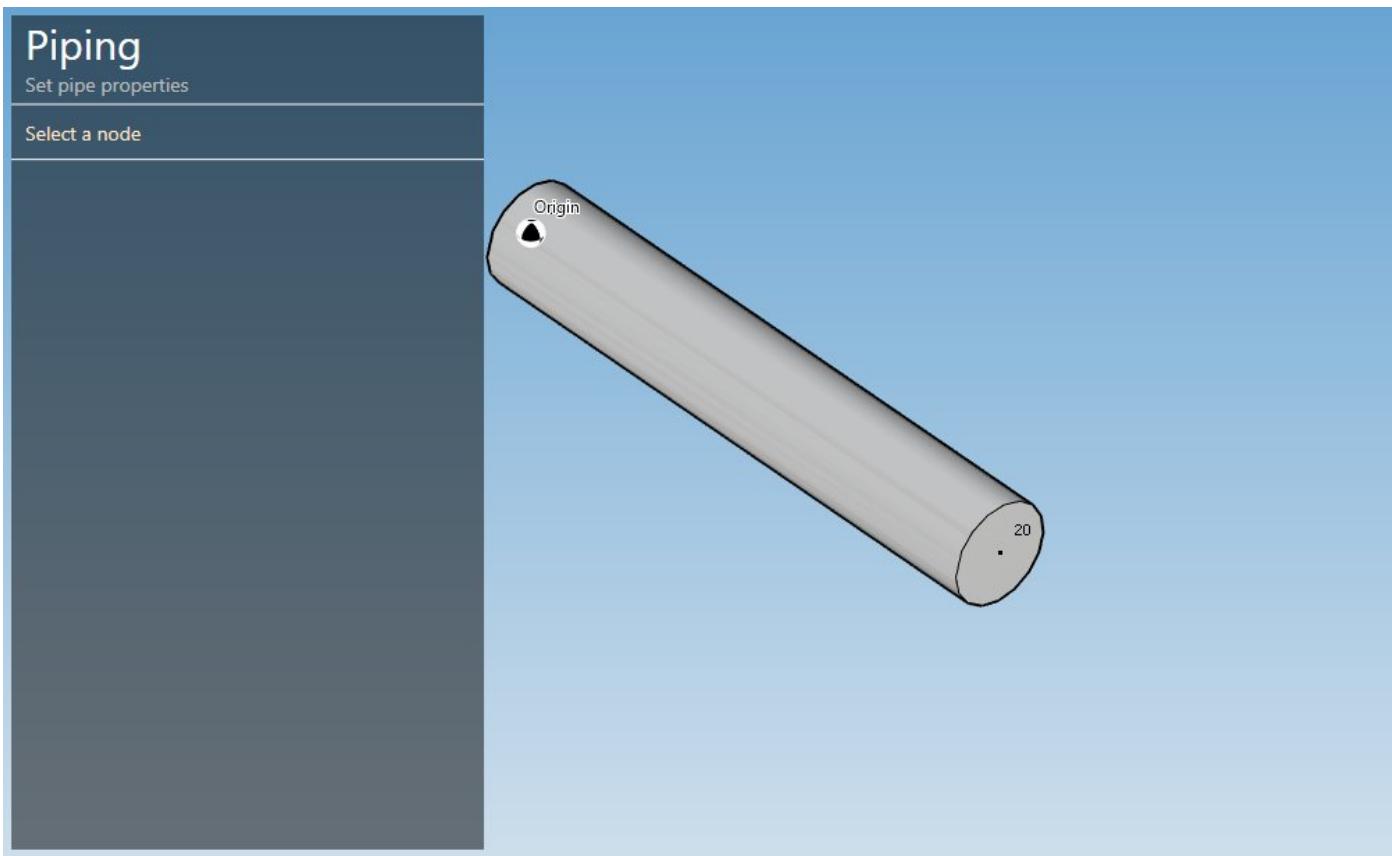
The only thing you have to do is to select how to draw the pipe.

Click [here](#) for more information about the orientation tool.

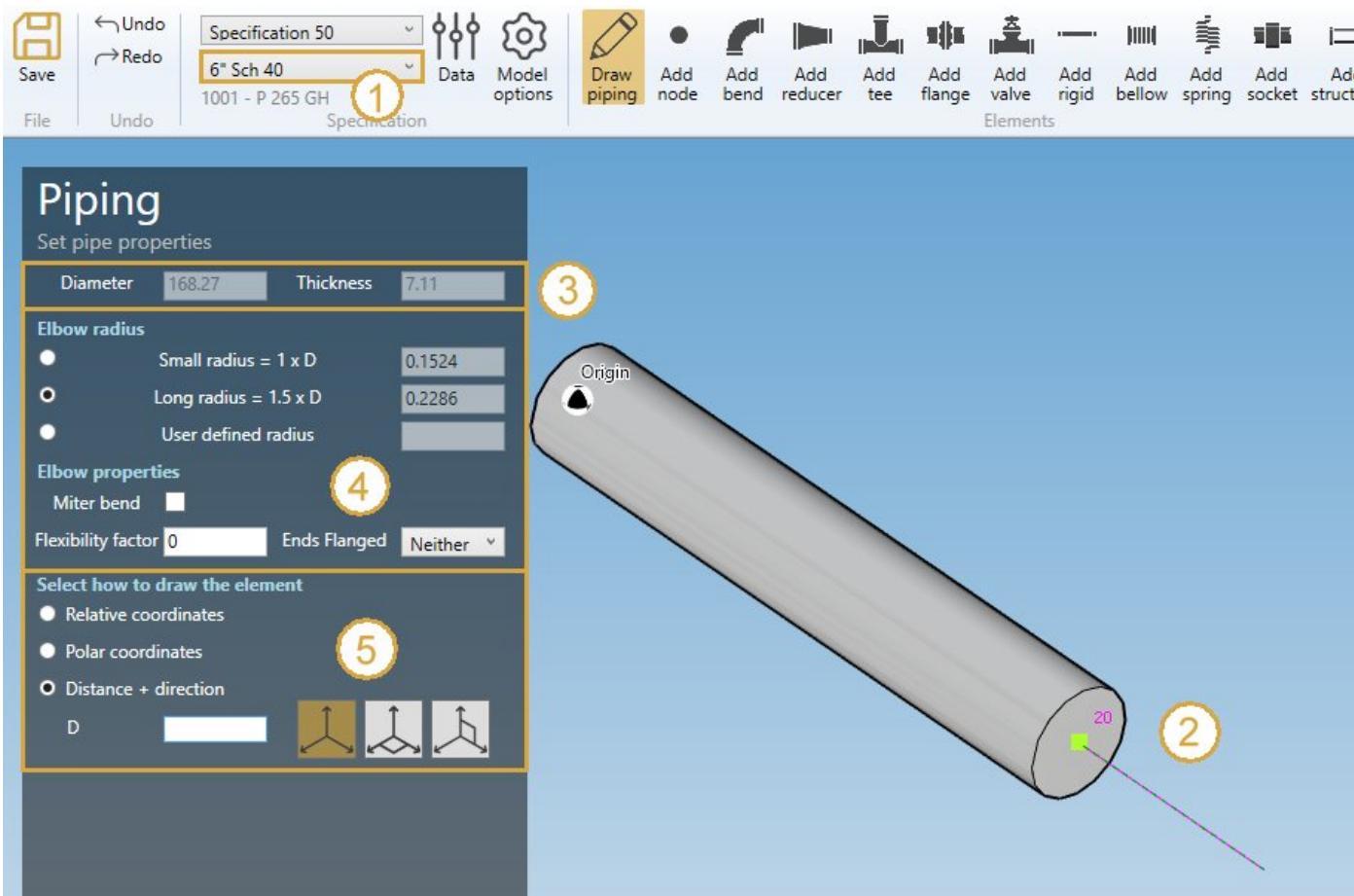
2. Draw piping

When you click on the **Draw piping** button without selection, the left panel shows a message :

Select a node



The **selection mode** is automatically set to **POINT**. You can so directly select a node :



- 1 Select the current **section/material** in the specification box.
- 2 Select a **node**.
- 3 The current diameter and thickness is shown on top of the left panel.

Click [here](#) for more information about the selection tool.

MetaPiping checks the section of the element on the selected node.

2.1 Same section

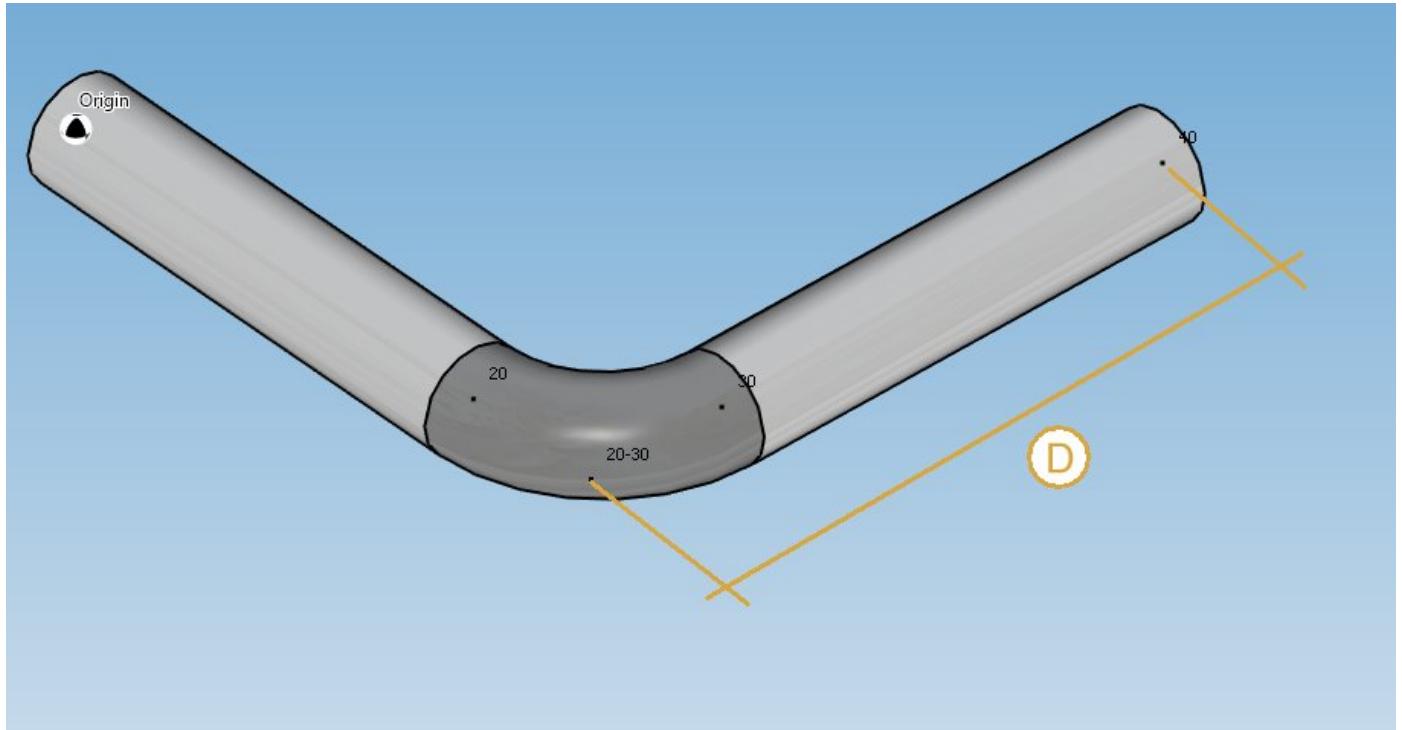
If the section of the element on the selected node is the same as the **current section** (1), the left panel shows **ELBOW PROPERTIES** (4).

Click [here](#) for more information about the elbow properties.

You can then define the second point of the pipe thanks to the **Orientation tool** (5).

Click [here](#) for more information about the orientation tool.

Based on the elbow properties and the second point definition (distance **D** in **X** direction for example), the pipe is created with a bend :



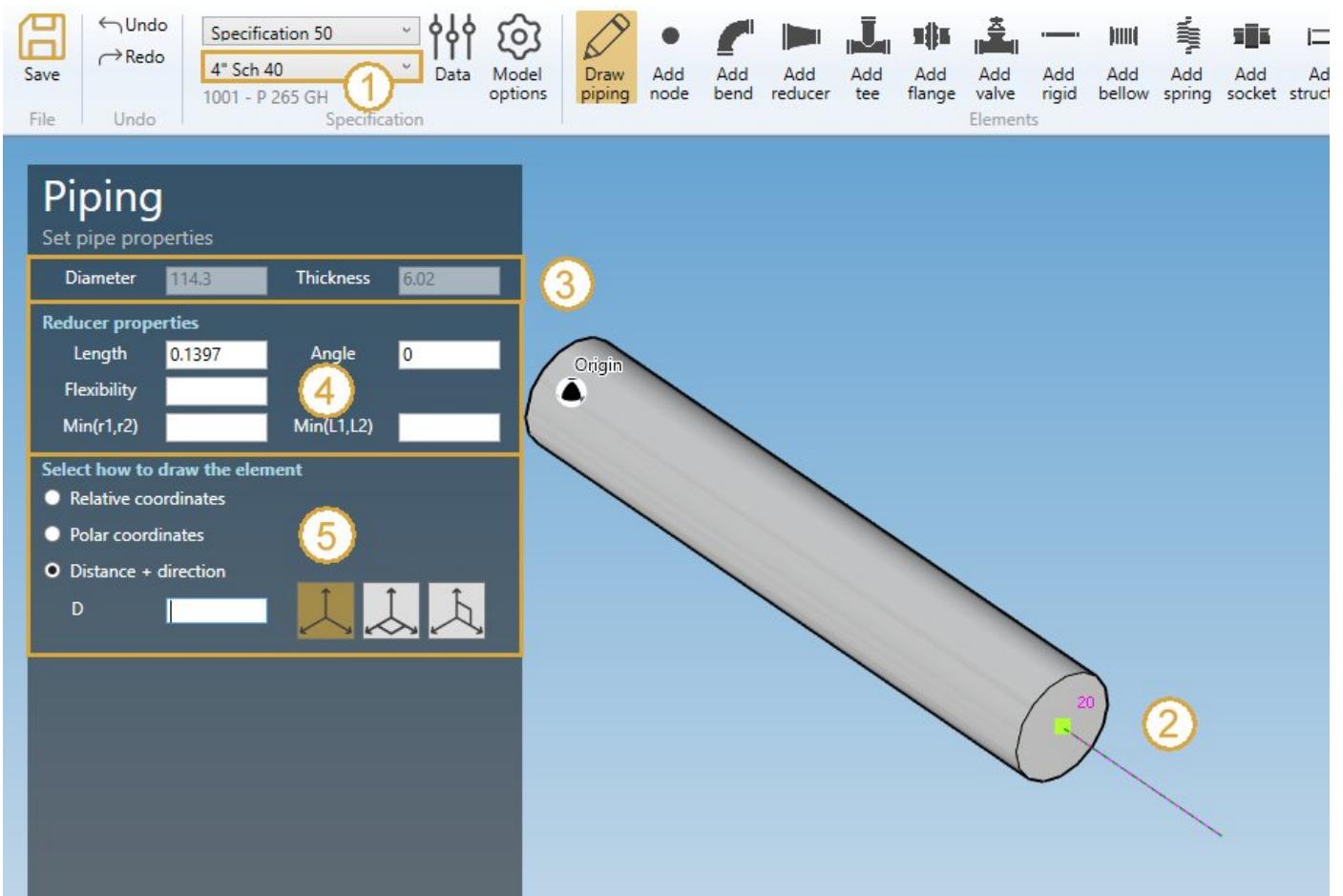
The distance **D** starts from old point 20 to point 40. An elbow is then inserted from new point 20 - point 20-30 to point 30.

The previous pipe is reduced from old point 20 (actually point 20-30) to new point 20.

You can **undo** this command.

2.2 Different section

If the section of the element on the selected node is different from the **current section** (1), the left panel shows **REDUCER PROPERTIES** (4).



WARNING

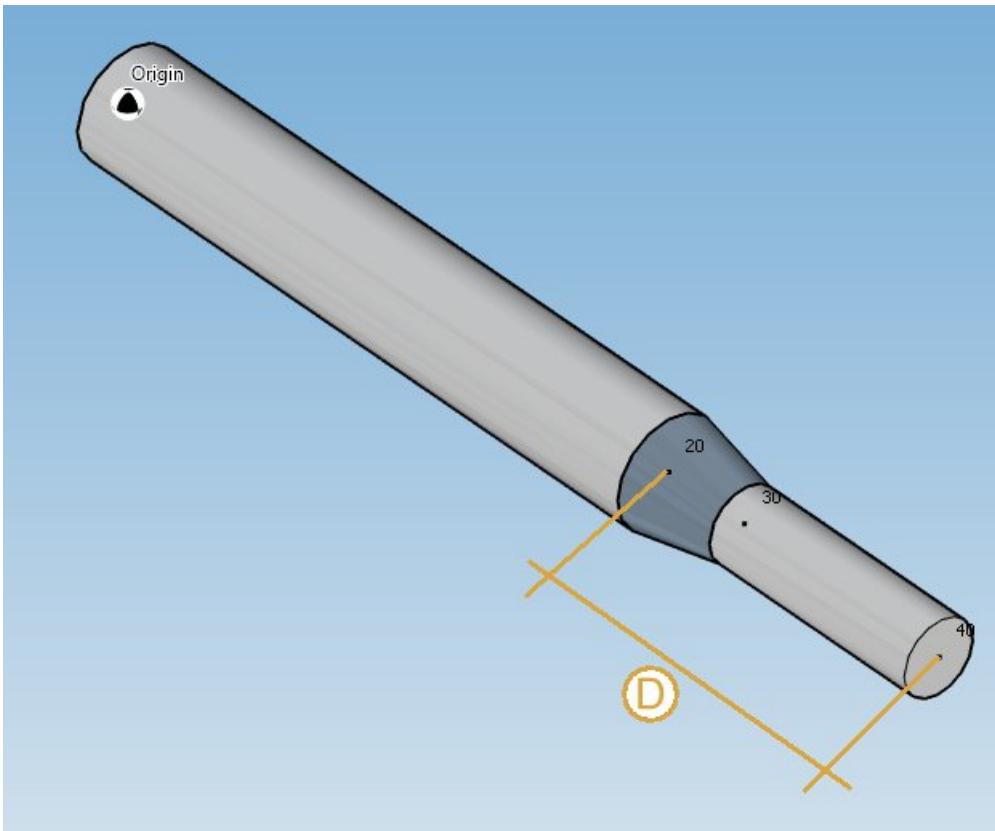
ATTENTION, you are supposed to draw the pipe tangent from the previous one.

Click [here](#) for more information about the reducer properties.

You can then define the second point of the pipe thanks to the **Orientation tool** (5).

Click [here](#) for more information about the orientation tool.

Based on the reducer properties and the second point definition (distance **D** in **TANGENT** direction for example), the pipe is created with a reducer :

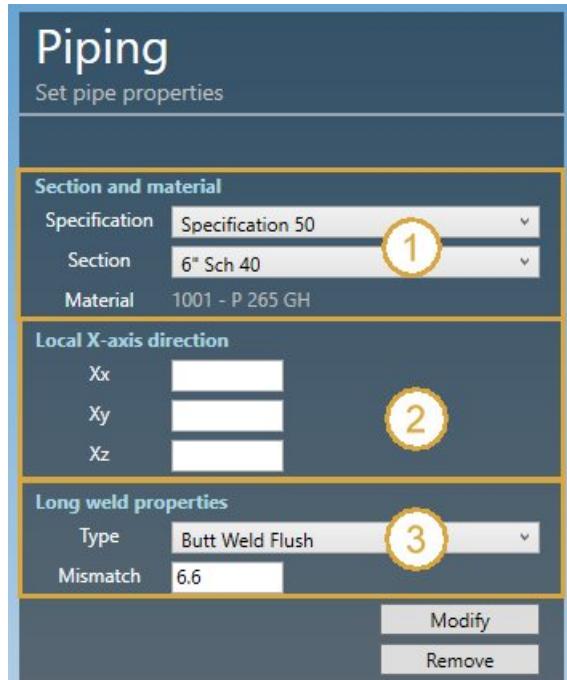


The previous pipe is NOT affected.

You can **undo** this command.

3. Modify/Remove a pipe

Change the **Selection mode** to ELEMENT and select a pipe :



Click [here](#) for more information about the selection tool.

SECTION AND MATERIAL (1) :

You can change the **specification** and **section/material** of the pipe.

X-AXIS DIRECTION (2) :

You can change the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

Label :

You can define a label to this element. The labels are shown with the node names view button.



LONG WELD (3) :

You can change the **Long weld properties** :

For the type, you can choose between :

- None
- Butt weld flush
- Butt weld as welded

Based on this property, define the **Long weld mismatch** [mm or in].

In this example, the default values correspond to **current specification** preset :

Pipes												
Size	Schedule	Material	Node connection	Mismatch / fillet length	Long weld type	Long weld mismatch	Linear mass + insulation	Insulation thickness	External corrosion	Internal erosion	Bend thickness	Description
6"	Sch 40-Std-40S	P 265 GH	Butt weld - as w	6	Butt Weld Flush	6.6	11.3	0	0	0	7.11	6" Sch 40
4"	Sch 40-Std-40S	P 265 GH	Butt weld - as w	4	Butt Weld Flush	4.4	5.84	0	0	0	6.02	4" Sch 40
2"	Sch 40-Std-40S	P 280 GH	Butt weld - as w	2	Butt Weld Flush	2.2	2.39	0	0	0	3.91	2" Sch 40

Click on the **Modify** button to change the selected pipe with these new properties.

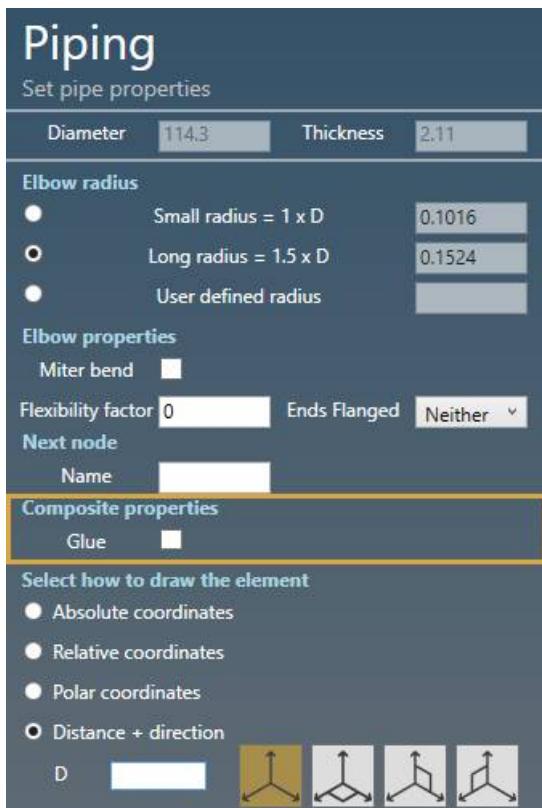
You can **undo** this command.

Click on the **Remove** button to delete the selected pipe.

You can **undo** this command.

COMPOSITE PIPE :

If the material of the pipe is **Composite**, a special property **Glue** appears :



This property indicates that the pipe is glued with the others.



Create bends

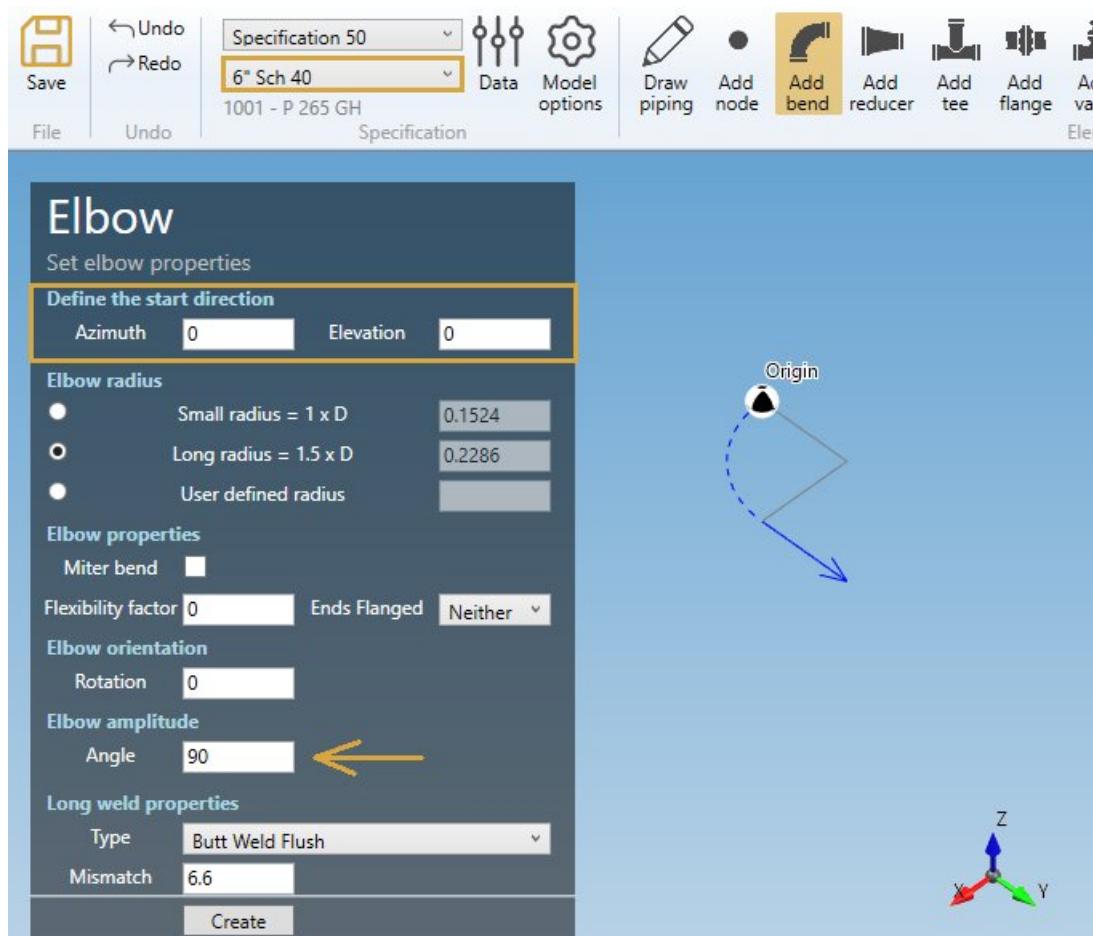
When you click on the **Add bend** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

1. Create first bend

Select an isolated node and click on the **Add bend** button :



MetaPiping asks you to define the **start direction** with 2 angles :

- Azimuth = Horizontal angle in °
- Elevation = Vertical angle in °

The default direction is the global **X axis**.

But with the default **Amplitude** of 90°, the elbow will have the global **Y axis** as final direction.

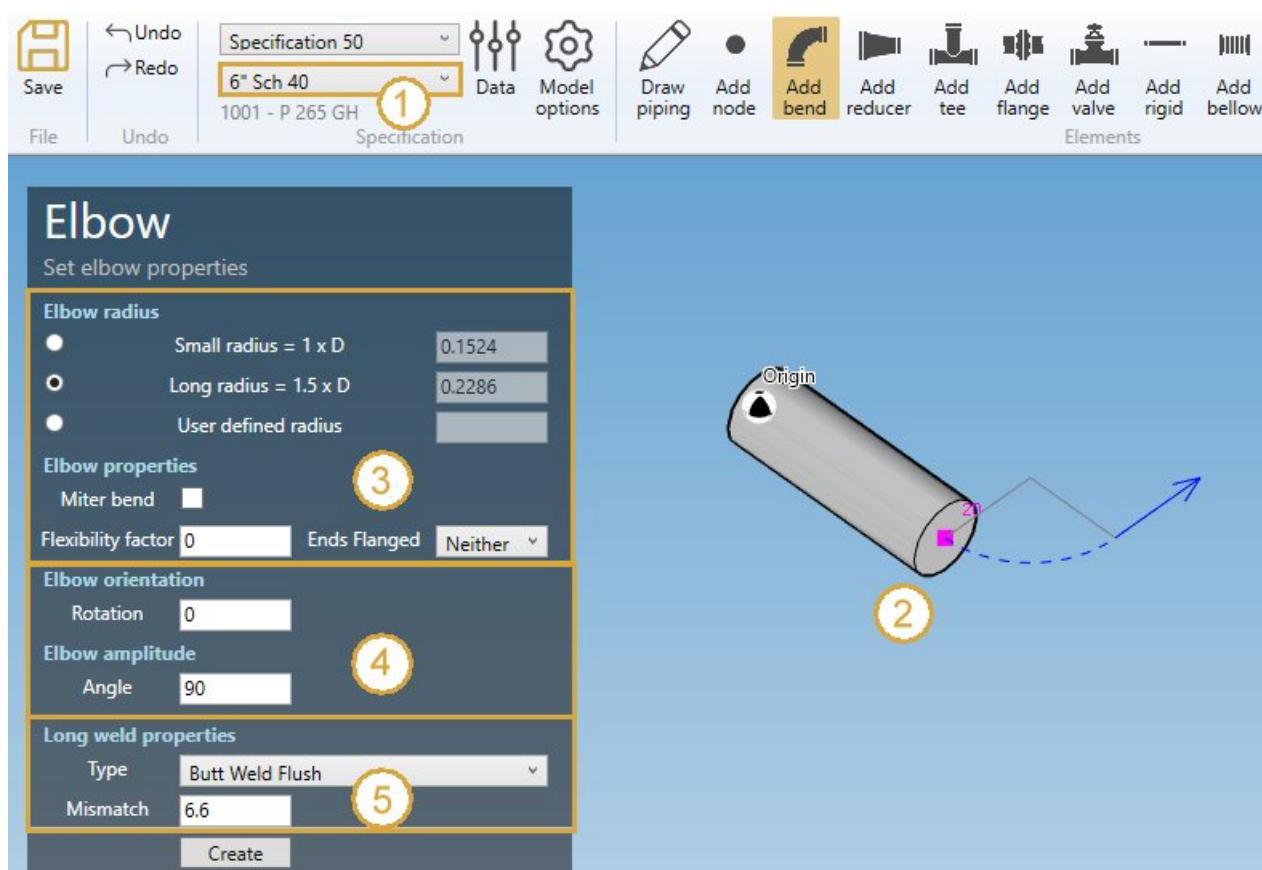
The properties are explained in §2.

Click on the **Create** button to add a bend.

You can **Undo** this command.

2. Create a bend from another element

- 1 Select the current **section/material** in the specification box.
- 2 Select a **node**.
- 3 Click the **Add bend** button



2.1 Elbow properties (3)

In this example, the values correspond to **current specification** preset.

RADIUS :

Bends			
Size	Long radius	Small radius	Mass
6"	0.2286	0.1524	0
4"	0.1524	0.1016	0
2"	0.0762	0.0508	0

MetaPiping sets the **Long radius** by default but you can change it (Small radius or user defined radius)

Property	Unit Metric	Unit USA
Radius	m	ft

To know the UNIT of the value, just let the mouse over the cell.

MITER :

You can define a miter bend by checking the checkbox and define the number of cuts and if it is based on the radius or on a spacing.

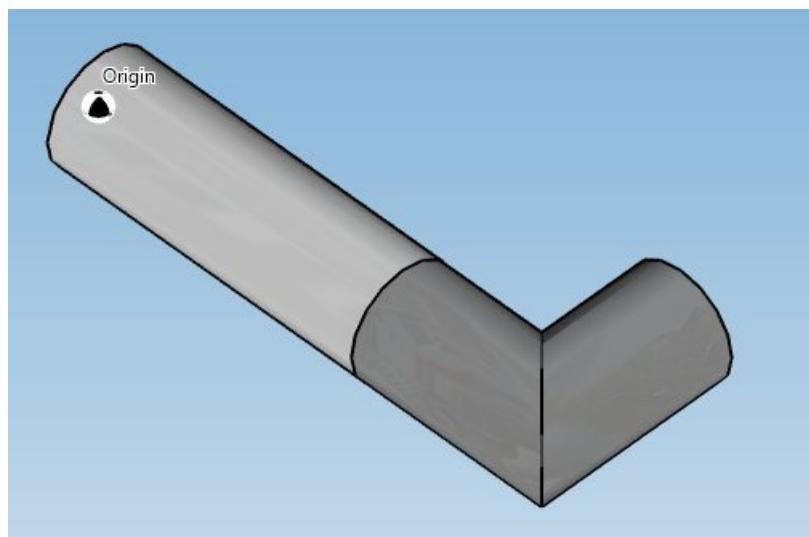
Property	Unit Metric	Unit USA
Spacing	m	ft

Here are some examples.

Radius & cuts = 1 :

Elbow properties

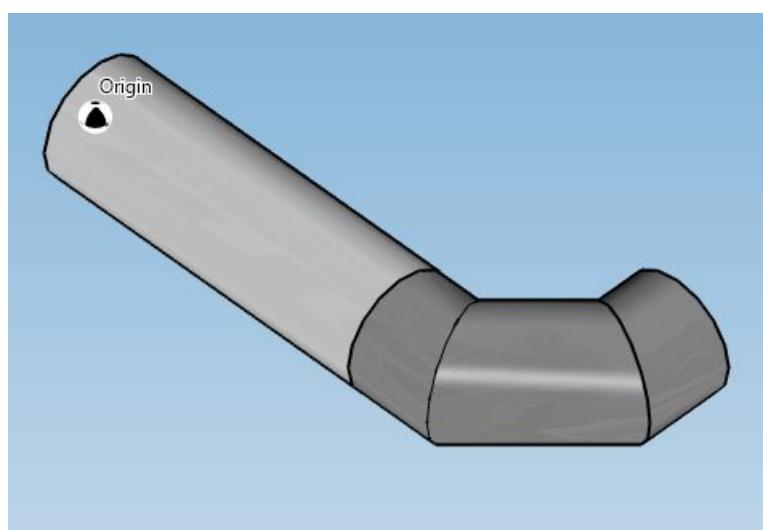
Miter bend	<input checked="" type="checkbox"/>	Number of cuts	1
Miter option	Radius	Radius	0.056095
Flexibility factor		Ends Flanged	Neither



Radius & cuts = 2 :

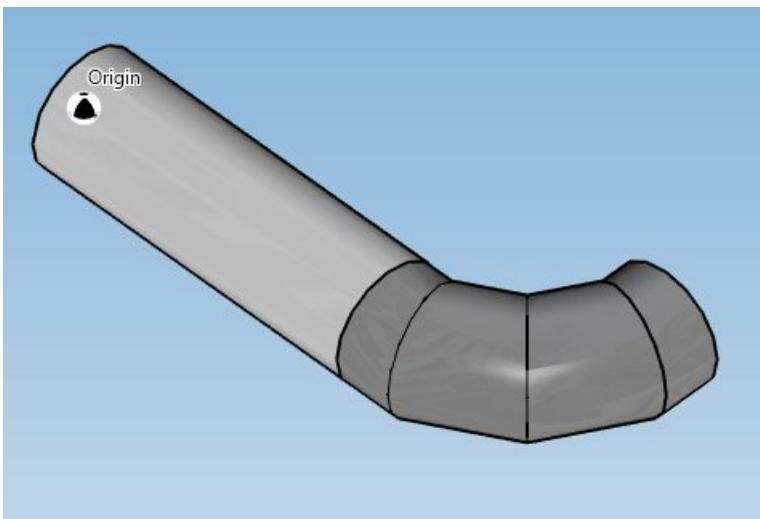
Elbow properties

Miter bend	<input checked="" type="checkbox"/>	Number of cuts	2
Miter option	Radius	Radius	0.1524
Flexibility factor		Ends Flanged	Neither



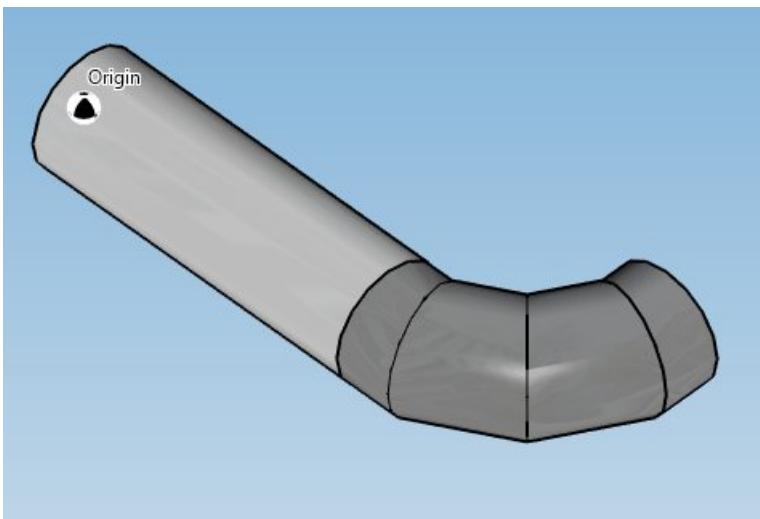
Radius & cuts = 3 :

Elbow properties			
Miter bend	<input checked="" type="checkbox"/>	Number of cuts	3
Miter option	Radius	Radius	0.1524
Flexibility factor		Ends Flanged	Neither



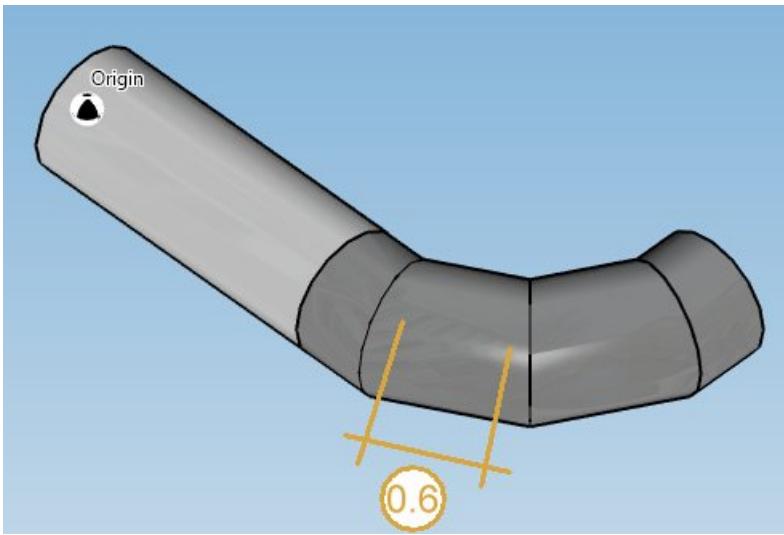
Spacing & cuts = 3, MetaPiping calculates the default spacing for an identical result :

Elbow properties			
Miter bend	<input checked="" type="checkbox"/>	Number of cuts	3
Miter option	Spacing	Spacing	0.4572
Flexibility factor		Ends Flanged	Neither



Spacing = 0.6 :

Elbow properties			
Miter bend	<input checked="" type="checkbox"/>	Number of cuts	3
Miter option	Spacing	Spacing	0.6
Flexibility factor		Ends Flanged	Neither



WARNING

ATTENTION, in this case, the node extremity of the previous pipe is moved to respect the bend construction

FLEXIBILITY FACTOR :

User can define the flexibility factor of the bend.

This flexibility factor will be used for all load cases.

Default = 0 or blank = default special k Factor in the code.

ENDS FLANGED :

You can choose between :

- Neither end flanged
- One end flanged
- Both ends flanged

2.2 Elbow orientation (4)

Here are some examples of **Rotation** and **Amplitude** values :

Elbow orientation	Rotation 0
Elbow amplitude	Angle 90

Elbow orientation	Rotation 0
Elbow amplitude	Angle 45

Elbow orientation	Rotation 90
Elbow amplitude	Angle 90

Rotation and Angle are in °.

2.3 Long weld properties (5)

Long weld properties
Type Butt Weld Flush
Mismatch 6.6

For the type, you can choose between :

- None
- Butt weld flush
- Butt weld as welded

Based on this property, define the **Long weld mismatch** [mm or in].

In this example, the default values correspond to **current specification** preset :

Pipes		[+]	Material	Node connection	Mismatch / fillet length	Long weld type	Long weld mismatch	Linear mass + insulation	Insulation thickness	External corrosion	Internal erosion	Bend thickness	Description	
6"	Sch 40-Std-40S	v	P 265 GH	v	Butt weld - as w	6	Butt Weld Flush	6.6	11.3	0	0	7.11	6" Sch 40	
4"	Sch 40-Std-40S	v	P 265 GH	v	Butt weld - as w	4	Butt Weld Flush	4.4	5.84	0	0	6.02	4" Sch 40	
2"	Sch 40-Std-40S	v	P 280 GH	v	Butt weld - as w	2	Butt Weld Flush	2.2	2.39	0	0	3.91	2" Sch 40	

2.4 Label

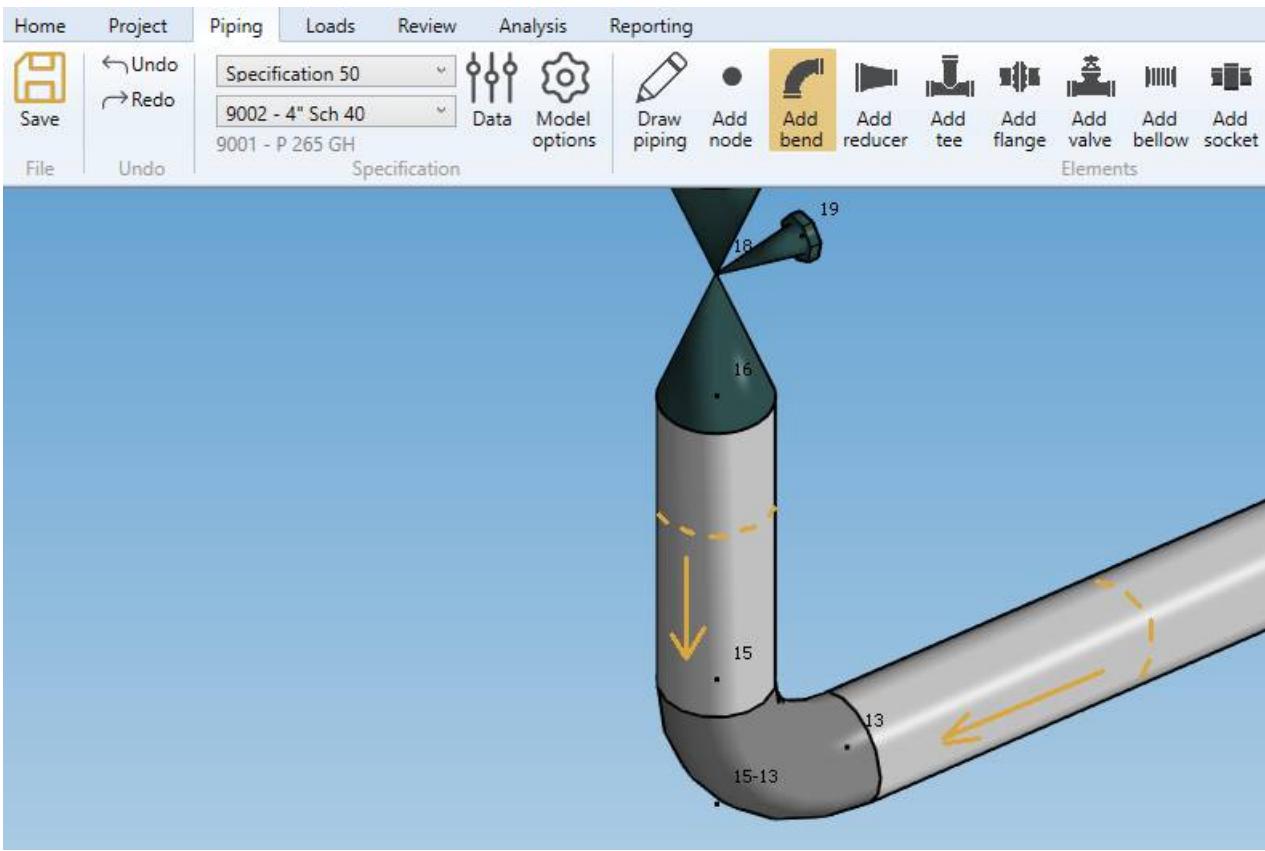
You can define a label to this element. The labels are shown with the node names view button.



3. Create a bend with 2 points

The screenshot shows the software interface for creating a bend. On the left, there is a sidebar with various settings and properties for an 'Elbow'. The 'Elbow radius' section has three options: 'Small radius = 1 x D' (selected), 'Long radius = 1.5 x D', and 'User defined radius'. The 'Elbow properties' section includes 'Miter bend' (unchecked) and 'Flexibility factor' (set to 0). The 'Long weld properties' section shows 'Type' as 'Butt Weld Flush' and 'Mismatch' as '2'. At the bottom of this sidebar is a 'Create' button. On the right, a 3D model of a pipe with an elbow is shown. Two nodes on the pipe are highlighted with yellow circles and numbered '1' and '2'. A yellow arrow points from the 'Create' button to the nodes. A third yellow circle with the number '3' is placed at the bottom of the sidebar, pointing towards the 'Create' button.

- 1 Click the **Add bend** button
- 2 Select two **nodes** of coplanar elements
- 3 Click the **Create** button after defining the section and the bend properties



This will extend the 2 adjacent elements and place a bend with the desired radius.

4. Modify/Remove a bend

Change the **Selection mode** to ELEMENT and select a bend :

Elbow

Set elbow properties

Elbow radius

- Small radius = $1 \times D$ 0.1524
- Long radius = $1.5 \times D$ 0.2286
- User defined radius (disabled)

Elbow properties

Miter bend

Flexibility factor Ends Flanged

Section and material

Specification Section Material

Long weld properties

Type Mismatch

Click [here](#) for more information about the selection tool.

You can modify the radius of the bend. The lengths of the adjacent pipes will be adapted.

You can also modify all bend properties.

SECTION AND MATERIAL :

You can change the **specification** and **section/material** of the bend.

LONG WELD :

You can change the **Long weld properties** :

For the type, you can choose between :

- None
- Butt weld flush
- Butt weld as welded

Based on this property, define the **Long weld mismatch** [mm or in].

In this example, the default values correspond to **current specification** preset :

Pipes												
Size	Schedule	Material	Node connection	Mismatch / fillet length	Long weld type	Long weld mismatch	Linear mass + insulation	Insulation thickness	External corrosion	Internal erosion	Bend thickness	Description
6"	Sch 40-Std-40S	P 265 GH	Butt weld - as w	6	Butt Weld Flush	6.6	11.3	0	0	0	7.11	6" Sch 40
4"	Sch 40-Std-40S	P 265 GH	Butt weld - as w	4	Butt Weld Flush	4.4	5.84	0	0	0	6.02	4" Sch 40
2"	Sch 40-Std-40S	P 280 GH	Butt weld - as w	2	Butt Weld Flush	2.2	2.39	0	0	0	3.91	2" Sch 40

Click on the **Modify** button to change the selected bend with these new properties.

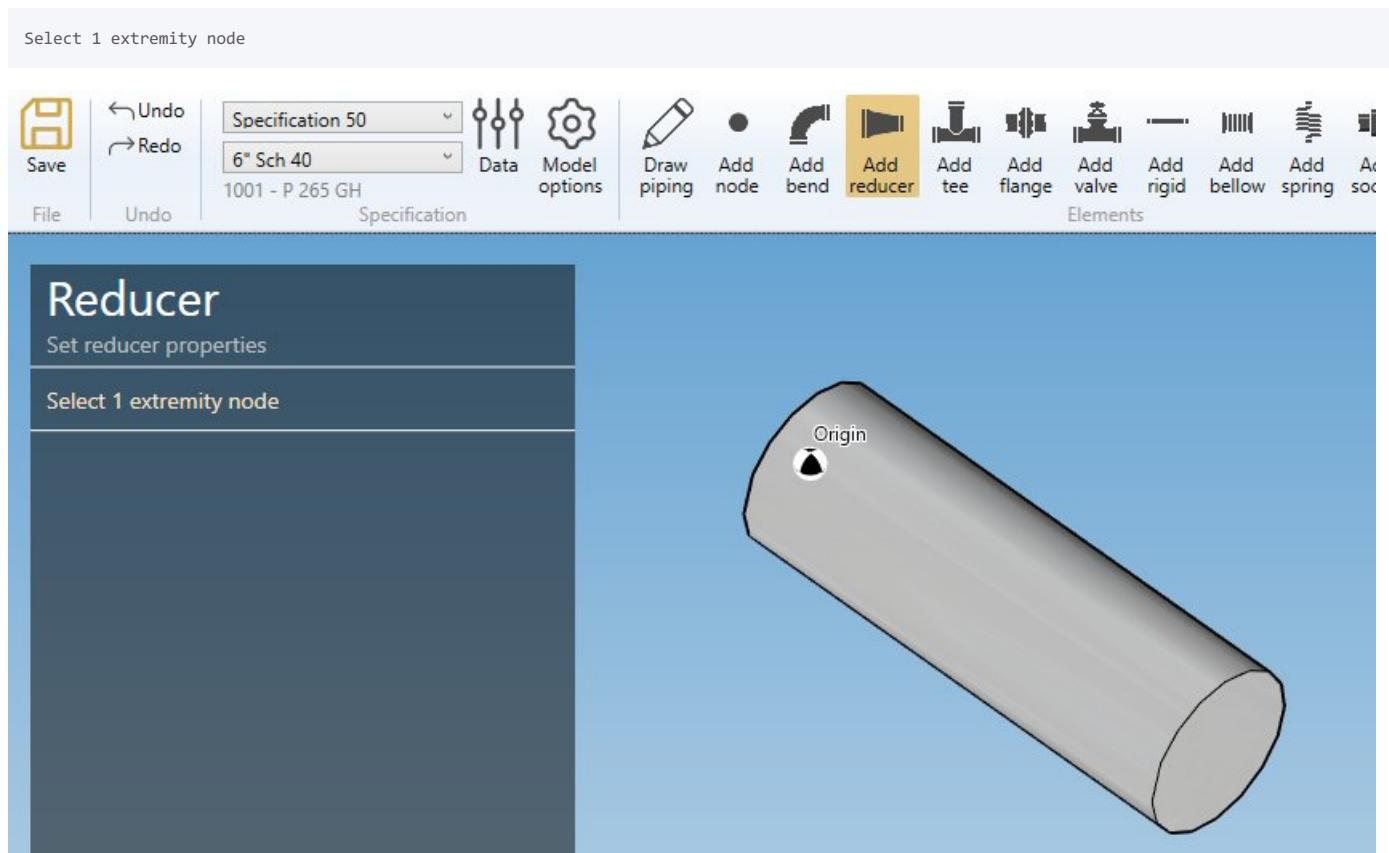
You can **undo** this command.

Click on the **Remove** button to delete the selected bend.

You can **undo** this command.

Create reducers

When you click on the **Add reducer** button without selection, the left panel shows a message :



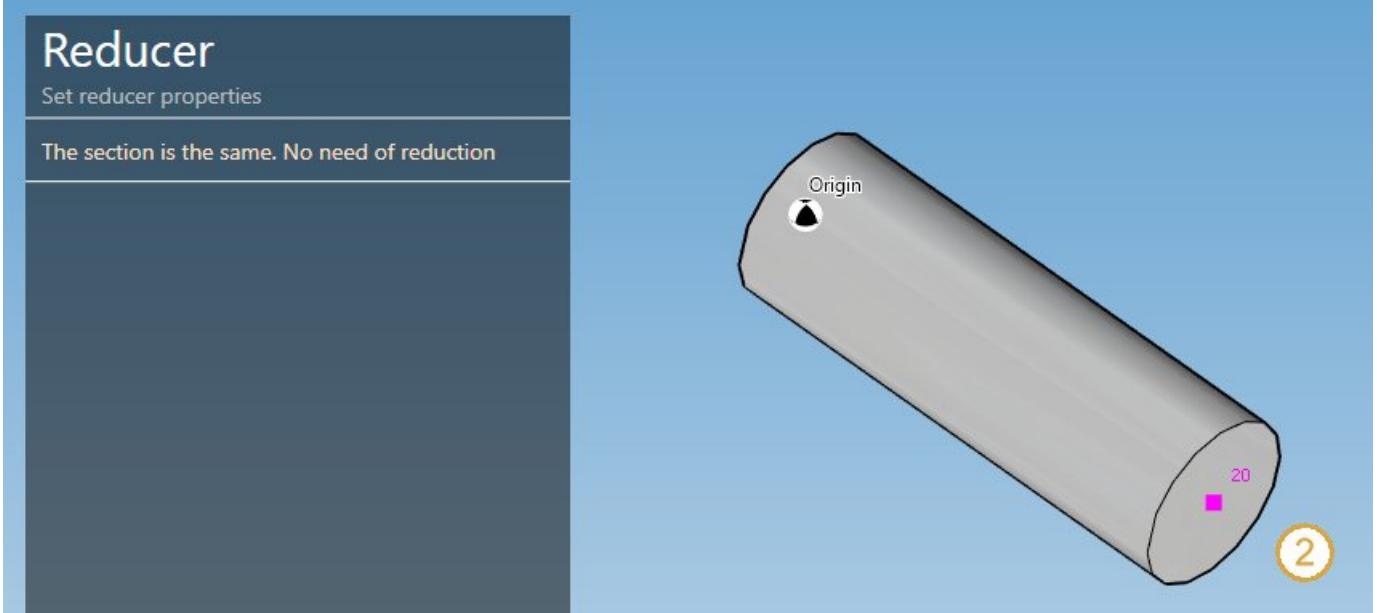
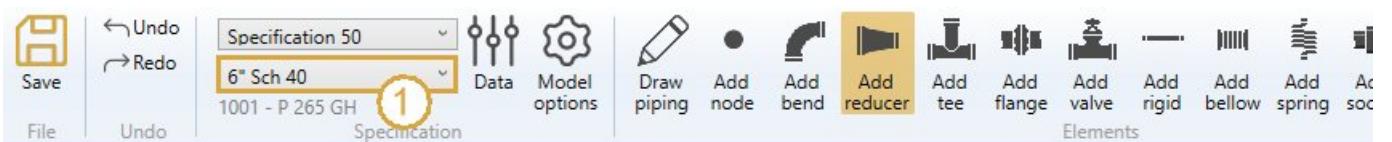
The **selection mode** is automatically set to **POINT**. You can so directly select a node.

WARNING

ATTENTION, you cannot start a reducer from an isolated node, only from the extremity of an element. If you have to start your piping line by a reducer, first create a small pipe in opposite direction, create a reducer from an extremity, then remove the pipe.

1. Create a reducer from another element

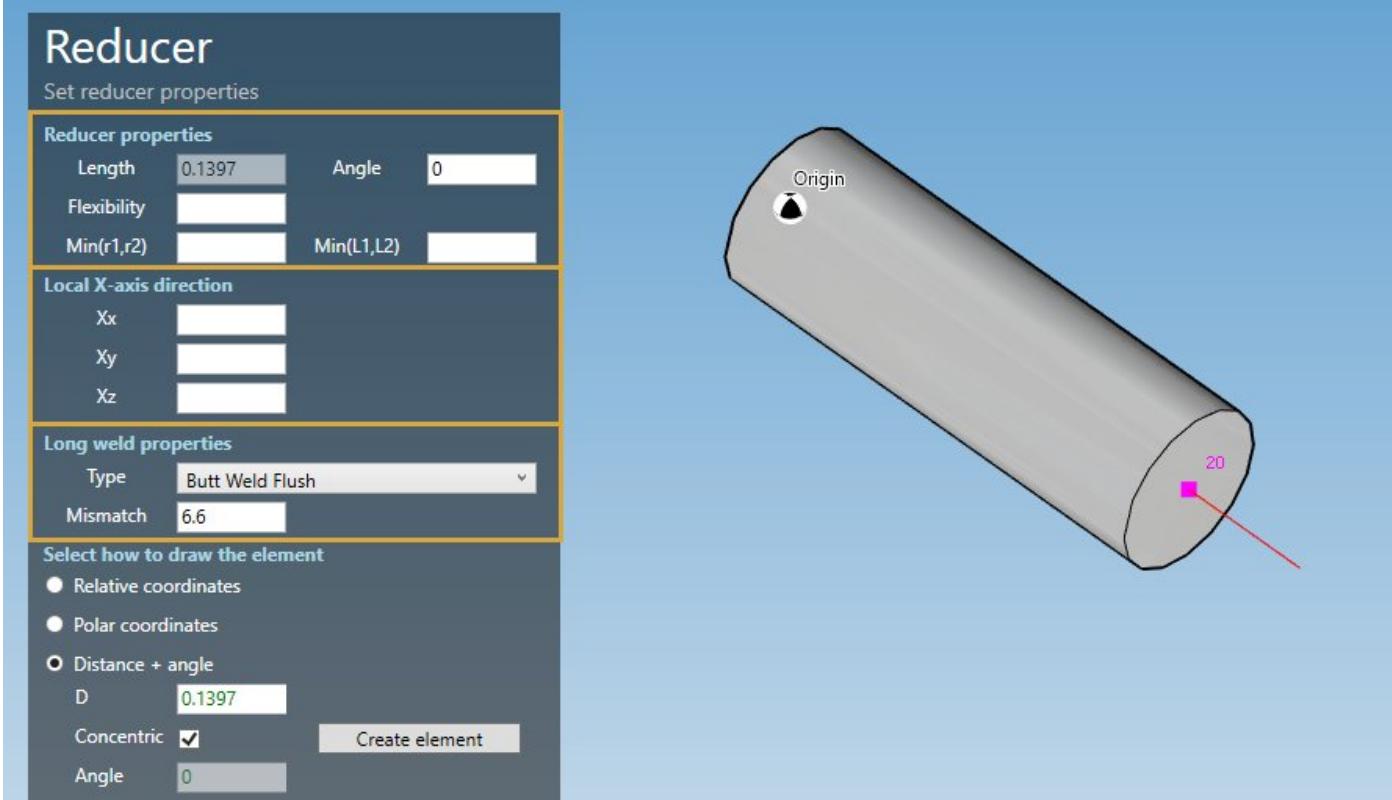
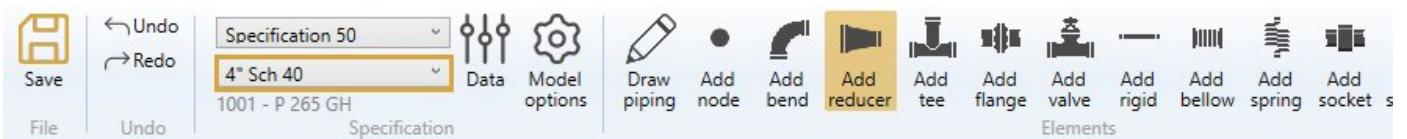
- Select the current **section/material** in the specification box (1).
- Select a **node** (2).
- Click the **Add reducer** button



If the section of the element on node has the same section than the **current specification** (1), the left panel shows a message :

The section is the same. No need of reduction

If the section of the element on node is different from the **current specification** (Ex : 4" Sch 40), you can define a new reducer :



REDUCER PROPERTIES :

Define the properties of the new reducer :

Property	Unit Metric	Unit USA	Remark
Length	m	ft	
Angle	°	°	
Flexibility	-	-	
Min(r1,r2)	mm	in	Minimum radius r1,r2 : B31J, Class 1 and Classes 2 & 3 Revision A94 for C2 of equations X and Z
Min(L1,L2)	m	ft	Minimum tangent L1, L2 : for B31J and Class 1 only

The Length and Angle come from the **current specification** :

Reducers
6"
Reduce to
4"
Length: 0.1397
Mass: 0
Angle: 0

You can change the **flexibility factor** of the reducer. If blank, default value = 1.

X-AXIS DIRECTION :

You can change the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

Label :

You can define a label to this element. The labels are shown with the node names view button.



LONG WELD :

You can change the **Long weld properties** :

For the type, you can choose between :

- None
- Butt weld flush
- Butt weld as welded

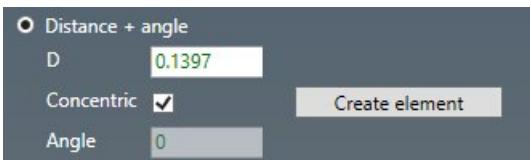
Based on this property, define the **Long weld mismatch** [mm or in].

In this example, the default values correspond to the first section preset.

You can then define the second point of the reducer thanks to the **Orientation tool**.

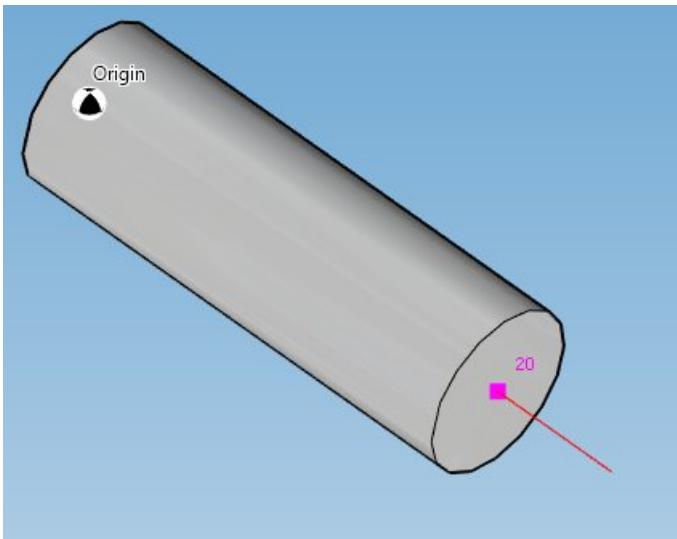
Click [here](#) for more information about the orientation tool.

By default, the **Distance + angle** tool is activated with the default **length** and in **concentric** mode :



Property	Unit Metric	Unit USA
D	m	ft
Angle	°	°

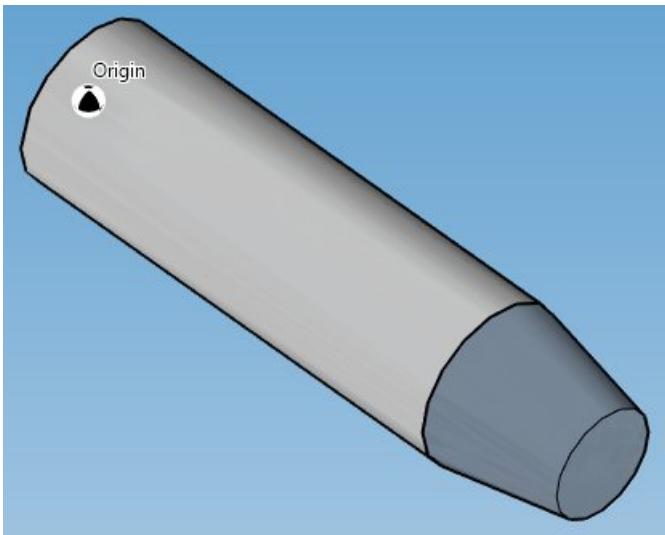
A red line shows a preview of the second point of the reducer :



WARNING

ATTENTION, you can change the length of the reducer but it will no more be standard

You can click on the **Create element** to create a concentric reducer :



You can **undo** this command.

ECCENTRIC REDUCER :

Uncheck the **Concentric** checkbox and define at which side of the pipe you want to be tangent (thanks to the **Angle** property) :

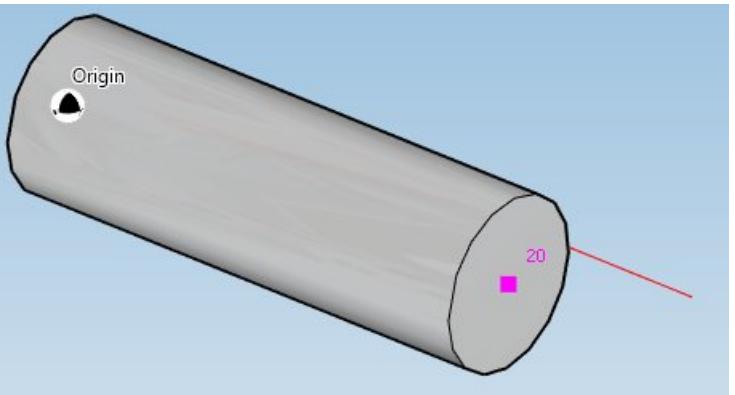
Select how to draw the element

- Relative coordinates
- Polar coordinates
- Distance + angle

D

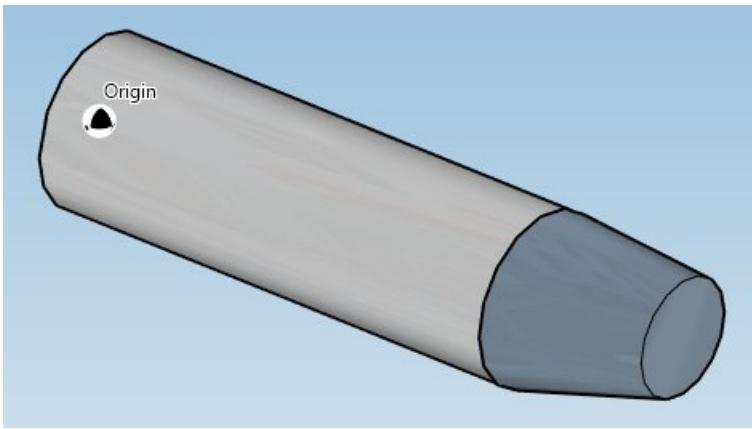
Concentric Create element

Angle



A red line shows a preview of the tangent side and length of the reducer.

You can click on the **Create element** to create a right aligned eccentric reducer :

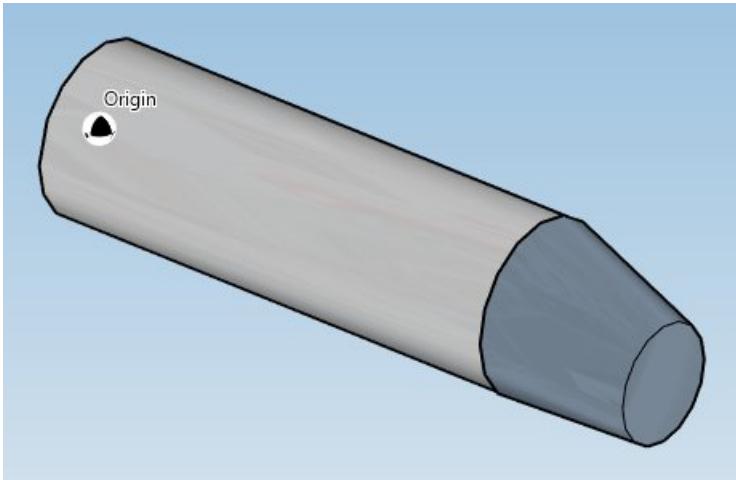


You can **undo** this command.

Another example with **Angle = 270 °** :

The screenshot shows a CAD software interface. On the left, a dark gray panel titled "Select how to draw the element" contains three options: "Relative coordinates", "Polar coordinates", and "Distance + angle". The third option is selected, indicated by a radio button. Below this are two input fields: "D" with the value "0.1397" and "Angle" with the value "270". To the right of these fields is a yellow square icon labeled "Concentric". At the bottom right of the panel is a blue "Create element" button. To the right of the panel is the 3D model of the pipe. A small red square marker is placed on the pipe's surface, and the number "20" is written next to it, indicating the distance from the "Origin" hole to the marker.

You can click on the **Create element** to create a bottom aligned eccentric reducer :



You can **undo** this command.

2. Modify/Remove a reducer

Change the **Selection mode** to ELEMENT and select a reducer :

Reducer

Set reducer properties

Reducer properties

Length	0.1397	Angle	0
Flexibility	0		
Min(r1,r2)	0	Min(L1,L2)	0

Local X-axis direction

Xx	
Xy	
Xz	

Section and material

Specification	Specification 50
Section	6" Sch 40
Material	P 265 GH -

Section and material

Specification	Specification 50
Section	4" Sch 40
Material	P 265 GH -

Long weld properties

Type	Butt Weld Flush
Mismatch	4.4

Modify
Remove

Click [here](#) for more information about the selection tool.

REDUCER PROPERTIES :

You can change the properties of the selected reducer (except the length) :

Property	Unit Metric	Unit USA	Remark
Length	m	ft	
Angle	°	°	
Flexibility	-	-	
Min(r1,r2)	mm	in	Minimum radius r1,r2 : B31J, Class 1 and Classes 2 & 3 Revision A94 for C2 of equations X and Z
Min(L1,L2)	m	ft	Minimum tangent L1, L2 : for B31J, Class 1 only

You can change the **flexibility factor** of the reducer. If blank, default value = 1.

X-AXIS DIRECTION :

You can change the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

SECTION AND MATERIAL :

You can change the **specification** and **section/material** of the two extremities of the reducer.

LONG WELD :

You can change the **Long weld properties** :

For the type, you can choose between :

- None
- Butt weld flush
- Butt weld as welded

Based on this property, define the **Long weld mismatch** [mm or in].

In this example, the default values correspond to the first section preset.

Click on the **Modify** button to change the selected reducer with these new properties.

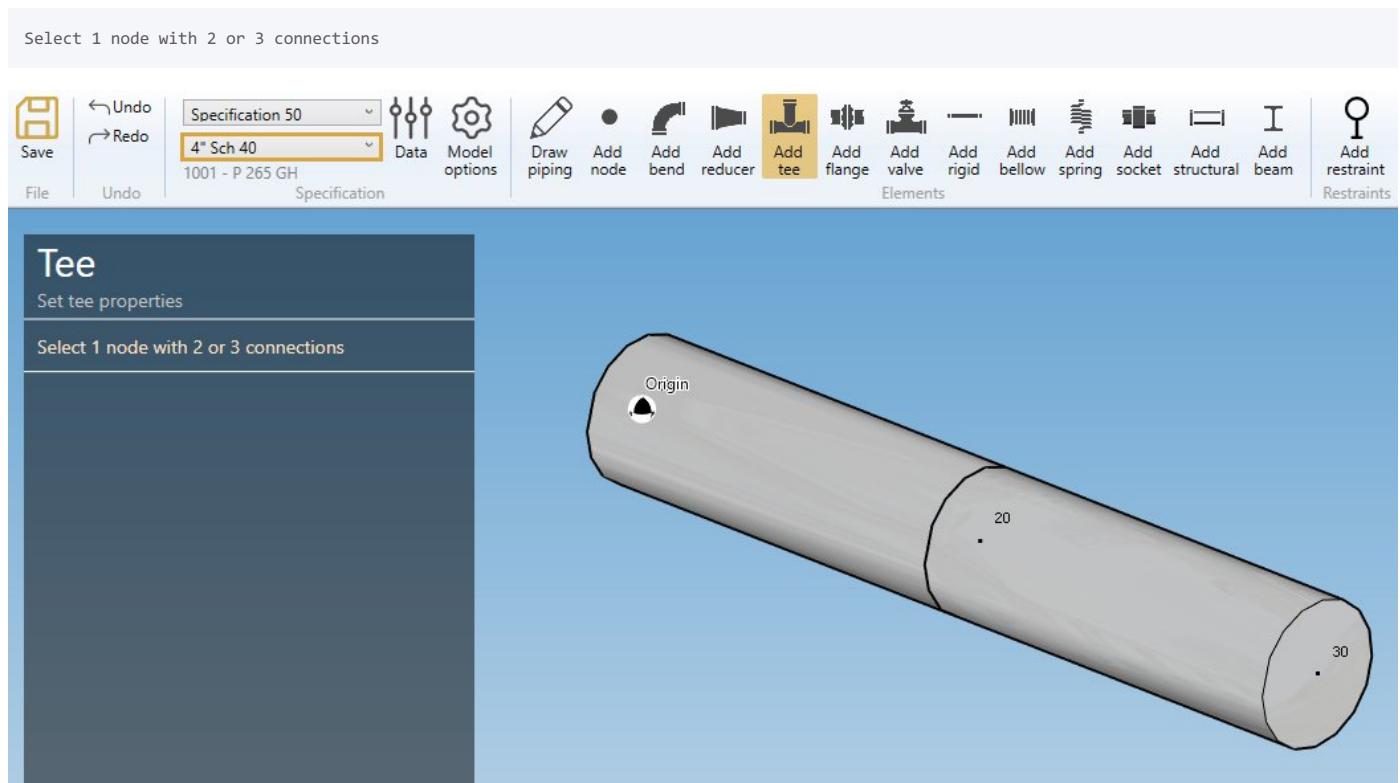
You can **undo** this command.

Click on the **Remove** button to delete the selected reducer.

You can **undo** this command.

Create tees

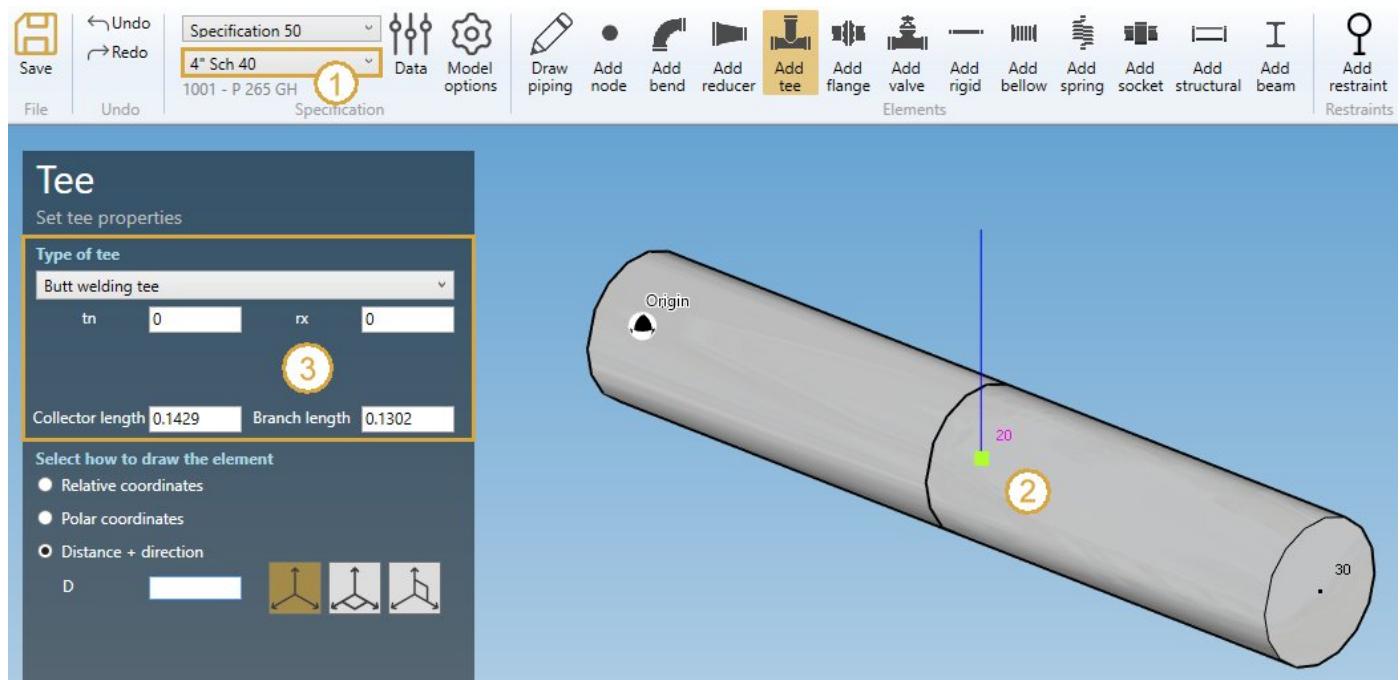
When you click on the **Add tee** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

1. Create a tee from an intermediate node

- Select the current **section/material** in the specification box (1).
- Select a **node** (2).
- Click the **Add tee** button



The left panel is the same if you click on the **Draw piping** button

TEE PROPERTIES :

Type of tee :

- Branch connection
- WeldingTee
- Non Standard
- Fabricated
- Sweepolet flush weld
- Sweepolet as welded
- Weldolet
- Extruded weldingTee
- Welded-in contour insert
- Branch welded-on fitting
- Lateral connection
- Partial penetration branch connection

Based on the type, some other properties must be filled :

Property	Description	Unit Metric	Unit USA
L1	Branch reinforcement length	mm	in
R2	Branch-to-run fillet radius	mm	in
TN	Branch connections and lateral connections CONSTANT thickness	mm	in
Y	Branch connections and lateral connections VARIABLE thickness	mm	in
Thickness type	Variable or constant	-	-
RE	Pad or saddle outer radius for reinforced fabricated tees	mm	in
RX	Transition radius	mm	in
PD	Pad thickness for reinforced fabricated tees	mm	in
Collector length	Half length of the header of welding tees	m	ft
Branch length	Branch length of welding tees	m	ft

MetaPiping automatically shows the necessary properties for each type.

The Type and properties come from the **current specification** :

Tees												
6"	Branch size	Branch type	L1	R2	TN or Y	Thickness type	RE	RX	PD	Collector length	Branch length	Mass
	6"	Branch connect	0	0	0	Constant						
	4"	WeldingTee			0			0		0.1429	0.1302	0
	2"	Non Standard										

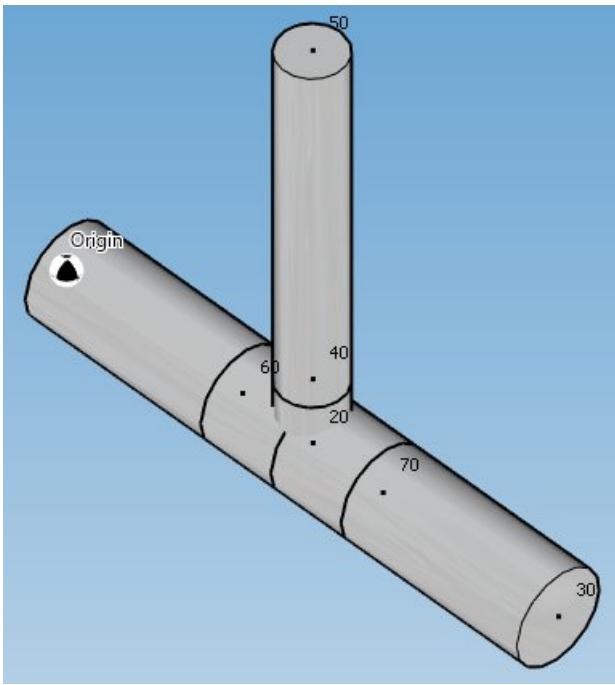
You can change all values before creating the tee.

ORIENTATION :

You can then define the second point of the tee thanks to the **Orientation tool**.

Click [here](#) for more information about the orientation tool.

After creation of the tee, you can **undo** the command.



Example of welding tee

2. Modify a tee

Change the **Selection mode** to TEE and select a node intersecting 3 or more pipes :

The image shows a screenshot of a software interface for modifying a tee. On the left, a dark blue dialog box titled 'Tee' contains the heading 'Set tee properties' and a section 'Type of tee' with a dropdown menu set to 'Butt welding tee'. Below this are input fields for 'tn' (0) and 'rx' (0). Further down are 'Collector length' (0.1429) and 'Branch length' (0.1302). At the bottom is a 'Modify' button. To the right of the dialog is the 3D model of the welding tee. A small square highlight is placed on the node where the vertical pipe meets the horizontal pipe, indicating it is selected for modification.

Click [here](#) for more information about the selection tool.

You can change the type of tee and the corresponding properties.

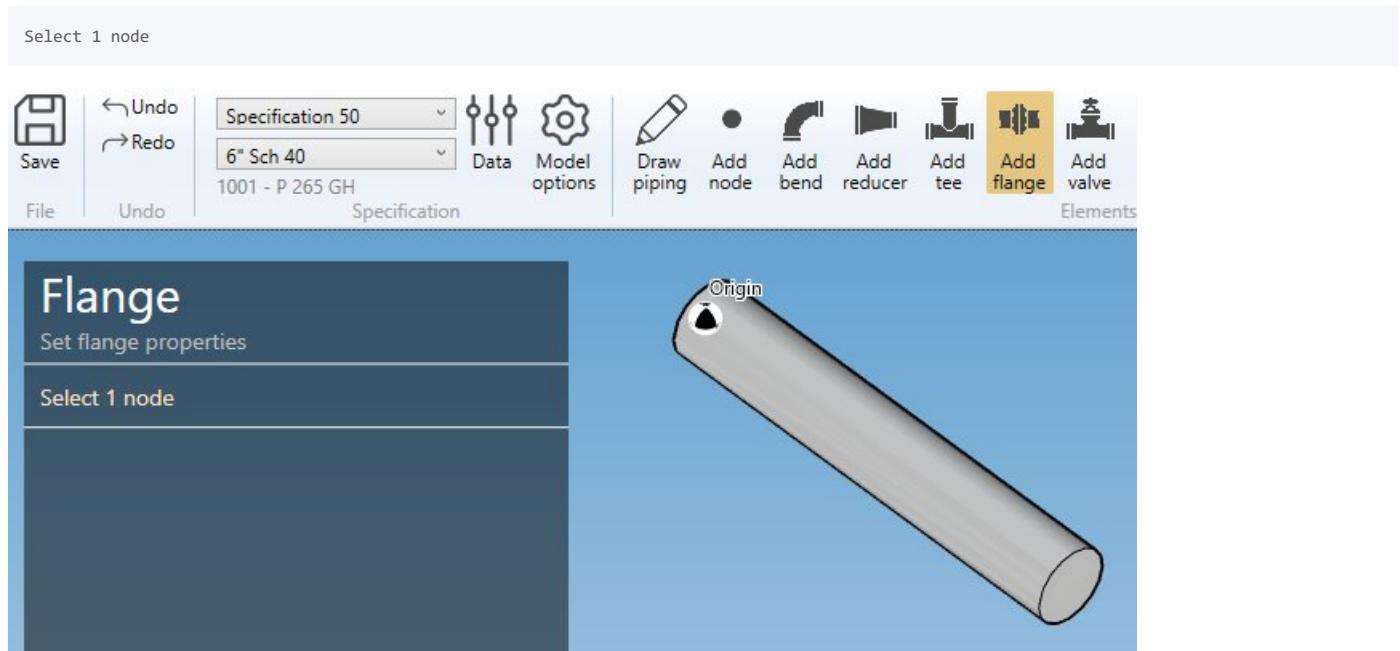
Click on the **Modify** button to modify the selected tee.

You can **undo** this command.



Create flanges

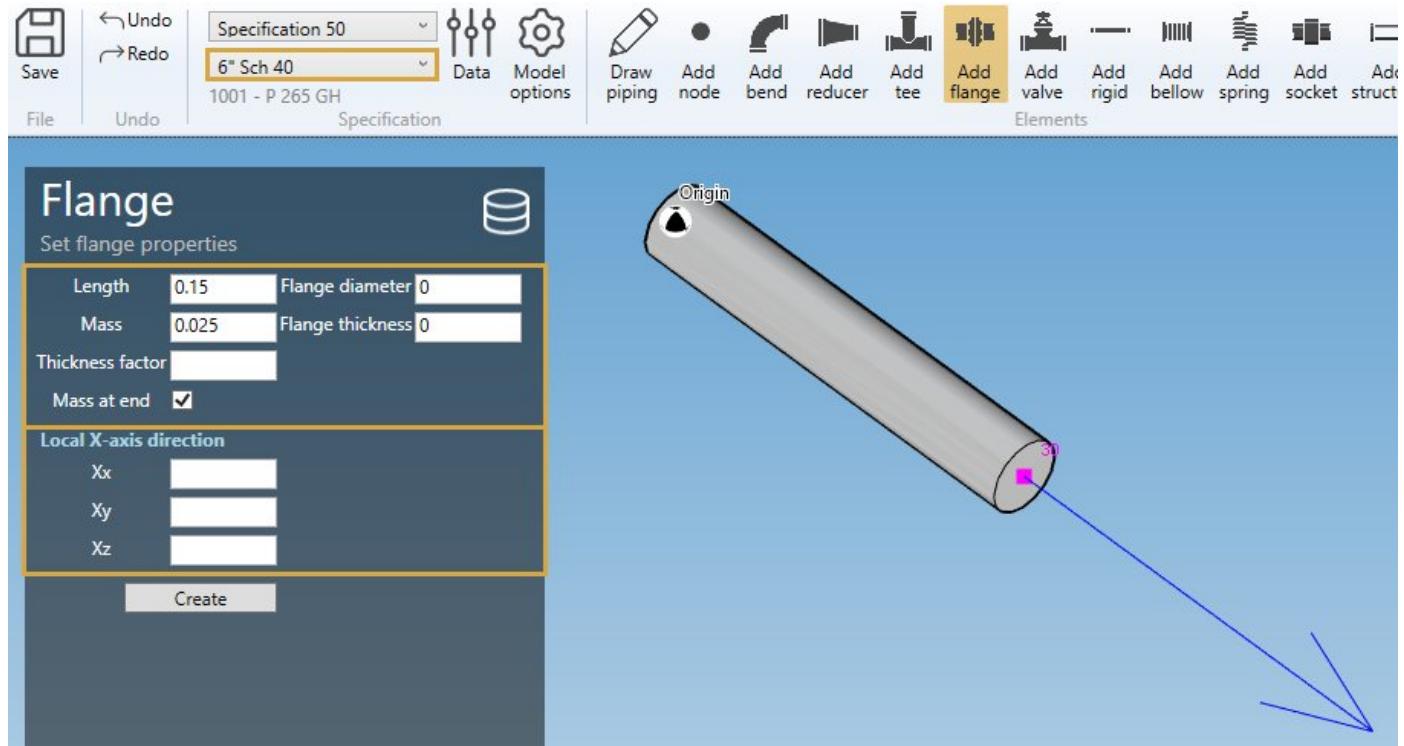
When you click on the **Add flange** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

1. Create a flange

- Select the current **section/material** in the specification box.
- Select a **node**.
- Click the **Add flange** button.



FLANGE PROPERTIES :

Define the properties of the new flange :

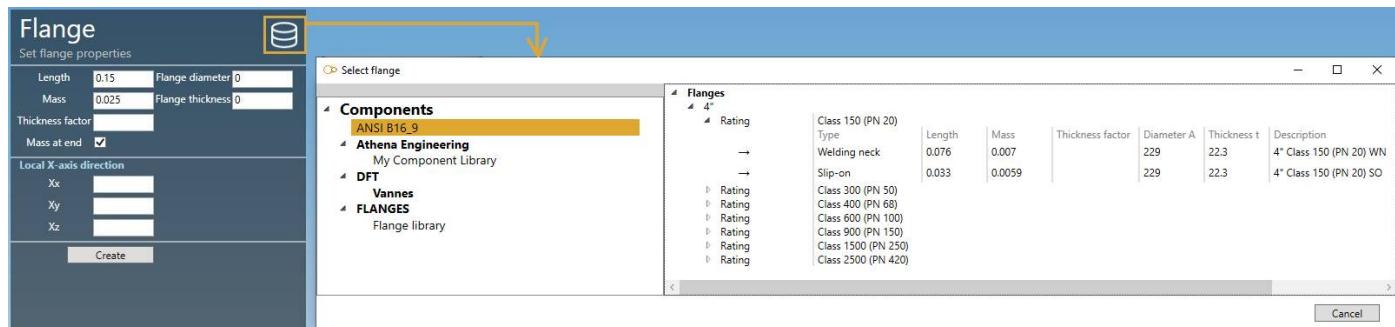
Property	Unit Metric	Unit USA
Length	m	ft
Mass	ton	kips
Thickness factor	-	-
Diameter of the flange	mm	in
Thickness of the flange	mm	in
Mass at end	-	-

The wall thickness is increased by the thickness factor by equally increasing the outer diameter and decreasing the inner diameter, keeping the

The Length and Mass come from the **current specification** :

Flanges	Size	Nominal pressure	Type	Length	Mass
	6"	50	Socket welding	0.15	0.025
	4"	50	Welding neck	0.08	0.003
	2"	50	Welding neck	0.07	0.0025

To save time, you can also directly select a flange from **database** :



Click on the **Database** button, select a **Library** and a **Flange**. The **OK** button will appear.

Flanges	4"	Rating	Class 150 (PN 20)	Type	Length	Mass	Thickness factor	Diameter A	Thickness t	Description
	→	Welding neck	0.076	0.007			229	22.3	22.3	4" Class 150 (PN 20) WN
	→	Slip-on	0.033	0.0059			229	22.3	22.3	4" Class 150 (PN 20) SO

The **Rating** and the **Type** is just informative.

The tables will be filled automatically :

Click [here](#) for more information about creation of library of components.

X-AXIS DIRECTION :

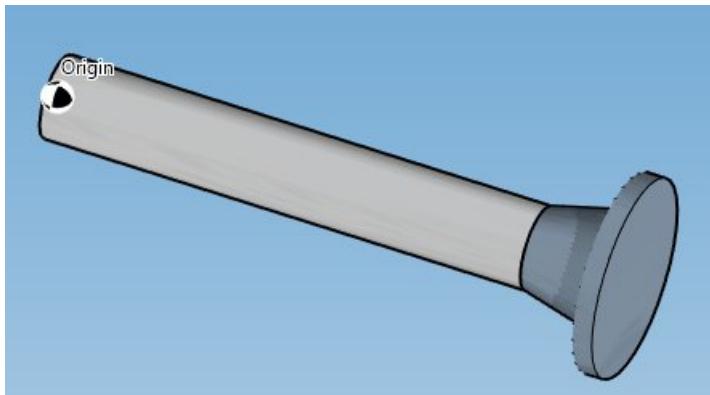
You can define the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

Label :

You can define a label to this element. The labels are shown with the node names view button.



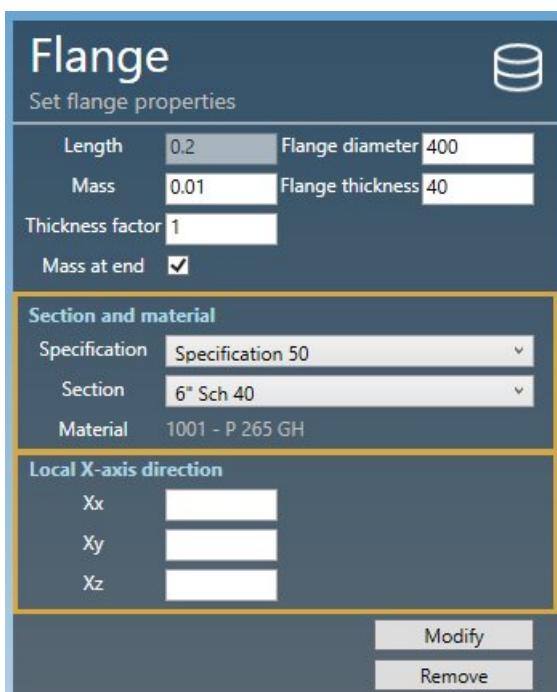
Click on the **Create** button to create the flange.



You can **undo** this command.

2. Modify/Remove a flange

Change the **Selection mode** to ELEMENT and select a flange :



Click [here](#) for more information about the selection tool.

You can change the properties of the selected flange (except the length).

SECTION AND MATERIAL :

You can change the **specification** and **section/material** of the flange.

X-AXIS DIRECTION :

You can change the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

Click on the **Modify** button to change the selected flange with these new properties.

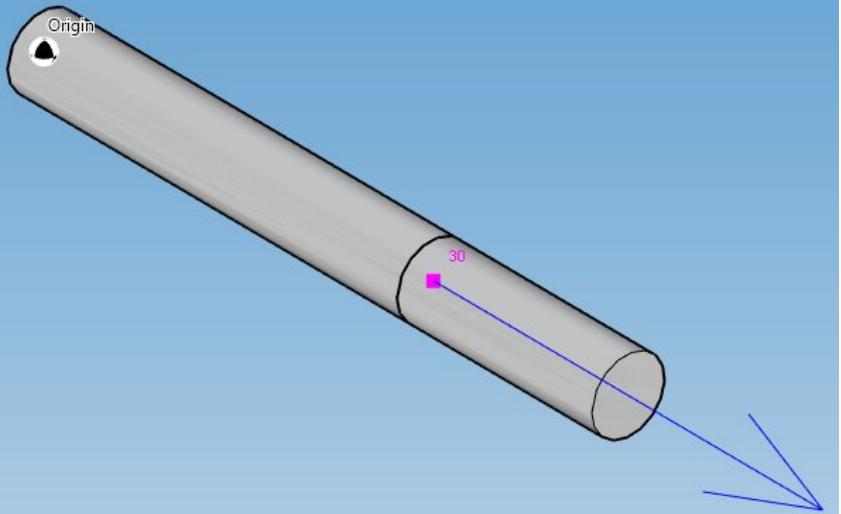
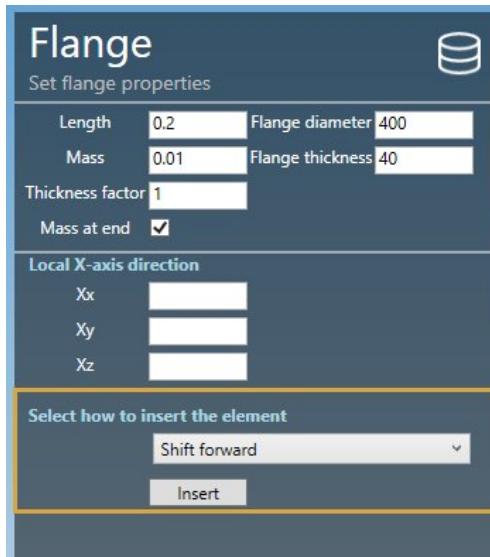
You can **undo** this command.

Click on the **Remove** button to delete the selected flange.

You can **undo** this command.

3. Insert a flange on an intermediate node

Click on the **Add flange** button and select an **intermediate node** between 2 elements.



Fill the properties (see §1) and select the **insertion mode** :

- Shift forward
- Shift backwards
- Reduce the next element
- Reduce the previous element
- Symmetrically reduce the neighboring elements

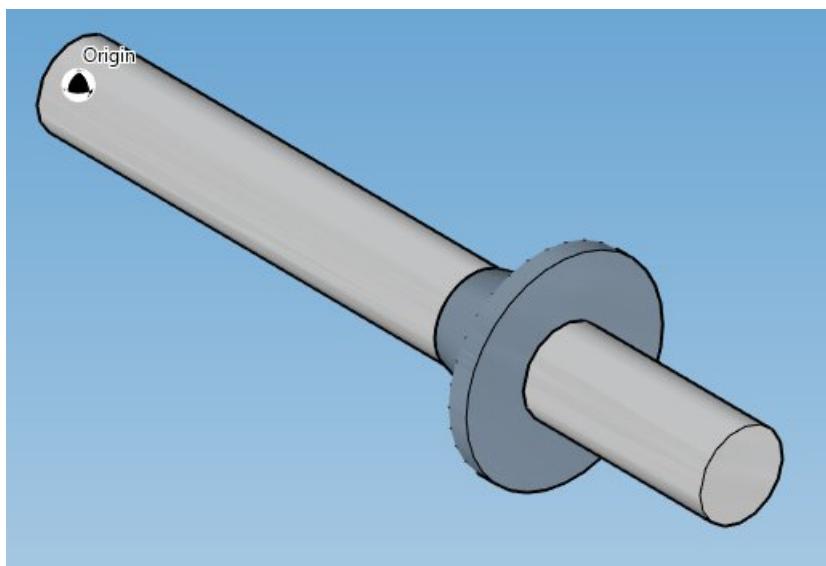
WARNING

ATTENTION, if the length is null, no mode will be proposed (empty list)

WARNING

Based on the length of the flange and the lengths of the neighboring elements, some mode could be hidden.

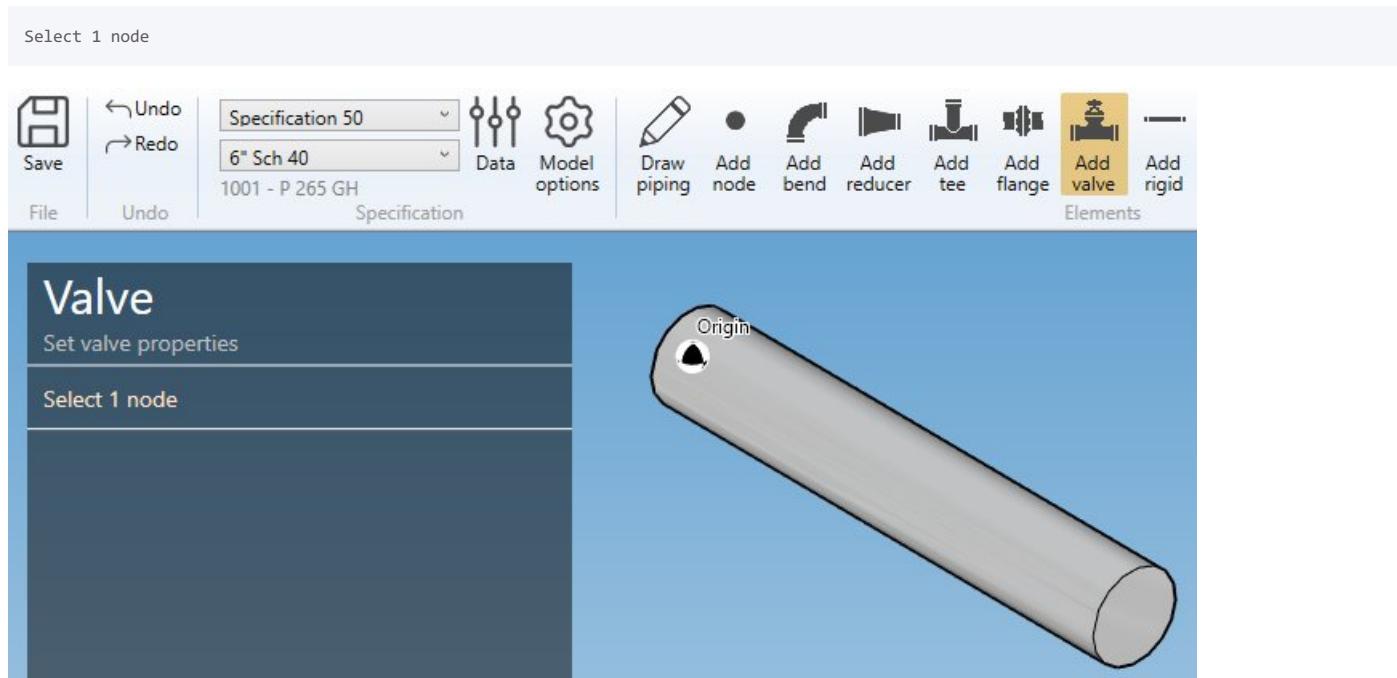
Select for example "Symmetrically reduce the neighboring elements" and click the **Insert** button :



You can **undo** this command.

Create valves

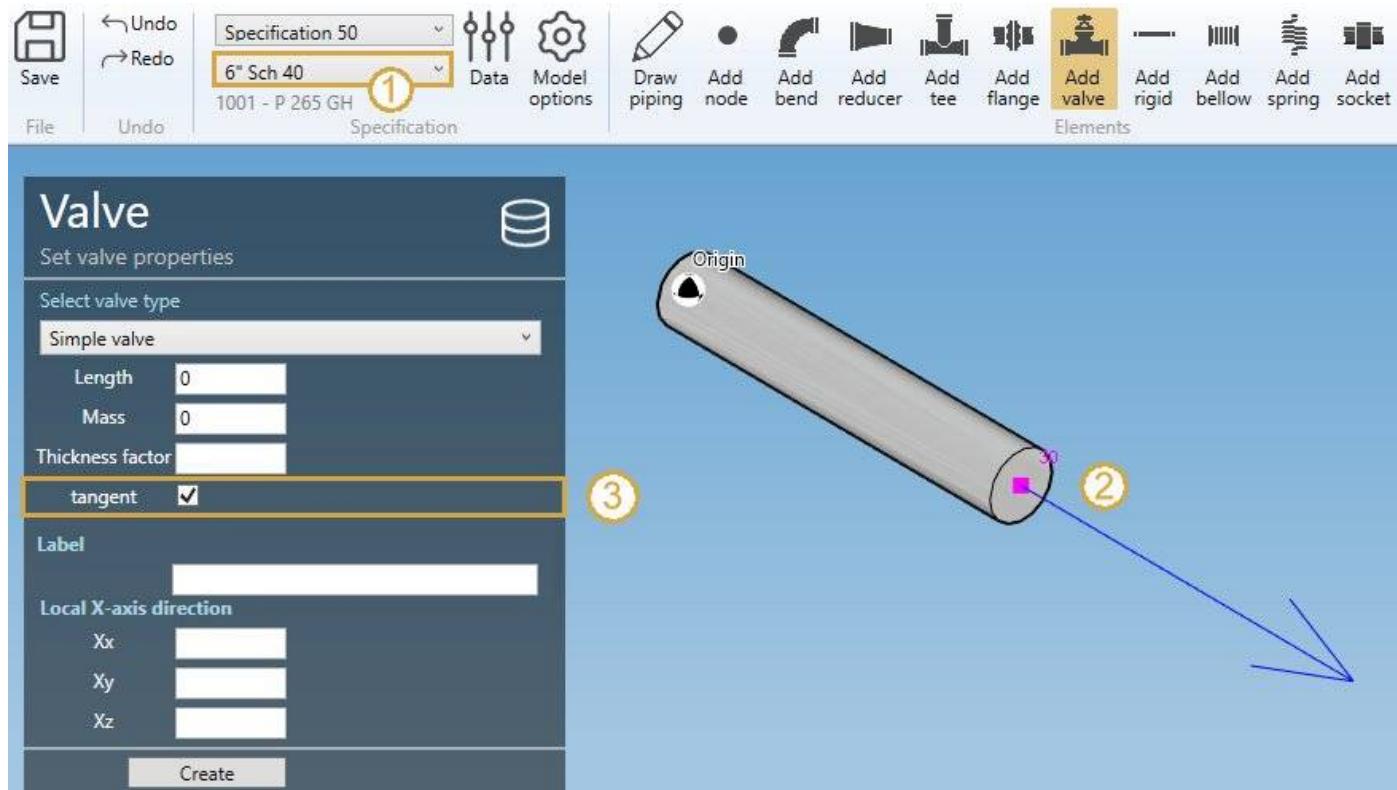
When you click on the **Add valve** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

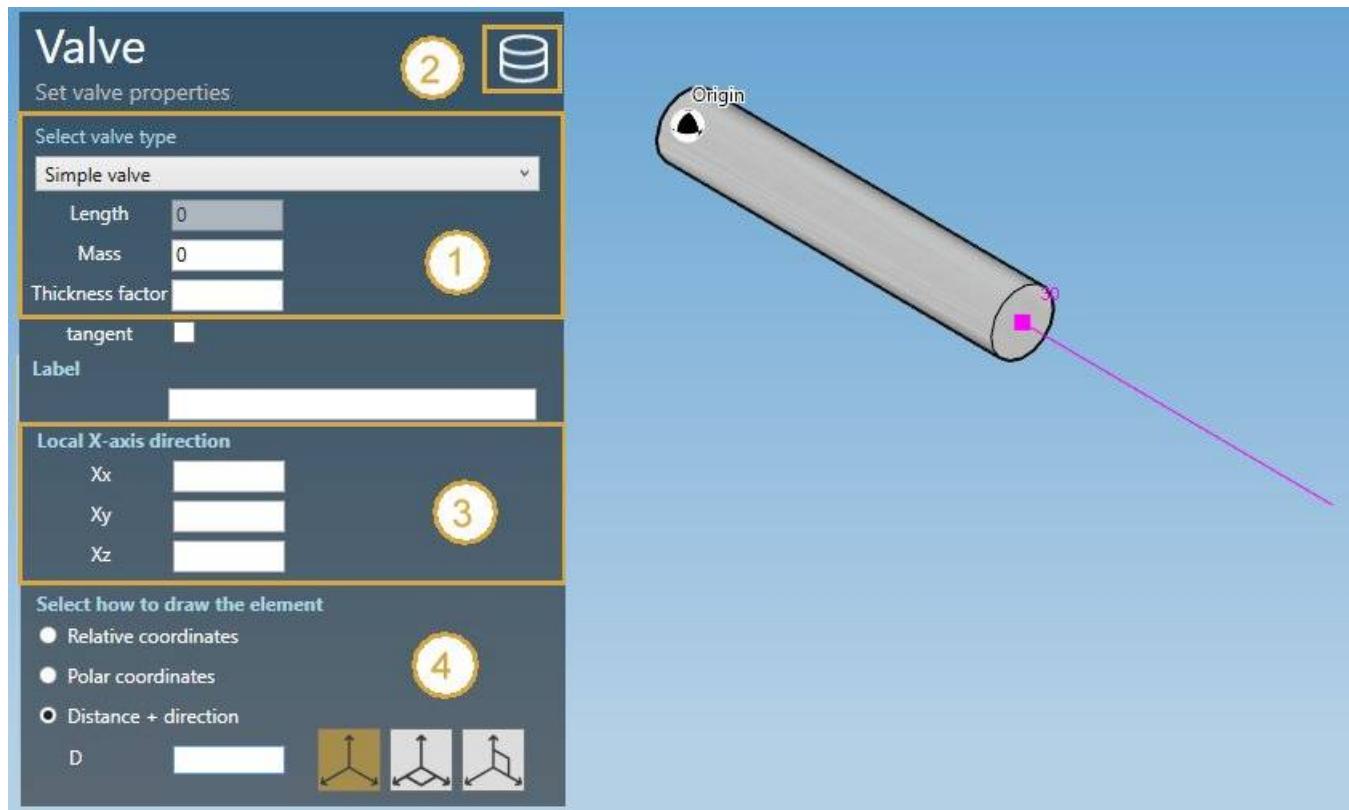
1. Create a valve

- Select the current **section/material** in the specification box.
- Select a **node**.
- Click the **Add valve** button.



By default, the orientation of the valve is defined to **Tangent** (3) represented by a blue arrow.

You can decide to use the **orientation tool** by unchecking the option :



VALVE PROPERTIES (1) :

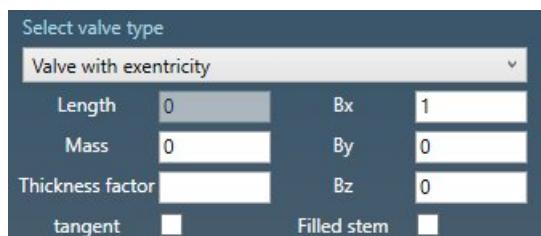
You have the choice between 3 types of valve :

- Simple valve
- Valve with middle point
- Valve with eccentricity

Property	Unit Metric	Unit USA
Length	m	ft
Mass	ton	kips
Thickness factor	-	-

The length must be defined by the **orientation tool**.

The valve with eccentricity has 4 more options :

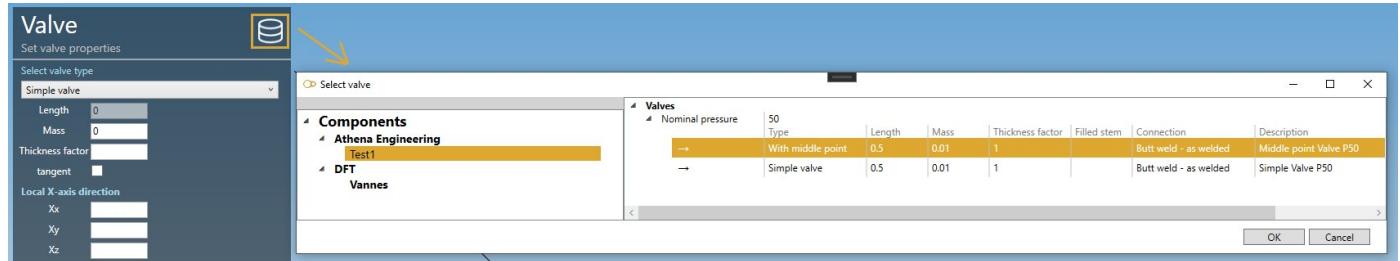


Property	Unit Metric	Unit USA
Bx	m	ft
By	m	ft
Bz	m	ft
Filled stem	-	-

The vector B (Bx, By, Bz) represents the eccentricity in **global coordinates**.

DATABASE (2) :

To save time, you can also directly select a valve from **database** :

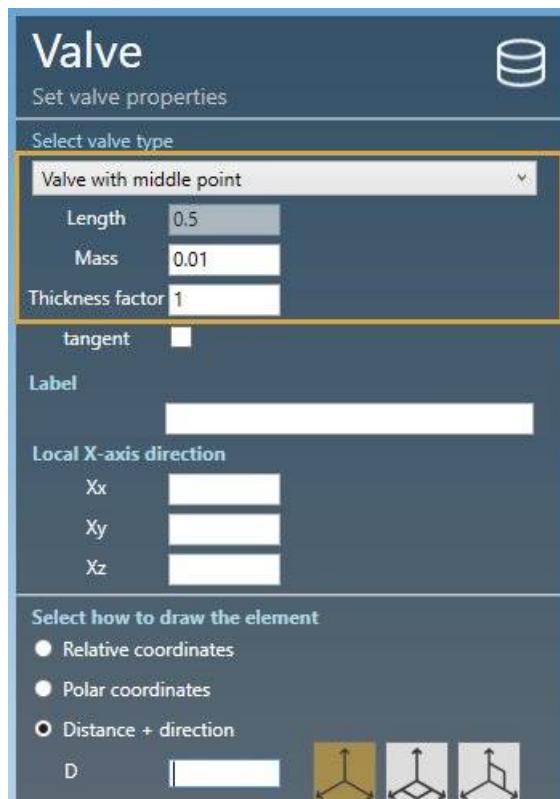


Click on the **Database** button, select a **Library** and a **Valve**. The **OK** button will appear.

Valves	Nominal pressure	Type	Length	Mass	Thickness factor	Filled stem	Connection	Description
→	50	With middle point	0.5	0.01	1		Butt weld - as welded	Middle point Valve P50
→		Simple valve	0.5	0.01	1		Butt weld - as welded	Simple Valve P50

The **Nominal pressure**, the **Type** and the **Connection** are just informative.

The tables will be filled automatically :



Click [here](#) for more information about creation of library of components.

LABEL :

You can define a label to the valve. The labels are shown with the node names view button.

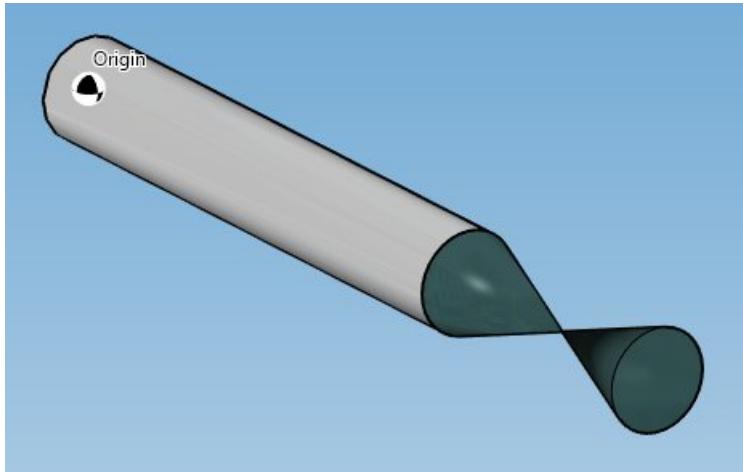
X-AXIS DIRECTION (3) :

You can define the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

You can then define the second point of the valve thanks to the **Orientation tool**.

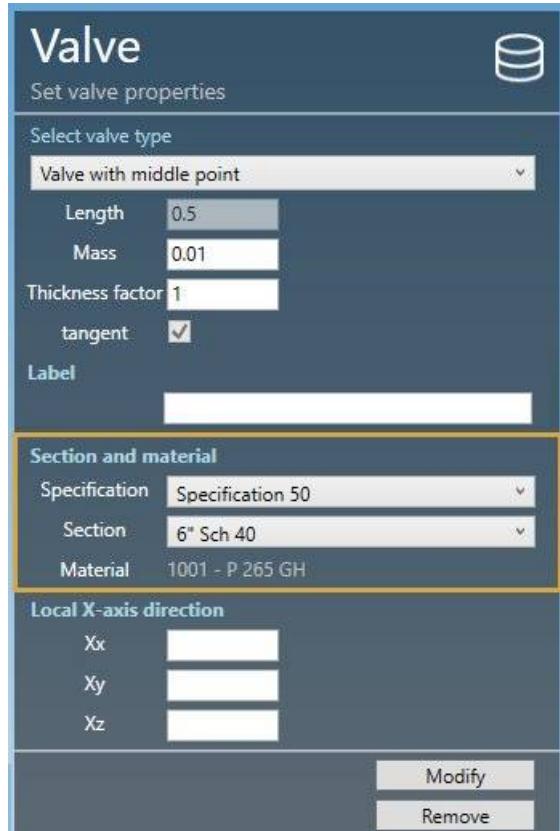
Click [here](#) for more information about the orientation tool.

Create the valve :



2. Modify/Remove a valve

Change the **Selection mode** to ELEMENT and select a valve :



Click [here](#) for more information about the selection tool.

You can change the properties of the selected valve (except the length).

LABEL :

You can modify the label of the valve. The labels are shown with the node names view button.

SECTION AND MATERIAL :

You can change the **specification** and **section/material** of the valve.

X-AXIS DIRECTION :

You can change the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

Click on the **Modify** button to change the selected valve with these new properties.

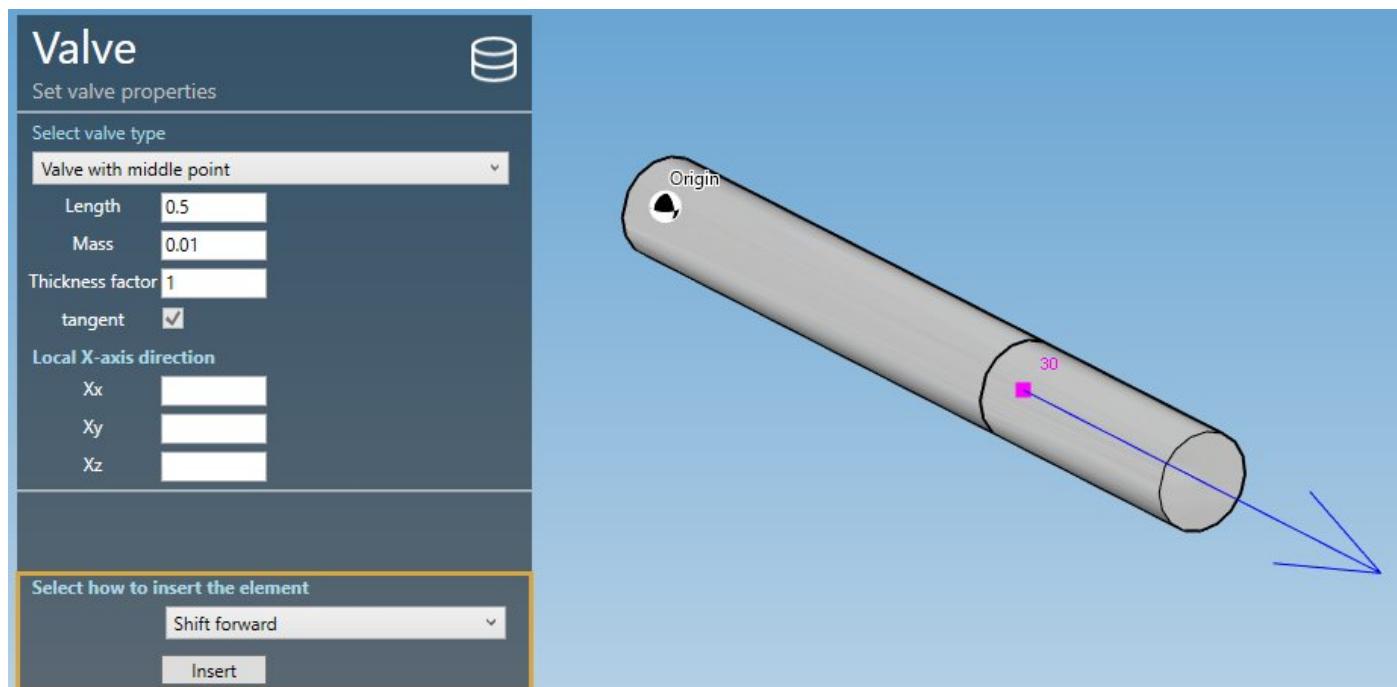
You can **undo** this command.

Click on the **Remove** button to delete the selected valve.

You can **undo** this command.

3. Insert a valve on an intermediate node

Click on the **Add valve** button and select an **intermediate node** between 2 elements.



Fill the properties (see §1) and select the **insertion mode** :

- Shift forward
- Shift backwards
- Reduce the next element
- Reduce the previous element
- Symmetrically reduce the neighboring elements

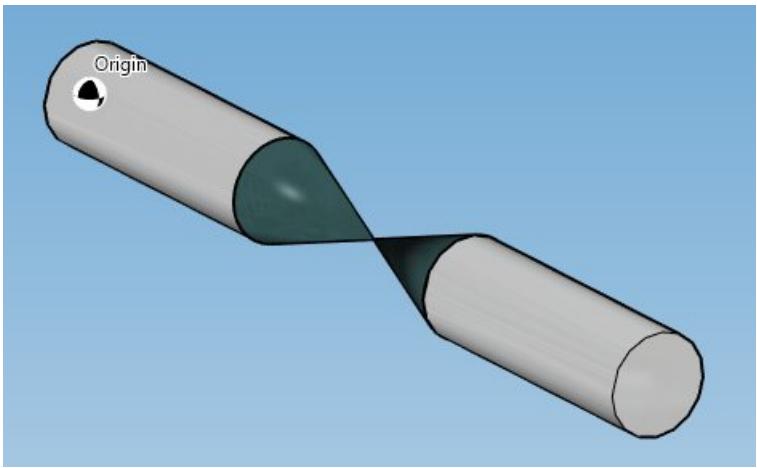
WARNING

ATTENTION, if the length is null, no mode will be proposed (empty list)

WARNING

Based on the length of the valve and the lengths of the neighboring elements, some mode could be hidden.

Select for example "Reduce the previous element" and click the **Insert** button :

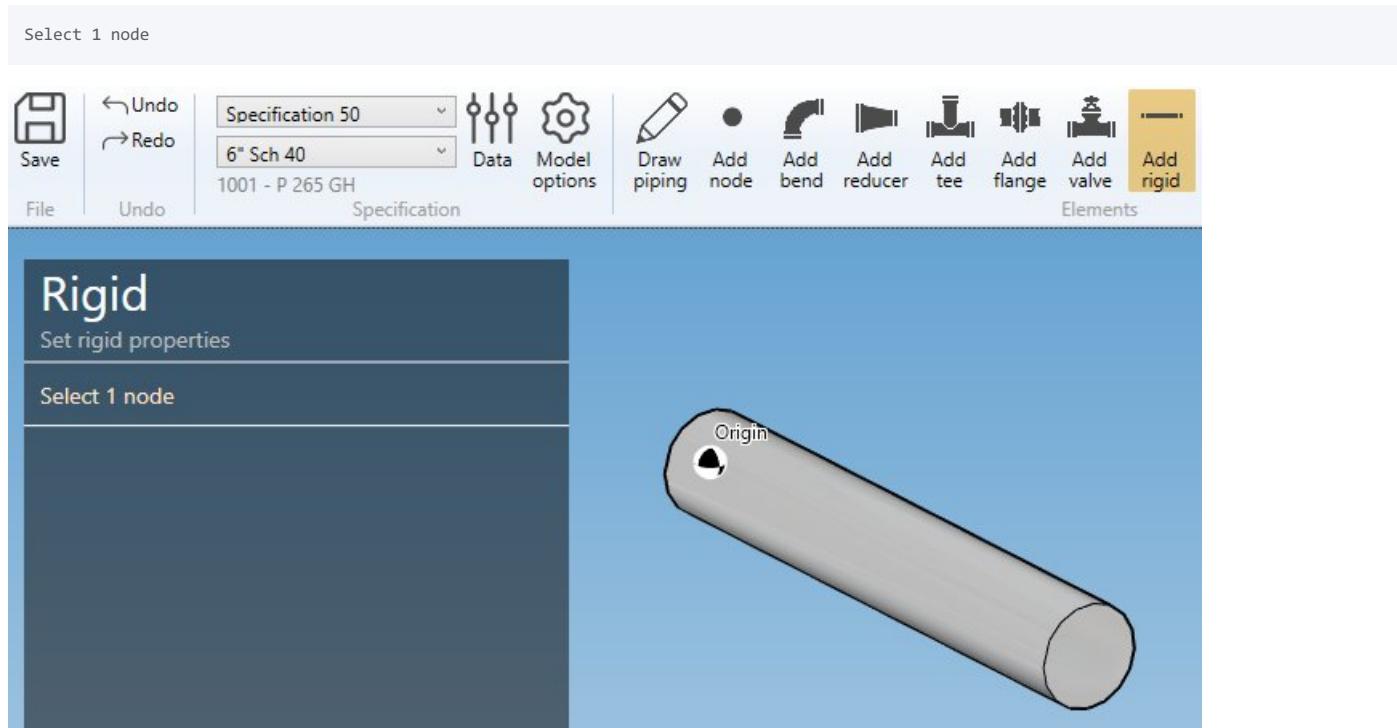


You can **undo** this command.

Create rigids

A rigid is a straight element sufficiently stiff to rigidly transmit all deflections and rotations.

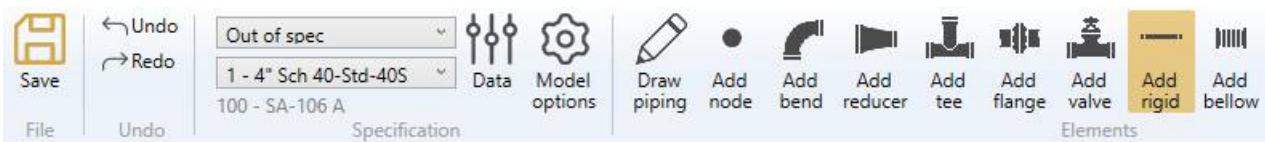
When you click on the **Add rigid** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

1. Create a rigid

- Select a **node**.
- Click the **Add rigid** button.



Rigid

Set rigid properties

Length: 0

Mass: 0

Material:

Name: 100 - SA-106 A

Local X-axis direction:

Xx, Xy, Xz

Select how to draw the element:

- Relative coordinates (radio button selected)
- Polar coordinates
- Distance + direction

D: [input field] [Orientation icons]

RIGID PROPERTIES :

Property	Unit Metric	Unit USA
Length	m	ft
Mass	ton	kips

The length must be defined by the **orientation tool**.

MATERIAL :

Select the material of the rigid.

Click [here](#) for more information about the materials definition.

X-AXIS DIRECTION :

You can define the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

Label :

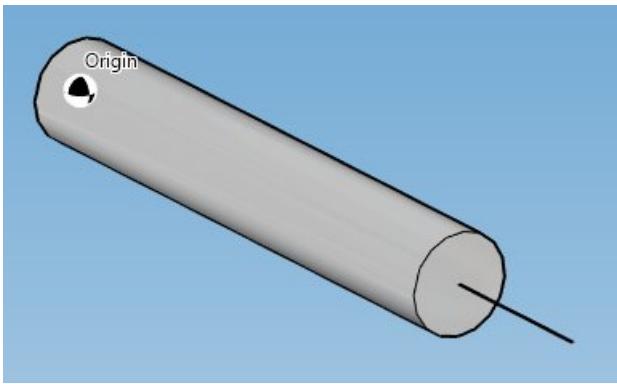
You can define a label to this element. The labels are shown with the node names view button.



You can then define the second point of the rigid thanks to the **Orientation tool**.

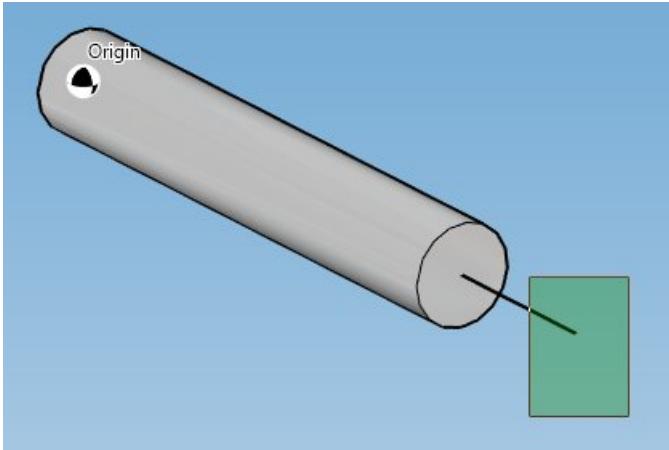
Click [here](#) for more information about the orientation tool.

Create the rigid (represented by a black line) :



2. Modify/Remove a rigid

Change the **Selection mode** to ELEMENT and select a rigid by dragging a selection rectangle:



Click [here](#) for more information about the selection tool.

You can change the properties of the selected rigid (except the length).

Rigid

Set rigid properties

Length	0.3
Mass	0

Material

Name	100 - SA-106 A
------	----------------

Local X-axis direction

Xx	<input type="button"/>
Xy	<input type="button"/>
Xz	<input type="button"/> Show/Hide

Buttons

Modify Remove

MATERIAL :

Change the material of the rigid.

X-AXIS DIRECTION :

You can change the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

Click on the **Modify** button to change the selected rigid with these new properties.

You can **undo** this command.

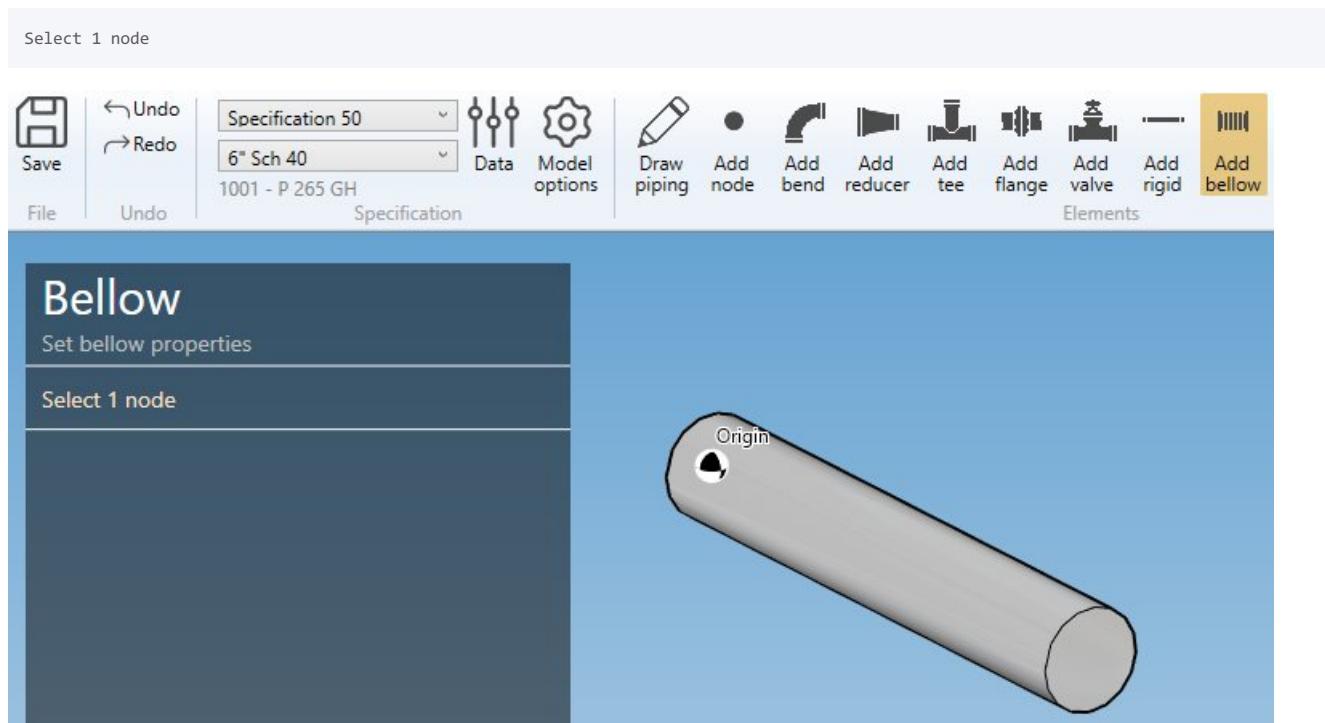
Click on the **Remove** button to delete the selected rigid.

You can **undo** this command.



Create bellows

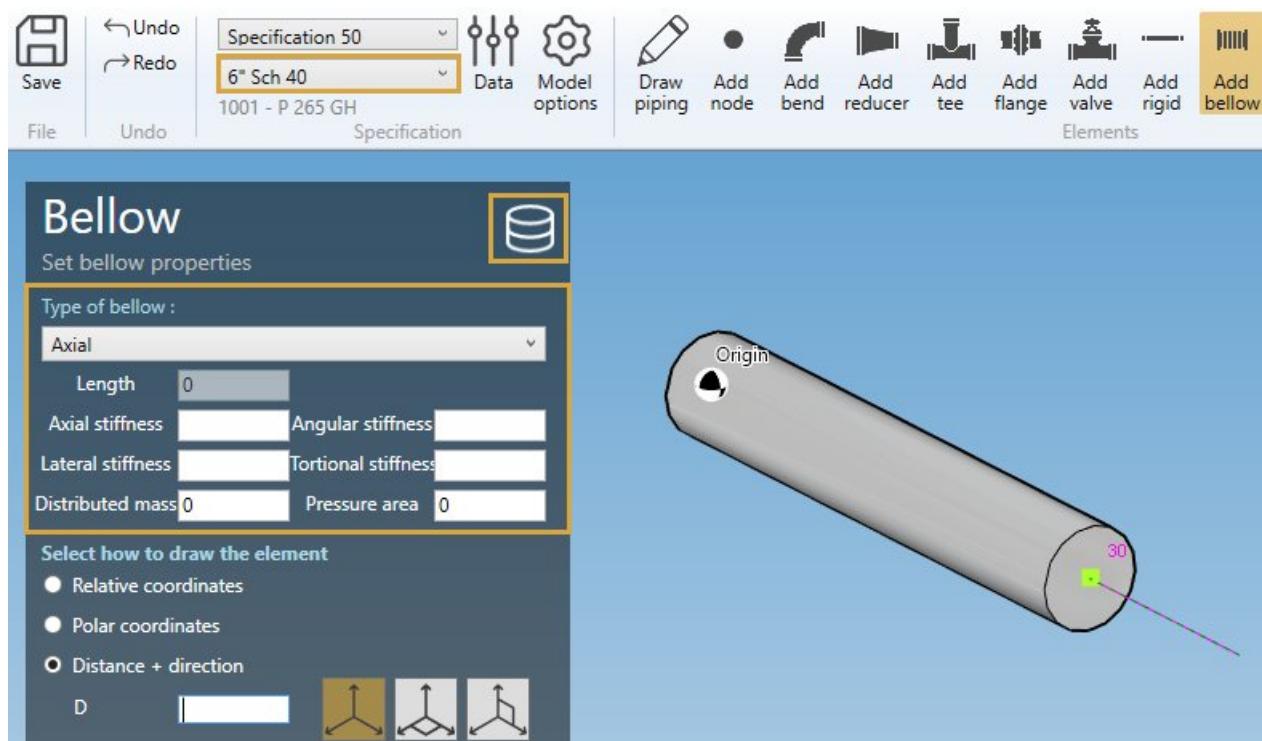
When you click on the **Add bellow** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

1. Create a bellow

- Select the current **section/material** in the specification box.
- Select a **node**.
- Click the **Add bellow** button.



BELLOW PROPERTIES :

- Axial : Axial and lateral displacements and rotations are permitted
- Lateral in a single plane : Lateral displacements are only permitted in the plane defined by the local axes Z' and X'. No axial displacements nor rotations are permitted
- Lateral in all planes : Lateral displacements are permitted in all directions perpendicular to the element axis Z'. No axial displacements nor rotations are permitted
- Angular in a single plane : Rotations are only permitted about the local axis Y'. The movements are thus restricted to the plane defined by the local axes Z' and X'
- Angular in all planes : Rotations are permitted about all axes perpendicular to Z'

The length must be defined by the **orientation tool**.

Based on **type**, you have to define several properties :

- Axial :

Type of bellow :

Axial		
Length	0	
Axial stiffness	<input type="text"/>	Angular stiffness <input type="text"/>
Lateral stiffness	<input type="text"/>	Torsional stiffness <input type="text"/>
Distributed mass	0	Pressure area <input type="text"/>

Property	Unit Metric	Unit USA
Length	m	ft
Axial stiffness	kN/mm	kips/in
Lateral stiffness	kN/mm	kips/in
Angular stiffness	kN.m/rad	kips.ft/rad
Torsional stiffness	kN.m/rad	kips.ft/rad
Distributed mass	ton	kips
Pressure area	mm ²	in ²

If Torsional stiffness is zero or blank, the joint is modeled as rigid in torsion and the user should verify that the moments about the element

One half of the mass will be concentrated at each end of the bellow.

- Lateral in a single plane

Type of bellow :

Lateral in a single plane		
Length	0	
Lateral stiffness	<input type="text"/>	Torsional stiffness <input type="text"/>
Distributed mass	0	
Local X-axis direction		
Xx	<input type="text"/>	
Xy	<input type="text"/>	
Xz	<input type="text"/>	

Property	Unit Metric	Unit USA
Length	m	ft
Lateral stiffness	kN/mm	kips/in
Torsional stiffness	kN.m/rad	kips.ft/rad
Distributed mass	ton	kips

If Torsional stiffness is zero or blank, the joint is modeled as rigid in torsion and the user should verify that the moments about the element centerline are balanced.

One half of the mass will be concentrated at each end of the bellow.

You can define the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

- Lateral in all planes

Type of bellow :

Lateral in all planes

Length 0

Lateral stiffness Torsional stiffness:

Distributed mass 0

Property	Unit Metric	Unit USA
Length	m	ft
Lateral stiffness	kN/mm	kips/in
Torsional stiffness	kN.m/rad	kips.ft/rad
Distributed mass	ton	kips

If Torsional stiffness is zero or blank, the joint is modeled as rigid in torsion and the user should verify that the moments about the element centerline are balanced.

One half of the mass will be concentrated at each end of the bellow.

- Angular in a single plane

Type of bellow :

Angular in a single plane

Length 0

Angular stiffness Torsional stiffness:

Distributed mass 0

Local X-axis direction

Xx

Xy

Xz

Property	Unit Metric	Unit USA
Length	m	ft
Angular stiffness	kN.m/rad	kips.ft/rad
Torsional stiffness	kN.m/rad	kips.ft/rad
Distributed mass	ton	kips

If Torsional stiffness is zero or blank, the joint is modeled as rigid in torsion and the user should verify that the moments about the element One half of the mass will be concentrated at each end of the bellow.

- Angular in all planes

Type of bellow :

Angular in all planes

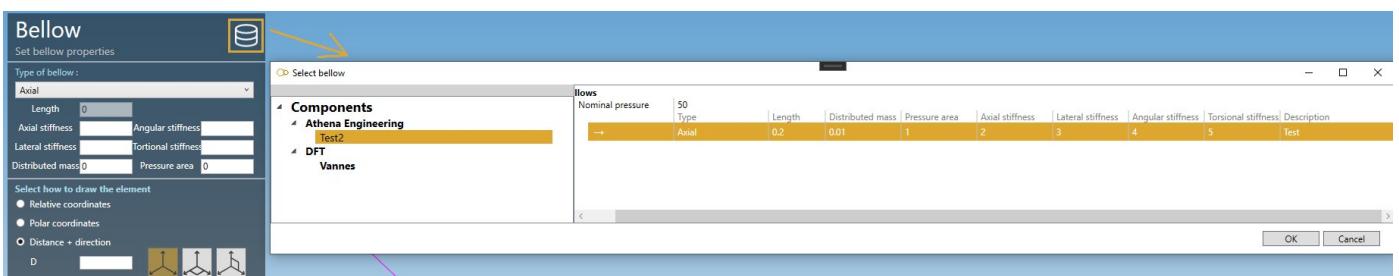
Length	0
Angular stiffness	
Torsional stiffness	
Distributed mass	0

Property	Unit Metric	Unit USA
Length	m	ft
Angular stiffness	kN.m/rad	kips.ft/rad
Torsional stiffness	kN.m/rad	kips.ft/rad
Distributed mass	ton	kips

If Torsional stiffness is zero or blank, the joint is modeled as rigid in torsion and the user should verify that the moments about the element One half of the mass will be concentrated at each end of the bellow.

DATABASE :

To save time, you can also directly select a bellow from **database** :



Click on the **Database** button, select a **Library** and a **Bellow**. The **OK** button will appear.

The tables will be filled automatically.

Click [here](#) for more information about creation of library of components.

Label :

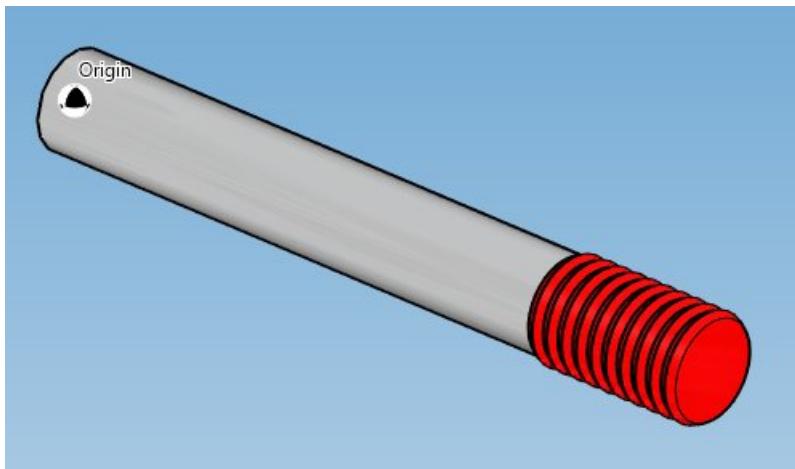
You can define a label to this element. The labels are shown with the node names view button.

Label

You can then define the second point of the bellow thanks to the **Orientation tool**.

Click [here](#) for more information about the orientation tool.

Create the bellow :



2. Modify/Remove a bellow

Change the **Selection mode** to ELEMENT and select a bellow :



Click [here](#) for more information about the selection tool.

You can change the type and the appropriate properties of the selected bellow (except the length).

SECTION AND MATERIAL :

You can change the **specification** and **section/material** of the bellow.

Click on the **Modify** button to change the selected bellow with these new properties.

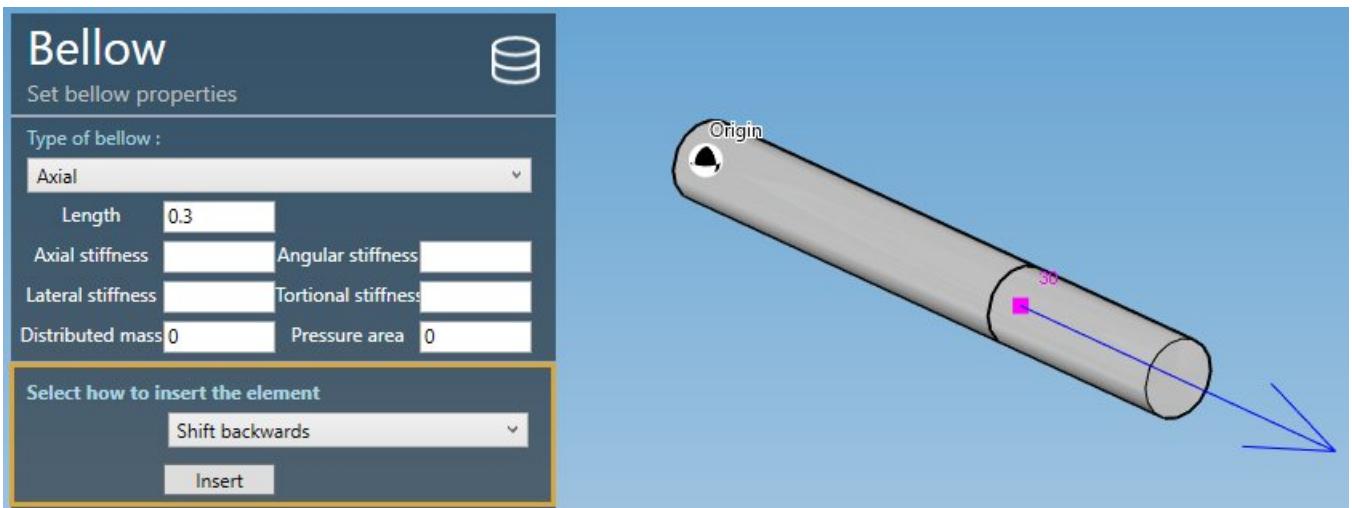
You can **undo** this command.

Click on the **Remove** button to delete the selected bellow.

You can **undo** this command.

3. Insert a bellow on an intermediate node

Click on the **Add bellow** button and select an **intermediate node** between 2 elements.



Fill the properties (see §1) and select the **insertion mode** :

- Shift forward
- Shift backwards
- Reduce the next element
- Reduce the previous element
- Symmetrically reduce the neighboring elements

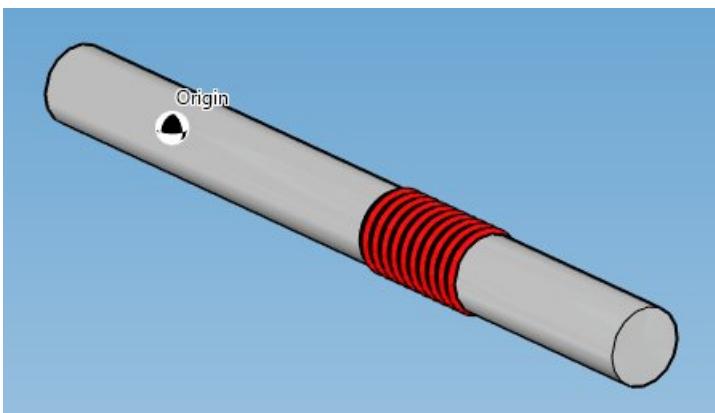
WARNING

ATTENTION, if the length is null, no mode will be proposed (empty list)

WARNING

Based on the length of the bellow and the lengths of the neighboring elements, some mode could be hidden.

Select for example "Shift backwards" and click the **Insert** button :

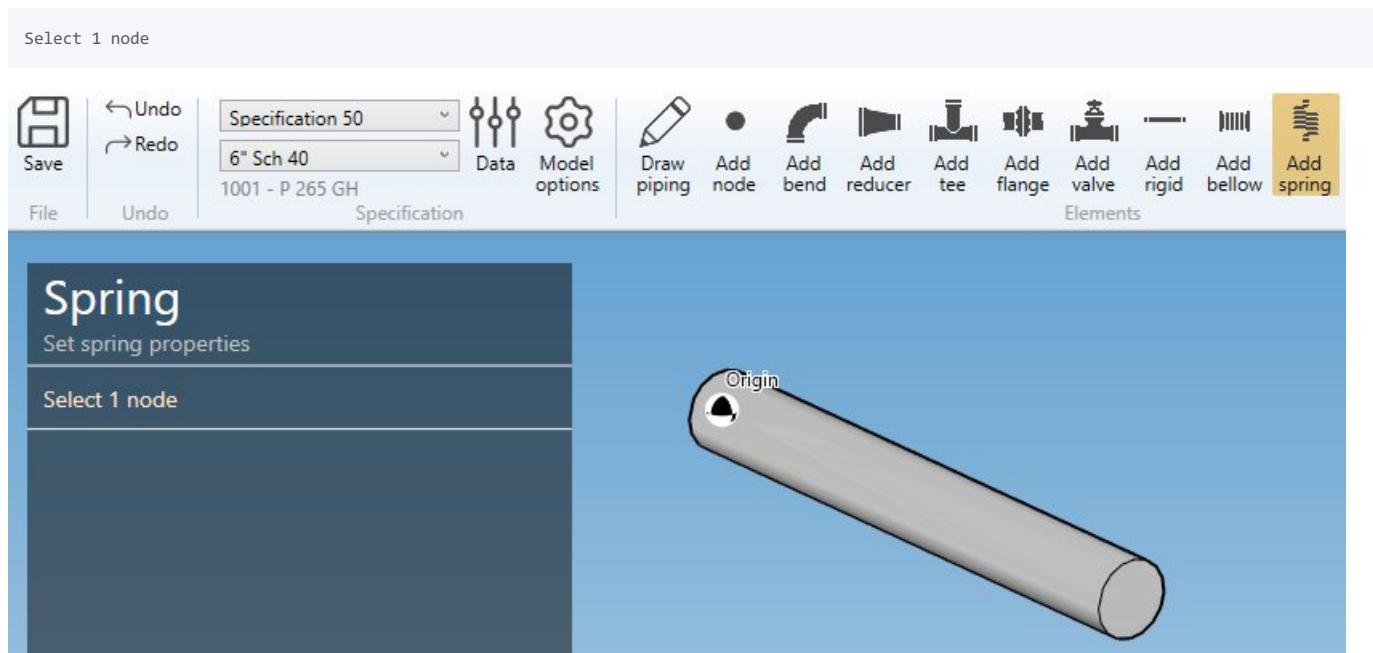


You can **undo** this command.



Create springs

When you click on the **Add spring** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

1. Create a spring

- Select the current **section/material** in the specification box.
- Select a **node**.
- Click the **Add spring** button.



Spring

Set spring properties

Length 0

Material
Name: 100 - SA-106 A

Local X-axis direction

Xx Xy Xz Use ← and →

Translation and rotation stiffness :

Kx	Krx
Ky	Kry
Kz	Krz

Select how to draw the element

- Relative coordinates
- Polar coordinates
- Distance + direction

D [] [] [] []

SPRING PROPERTIES :

Property	Description	Unit Metric	Unit USA
Length	-	m	ft
Kx	Translation stiffness in X direction	kN/mm	kips/in
Kx	Translation stiffness in Y direction	kN/mm	kips/in
Kx	Translation stiffness in Z direction	kN/mm	kips/in
Krx	Rotation stiffness in X direction	kN.m/rad	kips.ft/rad
Kry	Rotation stiffness in Y direction	kN.m/rad	kips.ft/rad
Krz	Rotation stiffness in Z direction	kN.m/rad	kips.ft/rad

The length must be defined by the **orientation tool**.

MATERIAL :

Select the material of the spring.

Click [here](#) for more information about the materials definition.

X-AXIS DIRECTION :

You can define the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

Label :

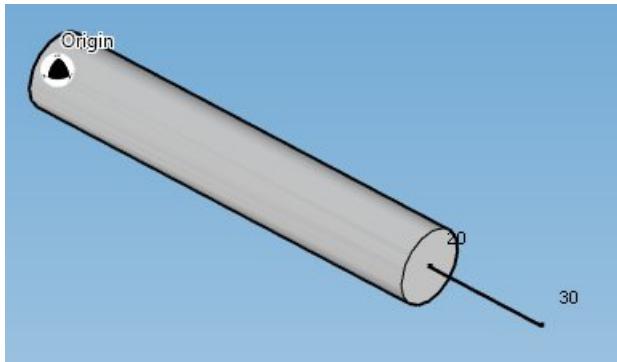
You can define a label to this element. The labels are shown with the node names view button.

Label []

You can then define the second point of the spring thanks to the **Orientation tool**.

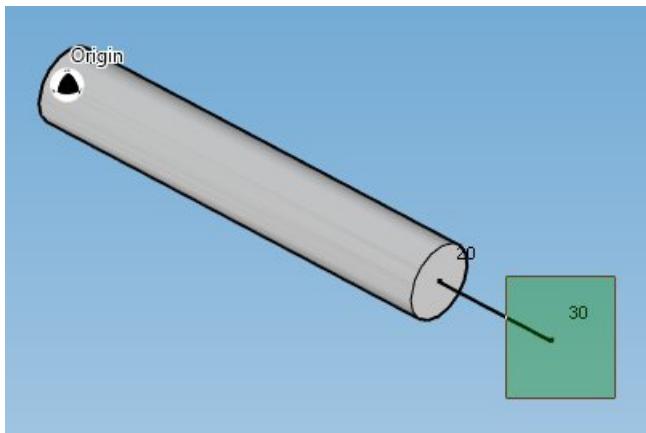
Click [here](#) for more information about the orientation tool.

Create the spring (represented by a black line) :



2. Modify/Remove a spring

Change the **Selection mode** to ELEMENT and select a spring by dragging a selection rectangle :



Click [here](#) for more information about the selection tool.

You can change the properties of the selected spring (except the length) :



Click on the **Modify** button to change the selected spring with these new properties.

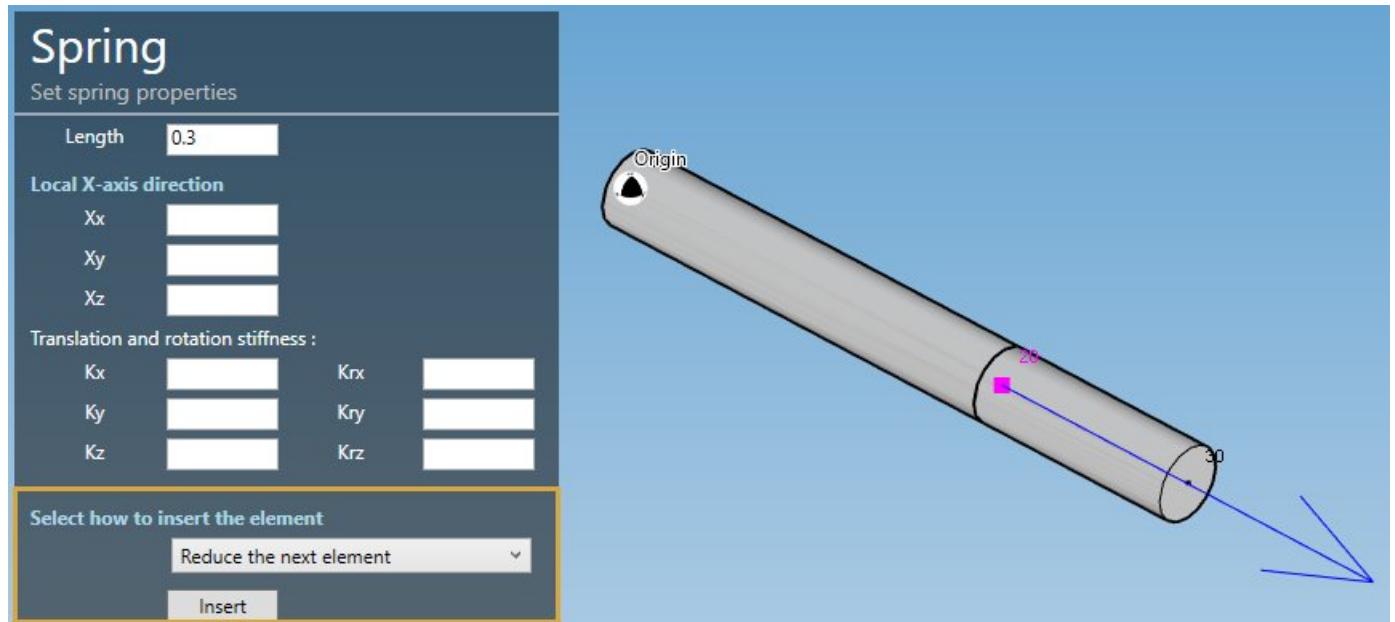
You can **undo** this command.

Click on the **Remove** button to delete the selected spring.

You can **undo** this command.

3. Insert a spring on an intermediate node

Click on the **Add spring** button and select an **intermediate node** between 2 elements.



Fill the properties (see §1) and select the **insertion mode** :

- Shift forward
- Shift backwards
- Reduce the next element
- Reduce the previous element
- Symmetrically reduce the neighboring elements

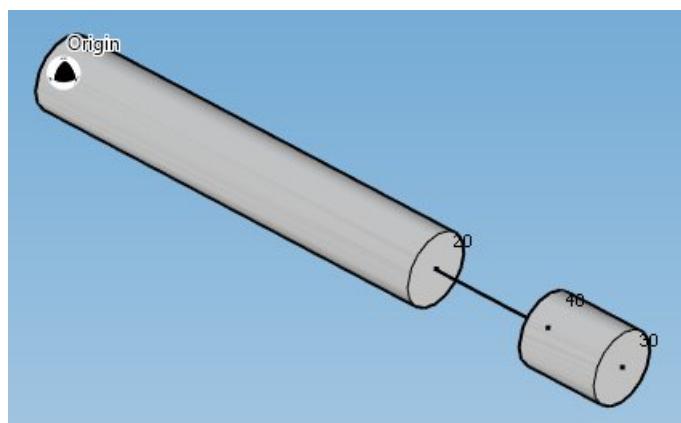
WARNING

ATTENTION, if the length is null, no mode will be proposed (empty list)

WARNING

Based on the length of the spring and the lengths of the neighboring elements, some mode could be hidden.

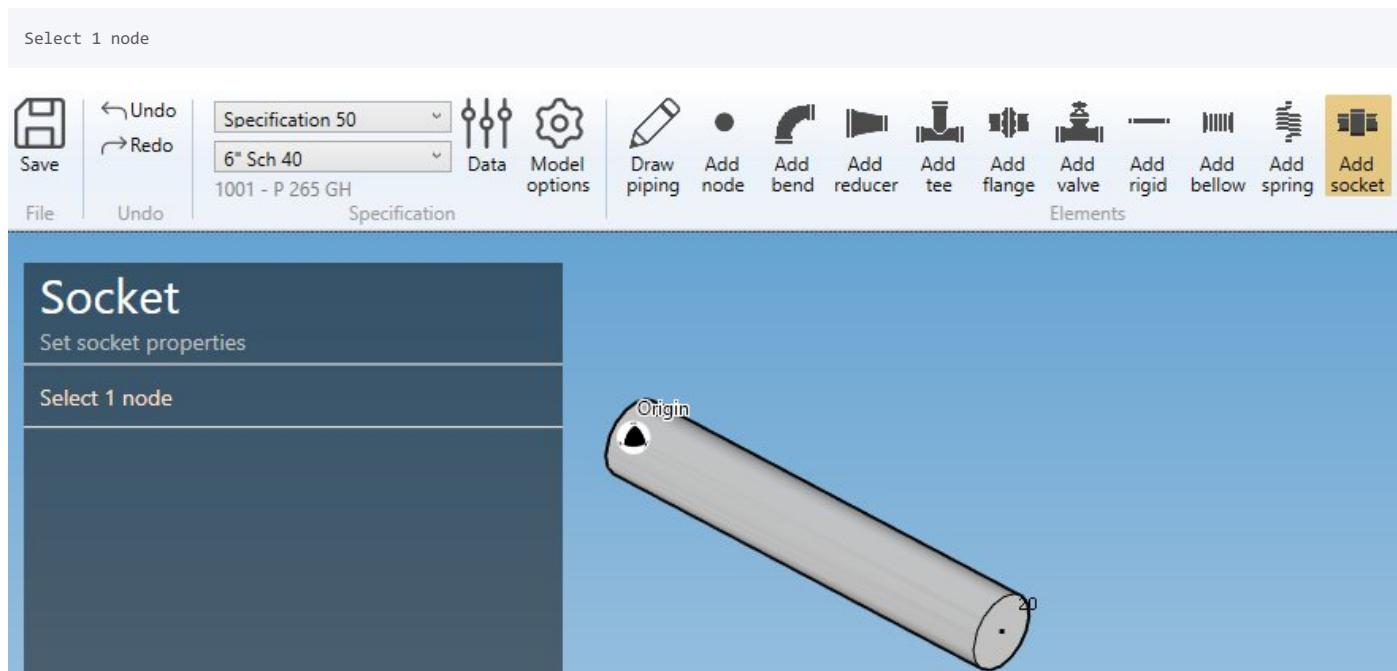
Select for example "Reduce the next element" and click the **Insert** button :



You can **undo** this command.

Create sockets

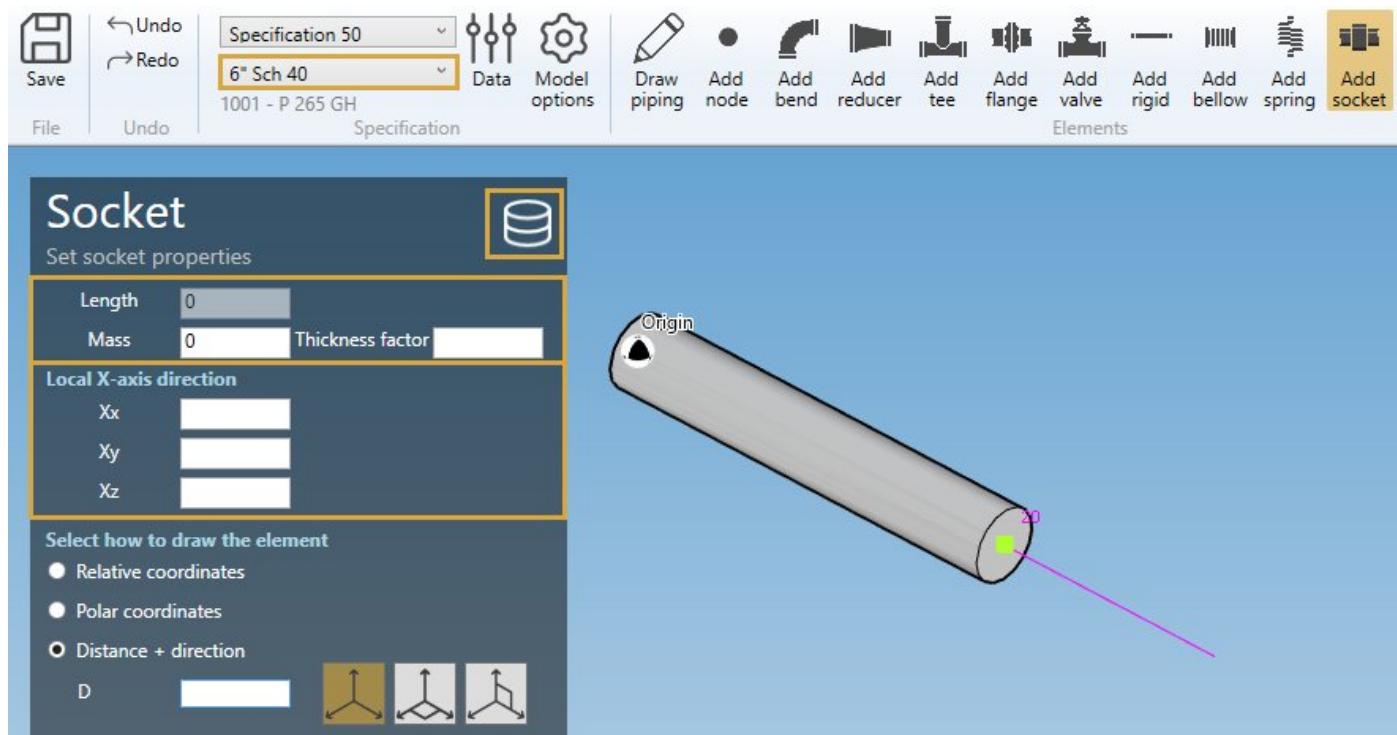
When you click on the **Add socket** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

1. Create a socket

- Select the current **section/material** in the specification box.
- Select a **node**.
- Click the **Add socket** button.



SOCKET PROPERTIES :

Property	Unit Metric	Unit USA
Length	m	ft
Mass	ton	kips
Thickness factor	-	-

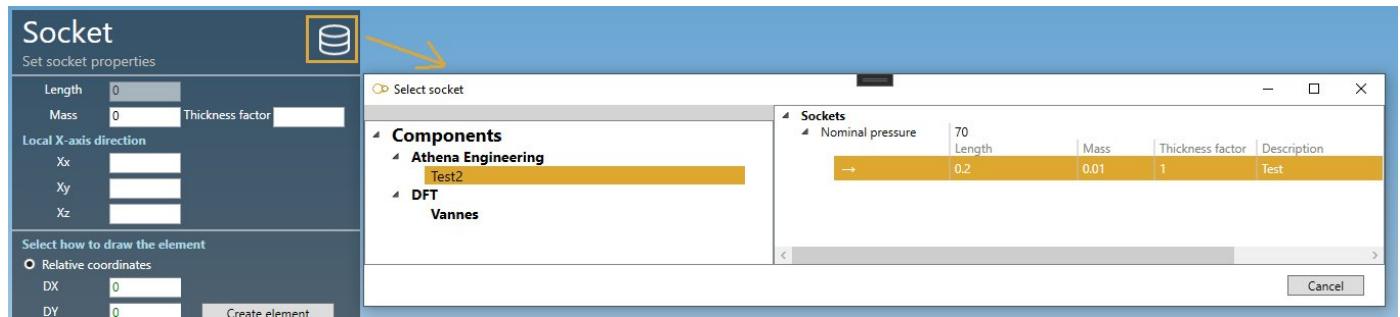
The length must be defined by the **orientation tool**.

X-AXIS DIRECTION :

You can define the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

DATABASE :

To save time, you can also directly select a socket from **database** :



Click on the **Database** button, select a **Library** and a **Socket**. The **OK** button will appear.

The tables will be filled automatically.

Click [here](#) for more information about creation of library of components.

Label :

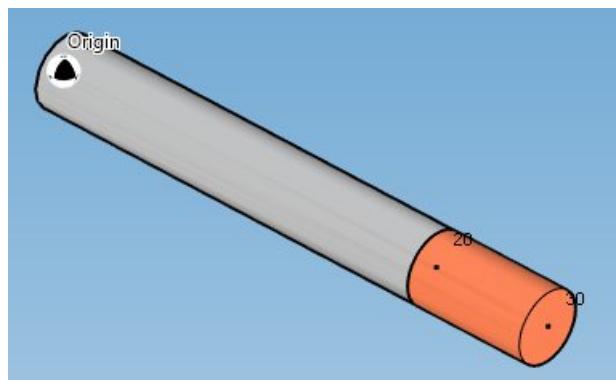
You can define a label to this element. The labels are shown with the node names view button.



You can then define the second point of the socket thanks to the **Orientation tool**.

Click [here](#) for more information about the orientation tool.

Create the socket :



2. Modify/Remove a socket

Change the **Selection mode** to ELEMENT and select a socket :



You can change the properties of the selected socket (except the length).

SECTION AND MATERIAL :

You can change the **specification** and **section/material** of the socket.

Click on the **Modify** button to change the selected spring with these new properties.

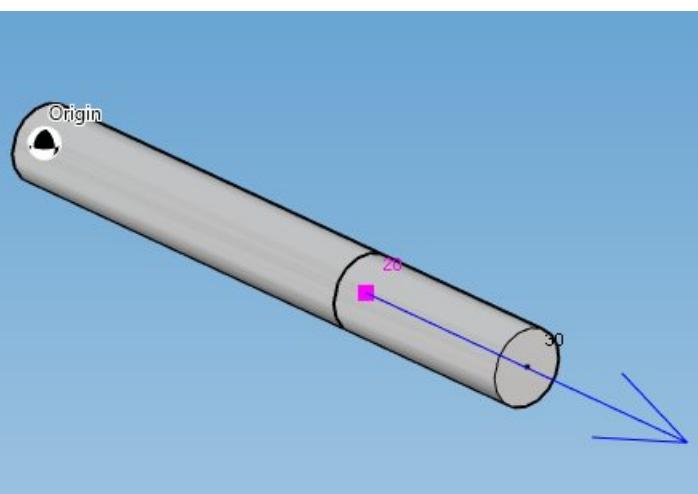
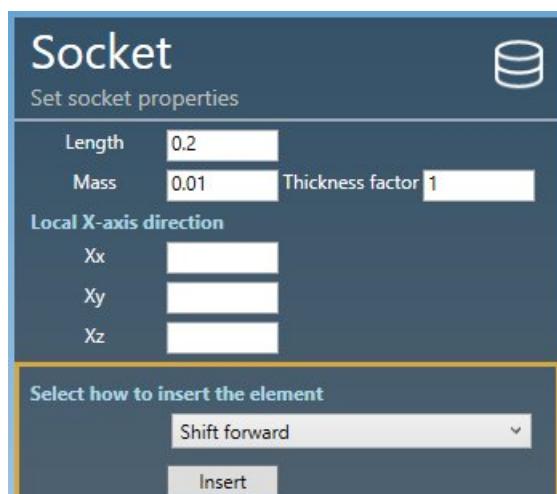
You can **undo** this command.

Click on the **Remove** button to delete the selected spring.

You can **undo** this command.

3. Insert a socket on an intermediate node

Click on the **Add socket** button and select an **intermediate node** between 2 elements.



Fill the properties (see §1) and select the **insertion mode** :

- Shift forward
- Shift backwards
- Reduce the next element
- Reduce the previous element
- Symmetrically reduce the neighboring elements

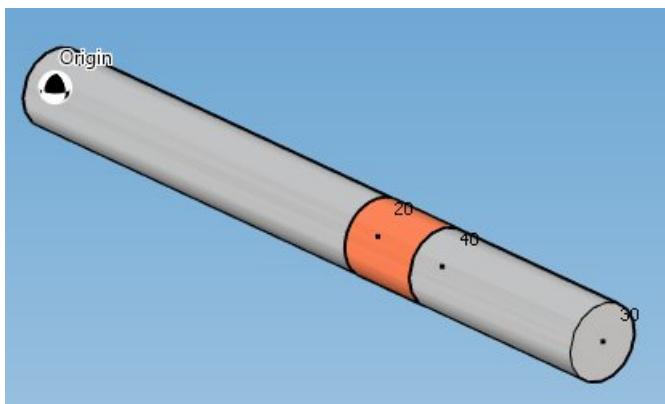
WARNING

ATTENTION, if the length is null, no mode will be proposed (empty list)

WARNING

Based on the length of the socket and the lengths of the neighboring elements, some mode could be hidden.

Select for example "Shift forward" and click the **Insert** button :

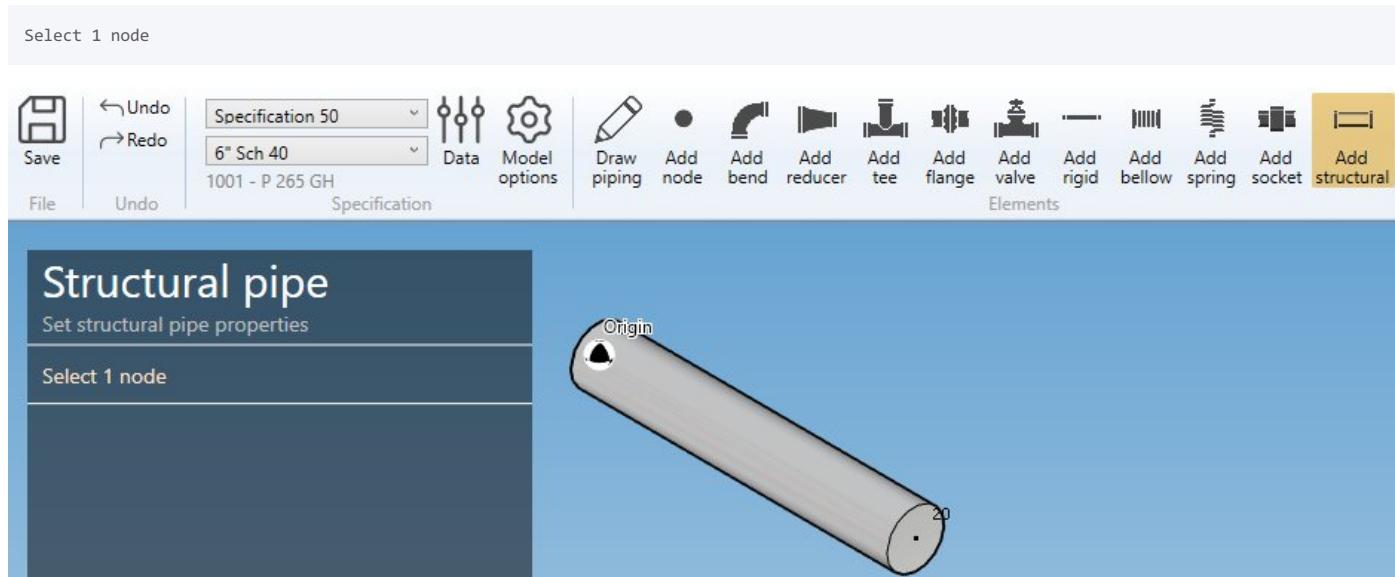


You can **undo** this command.

Create structurals

A structural is an element which is modeled as a straight pipe but which does not contain any fluid and is not counted as part of a tee connection.

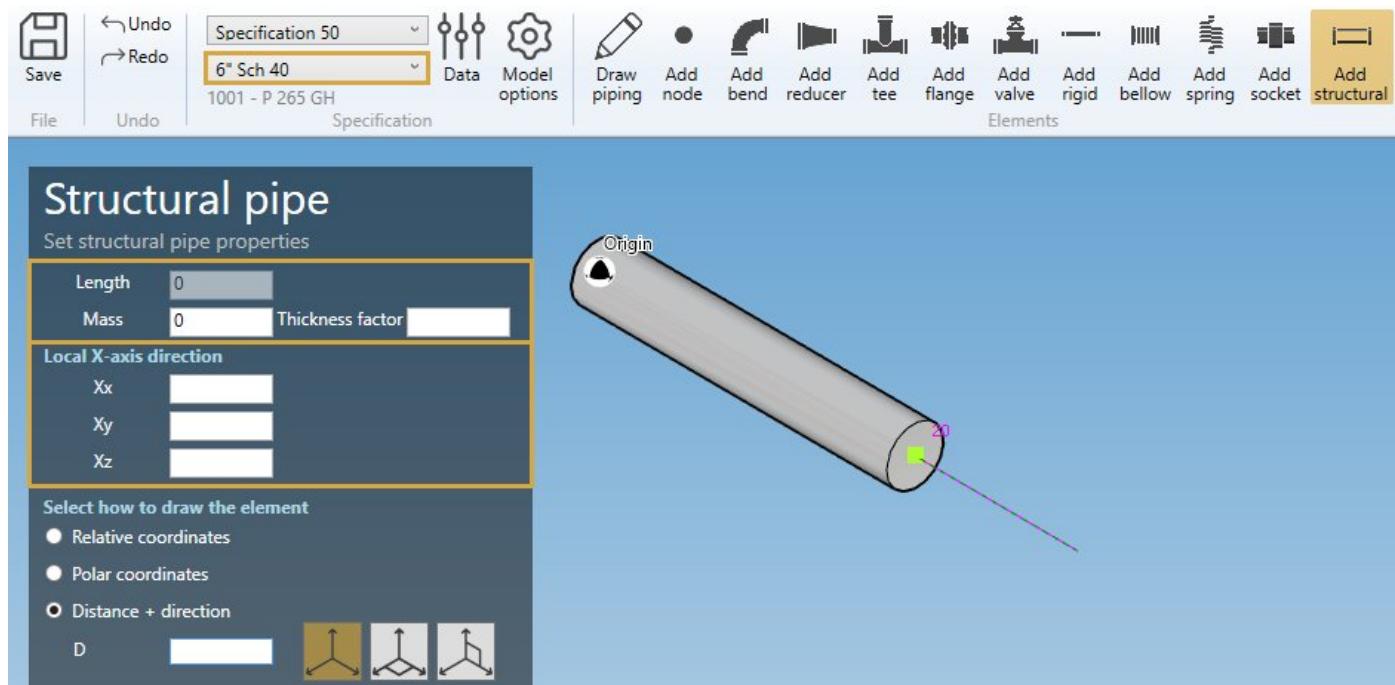
When you click on the **Add structural** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

1. Create a structural

- Select the current **section/material** in the specification box.
- Select a **node**.
- Click the **Add structural** button.



STRUCTURAL PROPERTIES :

Property	Unit Metric	Unit USA
Length	m	ft
Mass	ton	kips
Thickness factor	-	-

The length must be defined by the **orientation tool**.

X-AXIS DIRECTION :

You can define the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

Label :

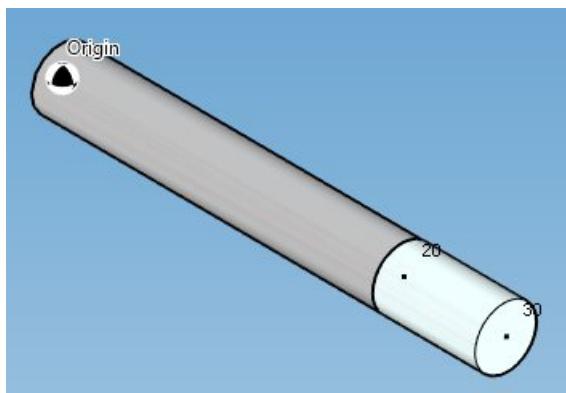
You can define a label to this element. The labels are shown with the node names view button.



You can then define the second point of the structural thanks to the **Orientation tool**.

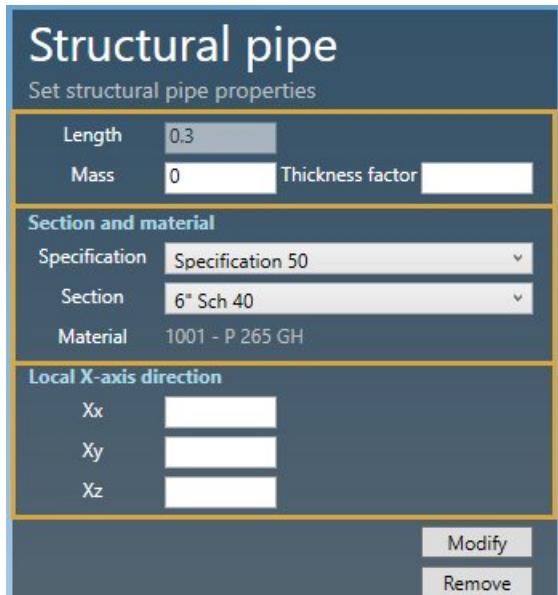
Click [here](#) for more information about the orientation tool.

Create the structural :



2. Modify/Remove a structural

Change the **Selection mode** to ELEMENT and select a structural :



You can change the properties of the selected structural (except the length).

SECTION AND MATERIAL :

You can change the **specification** and **section/material** of the structural.

Click on the **Modify** button to change the selected structural with these new properties.

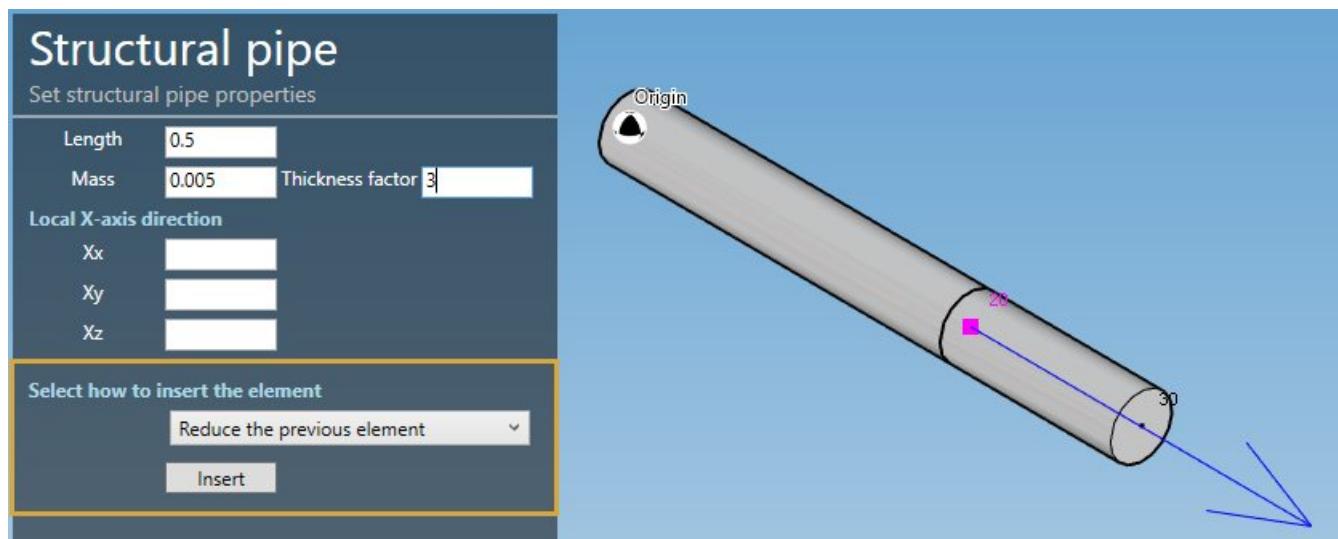
You can **undo** this command.

Click on the **Remove** button to delete the selected structural.

You can **undo** this command.

3. Insert a structural on an intermediate node

Click on the **Add structural** button and select an **intermediate node** between 2 elements.



Fill the properties (see §1) and select the **insertion mode** :

- Shift forward
- Shift backwards
- Reduce the next element
- Reduce the previous element
- Symmetrically reduce the neighboring elements

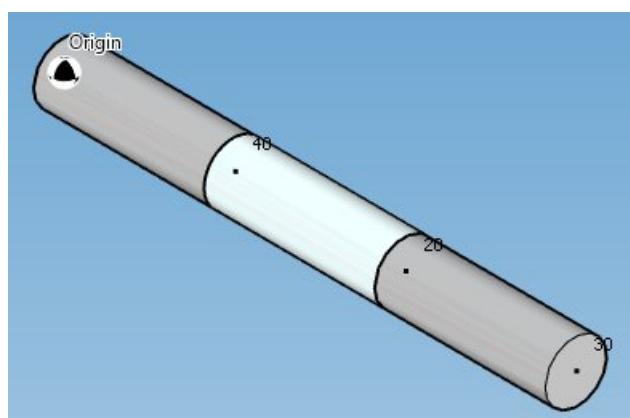
WARNING

ATTENTION, if the length is null, no mode will be proposed (empty list)

WARNING

Based on the length of the structural and the lengths of the neighboring elements, some mode could be hidden.

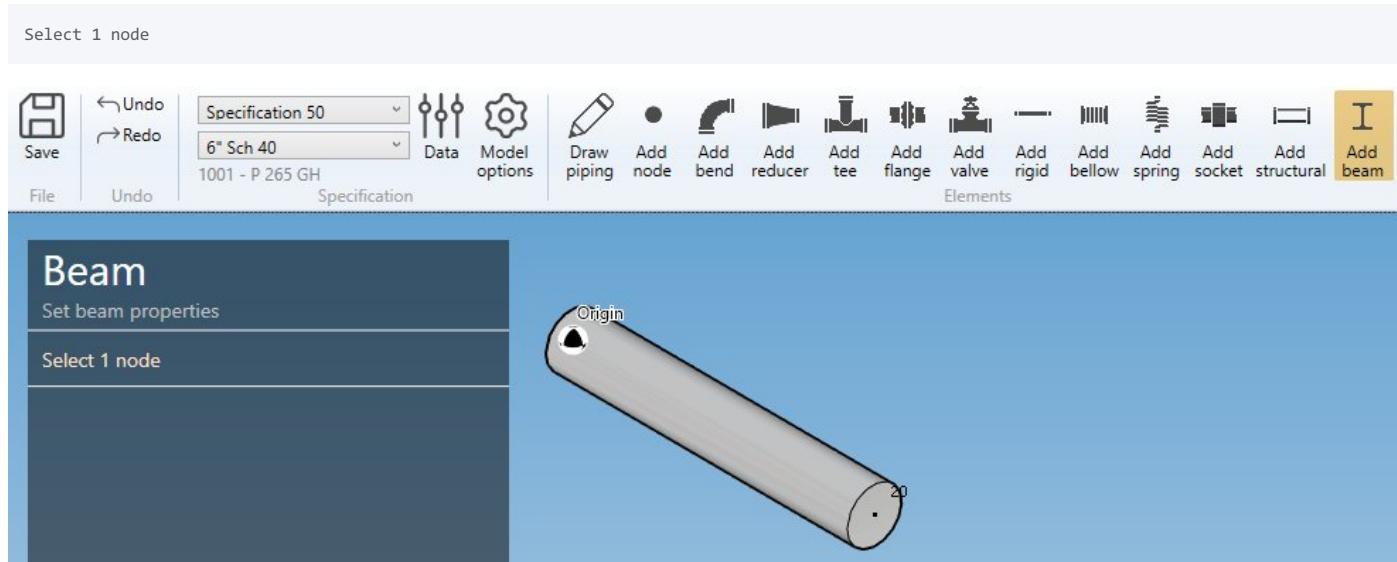
Select for example "Reduce the previous element" and click the **Insert** button :



You can **undo** this command.

Create beams

When you click on the **Add beam** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

1. Create a beam

- Select a **node**.
- Click the **Add beam** button.

Beam

Set beam properties

Type of beam
 Non standard

Length 0 Young modulus 0

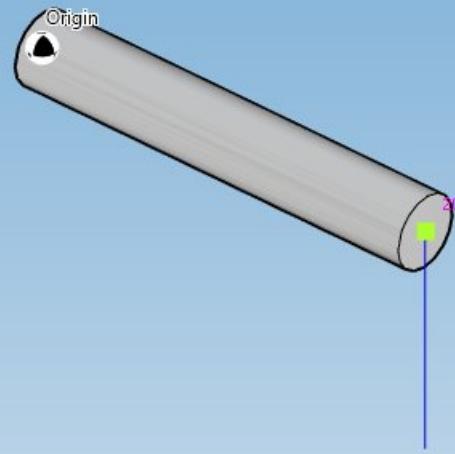
Flexural and torsional moments released
At start At end

Mass density
 Distributed linear mass density 0
 End point mass density 0
 Distributed volumetric mass density 0

Section properties
Area 0 Ix 0
X shear area 0 ly 0
Y shear area 0 It 0

Local X-axis direction
Xx
Xy
Xz

Select how to draw the element
 Relative coordinates
 Polar coordinates
 Distance + direction
D 



BEAM TYPE :

- Non standard
- I
- Channel
- Rectangular
- Tee
- Equal angle
- Unequal angle
- Round
- Plate

The length must be defined by the **orientation tool**.

Common property :

Property	Unit Metric	Unit USA
Young modulus	kN/mm ²	10 ⁶ .psi
Moments released	-	-
Distributed linear mass density	kg/m	lb/ft
End point mass density	kg/m	lb/ft
Distributed volumetric mass density	kg/m ³	lb/ft ³

Based on **type**, you have to define different section properties.

SECTION PROPERTIES :

- Non standard :

Section properties		
Area	0	Ix
X shear area	0	ly
Y shear area	0	It
Property	Unit Metric	Unit USA
Cross sectional area	mm ²	in ²
Shear area along X' axis	mm ²	in ²
Shear area along Y' axis	mm ²	in ²
Moment of inertia about X' axis : Ix	mm ⁴	in ⁴
Moment of inertia about Y' axis : ly	mm ⁴	in ⁴
Torsional inertia : It	mm ⁴	in ⁴

To know the UNIT of a property, just move the mouse over the property name.

- All other types :

Section properties		
Height	0	Web thickness
Flange width	0	Flange thickness
Property	Unit Metric	Unit USA
Height	mm	in
Web thickness	mm	in
Flange width	mm	in
Flange thickness	mm	in

X-AXIS DIRECTION :

You can define the **X-axis** vector by defining Xx, Xy, Xz in global coordinates.

Label :

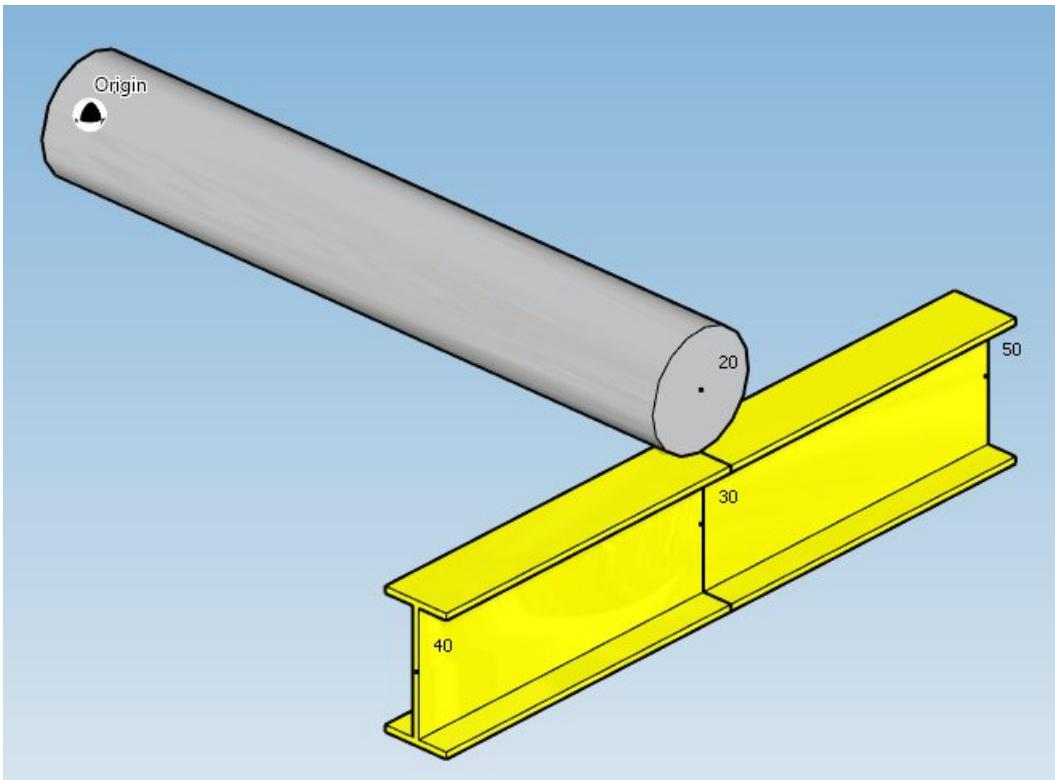
You can define a label to this element. The labels are shown with the node names view button.

Label	<input type="text"/>
-------	----------------------

You can then define the second point of the beam thanks to the **Orientation tool**.

Click [here](#) for more information about the orientation tool.

Create 2 **I beams** after creation of node 30 :



2. Modify/Remove a beam

Change the **Selection mode** to ELEMENT and select a beam :

Beam

Set beam properties

Type of beam
I

Length 0.5 Young modulus 0

Flexural and torsional moments released
At start At end

Mass density

- Distributed linear mass density 0
- End point mass density 0
- Distributed volumetric mass density 0

Section properties

Height	200	Web thickness	10
Flange width	100	Flange thickness	10

Local X-axis direction

Xx	
Xy	
Xz	

Buttons:

Modify Remove

Click [here](#) for more information about the selection tool.

You can change the type and the appropriate properties of the selected beam (except the length).

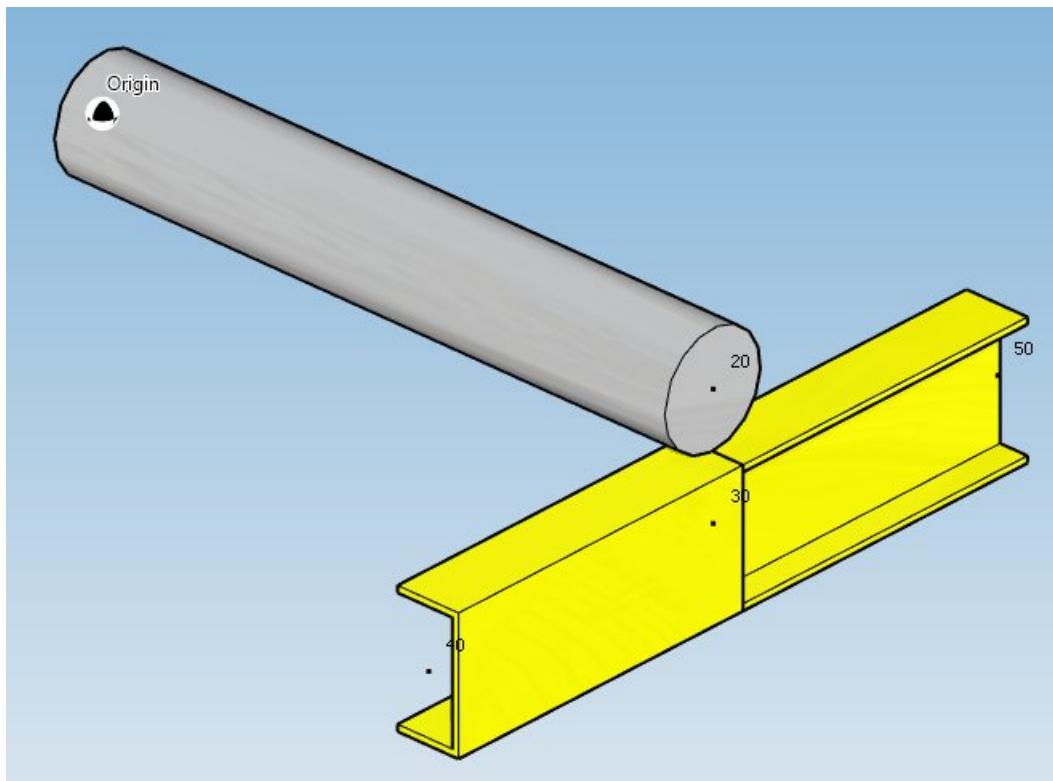
Click on the **Modify** button to change the selected beam with these new properties.

You can **undo** this command.

Click on the **Remove** button to delete the selected beam.

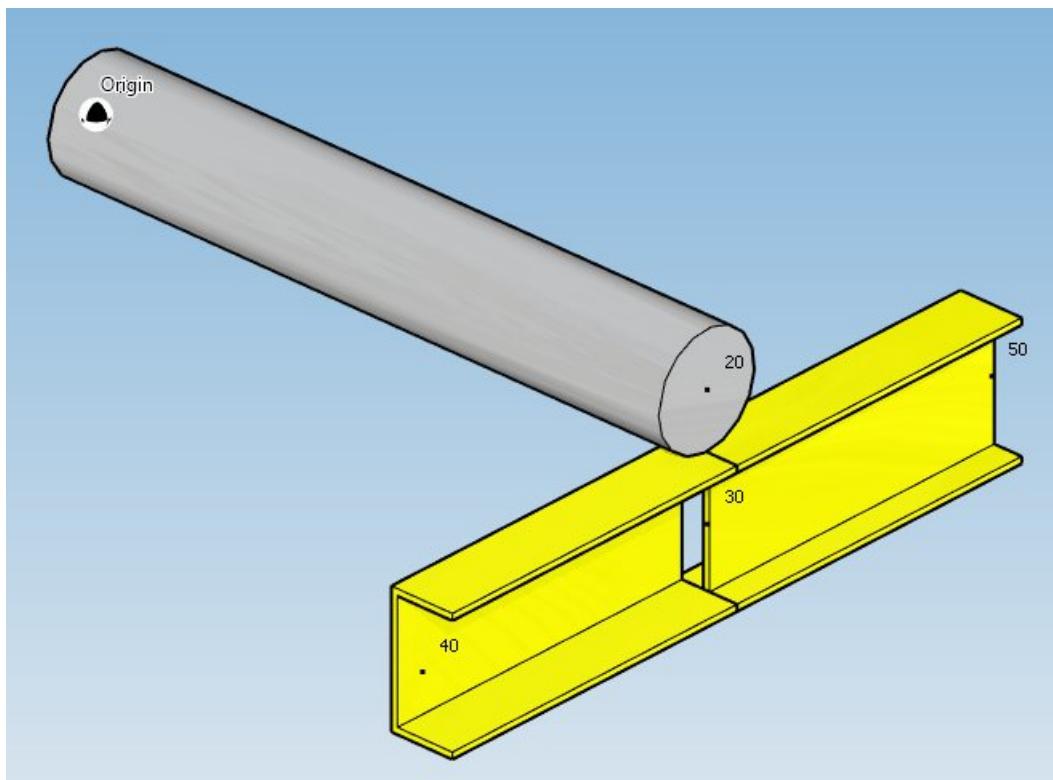
You can **undo** this command.

Example of modification, change **I** type to **Channel** type :



And by changing the **X-axis direction** :

Local X-axis direction	
Xx	0
Xy	0
Xz	-1



3. Insert a beam on an intermediate node

Click on the **Add beam** button and select an **intermediate node** between 2 beams.

Beam

Set beam properties

Type of beam

I	Length	0.8	Young modulus	0
---	--------	-----	---------------	---

Flexural and torsional moments released

At start

At end

Mass density

- Distributed linear mass density 0
- End point mass density 0
- Distributed volumetric mass density 0

Section properties

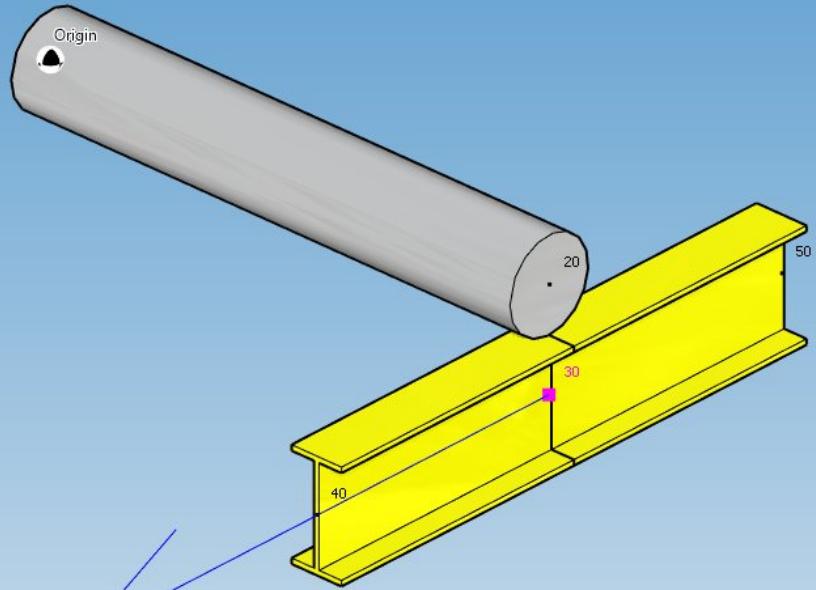
Height	200	Web thickness	10
Flange width	100	Flange thickness	10

Local X-axis direction

Xx
Xy
Xz

Select how to insert the element

Shift forward
Insert



Fill the properties (see §1) and select the **insertion mode** :

- Shift forward
- Shift backwards
- Reduce the next element
- Reduce the previous element
- Symmetrically reduce the neighboring elements

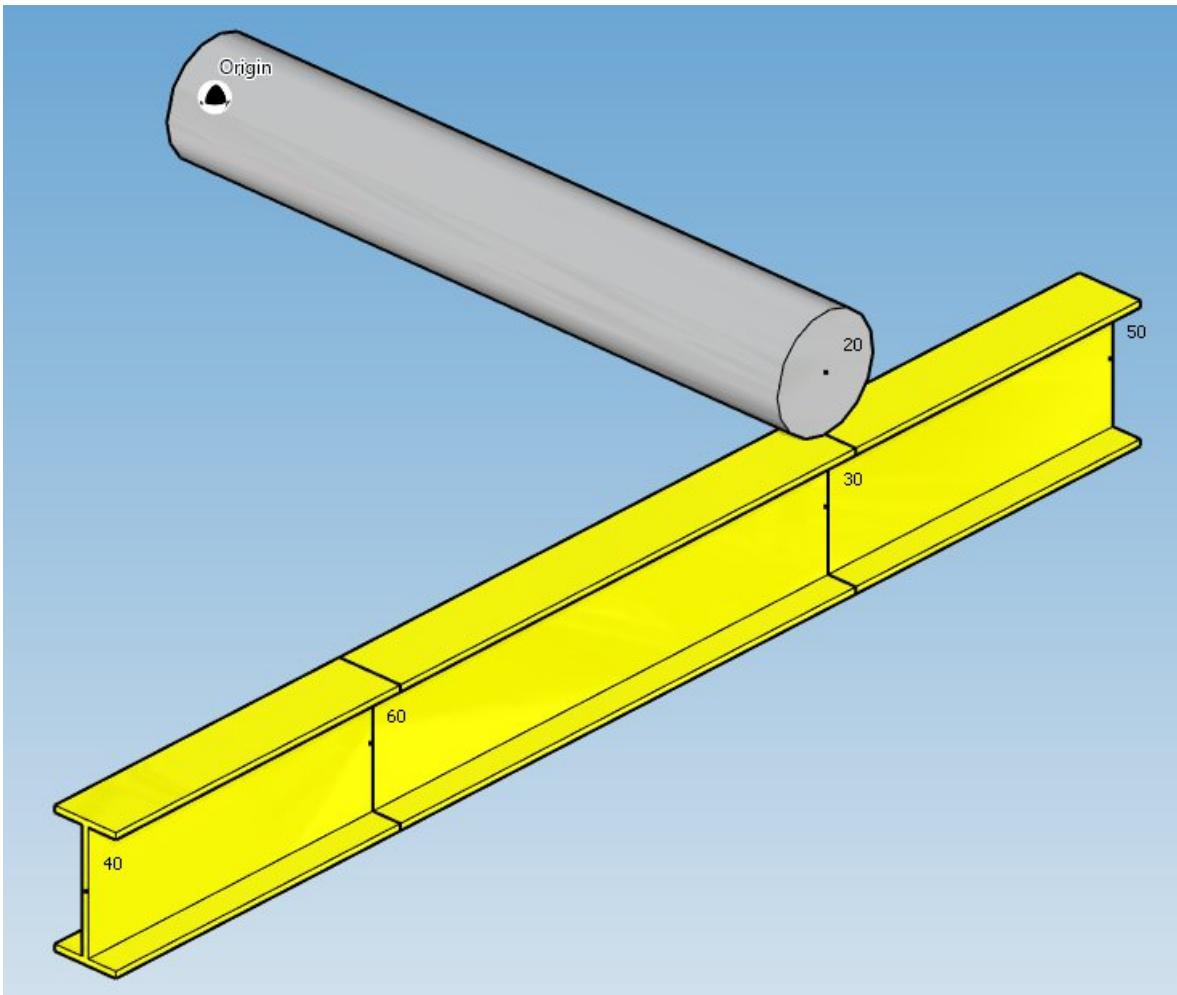
WARNING

ATTENTION, if the length is null, no mode will be proposed (empty list)

WARNING

Based on the length of the beam and the lengths of the neighboring elements, some mode could be hidden.

Select for example "Shift forward" and click the **Insert** button :



You can **undo** this command.

TABLE OF CONTENTS

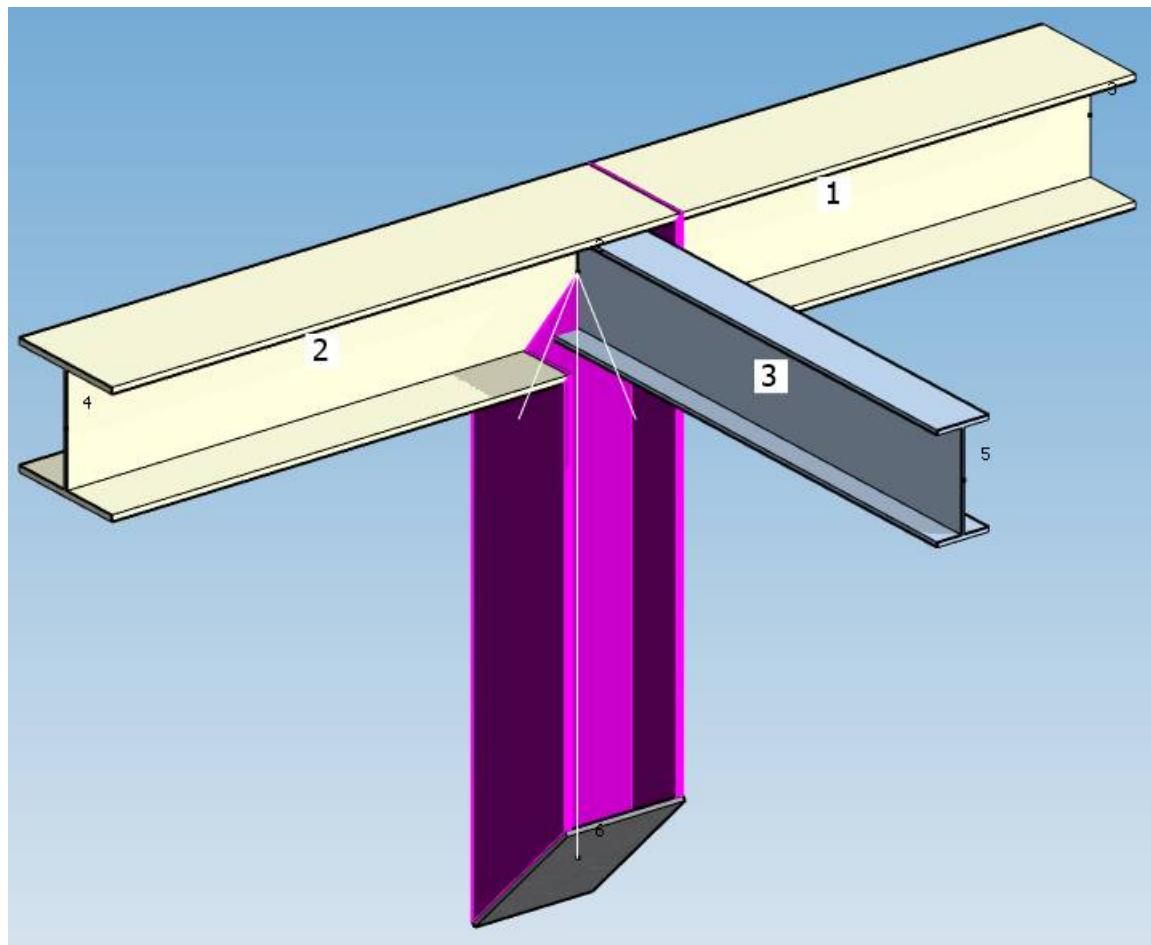
- [Graphical ending](#)
- [Joint](#)

Graphical ending

MetaStructure proposes several endings for each extremity of a beam :

Type	Description
None	Ending plane = plane perpendicular to the current beam axis at the node
Front	Ending plane = nearest plane encountered at the specified beam
Back	Ending plane = furthest plane encountered at the specified beam
Miter	Ending plane = bisector plane between current beam and specified beam
Plate	Ending plane = plane of the anchor plate

1. Definition



The direction of a selected beam is represented by a **white arrow** (Node1 to Node2).

All beams (not parallel to current one) of each extremity are numbered.

At extremity 1 (Node1), the numbers are drawn inside a **black square**.

At extremity 2 (Node2), the numbers are drawn inside a **white square**.

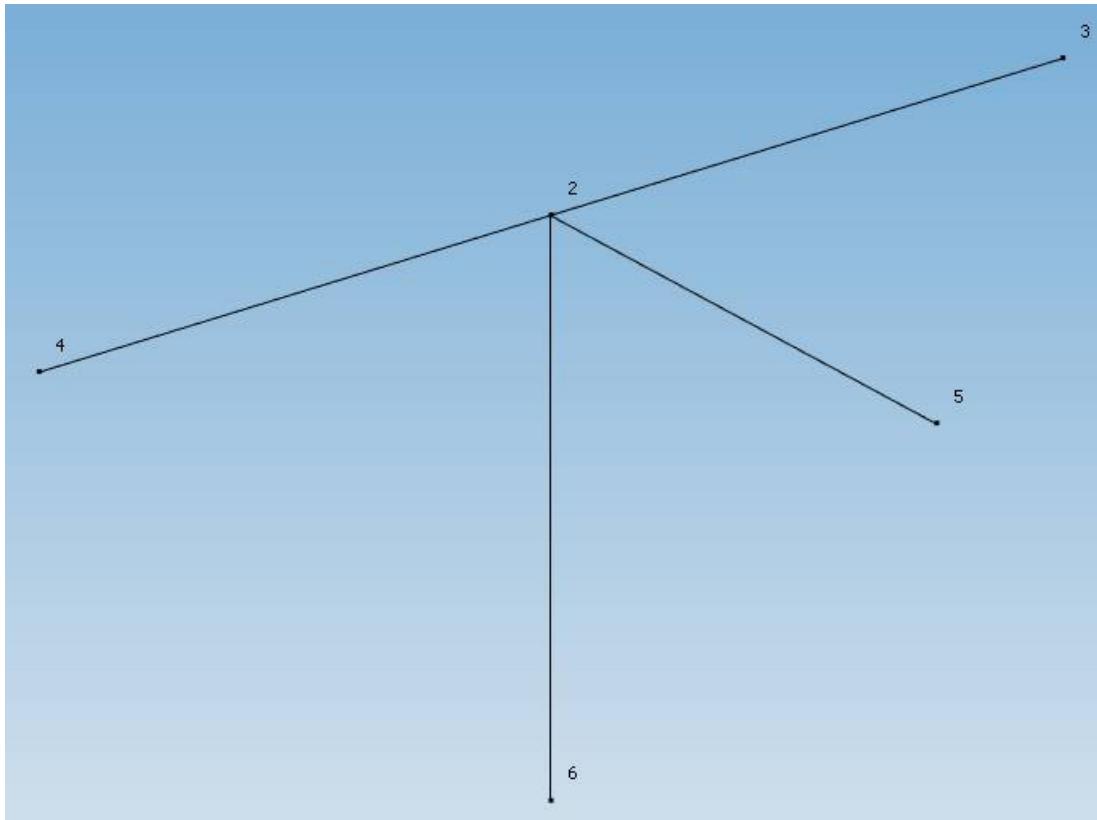
You can specify the beam with which you want a particular ending.

The **Offset** adds (or subtracts if negative value) an offset distance to the ending plane (parallel to the plane).

Property	Unit Metric	Unit USA
Offset	mm	in

WARNING

ATTENTION, the endings are only graphical and have no impact on the calculation. It helps for visualization and for the calculation of plate dimensions. In fact the beams do intersect at the node :



1.1 None ending

Ending plane = plane perpendicular to the current beam at the node :

Graphical ending

Extremity 1	None	Extremity 2	None
Beam 1	1	Beam 2	1
Offset 1	0	Offset 2	0

Joints

Extremity 1	Fixed	Extremity 2	Fixed
-------------	-------	-------------	-------

1.2 Front ending

Ending plane = nearest plane encountered at the specified beam :

Graphical ending

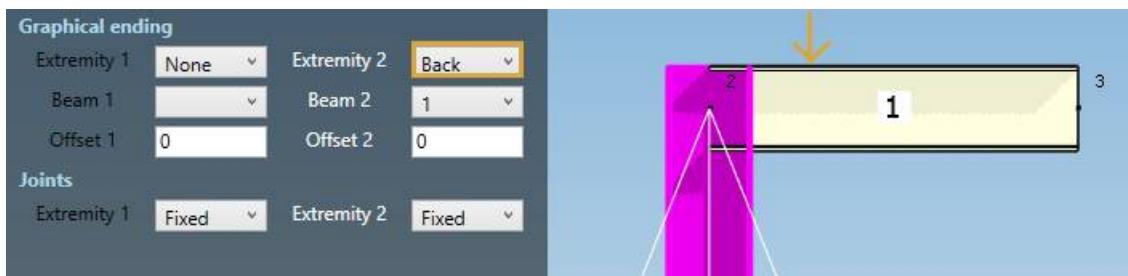
Extremity 1	None	Extremity 2	Front
Beam 1	1	Beam 2	1
Offset 1	0	Offset 2	0

Joints

Extremity 1	Fixed	Extremity 2	Fixed
-------------	-------	-------------	-------

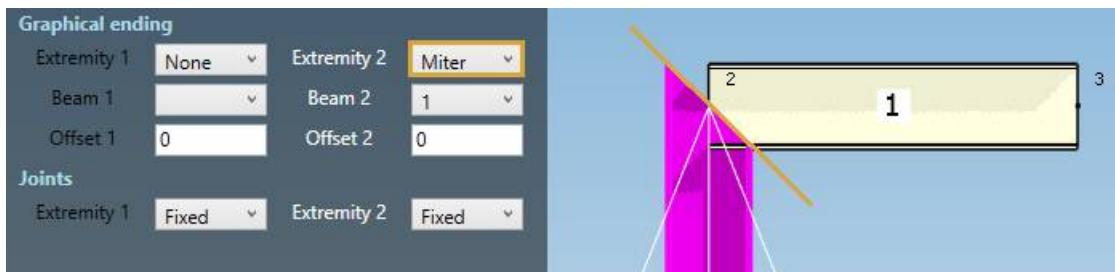
1.3 Back ending

Ending plane = furthest plane encountered at the specified beam :



1.4 Miter ending

Ending plane = bisector plane between current beam and specified beam :



1.5 Plate ending

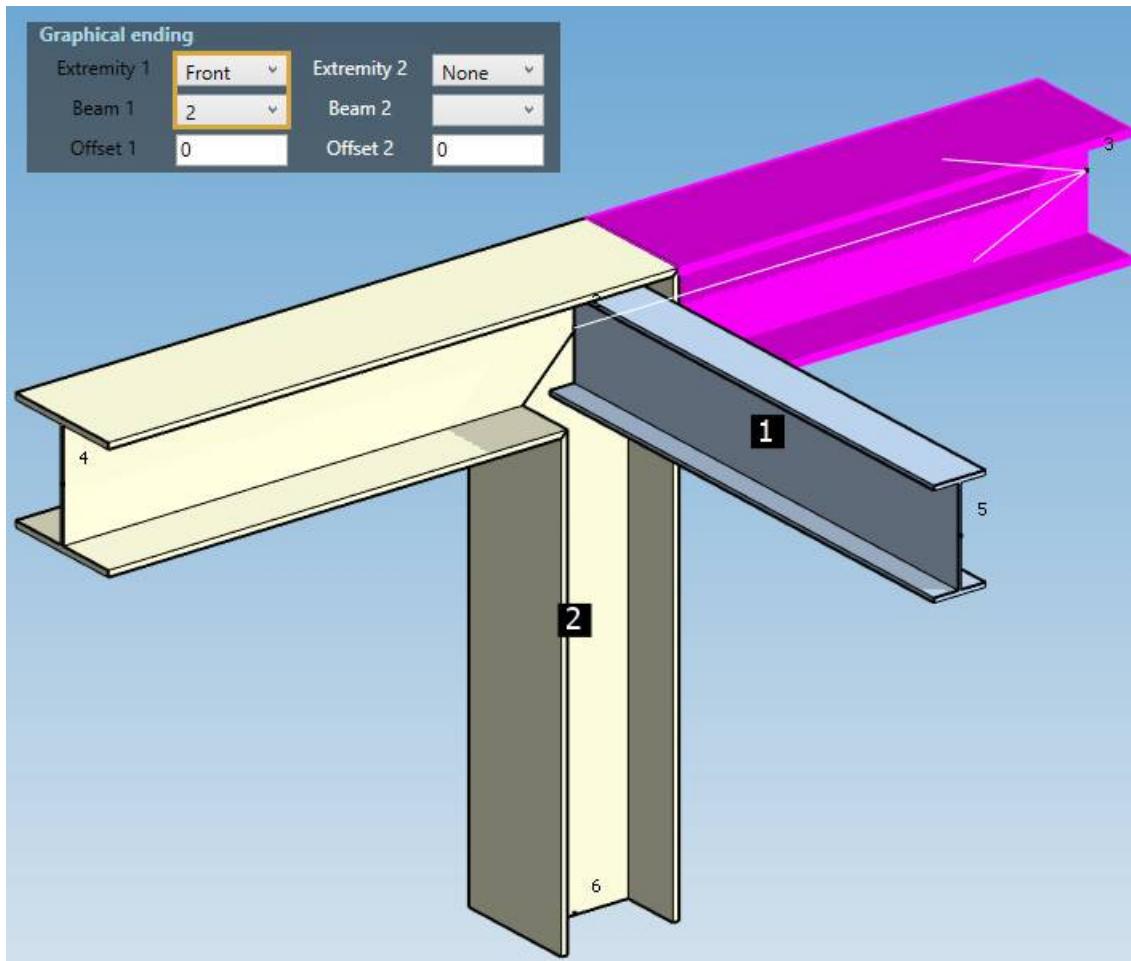
Ending plane = plane of the anchor plate.

An **anchor plate** must exists at the extremity node.

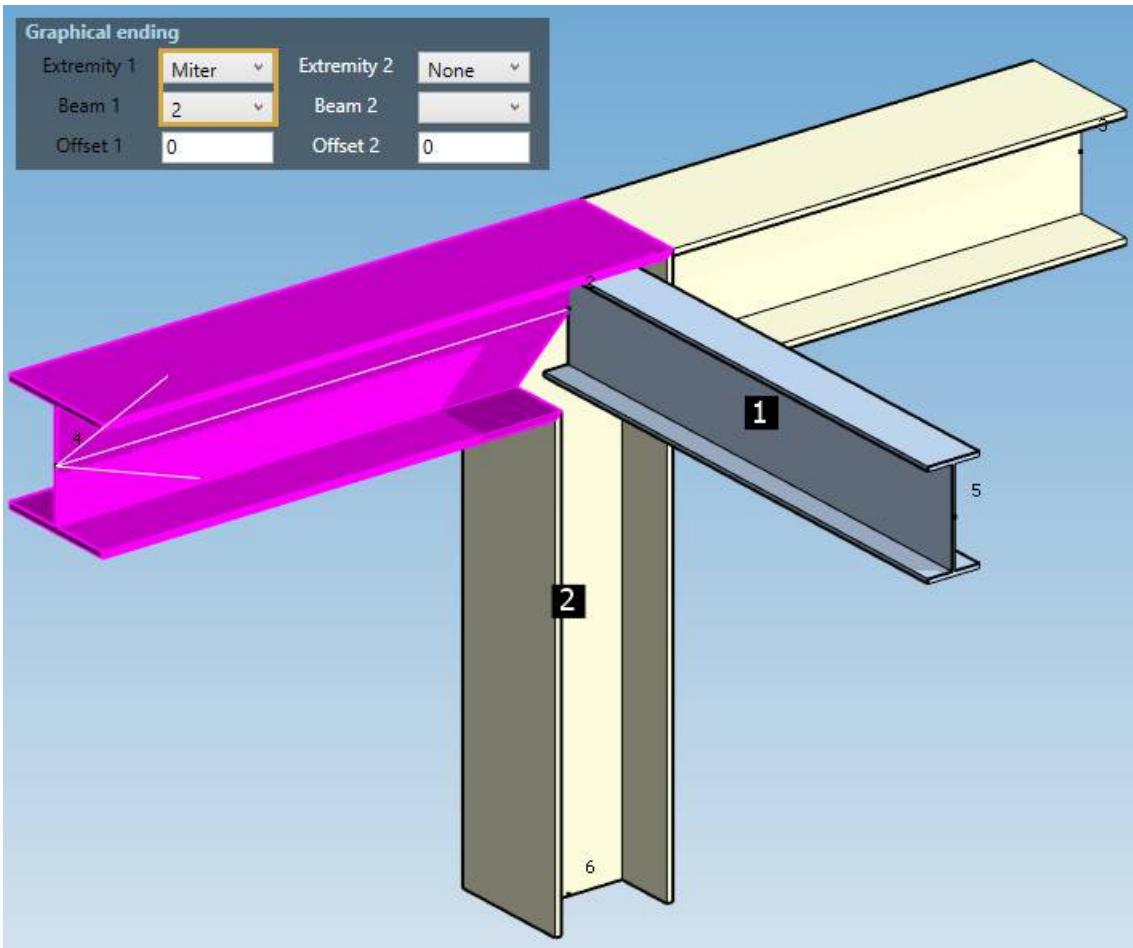
See §2. for an example of plate ending.

2. Examples

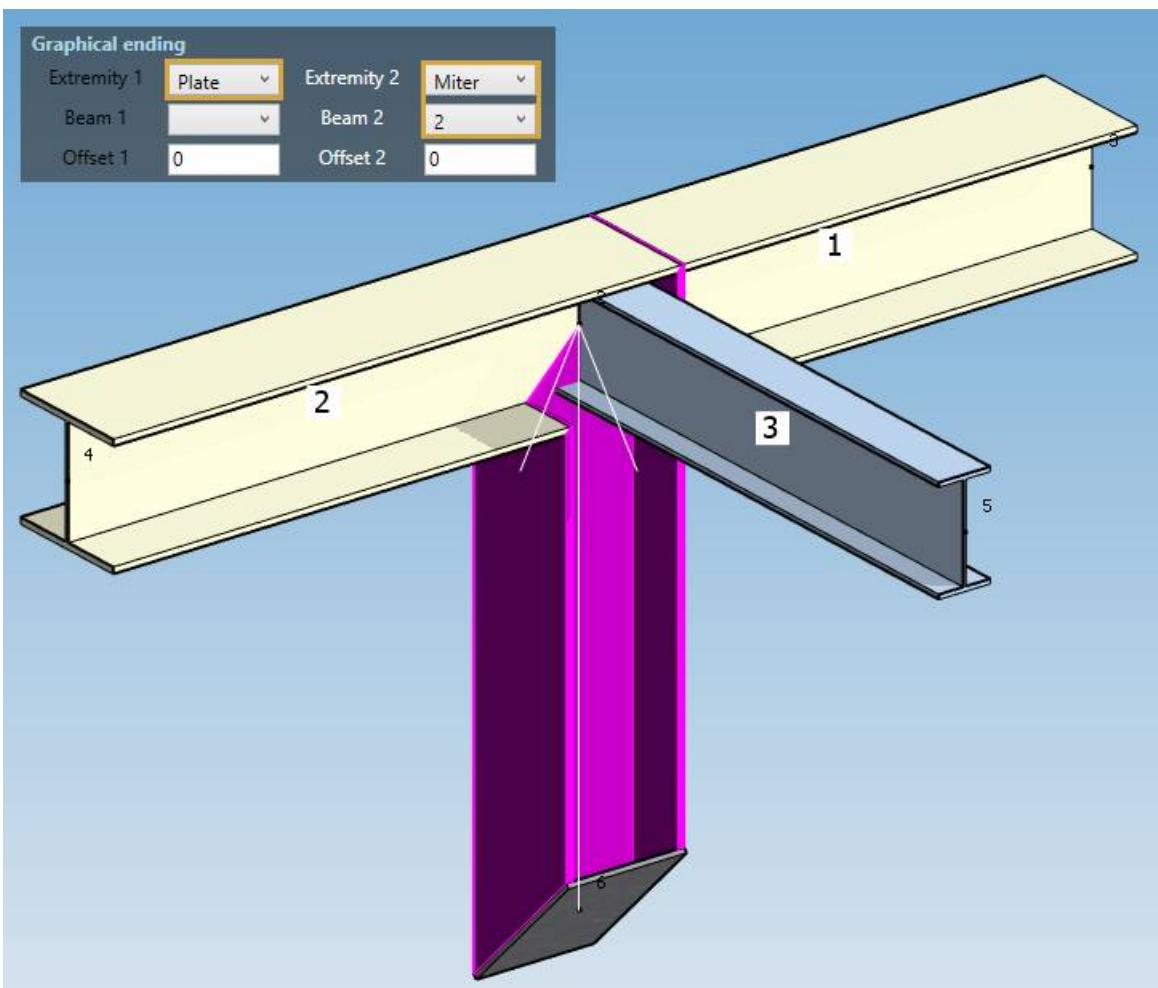
In this example, the selected beam has a **front** ending with beam 2 at its first extremity (black column):



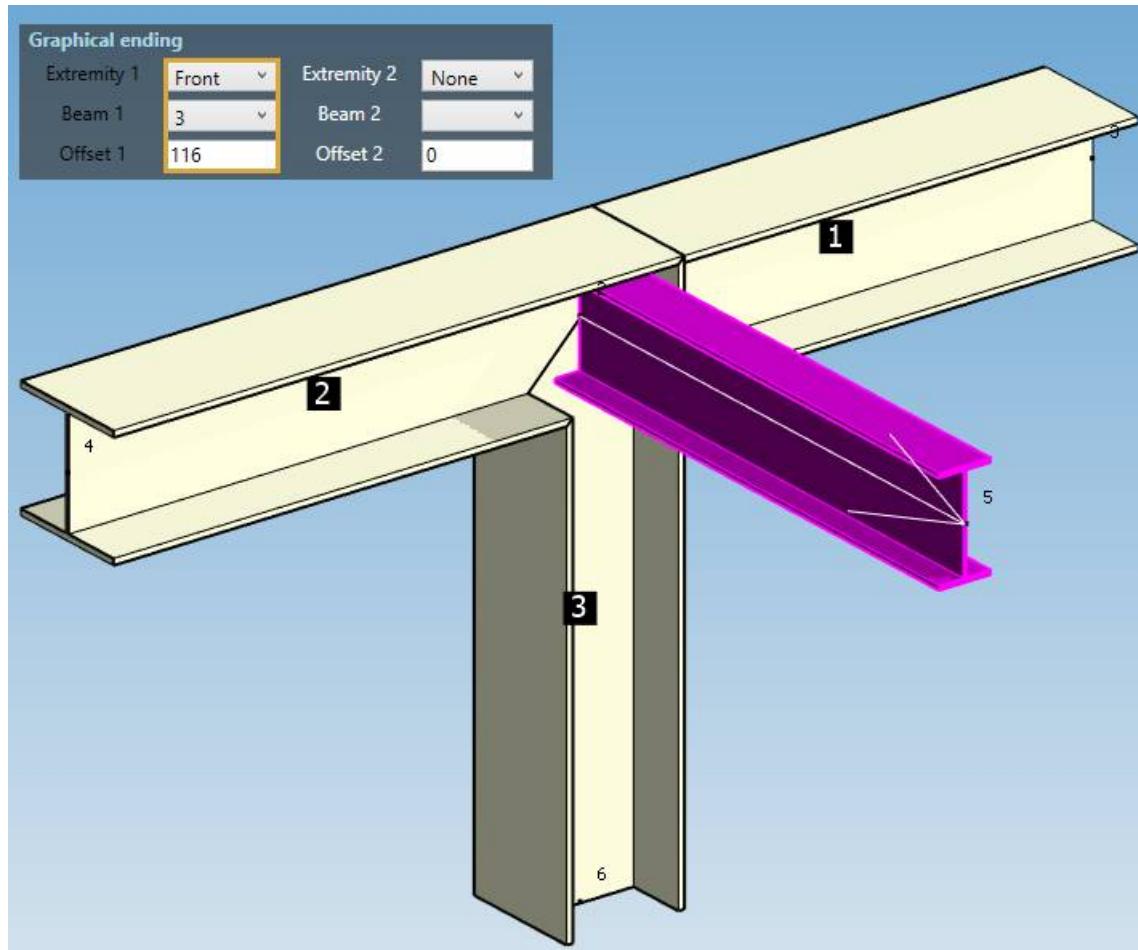
In this example, the selected beam has a **miter** ending with beam 2 at its first extremity (black column) :



In this example, the selected beam has a **plate** ending at its first extremity (black column) and a **miter** ending with beam 2 at its second extremity (white column) :



In this example, the selected beam has a **front** ending with beam 3 with an offset of 116 mm at its first extremity (black column) :



The **offset** corresponds to the distance from the first face encountered to the web = $(240 - 7.5) / 2 = +/- 116$ mm :

Current specification

- Specifications
- Beams
- Materials

Beam sections

Name: HEA 240

Description: Hot rolled

Dimensions:

- h = 230 mm
- b = 240 mm
- tw = 7.5 mm
- tf = 12 mm
- A = 76.84 cm²
- Ax = 17.25 cm²
- Ay = 57.6 cm²
- Ix = 2769 cm⁴
- ly = 7763 cm⁴
- It = 41.55 cm⁴
- lw = 328500 cm⁶
- WT = 34.625 cm³
- Wy = 48 cm²
- Wx = 14.375 cm²
- WEly = 675.1 cm³
- WElx = 230.7 cm³

Click [here](#) to return to **beam** page.



Joint

MetaStructure proposes several **assemblies** for each extremity of a beam :

Type	Description
Fixed	Rigid joint (transmission of moments)
Detailed	User can specify the 3 translation stiffnesses and 3 rotation stiffnesses
Bolted	User can define a bolted joint
Welded	User can define a welded joint

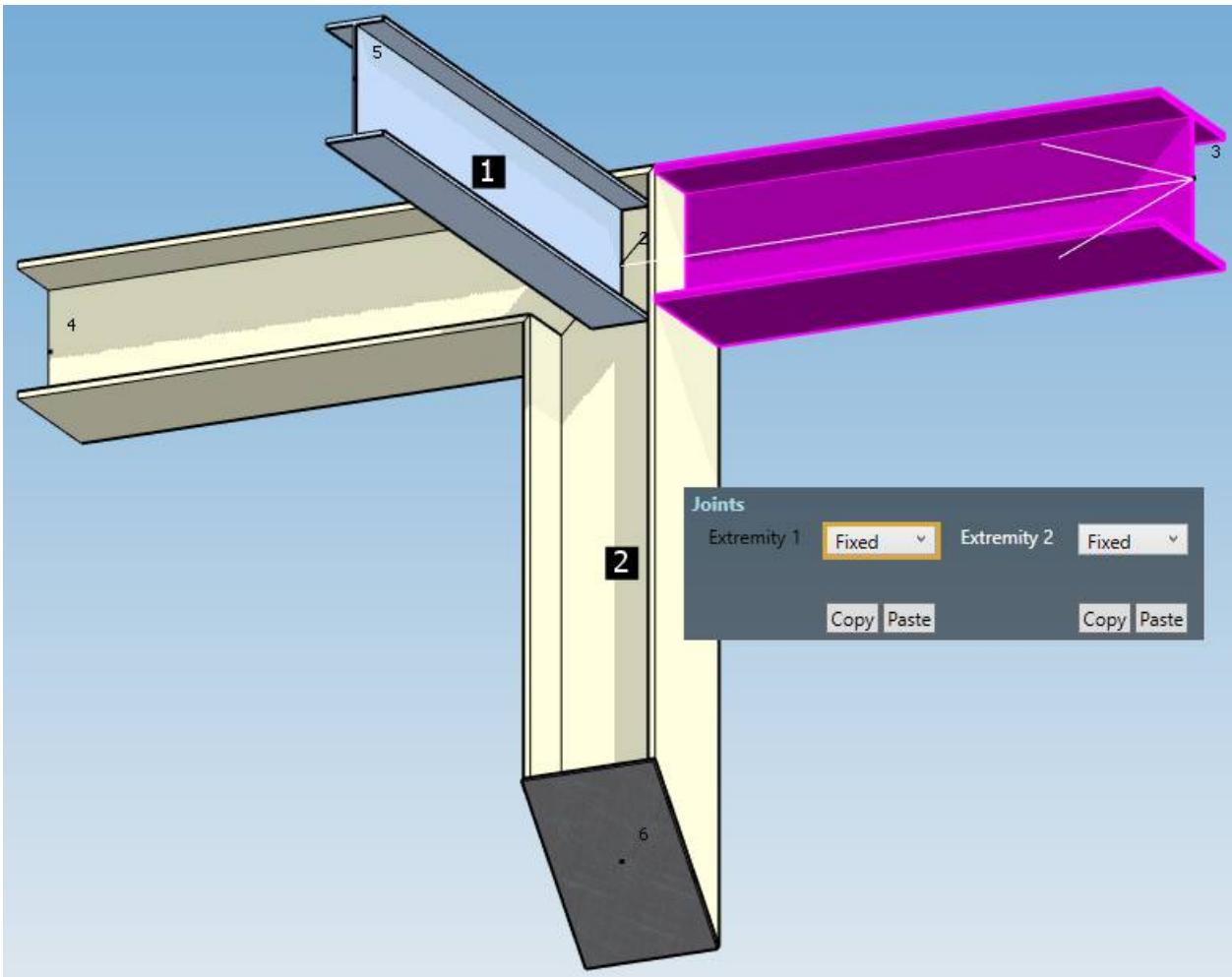
1. Definition

All joint types are at least defined by 6 stiffnesses :

Property	Description	Unit Metric	Unit USA
Kx	Translation stiffness X	kN/mm	kips/in
Ky	Translation stiffness Y	kN/mm	kips/in
Kz	Translation stiffness Z	kN/mm	kips/in
Krx	Rotation stiffness X	kN.m/rad	kips.ft/rad
Kry	Rotation stiffness Y	kN.m/rad	kips.ft/rad
Krz	Rotation stiffness Z	kN.m/rad	kips.ft/rad

1.1 Fixed joint

This is the default rigid connection.



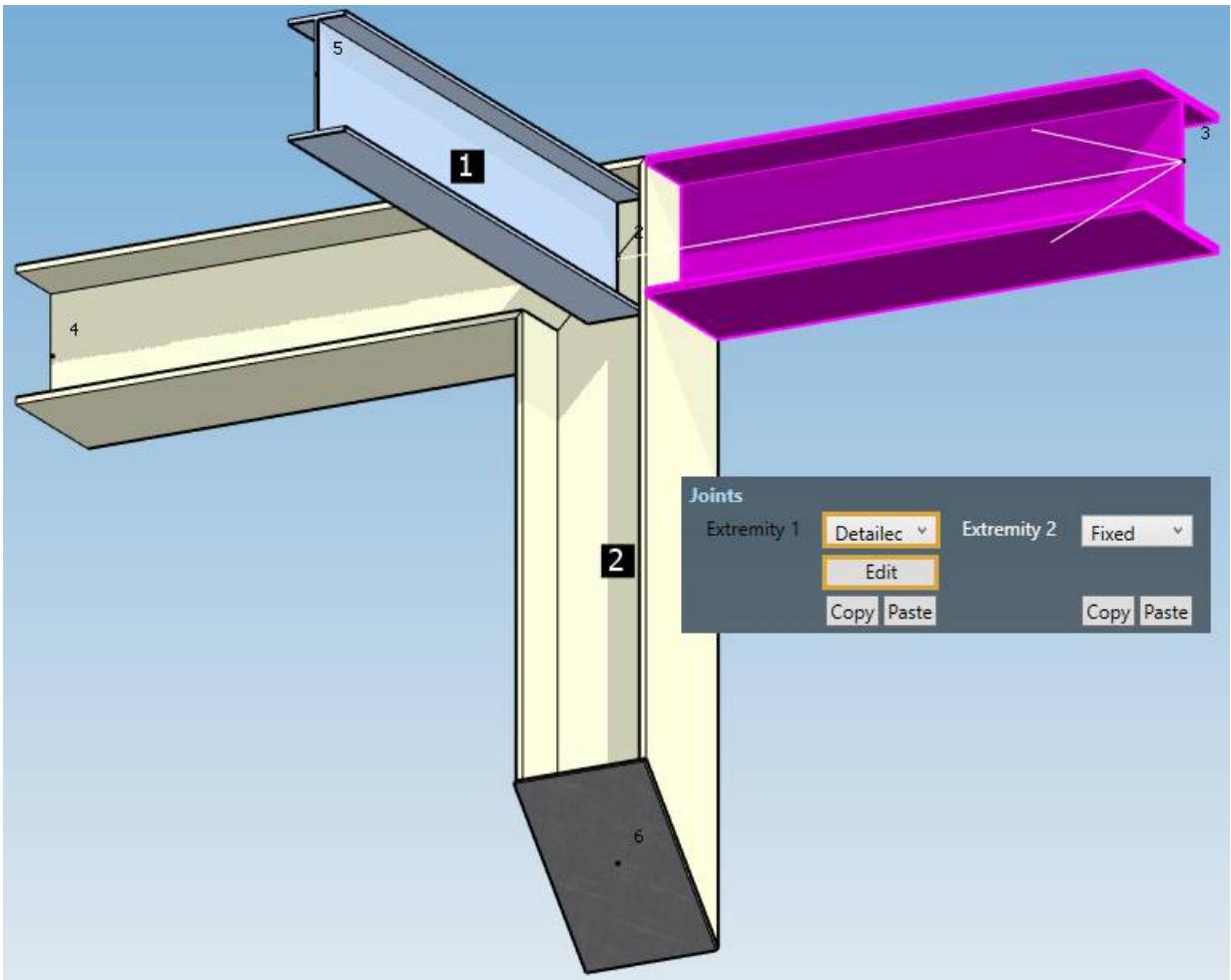
Example of fixed joint on extremity 1 (black side)

Property	Value (MKS units)
Kx, Ky, Kz	1.75e13
Krx, Kry, Krz	1.13e12

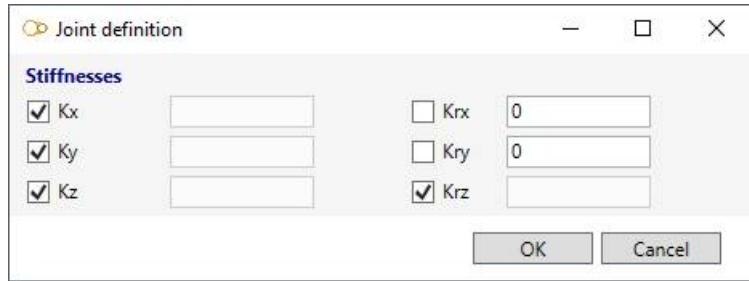
If user wants to define other stiffnesses, he has to set the joint to **detailed**.

1.2 Detailed joint

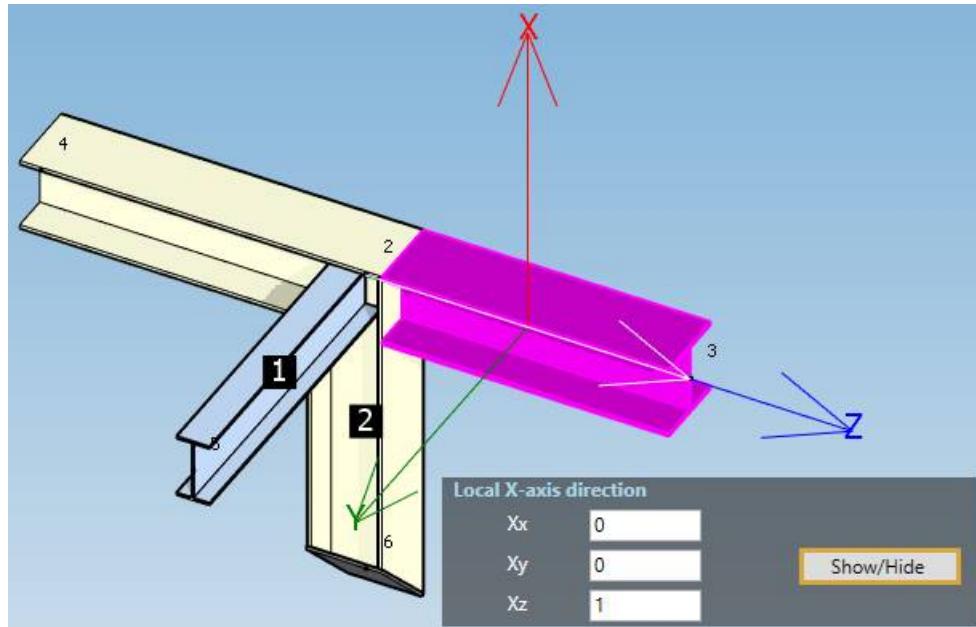
This is the simplest joint where user can define the stiffnesses :



Press the **Edit** button and disable the transmission of the moments (stiffness = 0) around X and Y axis :



To see the local axis, press the **Show/Hide** button of **Local X-axis direction** :

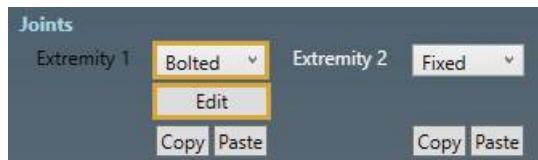


WARNING

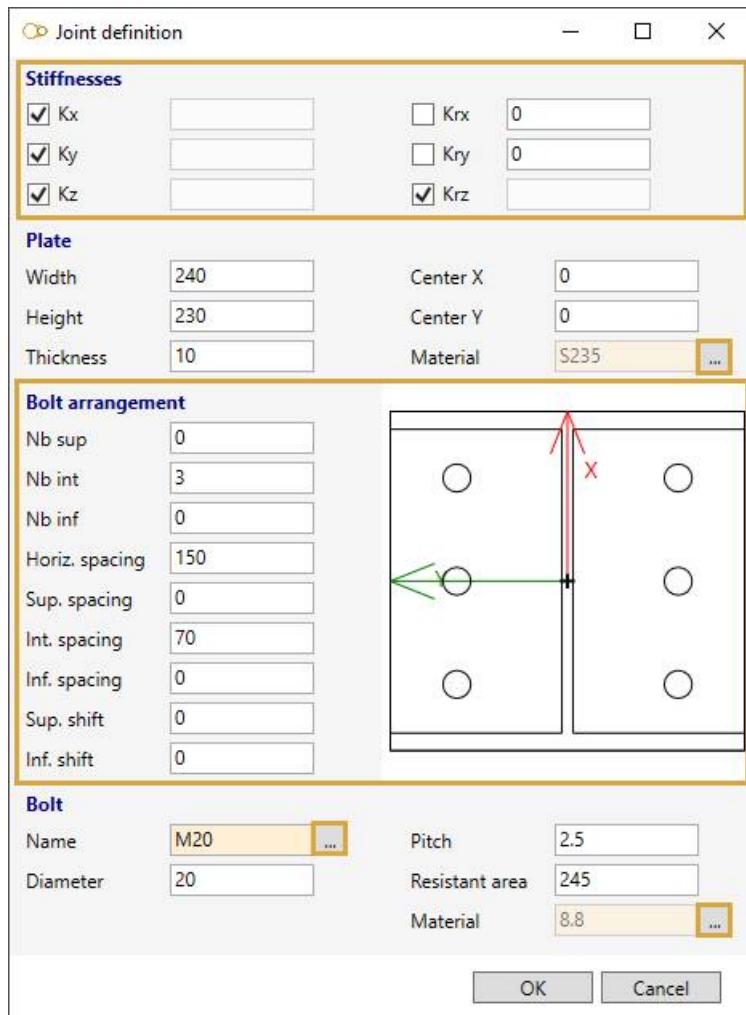
Z-axis is always the axis of the beam in MetaStructure and X-axis is the **weak** axis.

1.3 Bolted joint

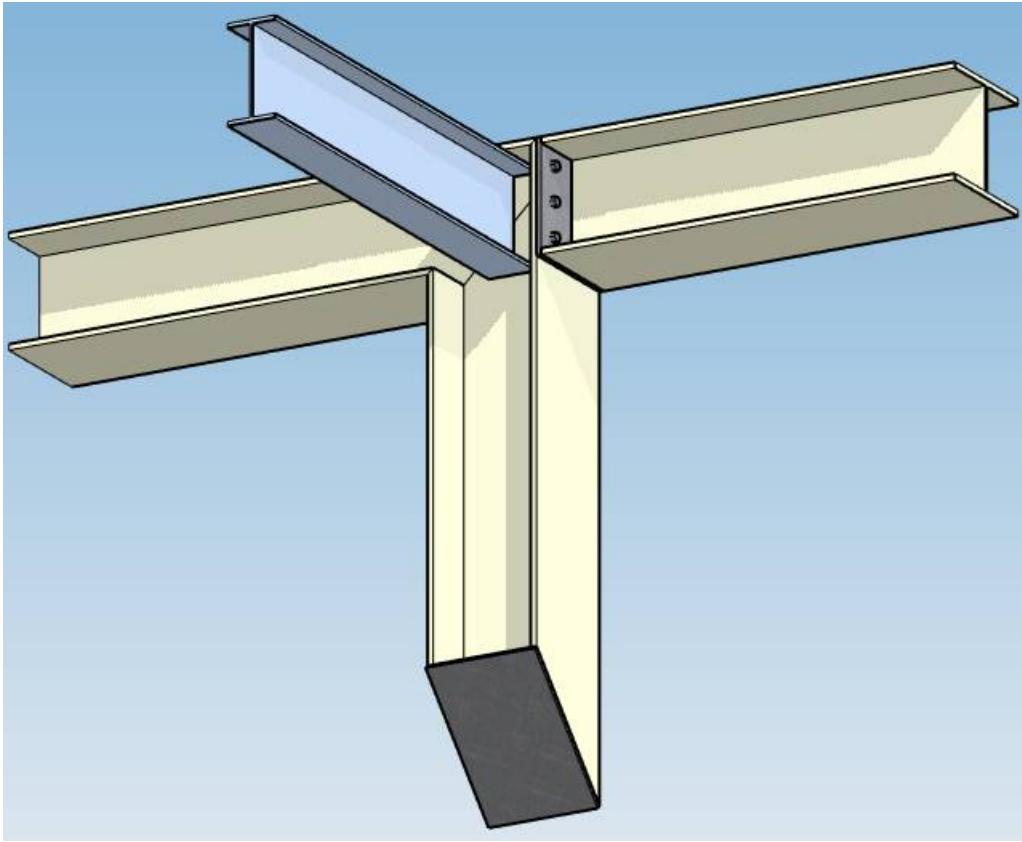
The assembly of a beam to another beam can be defined by a bolted plate :



Press the **Edit** button and set the plate properties :



Press **OK** to validate the bolted plate and press **Modify** to validate the beam :



Stiffnesses :

Property	Description	Unit Metric	Unit USA
Kx	Translation stiffness X	kN/mm	kips/in
Ky	Translation stiffness Y	kN/mm	kips/in
Kz	Translation stiffness Z	kN/mm	kips/in
Krx	Rotation stiffness X	kN.m/rad	kips.ft/rad
Kry	Rotation stiffness Y	kN.m/rad	kips.ft/rad
Krz	Rotation stiffness Z	kN.m/rad	kips.ft/rad

See §1.2 for more information.

Plate :

Explanation of the fields on the picture :

Joint definition

Stiffnesses

<input checked="" type="checkbox"/> Kx	<input type="text"/>	<input type="checkbox"/> Krx	<input type="text"/> 0
<input checked="" type="checkbox"/> Ky	<input type="text"/>	<input type="checkbox"/> Kry	<input type="text"/> 0
<input checked="" type="checkbox"/> Kz	<input type="text"/>	<input checked="" type="checkbox"/> Krz	<input type="text"/>

Plate

Width	260	Center X	30
Height	400	Center Y	0
Thickness	10	Material	S235 <input type="button" value="..."/>

Bolt arrangement

Nb sup	1
Nb int	3
Nb inf	2
Horiz. spacing	150
Sup. spacing	0
Int. spacing	70
Inf. spacing	50
Sup. shift	25
Inf. shift	30

Bolt

Name	M20 <input type="button" value="..."/>	Pitch	2.5
Diameter	20	Resistant area	245
		Material	8.8 <input type="button" value="..."/>

OK **Cancel**

The plate receives the dimensions of the beam when created.

Property	Description	Unit Metric	Unit USA
Width	Width of the plate	mm	in
Height	Height of the plate	mm	in
Thickness	Thickness of the plate	mm	in
Center X	Distance along X between center of plate and center of beam	mm	in
Center Y	Distance along Y between center of plate and center of beam	mm	in

Click on the ... button to select a *Structure* material :

Select material

Library
EN 10025-2

Material
S235

OK **Cancel**

Click [here](#) for more information about database of materials.

Bolt arrangement :

Explanation of the fields on the pictures :

Plate	
Width	260
Height	400
Thickness	10
Center X	30
Center Y	0
Material	S235
Bolt arrangement	
Nb sup	1
Nb int	3
Nb inf	2
Horiz. spacing	150
Sup. spacing	0
Int. spacing	70
Inf. spacing	50
Sup. shift	25
Inf. shift	30
Bolt	
Name	M20
Diameter	20
Pitch	2.5
Resistant area	245
Material	8.8

Horiz. spacing

Diameter

Bolt arrangement	
Nb sup	1
Nb int	3
Nb inf	2
Horiz. spacing	150
Sup. spacing	0
Int. spacing	70
Inf. spacing	50
Sup. shift	25
Inf. shift	30

Property	Unit Metric	Unit USA
Spacing	mm	in
Shift	mm	in

Bolt :

Bolt	
Name	M20
Diameter	20
Pitch	2.5
Resistant area	245
Material	8.8

Click on the ... button to select a Bolt :

Select bolt

Bolts

ISO

Name	ISO Coarse		
Units	Metric		
Bolts			
Name	Diameter	Pitch	Resistant area
M10	10	1.5	58
M12	12	1.75	84.3
M14	14	2	115
M16	16	2	157
M18	18	2.5	192
M20	20	2.5	245
M22	22	2.5	303
M24	24	3	353
M27	27	3	459
M30	30	3.5	561

OK Cancel

All properties are automatically filled.

Click [here](#) for more information about database of bolts.

Property	Unit Metric	Unit USA
Diameter	mm	in
Pitch	mm	in
Resistant are	mm ²	in ²

Click on the ... button to select a *Bolting* material :

Select material

Library
Bolting

Material
8.8

OK Cancel

Click [here](#) for more information about database of materials.

1.4 Welded joint

The assembly of a beam on another beam can be defined by welding :

Joints

Extremity 1: Welded Extremity 2: Fixed

Edit Copy | Paste Copy | Paste

Press the **Edit** button and set the weld properties :

Joint definition

Stiffnesses

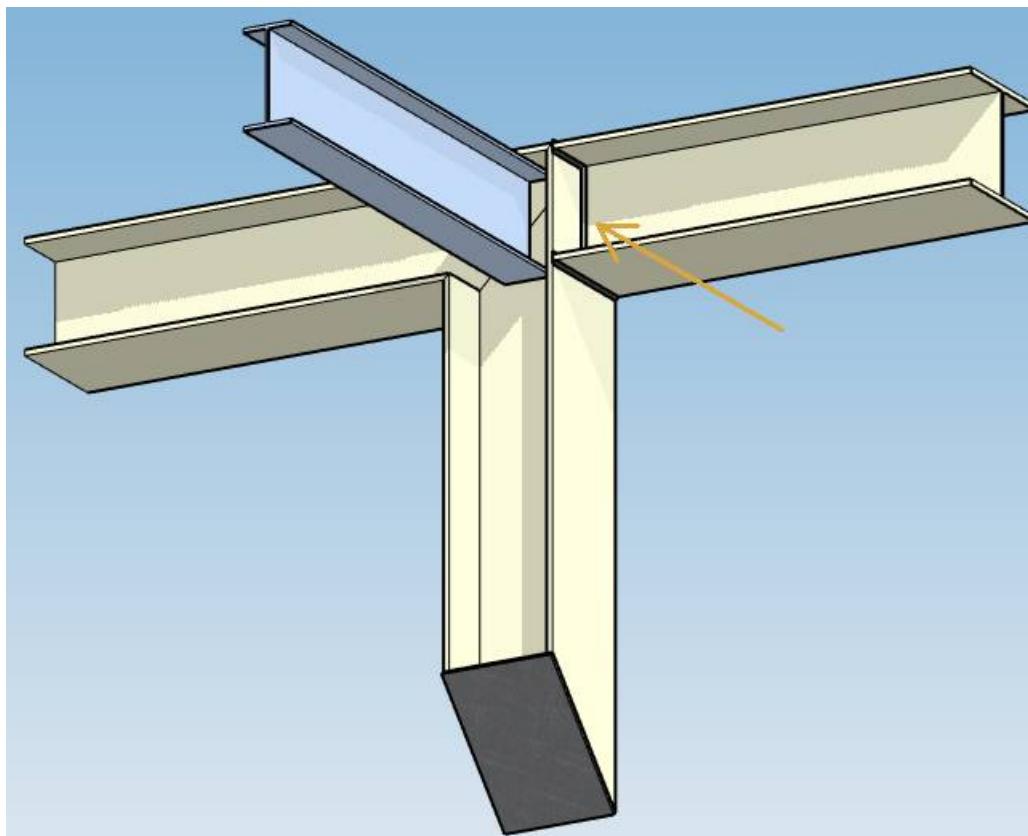
<input checked="" type="checkbox"/> Kx	<input type="text"/>	<input type="checkbox"/> Krx	<input type="text"/> 0
<input checked="" type="checkbox"/> Ky	<input type="text"/>	<input type="checkbox"/> Kry	<input type="text"/> 0
<input checked="" type="checkbox"/> Kz	<input type="text"/>	<input checked="" type="checkbox"/> Krz	<input type="text"/>

Weld

Flange throat	<input type="text"/> 5	Double flange	<input checked="" type="checkbox"/>
Web throat	<input type="text"/> 5	Material	<input type="text"/> Z_SDLX02 <input type="button"/> ...

OK Cancel

Press **OK** to validate the welding and press **Modify** to validate the beam :



Stiffnesses :

Property	Description	Unit Metric	Unit USA
Kx	Translation stiffness X	kN/mm	kips/in
Ky	Translation stiffness Y	kN/mm	kips/in
Kz	Translation stiffness Z	kN/mm	kips/in
Krx	Rotation stiffness X	kN.m/rad	kips.ft/rad
Kry	Rotation stiffness Y	kN.m/rad	kips.ft/rad
Krz	Rotation stiffness Z	kN.m/rad	kips.ft/rad

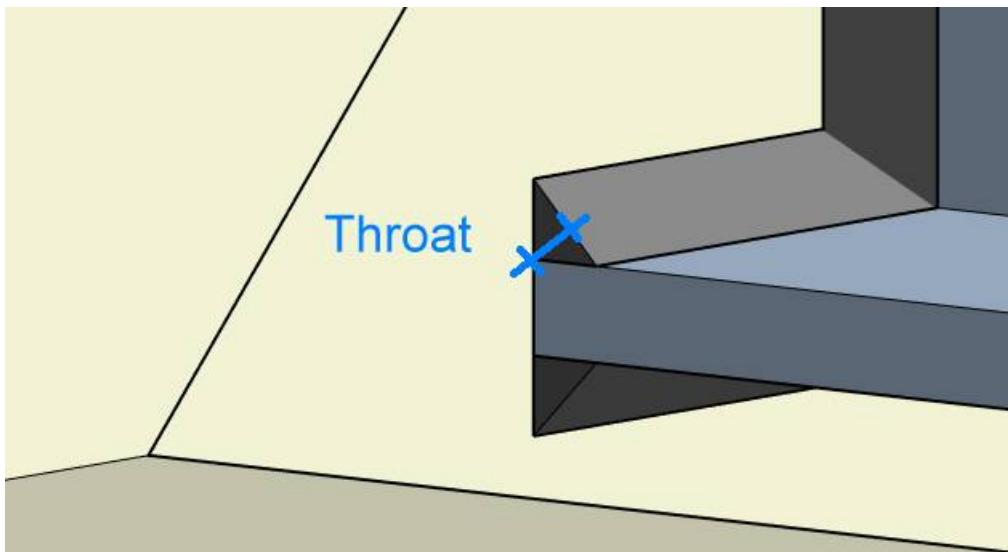
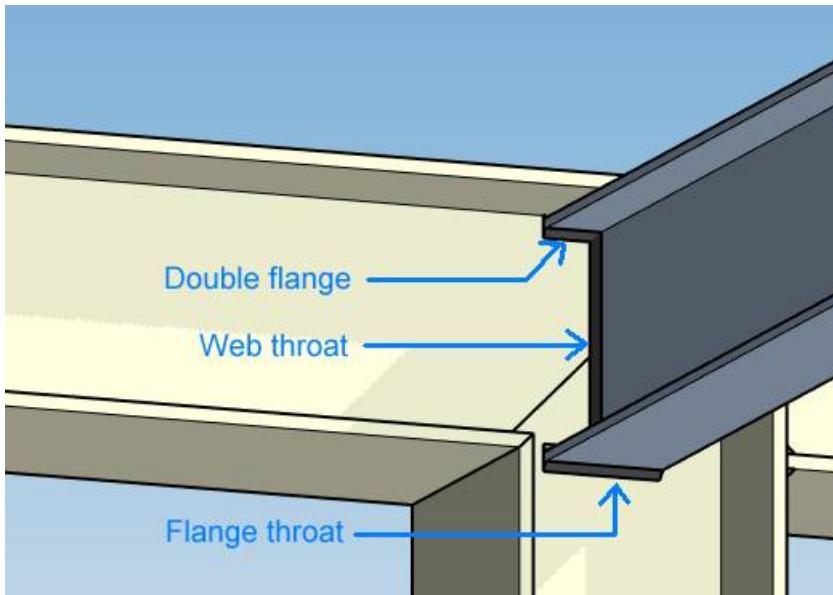
See §1.2 for more information.

Weld :

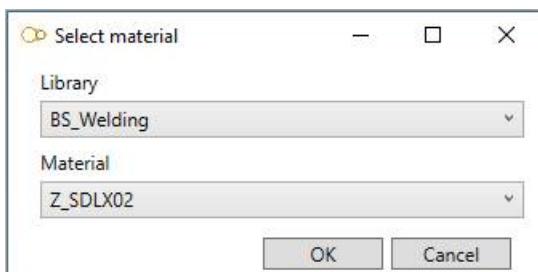
Weld

Flange throat	<input type="text"/> 5	Double flange	<input checked="" type="checkbox"/>
Web throat	<input type="text"/> 5	Material	<input type="text"/> Z_SDLX02 <input type="button"/> ...

Property	Description	Unit Metric	Unit USA
Flange throat	Apothem of the bead	mm	in



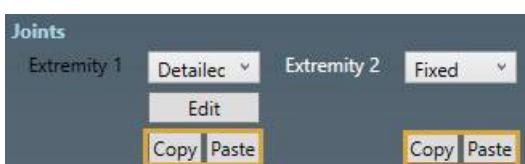
Click on the ... button to select a *Welding* material :



Click [here](#) for more information about database of materials.

1.5 Duplicate joint

The definition of a **joint** (fixed, detailed, bolted or welded) can be copied and pasted to the extremity of any beams :



WARNING

Don't forget to click to the **Modify** button to validate the duplication and see the result in 3D !

Click [here](#) to return to **beam** page.

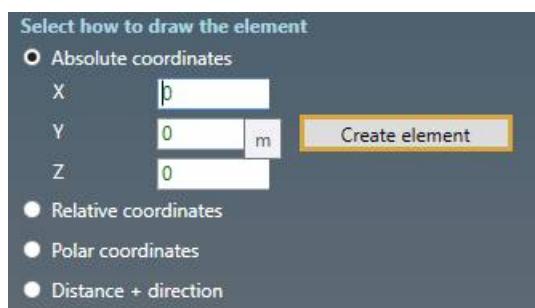
Orientation

This tool is common to all elements.

The goal is to help user to set the second extremity node, based on the first selected one.

1. Absolute coordinates

This mode is useful when you want to draw an element with the second node defined in **absolute global coordinates** :



Property	Unit Metric	Unit USA
X	m	ft
Y	m	ft
Z	m	ft

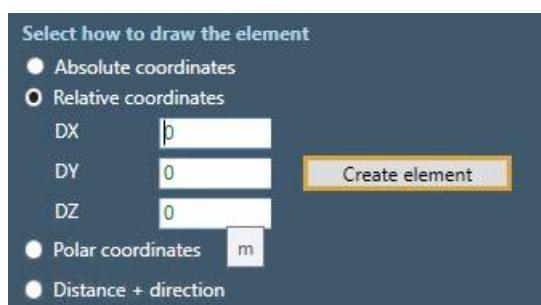
To know the UNIT of the value, just let the mouse over the cell.

Click the **Create element** button to create the element.

You can **Undo** this command.

2. Relative coordinates

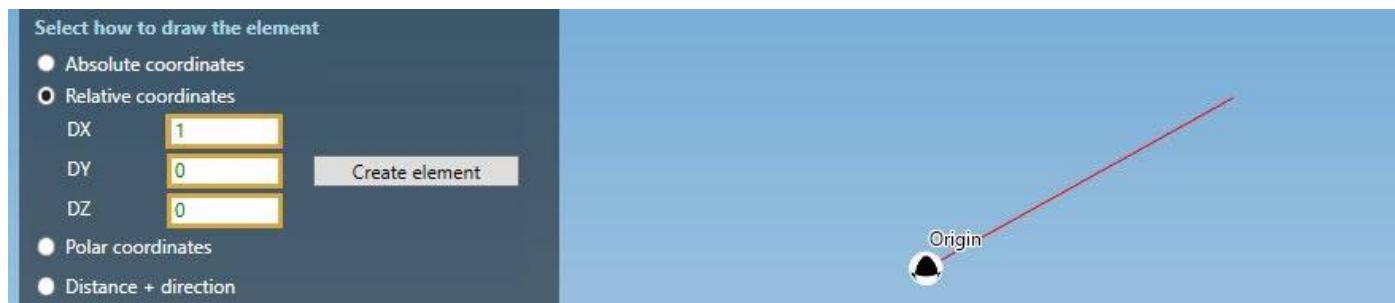
This mode is useful when you want to draw an element with the second node **relative** to the first one in the **global axis** directions :



Property	Unit Metric	Unit USA
DX	m	ft
DY	m	ft
DZ	m	ft

To know the UNIT of the value, just let the mouse over the cell.

TIP: a preview of the element length will be shown when entering values :

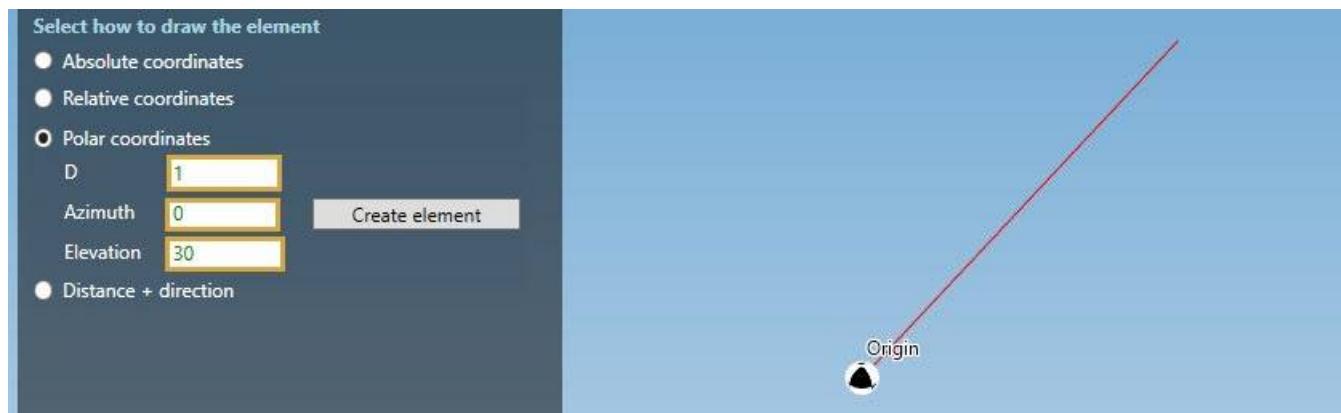


Click the **Create element** button to create the element.

You can **Undo** this command.

3. Polar coordinates

This mode is useful when you want to draw an element with the second point relative to the first one by a **distance** and **2 angles** :



Property	Description Unit	Metric	Unit USA
D	Distance	m	ft
Azimuth	Horizontal angle	°	°
Elevation	Vertical angle	°	°

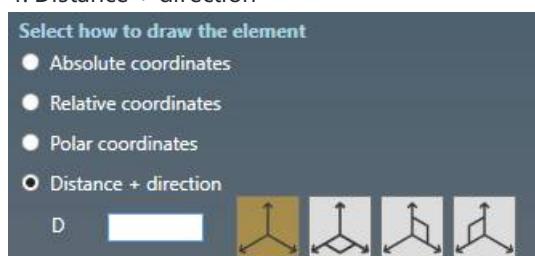
To know the UNIT of the value, just let the mouse over the cell.

TIP: a preview of the element length will be shown when entering values.

Click the **Create element** button to create the element.

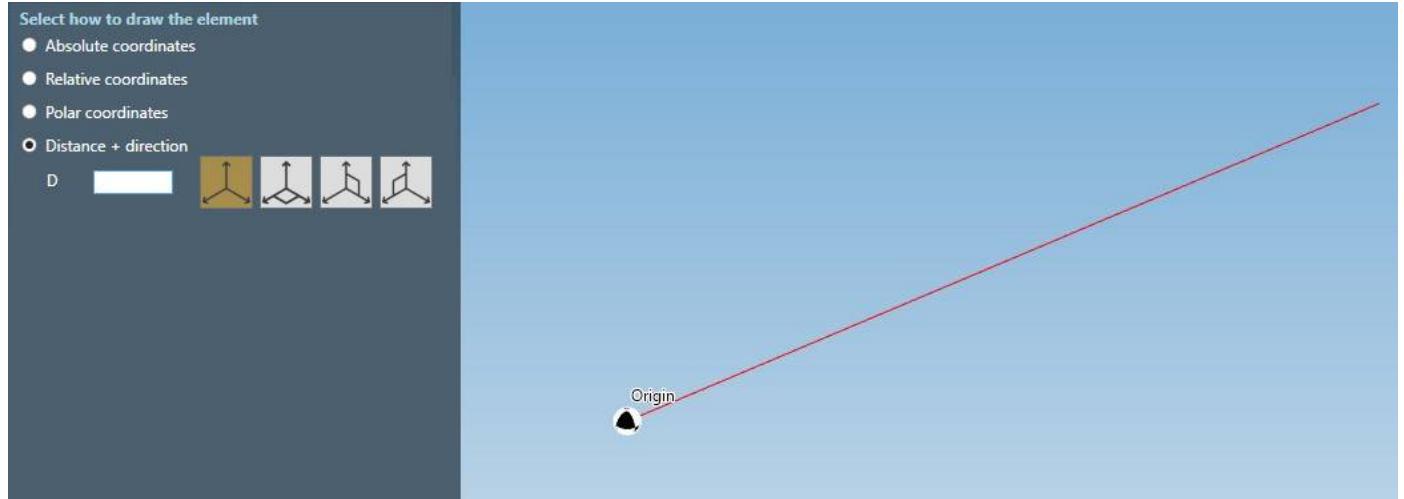
You can **Undo** this command.

4. Distance + direction



This mode is useful when you want to draw an element with a **standard direction** (+X, -X, +Y, -Y, +Z, -Z) and a **distance** from the first node :

Property	Unit Metric	Unit USA
Distance	m	ft



Move the mouse in the desired direction. The line will take the same color as the **global axis**. Red for **X**, green for **Y** and blue for **Z**.

Write the distance on the **keyboard** (m or ft) :



The value will automatically be entered in the corresponding cell (D).

Click on **Enter** to execute the creation.



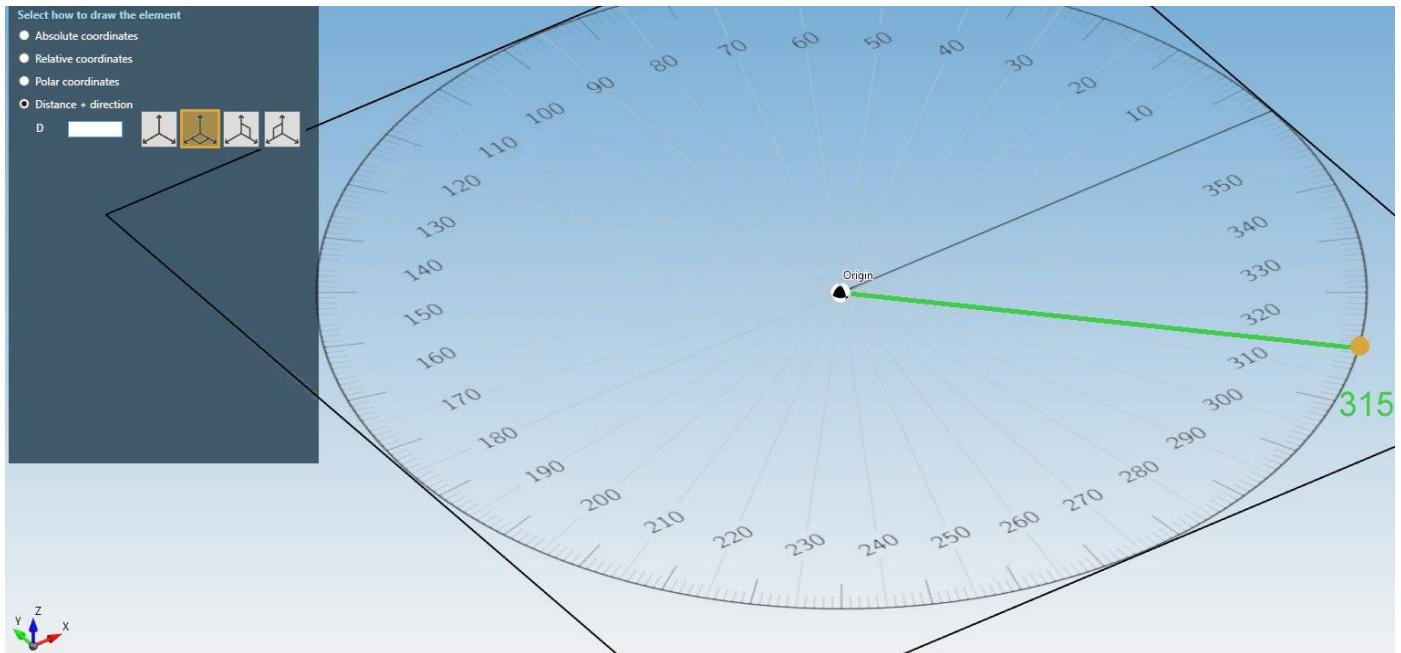
WARNING

ATTENTION, do not write the value in the cell directly !

You can **Undo** this command.

4.1 Horizontal plane

Click on the second button to display a **horizontal graduated wheel** :



Bring mouse closer to an angle to be magnetized (Ex : at 315 °).

Now that the direction is established, as in §3, write the distance on the **keyboard** :



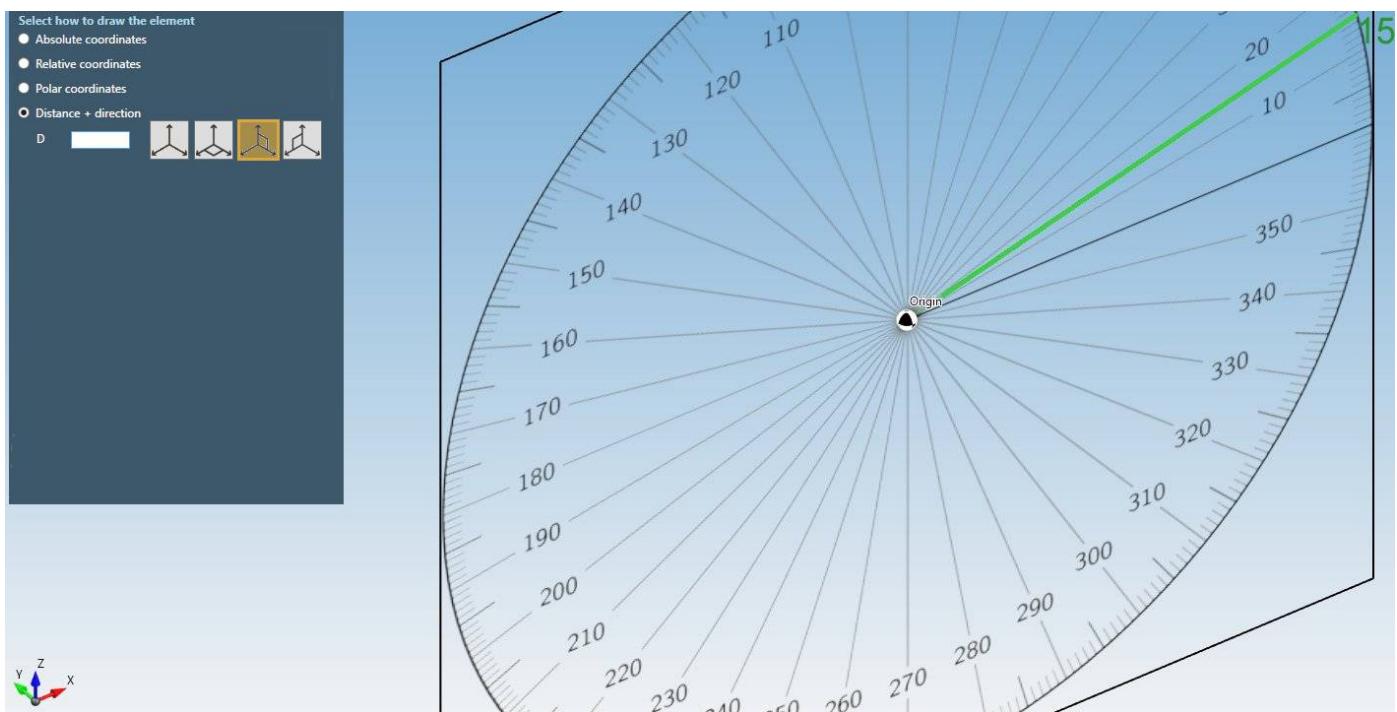
And validate :



The second node will be at "Distance" from the first node in the direction of the selected angle.

4.2 Vertical plane

Click on the third button to display a **vertical graduated wheel** :



Bring mouse closer to an angle to be magnetized (Ex : at 15 °).

Now the direction is established, as in §3., write the distance on the **keyboard** :



And validate :



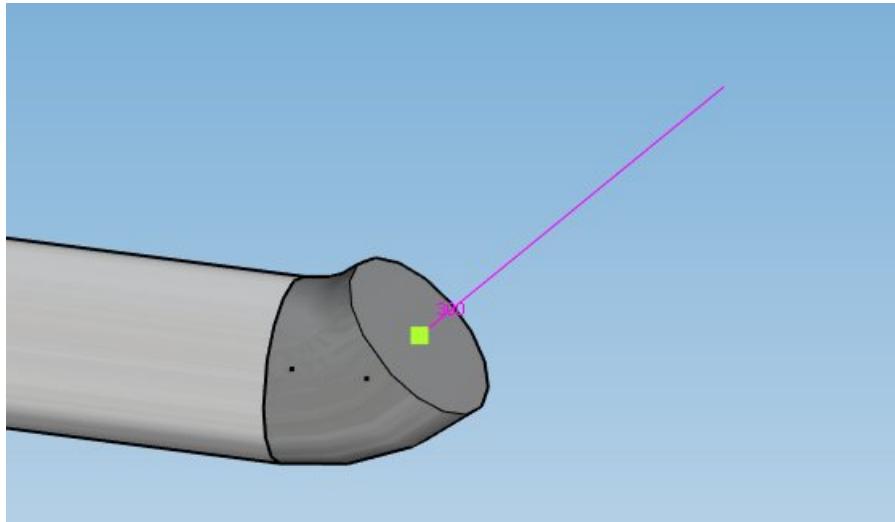
The second node will be at "Distance" from the first node in the direction of the selected angle.

You can **Undo** this command.

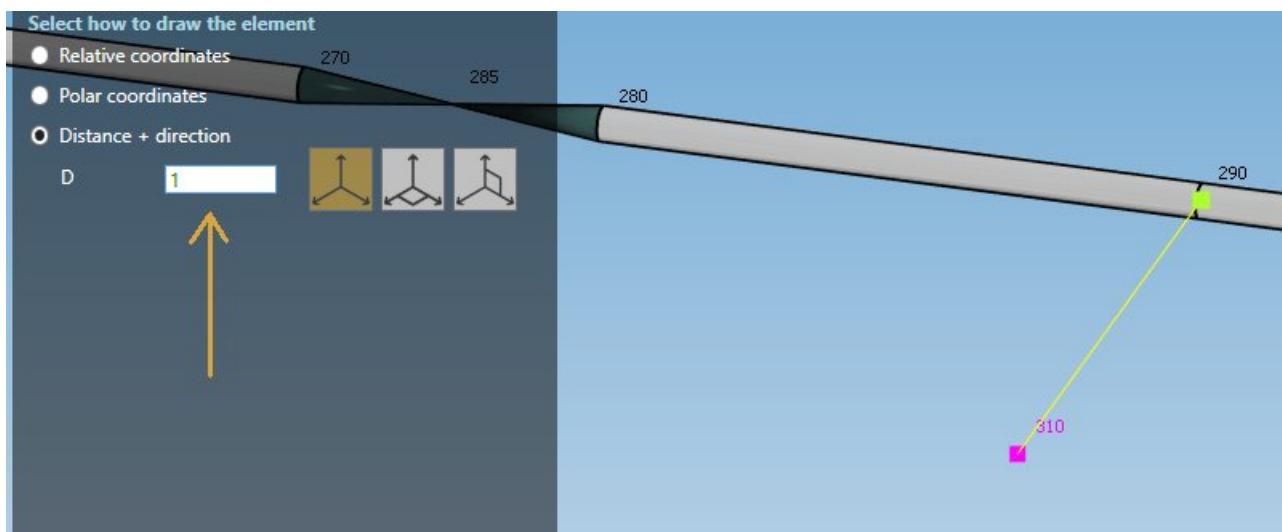
The last button is the **vertical plane** perpendicular to the element.

4.3 Special cases

- Tangent direction (in MAGENTA) :



-
- Snap to an existing node (for branch connection - in YELLOW) :



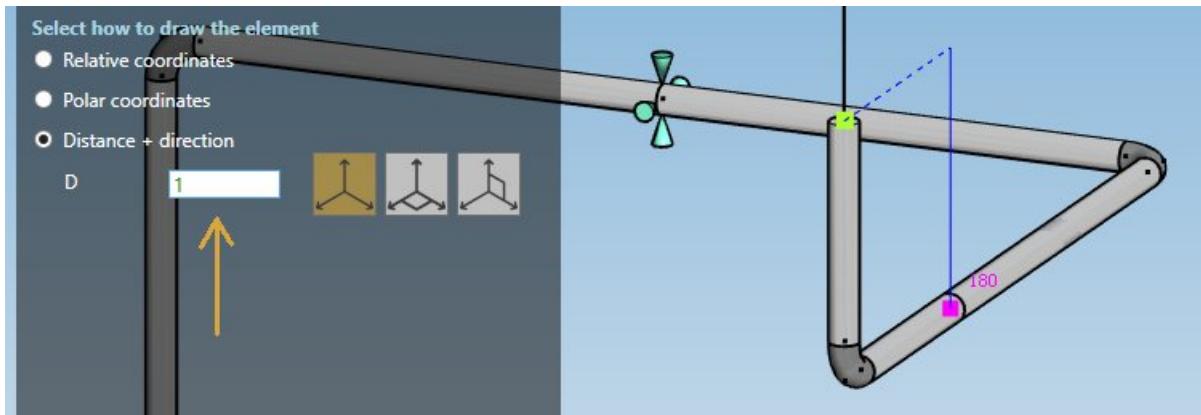
TIP : The distance is automatically filled in the cell. Press ENTER to create the element :



The point (GREEN) becomes a REFERENCE point

You can **Undo** this command.

- Indication of similar coordinate with a REFERENCE point :



- 1 Snap to a point (become the new REFERENCE POINT in GREEN)
- 2 Move mouse vertically from the first point and wait that the **dashed lines** appear.
- 3 The blue line indicates that the **Z** coordinate is similar to the coordinate of the REFERENCE point. Same for the **dashed lines**.
- 4 The distance is automatically filled in the cell.
- 5 Press **ENTER** to create the element :



You can **Undo** this command.

Restraints

MetaPiping lets you create several types of restraints on nodes :

- Anchors
- Multiple restraints
- Rotational restraints
- Translational restraints
- Snubbers
- Variable spring hangers
- Constant spring hangers
- Non-linear restraints

1. Database

You can create **Catalogs** of restraints in the database :

Name	My Support Library
Units	Metric
Rotational restraints	[+]
Translational restraints	[+]
Snubbers	[+]
Variable spring hanger series	[+]
Constant spring hanger series	[+]

Click [here](#) to have more information about the support catalog creation.

2. Anchors

Select a **Node** and click on **Add restraint** button :



Select the type **Anchor** :

Restraint

Set restraint definition

Select restraint type

Anchor

Label

Spring and rotational spring constants in X,Y,Z directions

Kx

K Θ x

Ky

K Θ y

Kz

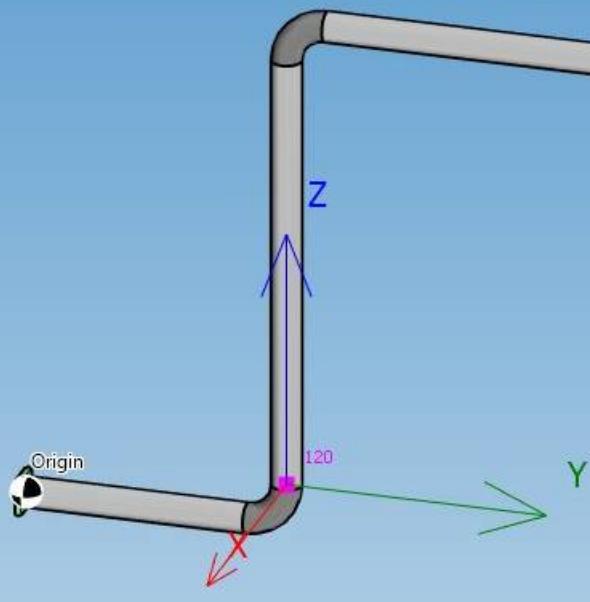
K Θ z

Coord. system

Global

Level

Create



Property	Description	Unit Metric	Unit USA
Label	Label of the restraint	-	-
Kx	Spring Constant acting in X direction	kN/mm	Kips/in
Ky	Spring Constant acting in Y direction	kN/mm	Kips/in
Kz	Spring Constant acting in Z direction	kN/mm	Kips/in
K Θ x	Rotational Spring Constant acting in X direction	kN.m/rad	Kips.ft/rad
K Θ y	Rotational Spring Constant acting in Y direction	kN.m/rad	Kips.ft/rad
K Θ z	Rotational Spring Constant acting in Z direction	kN.m/rad	Kips.ft/rad

To know the UNIT of the value, just move the mouse over the cell.

The labels are shown with the node names view button.

You can change the coordinate system that defines the directions X,Y,Z :

- Global (default)
- Local to adjacent element

The **Level** is used to identify the spectrum applied to this anchor for multilevel floor response spectrum analysis.

You can later modify or remove this anchor by selecting it and clicking on **Modify/Remove** buttons.

You can **Undo** this command.

3. Multiple restraints

Select a **Node** and click on **Add restraint** button :



Select the type **Multiple restraint** :

Restraint

Set restraint definition

Select restraint type

Multiple restraint

Label

Fixed rigid restraint in X,Y,Z directions

Dx	<input type="checkbox"/>	Rx	<input type="checkbox"/>
Dy	<input type="checkbox"/>	Ry	<input type="checkbox"/>
Dz	<input type="checkbox"/>	Rz	<input type="checkbox"/>

Coord. system **Global**

Level

Create

Property	Description
Label	Label of the restraint
Dx	Checked if there is a fixed rigid restraint acting in X direction
Dy	Checked if there is a fixed rigid restraint acting in Y direction
Dz	Checked if there is a fixed rigid restraint acting in Z direction
Rx	Checked if there is a fixed rigid rotational restraint about X axis
Ry	Checked if there is a fixed rigid rotational restraint about Y axis
Rz	Checked if there is a fixed rigid rotational restraint about Z axis

The labels are shown with the node names view button.

You can change the coordinate system that defines the directions X,Y,Z :

- Global (default)
- Local to adjacent element

The **Level** is used to identify the spectrum applied to this restraint for multilevel floor response spectrum analysis.

You can later modify or remove this restraint by selecting it and clicking on **Modify/Remove** buttons.

You can **Undo** this command.

4. Rotational restraints

Select a **Node** and click on **Add restraint** button :



Select the type **Rotational restraint** :

Restraint

Set restraint definition

Select restraint type: Rotational restraint

Label: [Input field]

Restraint directions and K-Spring constant:

Rx	1	Sp	[Input field]
Ry	0		kN.m/rad
Rz	0		

Coord. system: Global

Level: [Input field]

Create

Property	Description	Unit Metric	Unit USA
Label	Label of the restraint	-	-
Rx	X direction of the restraint	-	-
Ry	Y direction of the restraint	-	-
Rz	Z direction of the restraint	-	-
Sp	Rotational Spring Constant	kN.m/rad	Kips.ft/rad

if Sp is blank = rigid, which is modeled by using a value of 1.13×10^9 kN.m/rad or 8.33×10^8 Kips.ft/rad.

To know the UNIT of the value, just move the mouse over the cell.

The labels are shown with the node names view button.

The **Database** button on upper right corner gives an access to the libraries :



Select a library on the left treeview, then select a row that represents a predefined restraint. Some values are just informative. Click **OK**. The properties are copied to the restraint screen.

You can change the coordinate system that defines the directions X,Y,Z :

- Global (default)
- Local to adjacent element

The **Level** is used to identify the spectrum applied to this restraint for multilevel floor response spectrum analysis.

You can later modify or remove this restraint by selecting it and clicking on **Modify/Remove** buttons.

You can **Undo** this command.

5. Translational restraints

Select a **Node** and click on **Add restraint** button :



Select the type **Restraint** :

Restraint

Set restraint definition

Select restraint type: **Restraint**

Label:

Restraint directions and K-Spring constant:

Dx	1	Sp	<input type="text"/>
Dy	0		
Dz	0		

Coord. system: **Local to following element coordinate**

Level:

Create

Property	Description	Unit Metric	Unit USA
Label	Label of the restraint	-	-
Dx	X direction of the restraint	-	-
Dy	Y direction of the restraint	-	-
Dz	Z direction of the restraint	-	-
Sp	Translational Spring Constant	kN/mm	Kips/in

if Sp is blank = rigid, which is modeled by using a value of 8.75×10^5 N/mm or 5×10^6 lb/in.

To know the UNIT of the value, just move the mouse over the cell.

The labels are shown with the node names view button.

The **Database** button on upper right corner gives an access to the libraries.

You can change the coordinate system that defines the directions X,Y,Z :

- Global (default)
- Local to adjacent element (**in yellow in the picture**)

The **Level** is used to identify the spectrum applied to this restraint for multilevel floor response spectrum analysis.

You can later modify or remove this restraint by selecting it and clicking on **Modify/Remove** buttons.

You can **Undo** this command.

6. Snubbers

Select a **Node** and click on **Add restraint** button :



Select the type **Snubber** :

Restraint

Set restraint definition

Select restraint type
Snubber

Label

Restraint directions and K-Spring constant

Dx	1	Sp	
Dy	0		
Dz	0		

Coord. system
Local to preceding element coordinate

Level

Create

Property	Description	Unit Metric	Unit USA
Label	Label of the restraint	-	-
Dx	X direction of the restraint	-	-
Dy	Y direction of the restraint	-	-
Dz	Z direction of the restraint	-	-
Sp	Translational Spring Constant	kN/mm	Kips/in

if Sp is blank = rigid, which is modeled by using a value of 2.625×10^5 N/mm or 1.5×10^6 lb/in.

To know the UNIT of the value, just move the mouse over the cell.

The labels are shown with the node names view button.

The **Database** button on upper right corner gives an access to the libraries.

You can change the coordinate system that defines the directions X,Y,Z :

- Global (default)
- Local to adjacent element (**in yellow in the picture**)

The **Level** is used to identify the spectrum applied to this restraint for multilevel floor response spectrum analysis.

You can later modify or remove this restraint by selecting it and clicking on **Modify/Remove** buttons.

You can **Undo** this command.

7. Variable spring hangers

Select a **Node** and click on **Add restraint** button :



Select the type **Variable support spring hanger** :

Restraint

Set restraint definition

Select restraint type : Variable support spring hanger

Label

Restraint directions, K-Spring constant and force

Dx	1	Sp	
Dy	0	Force	0
Dz	0		kN

Coord. system : Global

Level :

Create

Property	Description	Unit Metric	Unit USA
Label	Label of the restraint	-	-
Dx	X direction of the restraint	-	-
Dy	Y direction of the restraint	-	-
Dz	Z direction of the restraint	-	-
Sp	Spring Constant	kN/mm	Kips/in
Force	Pre-compression/pre-tension force	kN	Kips

if Sp is blank = rigid, which is modeled by using a value of 8.75×10^5 N/mm or 5×10^6 lb/in.

To know the UNIT of the value, just move the mouse over the cell.

The labels are shown with the node names view button.

The **Database** button on upper right corner gives an access to the libraries.

You can change the coordinate system that defines the directions X,Y,Z :

- Global (default)
- Local to adjacent element

The **Level** is used to identify the spectrum applied to this restraint for multilevel floor response spectrum analysis.

You can later modify or remove this restraint by selecting it and clicking on **Modify/Remove** buttons.

You can **Undo** this command.

8. Constant spring hangers

Select a **Node** and click on **Add restraint** button :



Select the type **Constant support hanger** :

Restraint

Set restraint definition

Select restraint type
Constant support hanger

Label

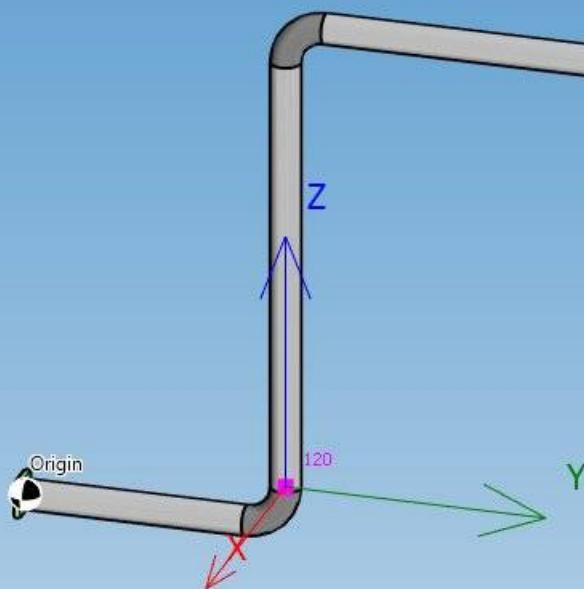
Restraint directions and force

Dx	1	Sp	0
Dy	0	Force	0
Dz	0		

Coord. system Global

Level

Create



Property	Description	Unit Metric	Unit USA
Label	Label of the restraint	-	-
Dx	X direction of the restraint	-	-
Dy	Y direction of the restraint	-	-
Dz	Z direction of the restraint	-	-
Force	Pre-compression/pre-tension force	kN	Kips

To know the UNIT of the value, just move the mouse over the cell.

The **Labels** are shown with the node names view button.

The **Database** button on upper right corner gives an access to the libraries.

You can change the coordinate system that defines the directions X,Y,Z :

- Global (default)
- Local to adjacent element

The **Level** is used to identify the spectrum applied to this restraint for multilevel floor response spectrum analysis.

You can later modify or remove this restraint by selecting it and clicking on **Modify/Remove** buttons.

You can **Undo** this command.

9. Non-linear restraint

Select a **Node** and click on **Add restraint** button :



Select the type **Non-linear restraint**:

Restraint

Set restraint definition

Select restraint type

Non-linear restraint

Restraint directions and K-Spring constant

Dx	1	Sp	<input type="text"/>
Dy	0		
Dz	0		

Properties

Negative gap 0

FLm 0

K1m

K2m

Positive gap 0

FLp 0

K1p

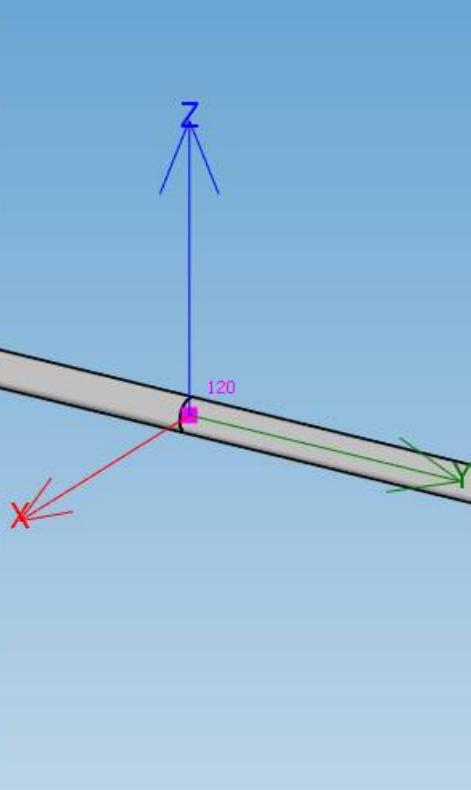
K2p

Coef. of friction 0

Coord. system Global

Level

Create



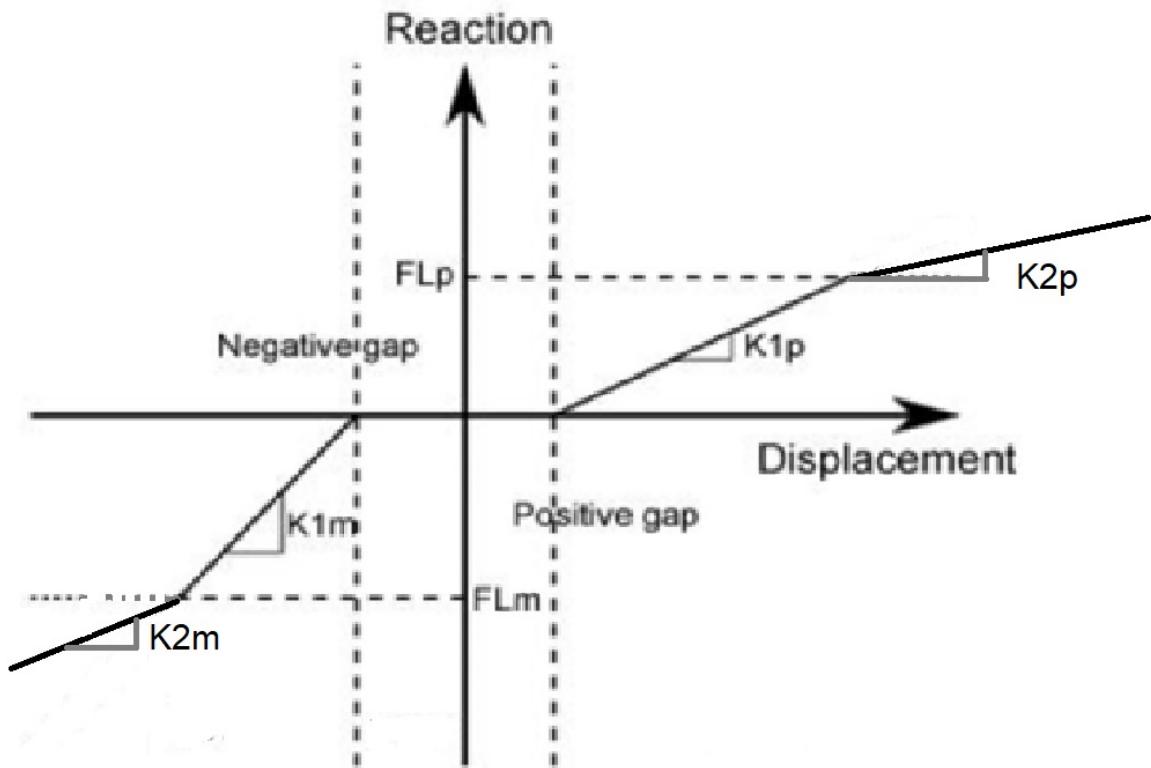
Property	Description	Unit Metric	Unit USA
Label	Label of the restraint	-	-
Dx	X direction of the restraint	-	-
Dy	Y direction of the restraint	-	-
Dz	Z direction of the restraint	-	-
Sp	Translational Spring Constant	kN/mm	Kips/in

if Sp is blank = rigid, which is modeled by using a value of 8.75×10^5 N/mm or 5×10^6 lb/in.

To know the UNIT of the value, just move the mouse over the cell.

The labels are shown with the node names view button.

Property	Description	Unit Metric	Unit USA
Negative gap	Must be ≤ 0	mm	in
K1m	First spring constant in the negative direction	kN/mm	Kips/in
K2m	Second spring constant in the negative direction	kN/mm	Kips/in
Positive gap	Must be ≥ 0	mm	in
K1p	First spring constant in the positive direction	kN/mm	Kips/in
K2p	Second spring constant in the positive direction	kN/mm	Kips/in
FLm	Reaction at stiffness transition from K1m to K2m (≤ 0 , infinite if blank or 0)	kN	Kips
FLp	Reaction at stiffness transition from K1p to K2p (≥ 0 , infinite if blank or 0)	kN	Kips
Coef. of friction	Coulomb coefficient of friction between the pipe and the support (≥ 0)	-	-



You can change the coordinate system that defines the directions X,Y,Z :

- Global (default)
- Local to adjacent element

The **Level** is used to identify the spectrum applied to this restraint for multilevel floor response spectrum analysis.

You can later modify or remove this restraint by selecting it and clicking on **Modify/Remove** buttons.

You can **Undo** this command.

Example :

Restraint

Set restraint definition

Select restraint type

Non-linear restraint

Restraint directions and K-Spring constant

Dx 0
Dy 0
Dz 1

Sp []

Properties

Negative gap -10 FLm 0
K1m 1E5 K2m 1E6
Positive gap 10 FLp 0
K1p 1E5 K2p 1E6

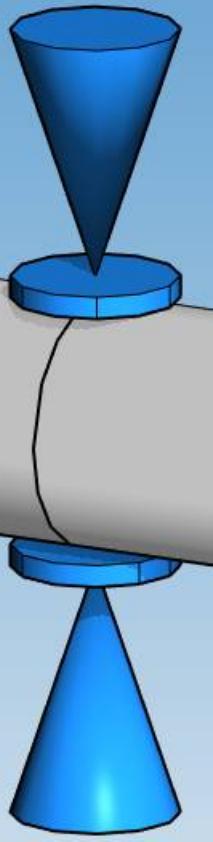
Coeff. of friction 0.5

Coord. system Global

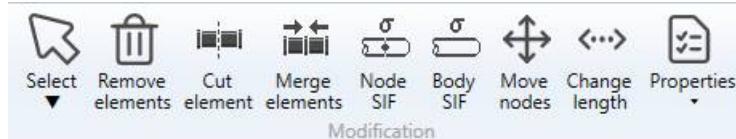
Level []

Modify

Remove

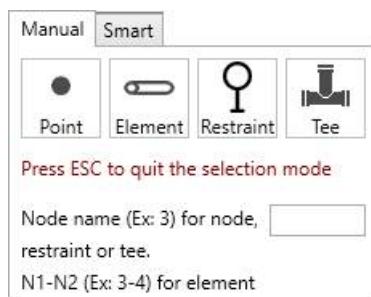


Modification



1. Selection

Click on **Select** button to open the selection window :



- Click on **Point** to select nodes
- Click on **Element** to select piping elements (pipe, bend, reducer, valve, etc.)
- Click on **Restraint** to select restraints
- Click on **Tee** to select nodes where a tee has been assigned
- Press **ESC** on keyboard to hide the selection window

You can now select objects of corresponding type.

The selection is done by pressing the left mouse button above the desired object

You can also select a node by entering the node name and click to the **Point** button.

You can also select an element by entering the two node names separate by “-” and click to the **Element** button.

You can also select a valve by entering its label and click to the **Element** button.

You can also select a restraint by entering the node name or its label and click to the **Restraint** button.

Click [here](#) to have more information about the selection.

2. Remove elements

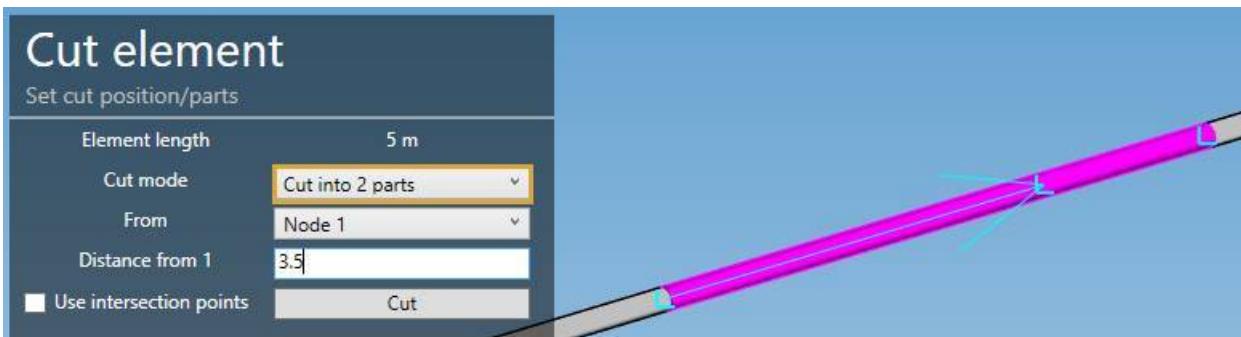
Select elements and click on the **Remove elements** button to delete these elements.

You can **Undo** this command.

3. Cut element

It is possible to cut **pipe** in several parts.

Select one pipe and click to the **Cut element** button :



Cut mode can be :

- Cut into 2 parts
- Cut into N parts

The current length of the pipe is shown on top.

By default, a distance from the first node of the pipe is calculated, which is the half length of the selected pipe (middle point).

You can choose from which node to start and the distance.

Property	Unit Metric	Unit USA
Length	m	ft
Distance	m	ft

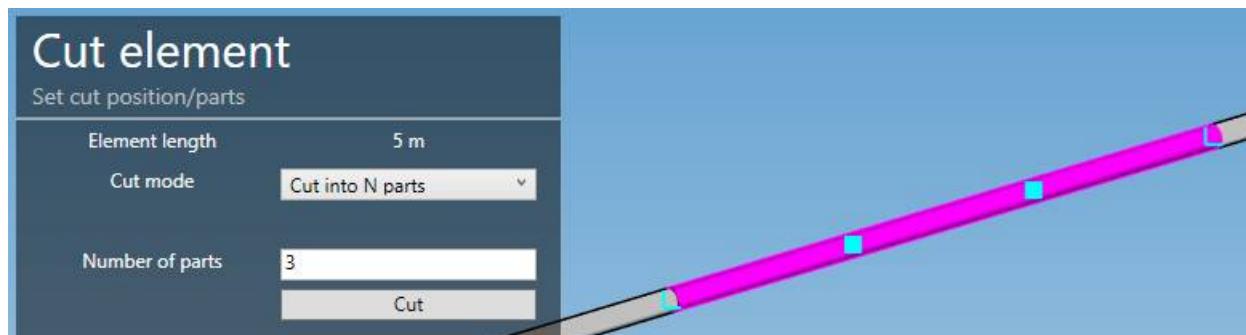
If the pipe has a bend before, next or both, you can check the cell *Use intersection point* to calculate the distance between these points instead of the extremities of the pipe.

Click on **Cut** to execute the command.

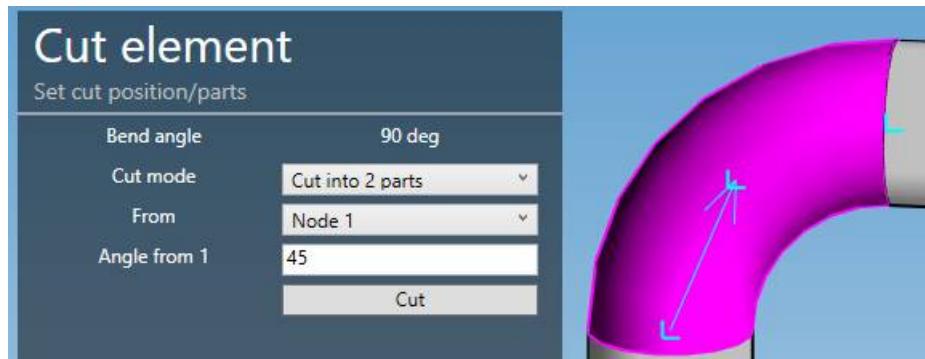
To know the UNIT of the value, just let the mouse over the cell.

You can **Undo** this command.

The second possibility is to cut in N parts with preview :



The tool is available only for pipes, beams and bends. For bends, length and distance are replaced by angles :



4. Merge elements

Select a **Node** between 2 similar colinear pipes or 2 successive bends with same center and radius, and click on the **Merge** button.

The 2 pipes/bends must have the same **Material** and the same **Section**.

The 2 pipes/bends will be replaced by one pipe/bend.

WARNING

Attention, the force, restraint, lumped mass or local coordinates defined on the node will be destroyed.

You can **Undo** this command.

5. Node SIF

MetaPiping lets you define **Stress Intensification Factors** and **Stress Indices** on nodes.

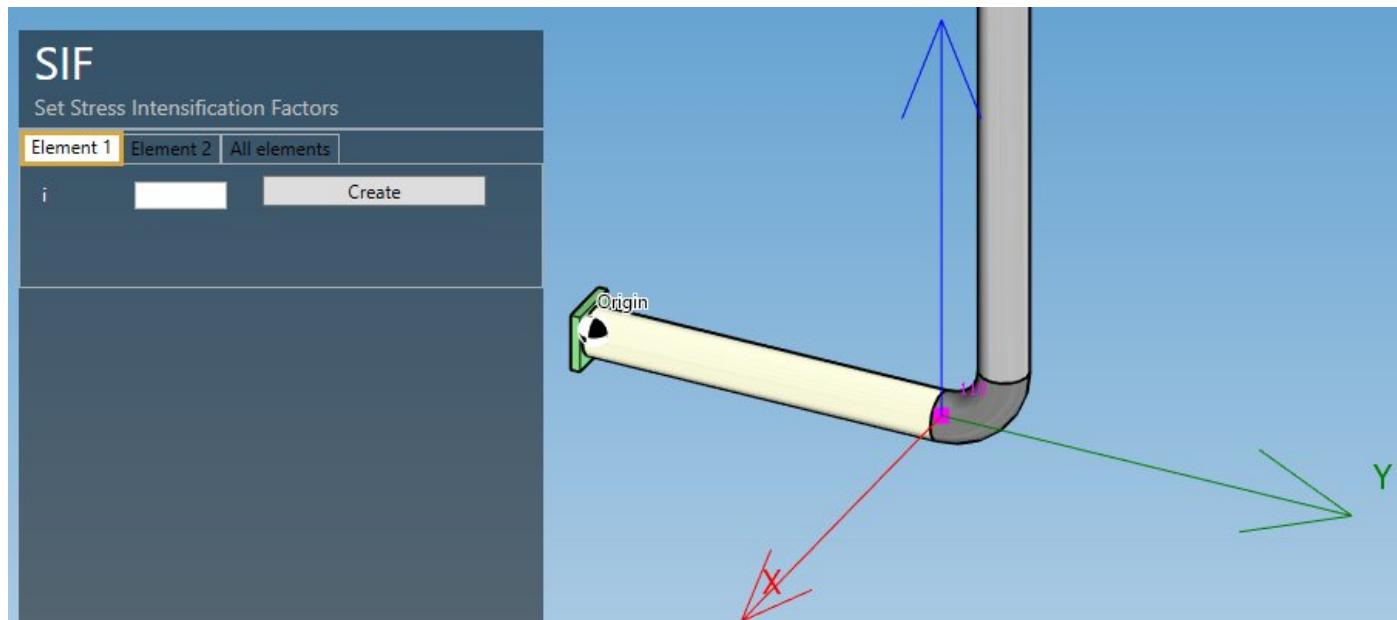
Each **piping code** has its own factors.

Code	SIFS
ASME Class 1	B1, C1, K1, B2, C2, K2, C3, C3', K3
ASME Class 2	i, B1, B2, B2', C2
B31.1	i
B31.1 B31.J	ii, io, it, ia, li, lo, lt, la
EN 13480	io, ii
RCCM Class 2	i

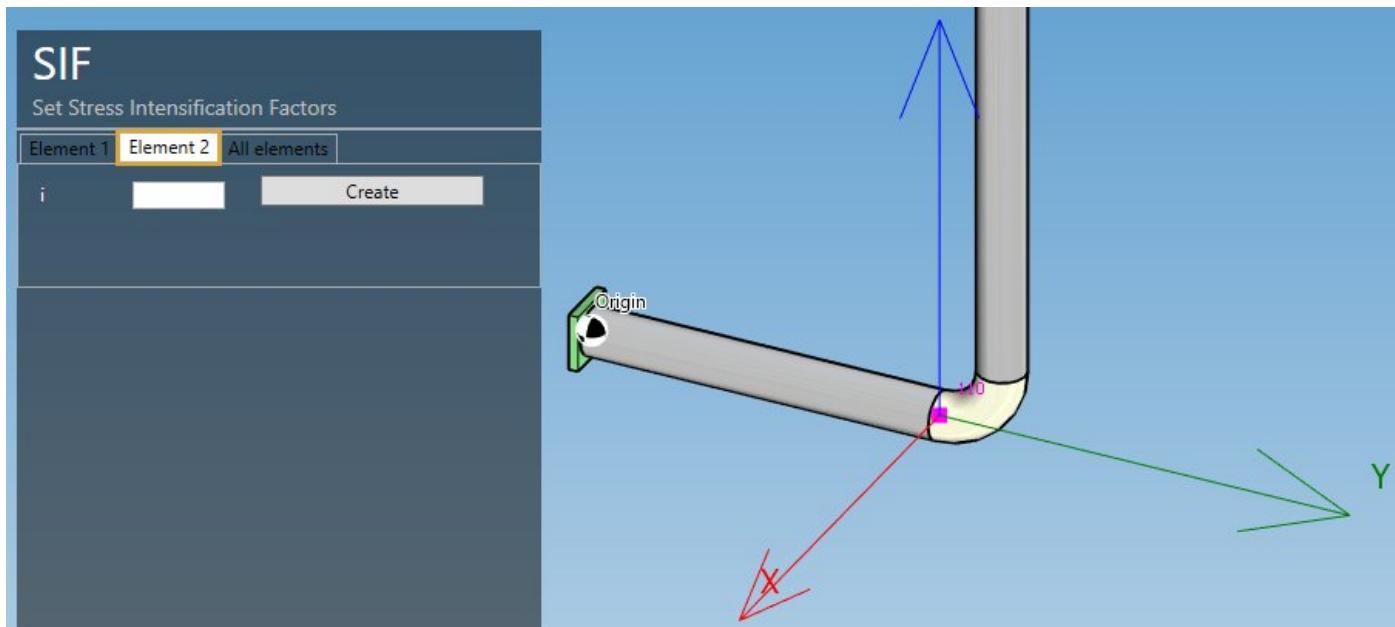
These factors are unitless.

Select a node and click on the **Node SIF** button.

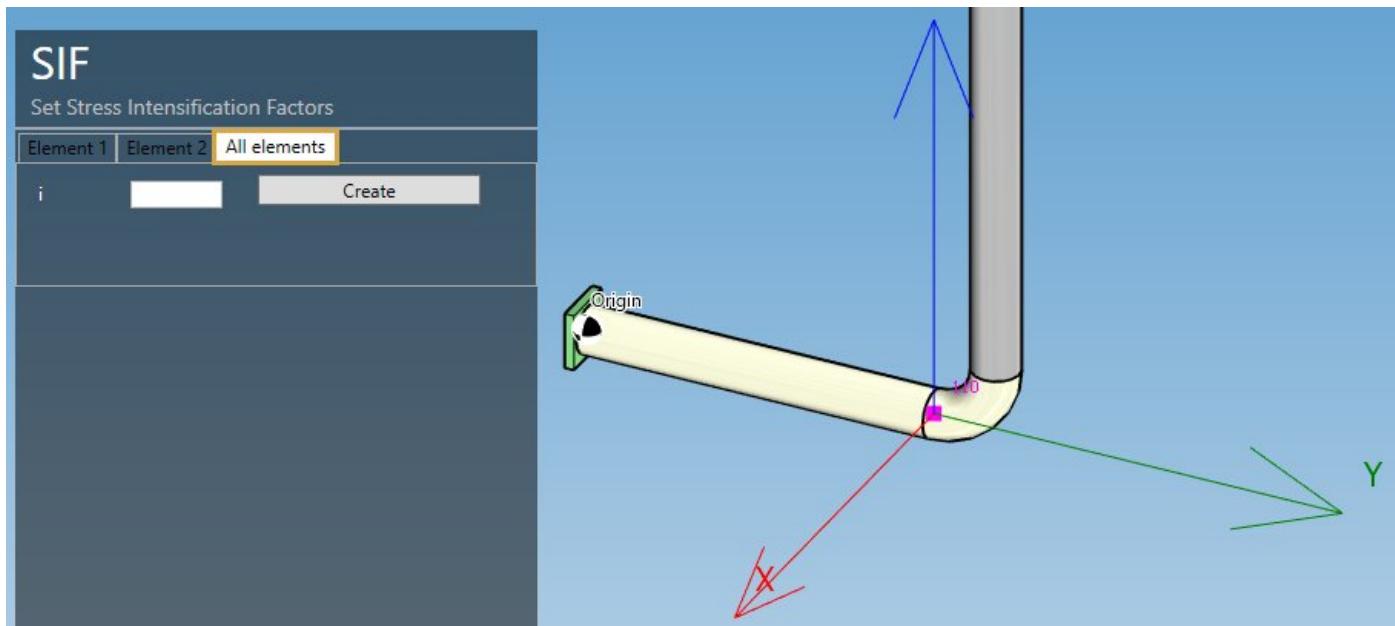
- You can change one (or more) factor(s) at the extremity of the first element connected to the node, e.g. in RCC-M Class 2 code :



- You can change one (or more) factor(s) at the **extremity** of the second element connected to the node :



- You can change one (or more) factor(s) of all elements :



Click on **Create** button to apply the SIFs to the model.

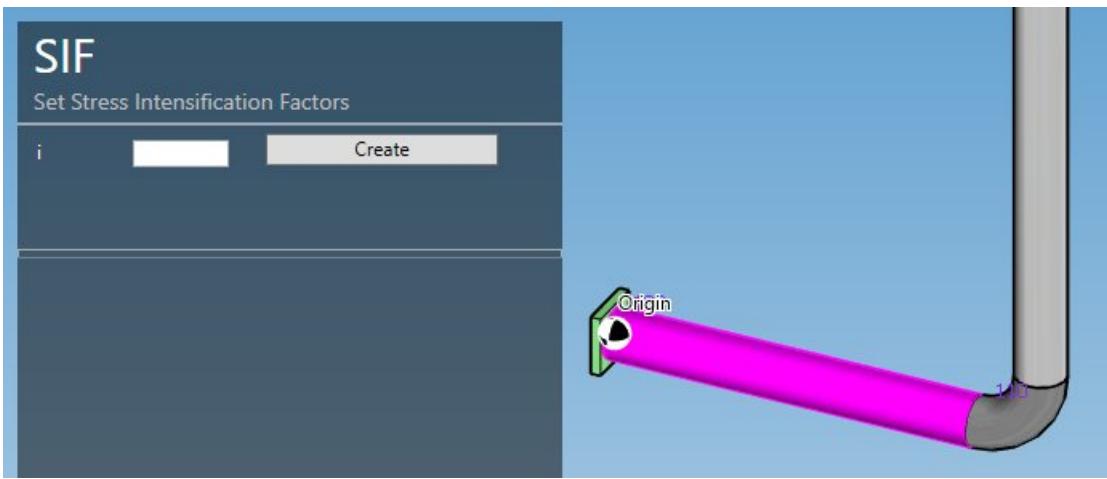
You can later modify or remove the SIFs by selecting the node and clicking on **Modify/Remove** buttons.

You can **Undo** this command.

6. Body SIF

MetaPiping lets you define **Stress Intensification Factors** and **Stress Indices** on elements.

Select an element and click on the **Body SIF** button :



Click on **Create** button to apply the SIFs to the model.

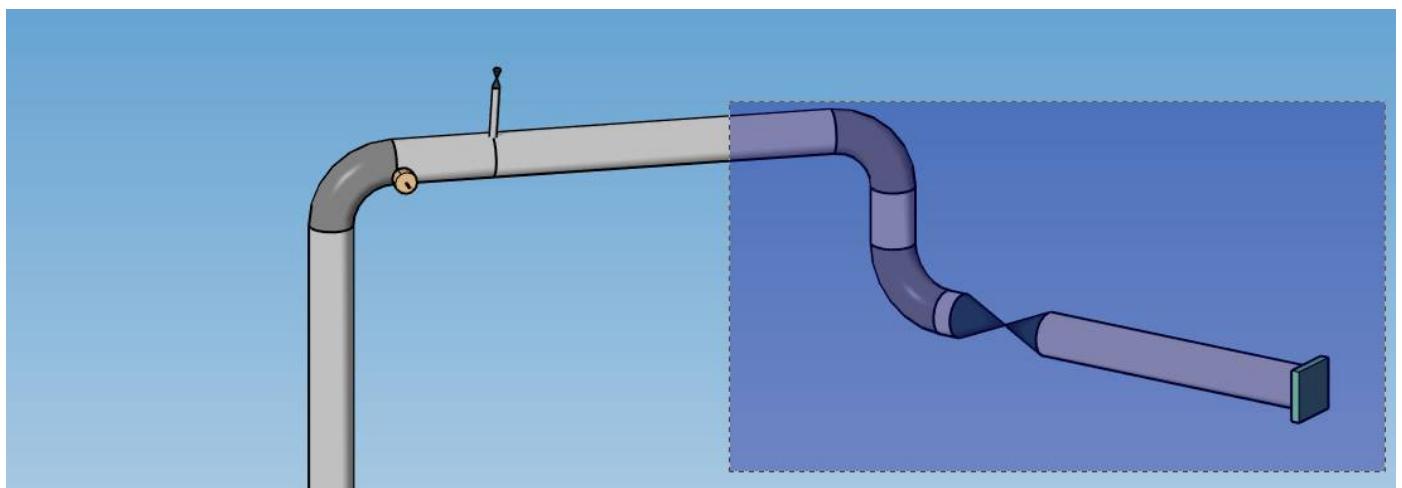
You can later modify or remove the SIFs by selecting the element and clicking on **Modify/Remove** buttons.

You can **Undo** this command.

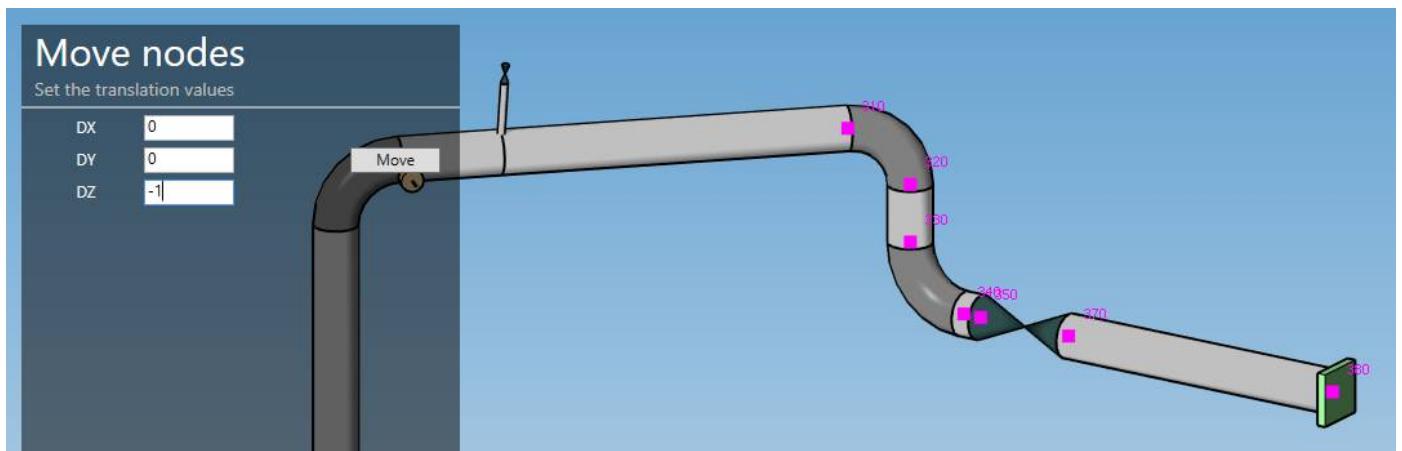
7. Move nodes

MetaPiping lets you move nodes.

Select the nodes one by one (with CTRL) or by a selection rectangle :

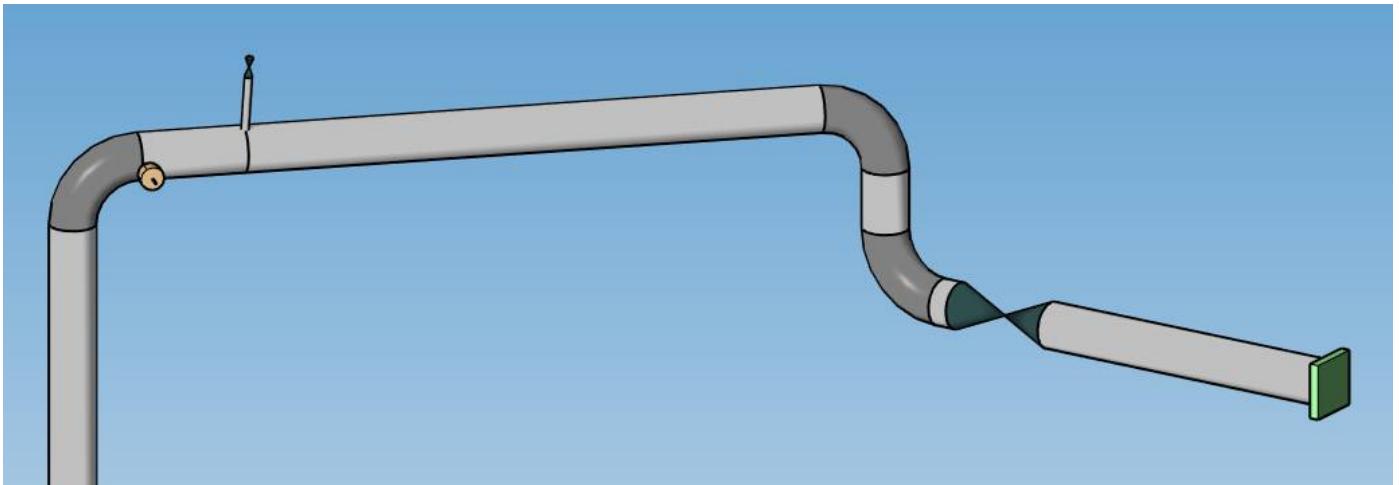


Set the offset to apply to the nodes :



Property	Unit Metric	Unit USA
DX	m	ft
DY	m	ft
DZ	m	ft

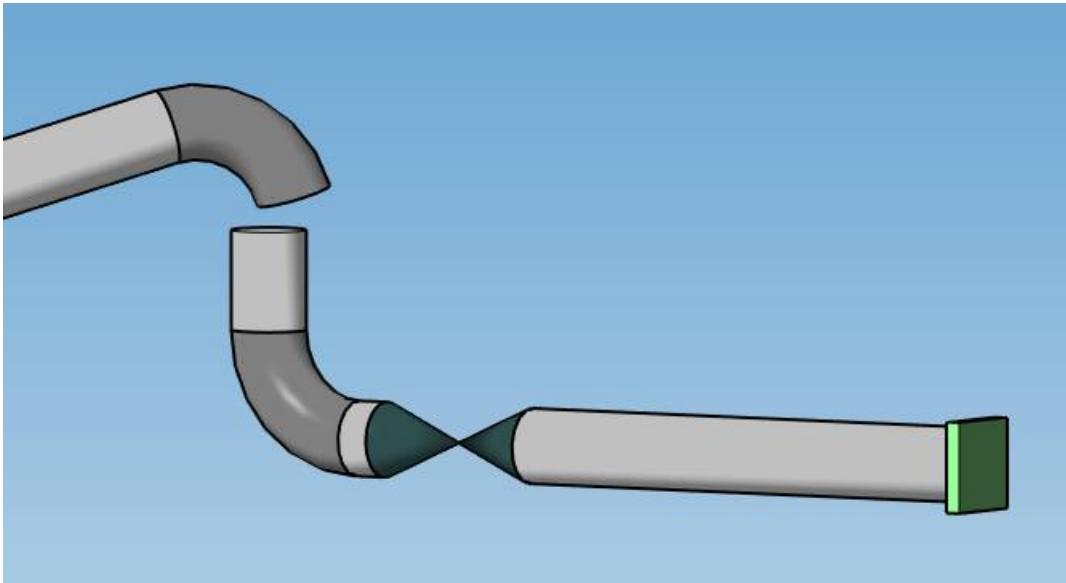
Validate the command by clicking on the **Move** button :



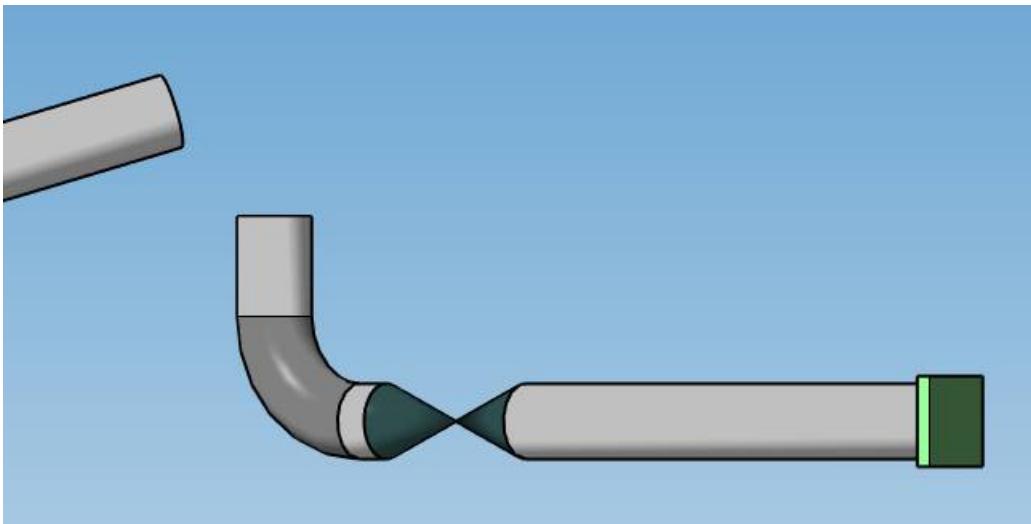
You can **Undo** this command.

WARNING

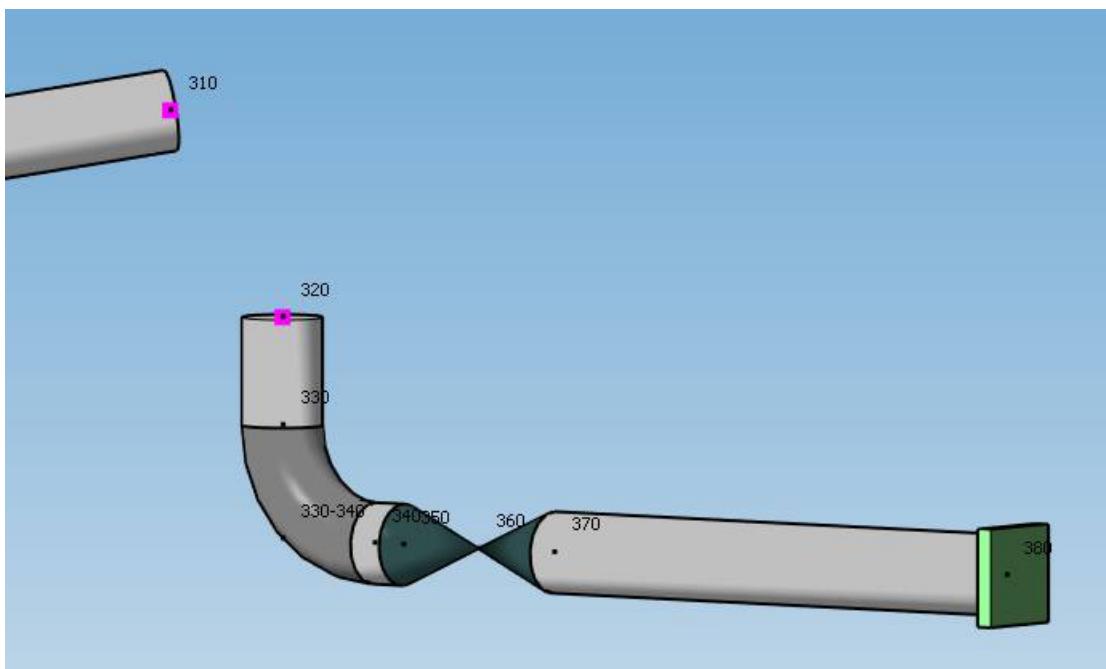
ATTENTION, sometimes this modification can create unwanted results, essentially due to bend definition no more compatible. You have to remove elements and rebuild the connection.



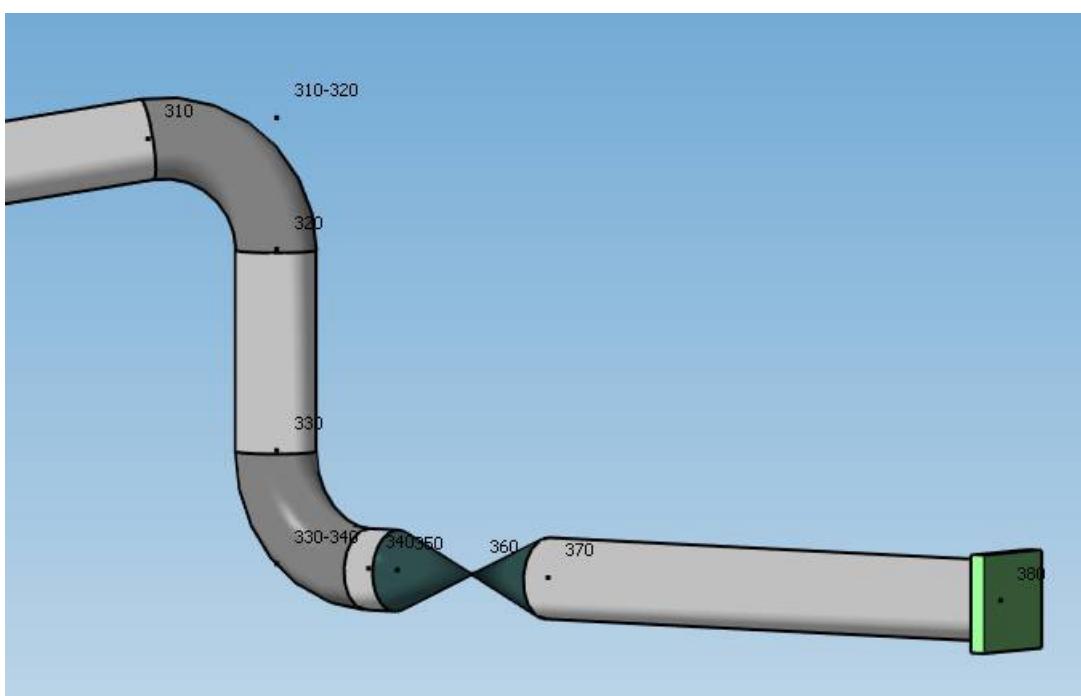
Remove bend :



Select the 2 nodes :



Click on the *Add bend* button, set the section and the radius, click on the *Create* button :



This will extend the 2 adjacent elements and place a bend with the desired radius.

8. Change length/direction

MetaPiping lets you change length of any element (no bend).

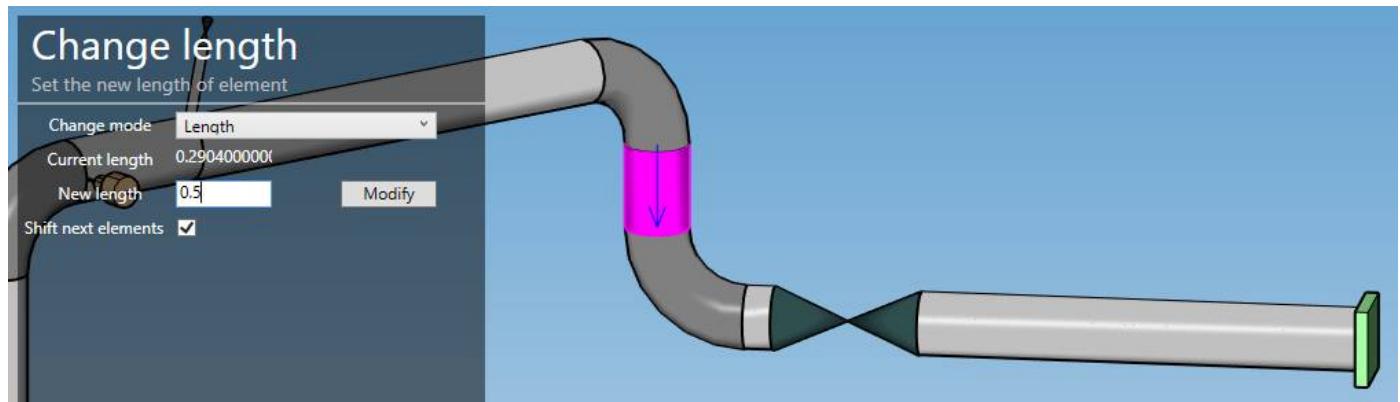
Change mode :

- Change length
- Change direction

You can select only one element.

8.1 Change length

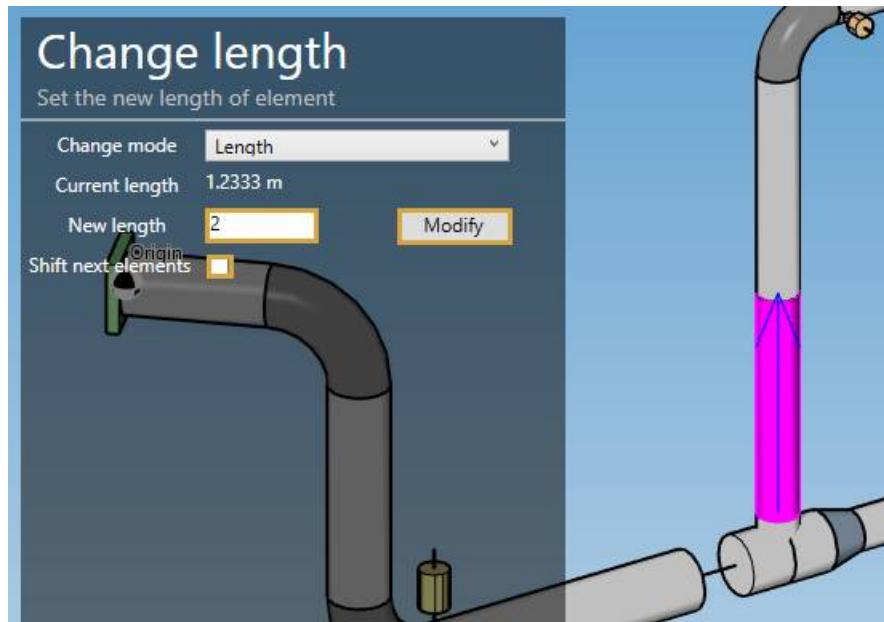
Select an element and set the new length :

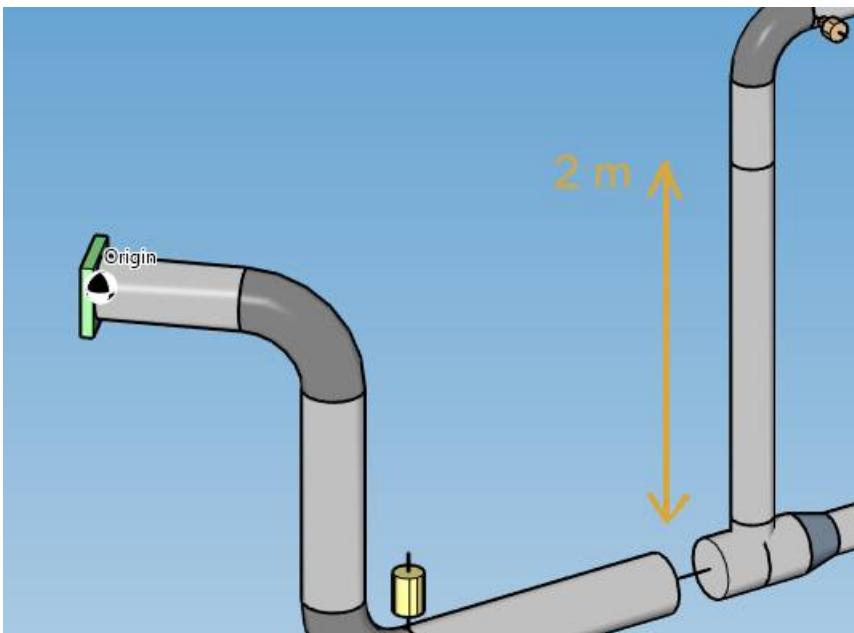


Property	Unit Metric	Unit USA
Length	m	ft

The **Shift** option offsets all following elements.

If **Shift** option is unchecked, MetaPiping will try to change the length of the selected element and modify the length of the next element.





You can **Undo** this command.

8.2 Change direction

Select an element and set the new length vector (DX, DY, DZ) :

Change length

Set the new length of element

Change mode	<input type="button" value="Direction"/>
DX	-0.5
DY	0
DZ	0
<input checked="" type="checkbox"/> Shift next elements	

Property	Unit Metric	Unit USA
DX	m	ft
DY	m	ft
DZ	m	ft

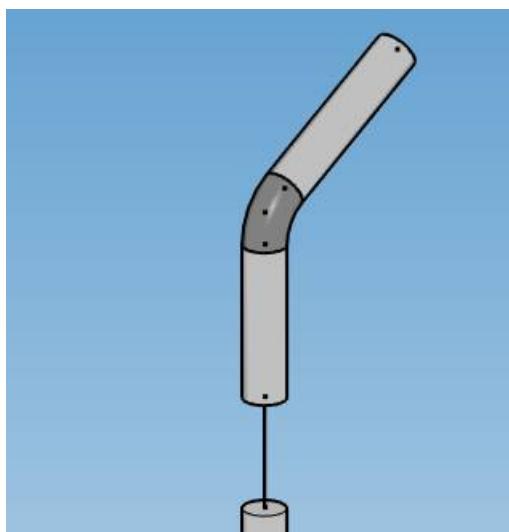
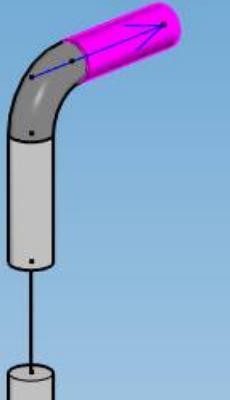
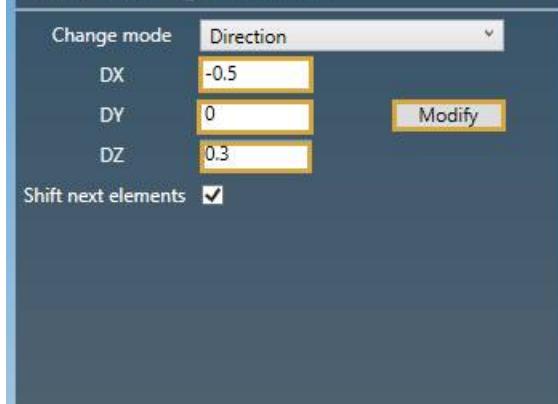
WARNING

ATTENTION, the vector (DX, DY, DZ) is calculated from intersection point to intersection point. On the last picture, the arrow starts from the intersection point on the bend and ends at the last point of the pipe.

Let's try this change of direction :

Change length

Set the new length of element



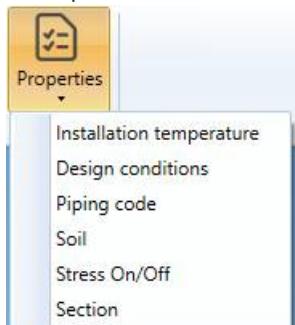
The **Shift** option offsets all following elements.

WARNING

ATTENTION, sometime this modification can create unwanted results, essentially due to bend definition no more compatible. You have to remove elements and rebuild the connection. See §7 that show an example of how to rebuild a bend.

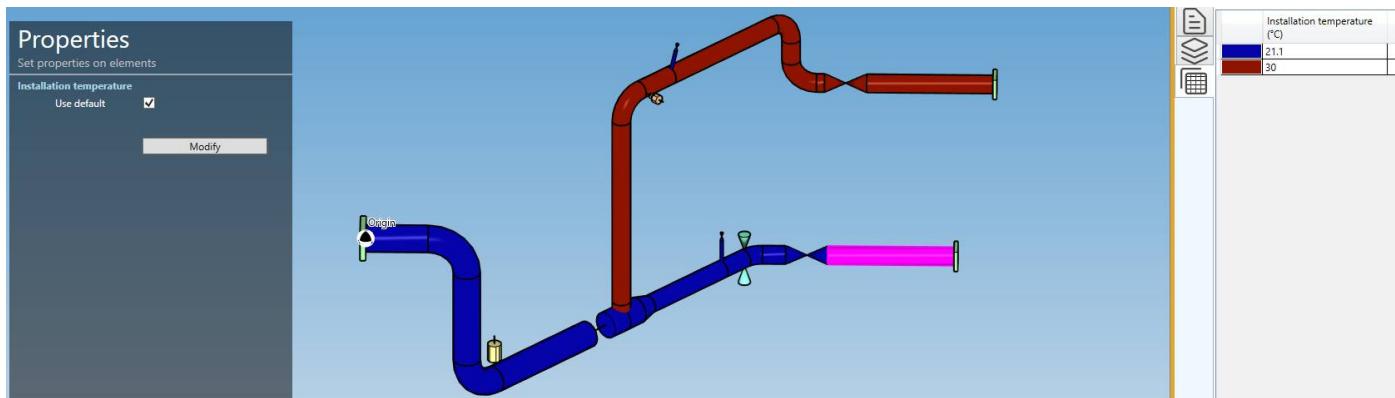
You can **Undo** this command.

9. Properties



9.1 Installation temperature

MetaPiping lets you define installation temperature of elements when different from default one :



Select multiple elements, uncheck **Use default** cell and set the new temperature.

Property	Unit Metric	Unit USA
Temperature	°C	°F

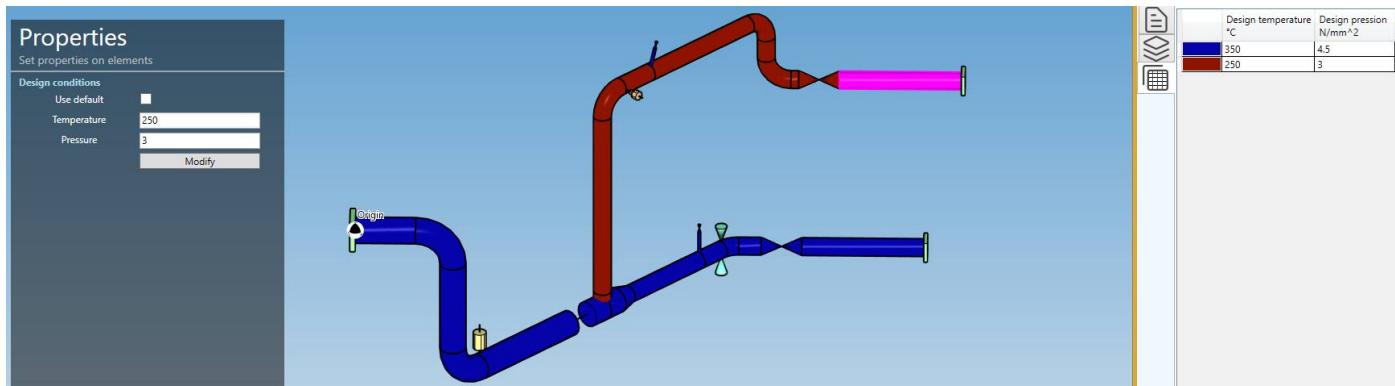
Click on Modify to validate.

You can see the elements with this property by a colored map and the legend in the right tab.

You can **Undo** this command.

9.2 Design conditions

MetaPiping lets you define the design conditions of elements in the case of different from default one :



Select multiple elements, uncheck **Use default** cell and set the temperature and pressure.

Property	Unit Metric	Unit USA
Temperature	°C	°F
Pressure	N/mm ²	psi

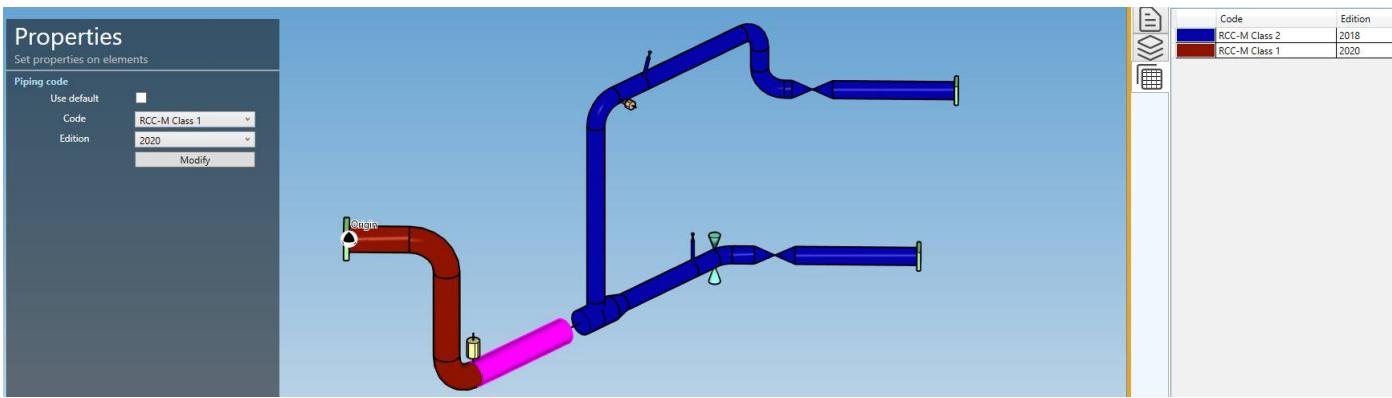
Click on Modify to validate.

You can see the elements with this property by a colored map and the legend in the right tab.

You can **Undo** this command.

9.3 Piping code

MetaPiping lets you define the piping code of elements when different from default one :



Select multiple elements, uncheck **Use default** cell and set the new code and edition.

Click on Modify to validate.

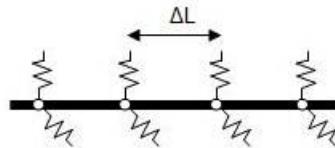
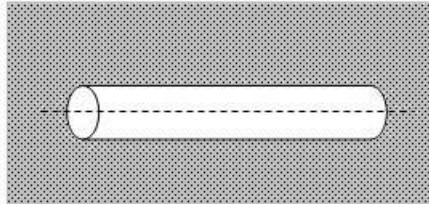
You can see the elements with this property by a colored map and the legend in the right tab.

You can **Undo** this command.

9.4 Soil

In **Buried piping**, soils must be defined for all elements.

The soil is modeled as a Winkler foundation. Discrete springs are placed at regular intervals ΔL along the piping in the transverse and axial directions.



Select multiple elements and choose **New soil**:



Give it a name and fill the soil properties :

Properties

Set properties on elements

Soil definition

Soil	Sand	<input type="button" value="Apply"/>
------	------	--------------------------------------

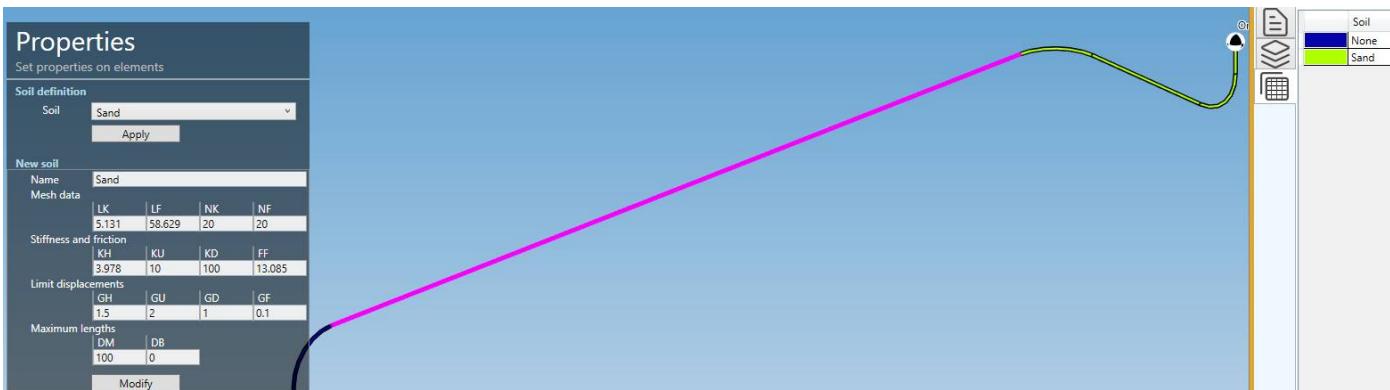
New soil

Name	Sand		
Mesh data			
LK	LF	NK	NF
5.131	58.629	20	20
Stiffness and friction			
KH	KU	KD	FF
3.978	10	100	13.085
Limit displacements			
GH	GU	GD	GF
1.5	2	1	0.1
Maximum lengths			
DM	DB		
100	0		

Property	Description	Unit Metric	Unit USA
LK	Influence length	m	in
LF	Slippage length	m	in
NK	Number of transverse springs along the influence length	-	-
NF	Number of axial springs along the slippage length	-	-
KH	Horizontal modulus of subgrade reaction	N/mm ²	ksi
KU	Vertical upwards modulus of subgrade reaction	N/mm ²	ksi
KD	Vertical downwards modulus of subgrade reaction	N/mm ²	ksi
FF	Maximum unit friction force at the pipe/soil interface	kN/m	kips/ft
GH	Pipe displacement at maximum horizontal resistance	mm	in
GU	Pipe displacement at maximum upwards resistance	mm	in
GD	Pipe displacement at maximum downwards resistance	mm	in
GF	Axial pipe displacement at maximum friction force	mm	in
DM	Maximum pipe length	m	ft
DB	Maximum bend length	m	ft

Click on Modify to validate.

You can see the elements with this property by a colored map and the legend in the right tab.

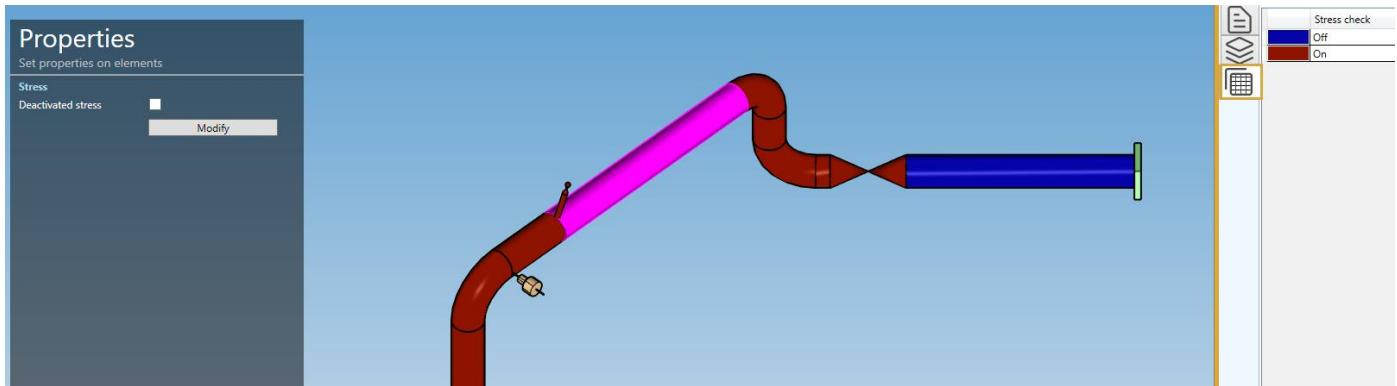


You can **Undo** this command.

9.5 Stress On/Off

MetaPiping lets you deactivate the stress report of a portion of the piping system.

Select multiple elements and click on the **Deactivated stress** button :



Click on the **Modify** button to deactivate the stress reporting of these elements.

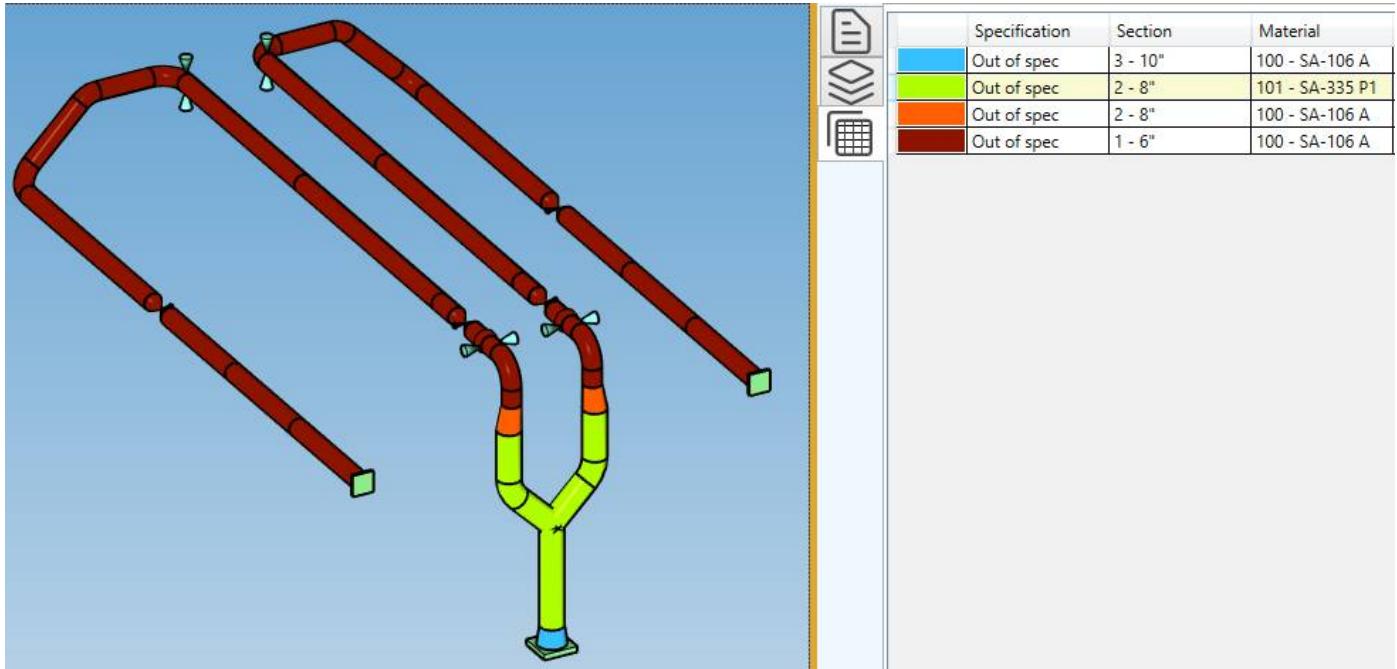
You can later modify this property by selecting the elements and activating or deactivating the checkbox.

You can see the elements with this property by a colored map and the legend in the right tab.

You can **Undo** this command.

9.6 Section

As in review, The **Section** property shows all sections used in the model with a colored map :



MetaPiping let you change the section of multiple elements in one operation.

Select multiple elements, a specification and a section, and click on the **Modify** button :

Properties

Set properties on elements

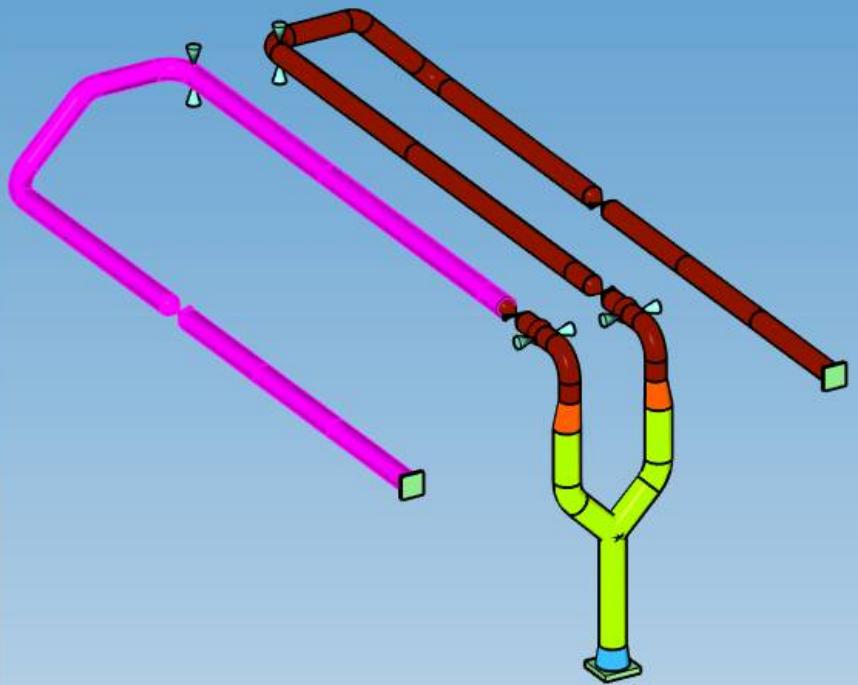
Section and material

Specification Out of spec

Section 1 - 6"

Material 100 - SA-106 A

Modify



You can **Undo** this command.

Selection

MetaPiping lets you select one or multiple objects of the same type.

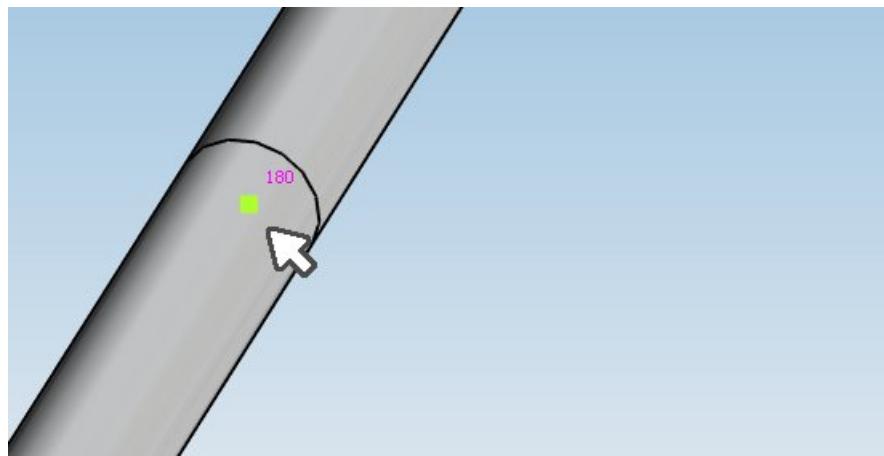
Click on the **Select** button to open the **Selection tool** :



1. Manual selection

POINT :

Click on the **Point** button to select **Nodes** or **element extremities**.

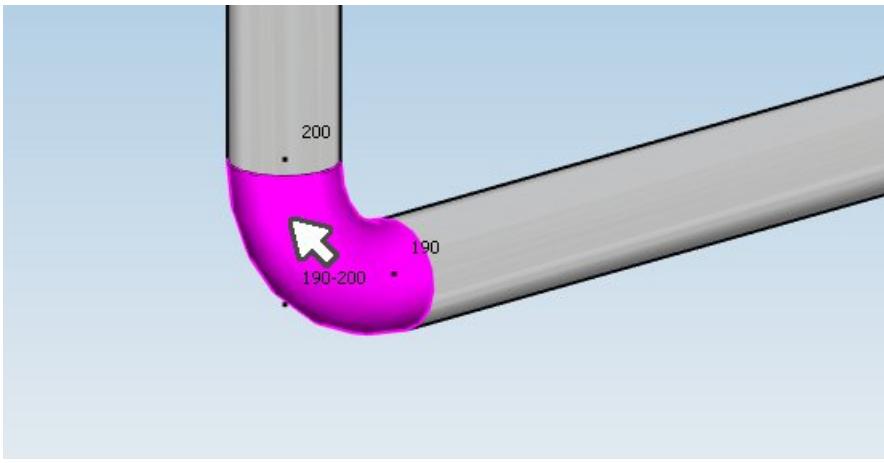


The node is highlighted and its **Name** appears in magenta.

You can also select a node by entering the node name (1) and click to the **Point** button.

ELEMENT :

Click on the **Element** button to select an **element**.



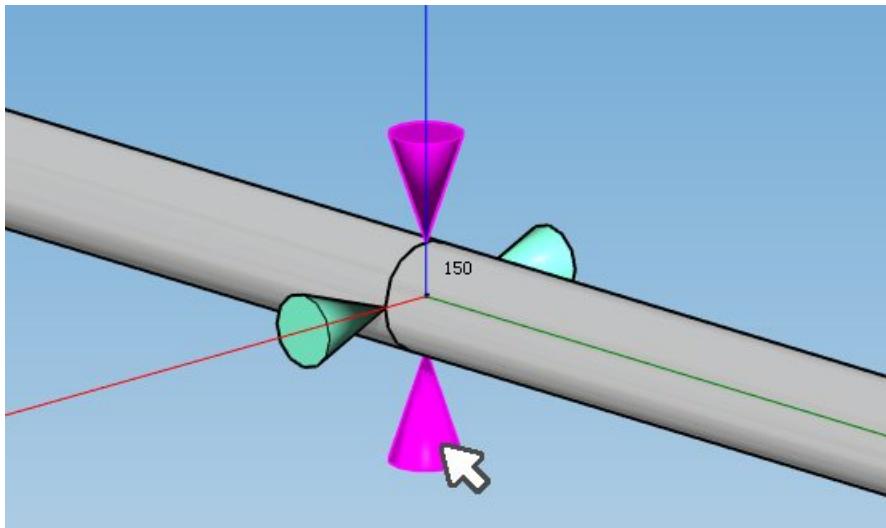
The element is highlighted in magenta.

You can also select an element by entering the two node names separate by “-” (1) and click to the **Element** button.

You can also select a valve by entering its label (1) and click to the **Element** button.

RESTRAINT :

Click on the **Restraint** button to select a restraint.

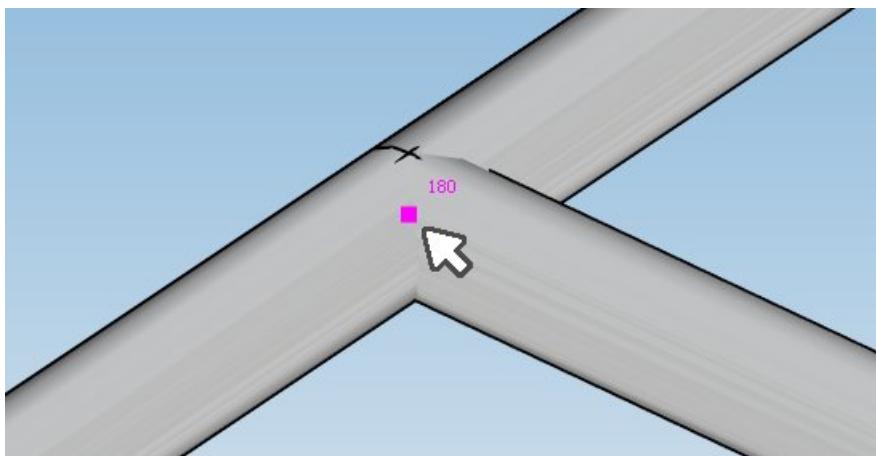


The restraint is highlighted in magenta.

You can also select a restraint by entering the node name or its label (1) and click to the **Restraint** button.

TEE :

Click on the **Tee** button to select a tee.



The node containing the tee is highlighted and its **Name** appears in magenta.

CLEAR SELECTION :

Clicking in the 3D view where no objects are present without **CTRL key** pressed will clear the selection.

CANCEL SELECTION :

Just press **ESC** on keyboard to hide the **selection tool**.

2. Smart selection

Not yet implemented

3. Selection type

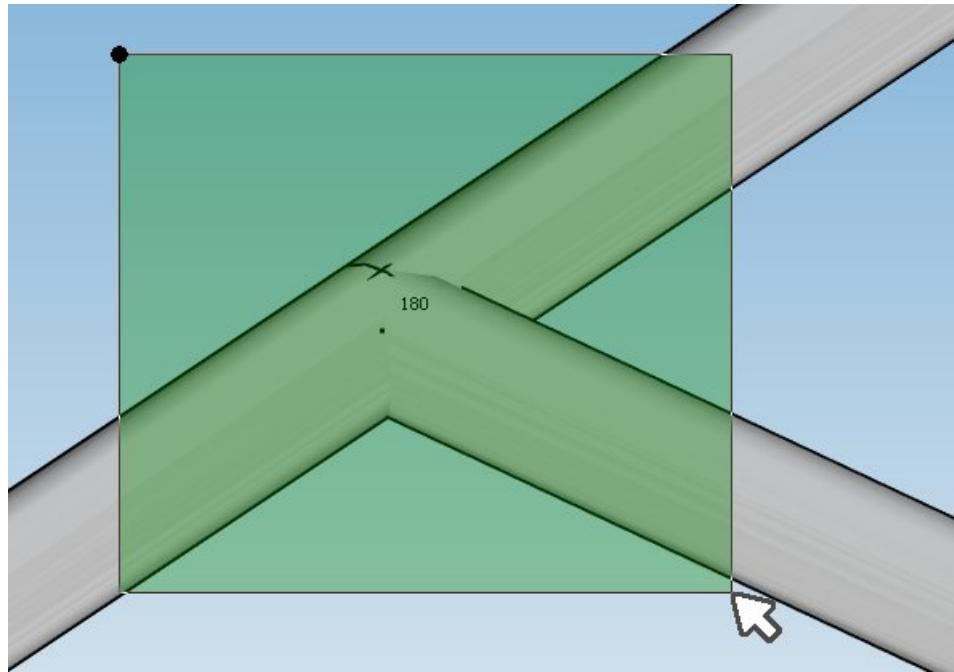
MOUSE CLICK :

To select **ONE** object, just make a click with the **left mouse button**:



RECTANGLE SELECTION :

It is also possible to make a rectangular selection :



1 Press the **left mouse button**

2 Move the mouse without releasing the left mouse button

3 Release the left mouse button

Every object that intersects the **rectangle** will be selected (in magenta).

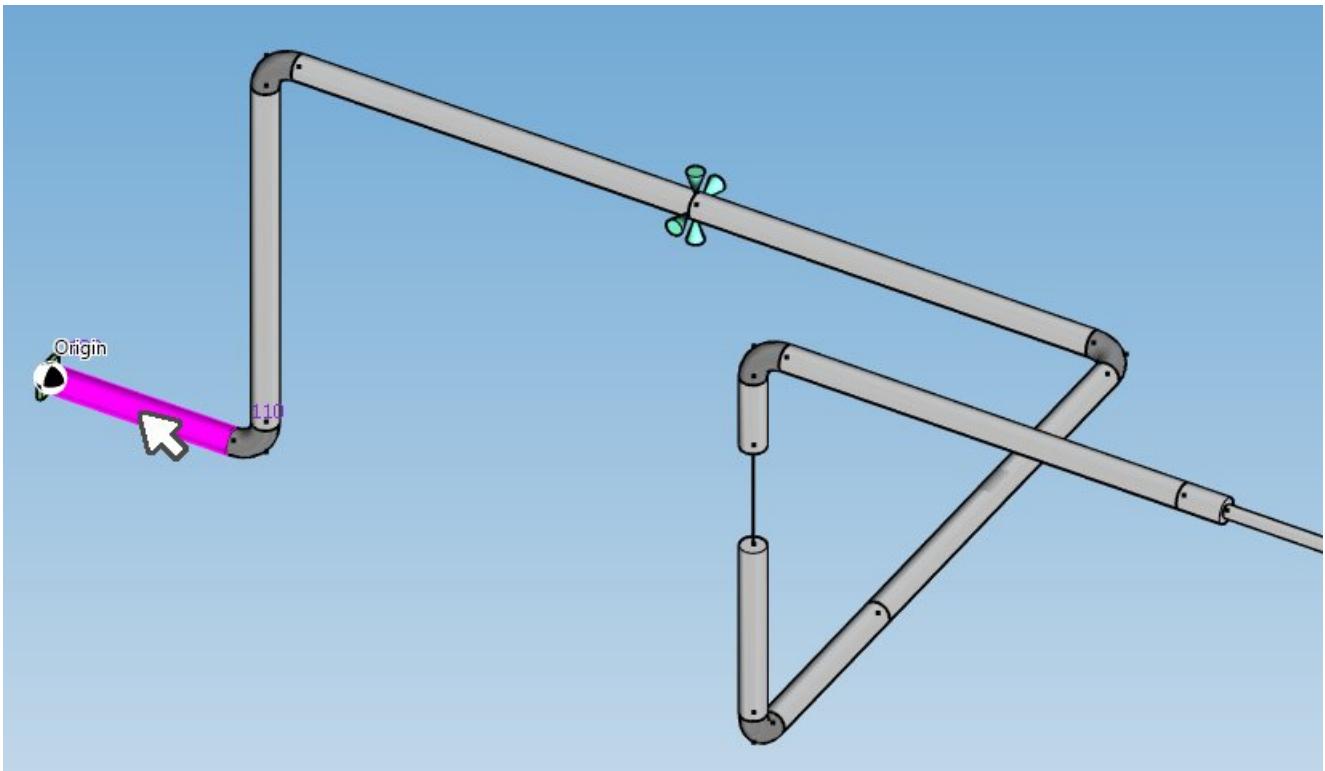
MULTIPLE SELECTION :

You can make multiple selection by pressing the **CTRL** key on keyboard.

MULTIPLE SELECTION BY CONNECTION :

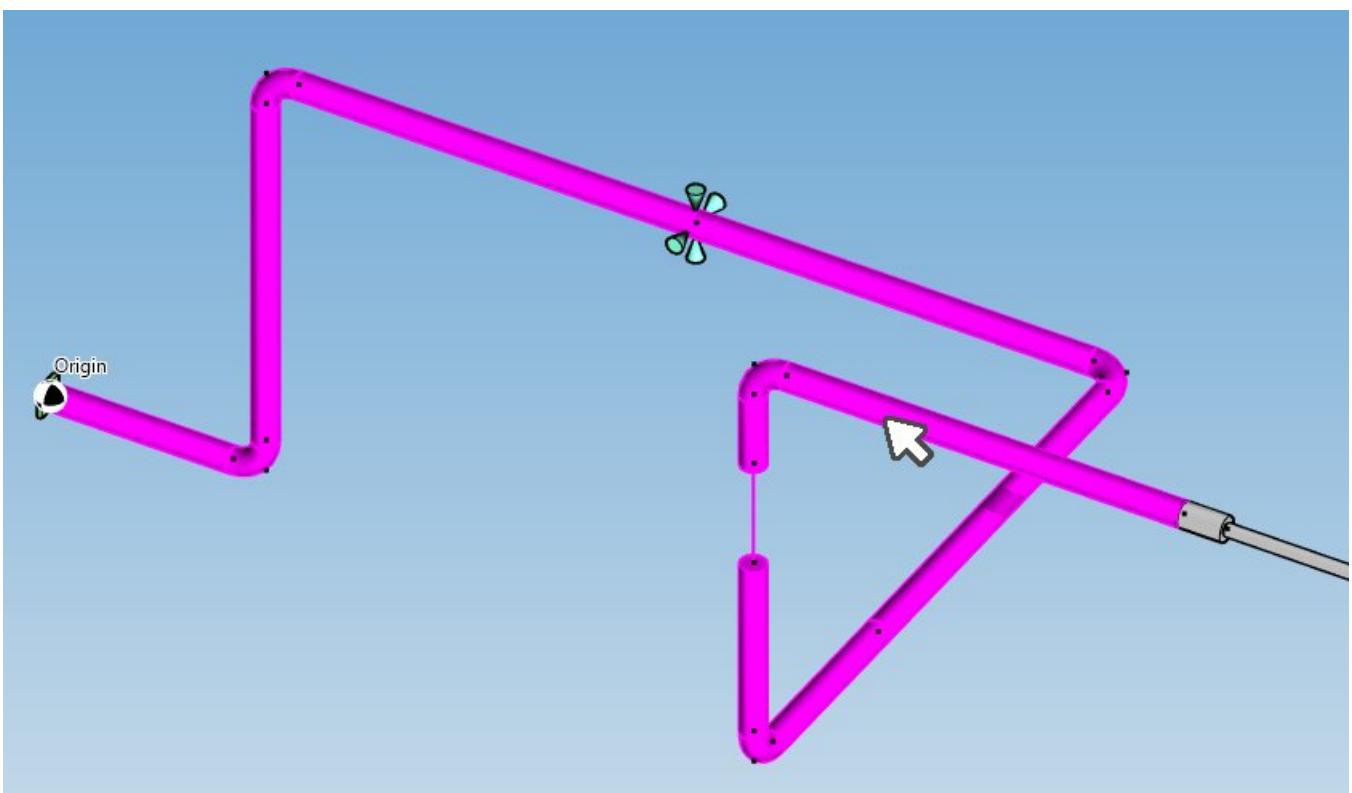
You can make continuous selection by pressing the **SHIFT** key on keyboard and 2 distant **elements**.

1 Select a first element :



1 Press **SHIFT**

2 Select a second element :



Every object between the 2 elements will be selected (in magenta).



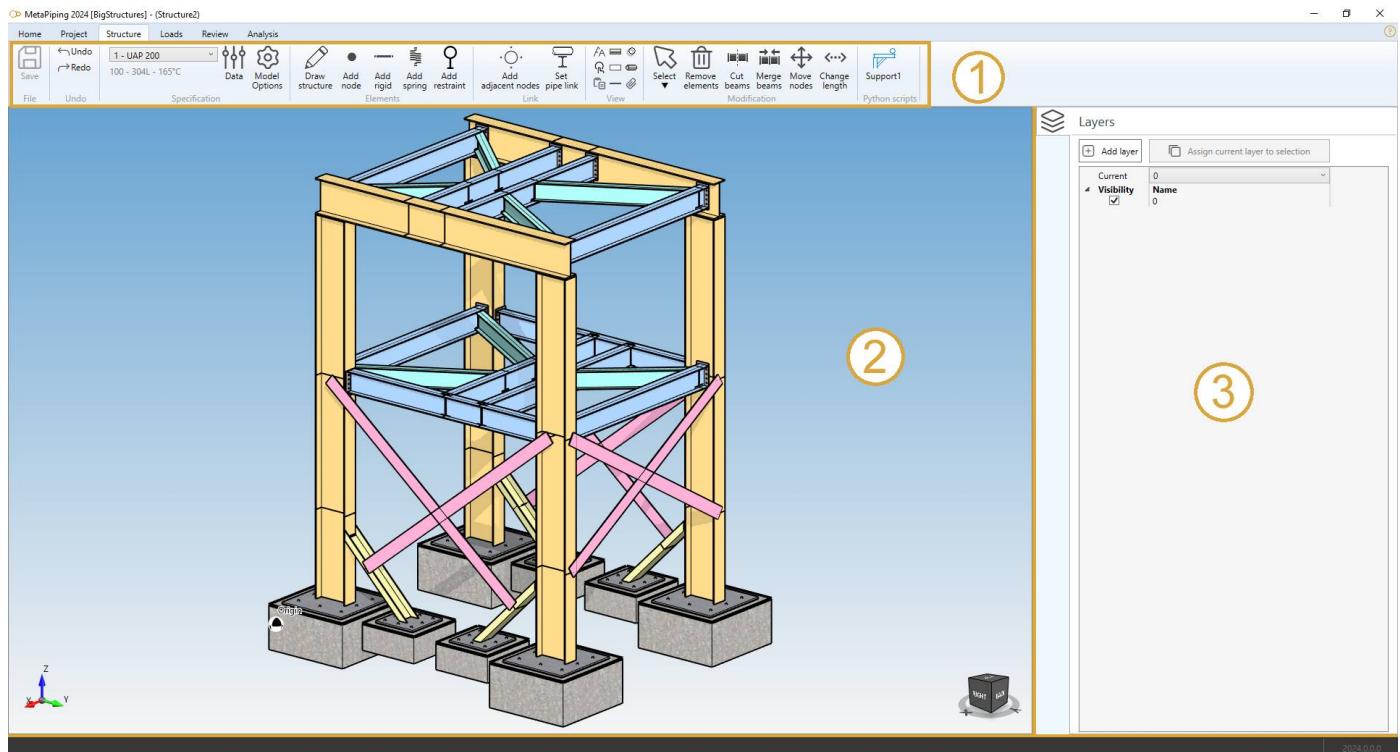
Structure

After creating a new **structure study** and pressing the **Edit** button, a window appear to define the calculation code :



Set the **Code** and **Edition** and press **OK** : new tabs appear in the ribbon menu on top of the application :

- Structure
- Load - click [here](#) to have more information about loading
- Review - click [here](#) to have more information about review tools
- Analysis - click [here](#) to have more information about analysis



The **Structure** screen consists of a ribbon menu (1), a 3D visualization engine (2) and a data panel on the right (3).

The **Data Panel** can be opened/closed with the shortcut **F2**

1. Ribbon menu



1.1 File

During the design of the model, every command is automatically saved in a temporary file (conception.~metal).

You can decide to save the last modifications or cancel it and return to the study screen.

A floppy disk on the left side of the ribbon indicates the state of the file on disk :



Save means that everything has been saved.



Save means that something has changed and the MetaL need to be saved.

WARNING

If the application crashes, MetaPiping will prompt to reopen the last modifications

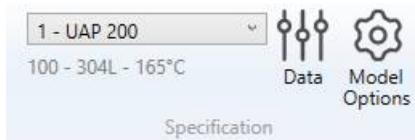
1.2 Undo/Redo



Every command in MetaStructure is stored in a **command list**.

You can *navigate* through this list by pressing the **Undo/Redo** buttons.

1.3 Specification



This group of controls let you define the current **Section/Material** for the next beams.

You have an access to a window that defines the beam sections and materials for the current model.

You have an access to the model options (code, edition, calculation options...).

Click [here](#) to have more information about the **Data** button and the **Model options** button.

1.4 Elements & Link



Click [here](#) to have more information about element creation and link between piping nodes and structure nodes.

1.5 View options



show/hide the node name

show/hide the node point

copy the 3D view to the clipboard

show the structure in shaded mode

show the structure in hidden lines mode

show the structure in wireframe mode

show/hide the shadows

show/hide the origin symbol

 show/hide the linked study models

 show/hide the concrete

1.6 Modification tools



Click [here](#) to have more information about all modification tools.

1.7 Python scripts

MetaStructure lets you create your own **tools**.

If a tool is missing, user can create his own **command** that can interact with the user and modify the selected elements or the whole model.

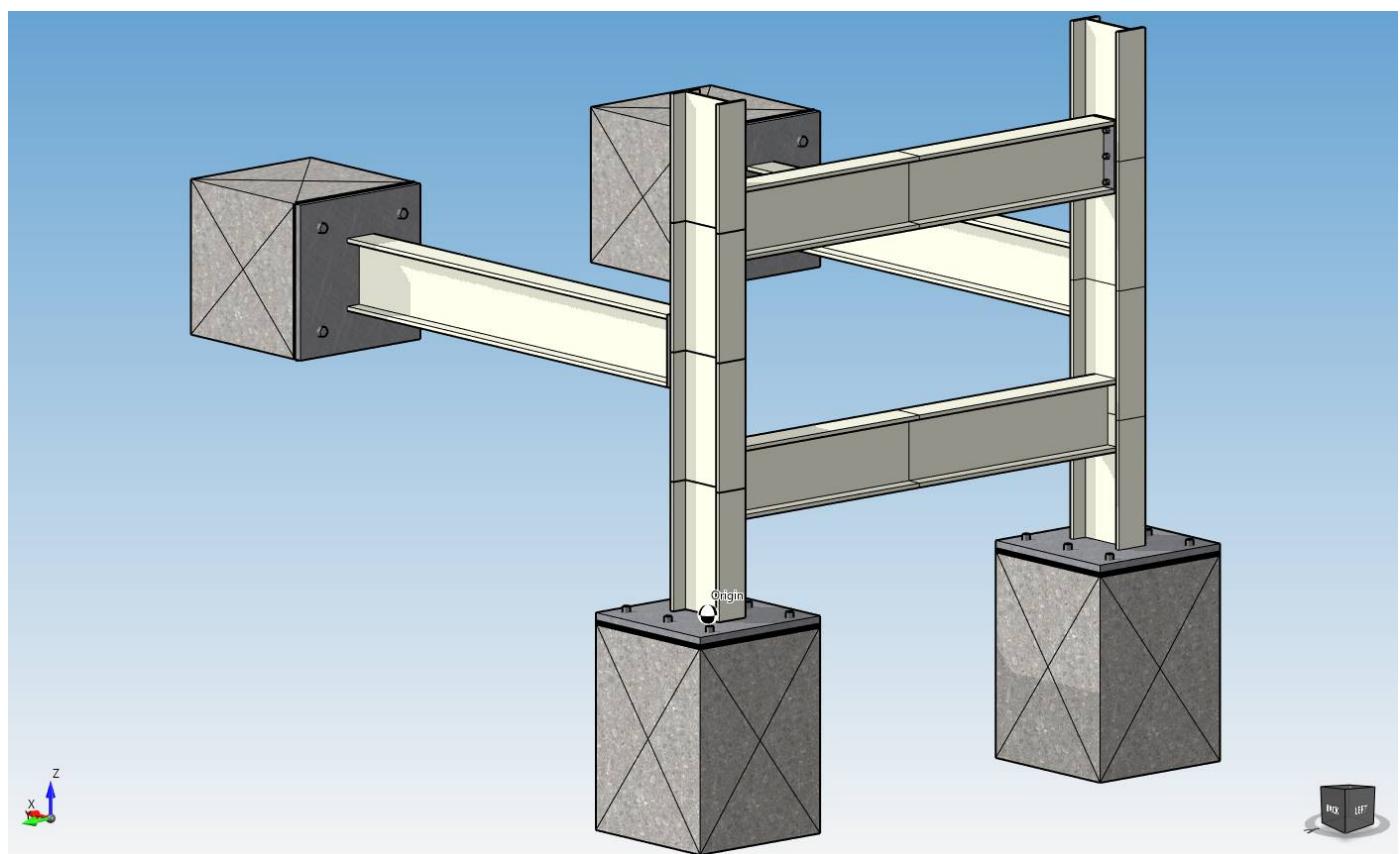
MetaStructure will take care of the Undo/Redo mechanism.

Click [here](#) to have more information about python script creation.

[See an example of a Design script](#) : Create a beam support of a piping on a node

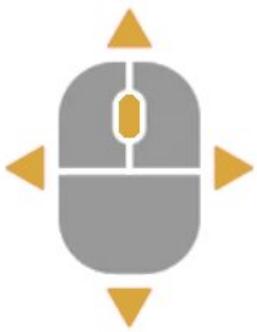
2. 3D Visualization

MetaStructure is equipped with a powerful 3D engine :



2.1 Navigation

You navigate inside the view with the **middle button** of the mouse pressed :



Hold down the **CTRL key** during navigation to translate the view.

You zoom in by scrolling the mouse wheel up :



You zoom out by scrolling the mouse wheel down :

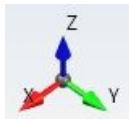


You zoom all by **double click** the mouse wheel button :



2.2 Axis

The **global axis** are shown in the bottom left corner of the screen :



2.3 Predefined views

An interactive **cube** with predefined views is shown in the bottom right corner of the screen :



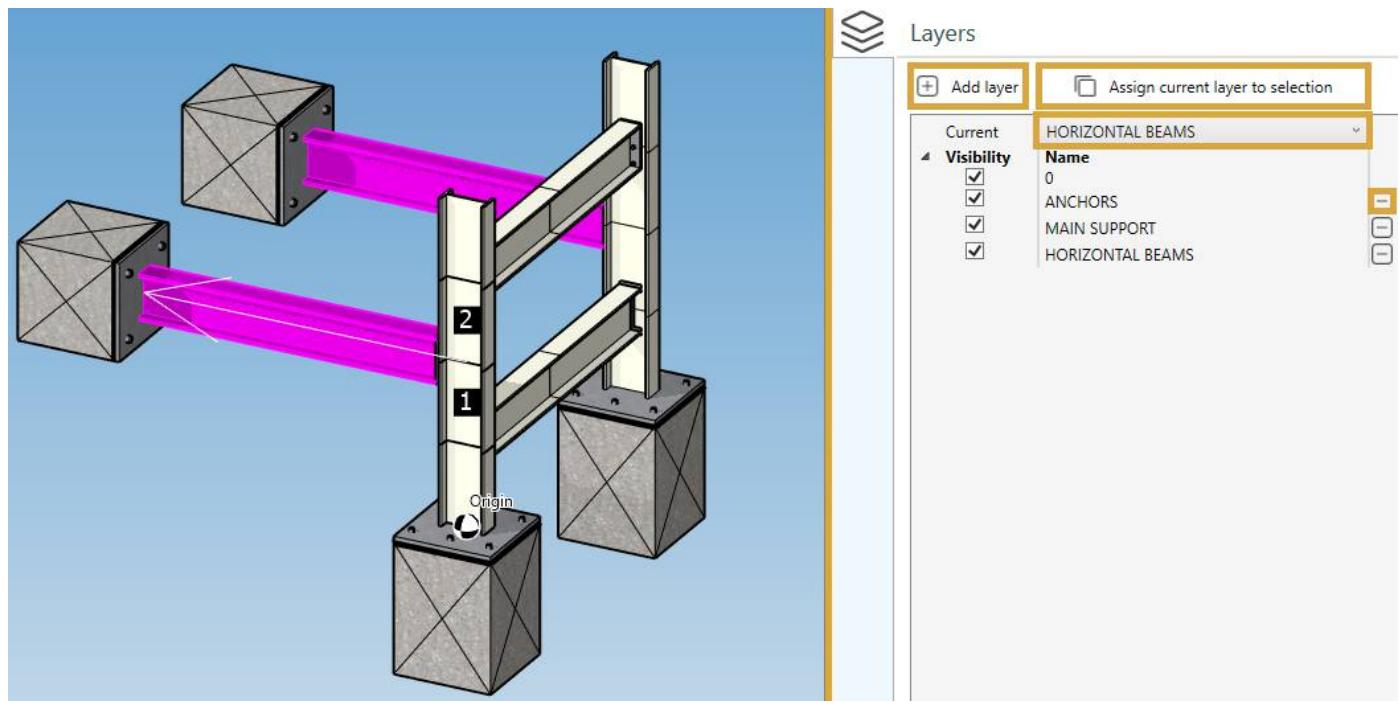
- Click on top corners of the cube to change to predefined axonometric view.
- Click twice on the faces of the cube to change to 2D views.

3. Data panel

The **Data Panel** can be opened/closed with the shortcut **F2**

3.1 Layers

MetaStructure lets you simplify your visualization by using the **Layer** concept :



By default, all elements are assigned to **Layer 0**.

To create a new layer, click on **Add layer** button.

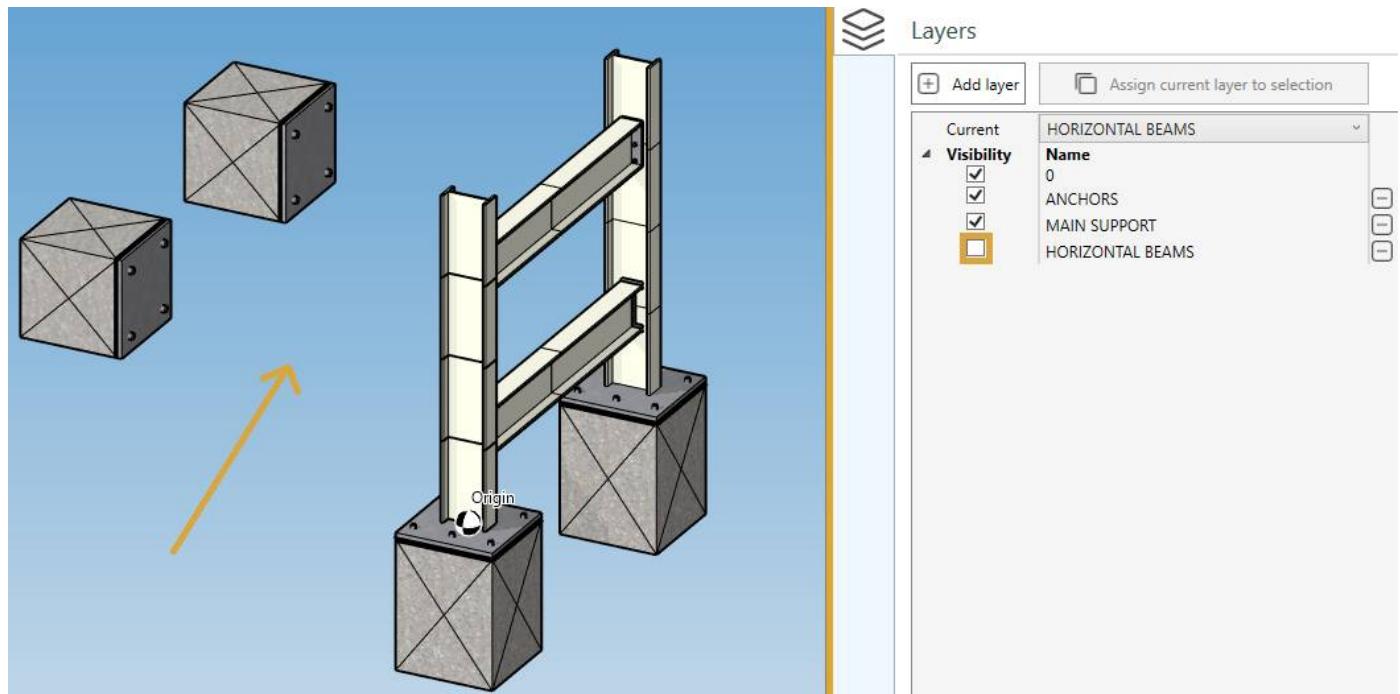
Give it a name, for example "HORIZONTAL BEAMS".

The layer "HORIZONTAL BEAMS" will appear in the list of layers and will become the current layer.

To modify the layer of objects, select elements and/or restraints and click on **Assign current layer to selection** button.

If you decide to remove a layer, click on - button next to the layer name. All elements and restraints of this layer will be assigned to the default **Layer 0**.

You can now show/hide elements and restraints by activating/deactivating the visibility buttons :



All layer manipulations support Undo/Redo mechanism.

TABLE OF CONTENTS

- [Specification](#)
- [Elements](#)
- [Modification](#)
- [Selection](#)



Specification



The specification tool lets you define the current beam section and material.

You can define one or more **specification** by clicking the **Data** button.

The goal of a specification is to predefine the possible sections for the model with associated materials, descriptions and colors.

You have also an access to the model options (code, edition, calculation options...) by clicking the **Model options** button.

TABLE OF CONTENTS

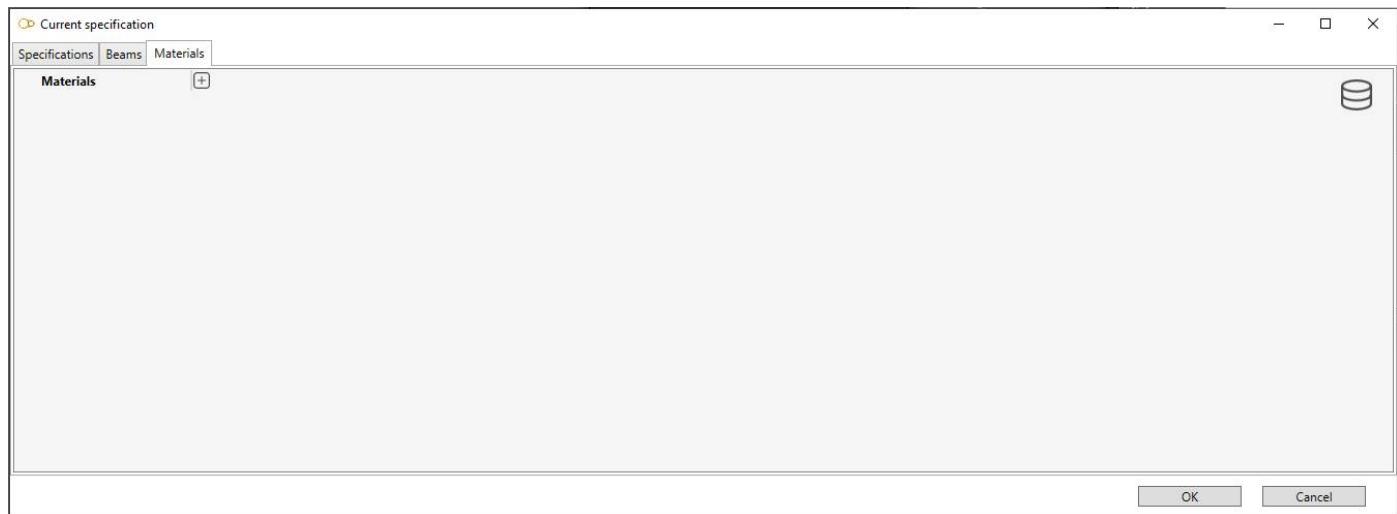
- [Data](#)
- [Options](#)

Data

The **Data** window lets you create all materials and all sections of the model.

1. New project

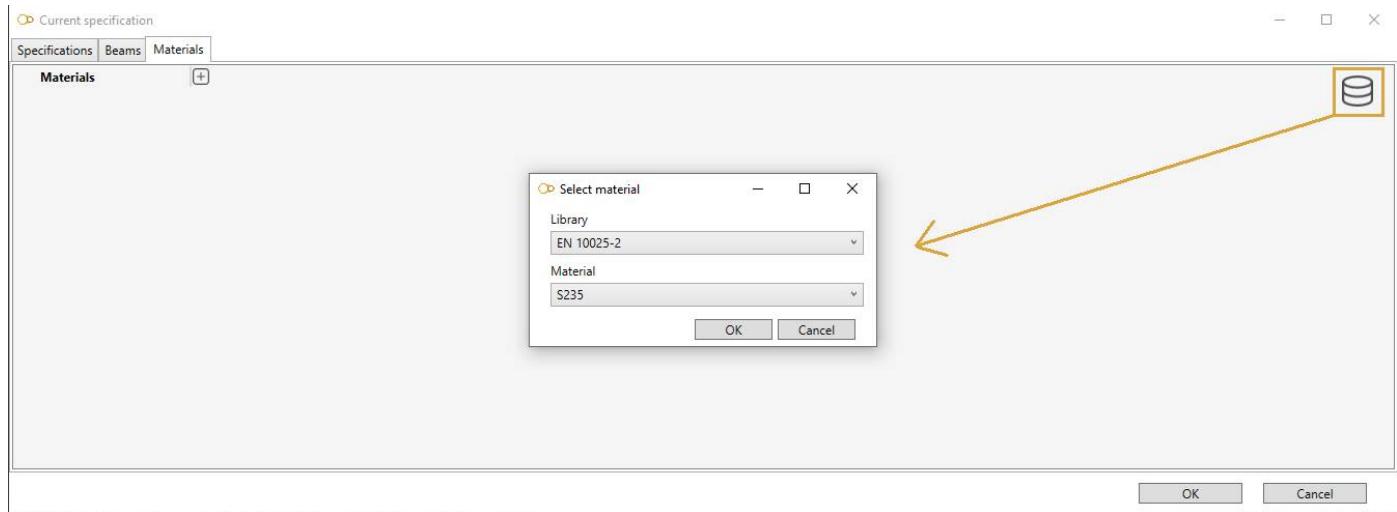
When starting a new model, the **Data** window opens :



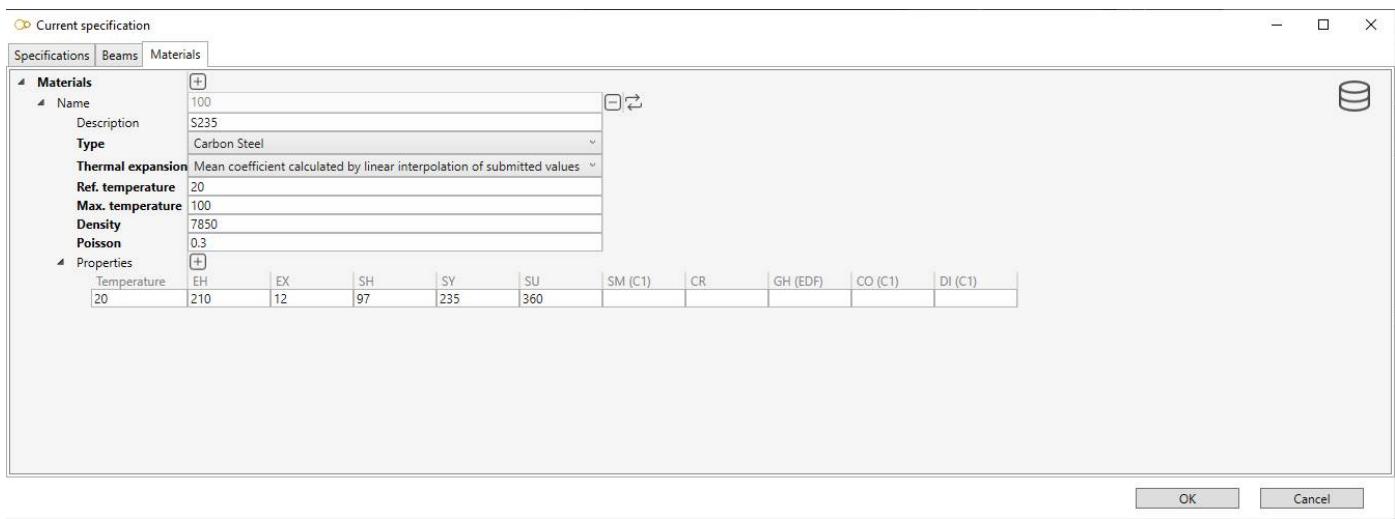
1.1 Material definition

Let's define a new material.

We recommend to select a material from **database** :



Click on the **Database** button, select a **Library** and a **Material**. The tables will be filled automatically :

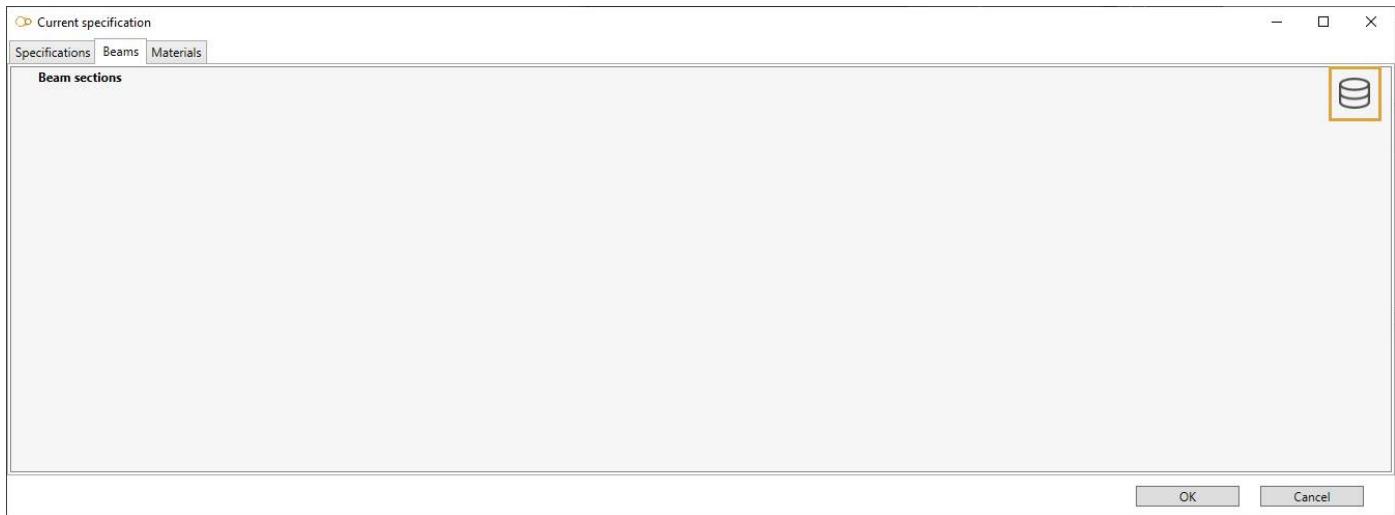


Only library with **Target = Structure** will be shown !

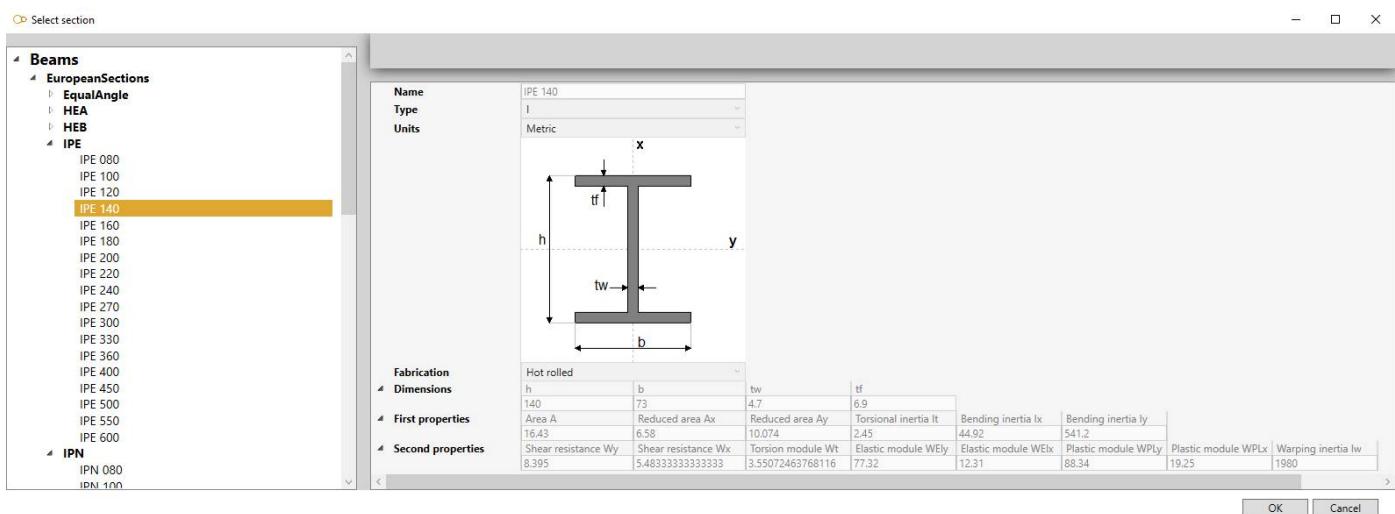
Click [here](#) for more information about creation of library of materials.

1.2 Beam definition

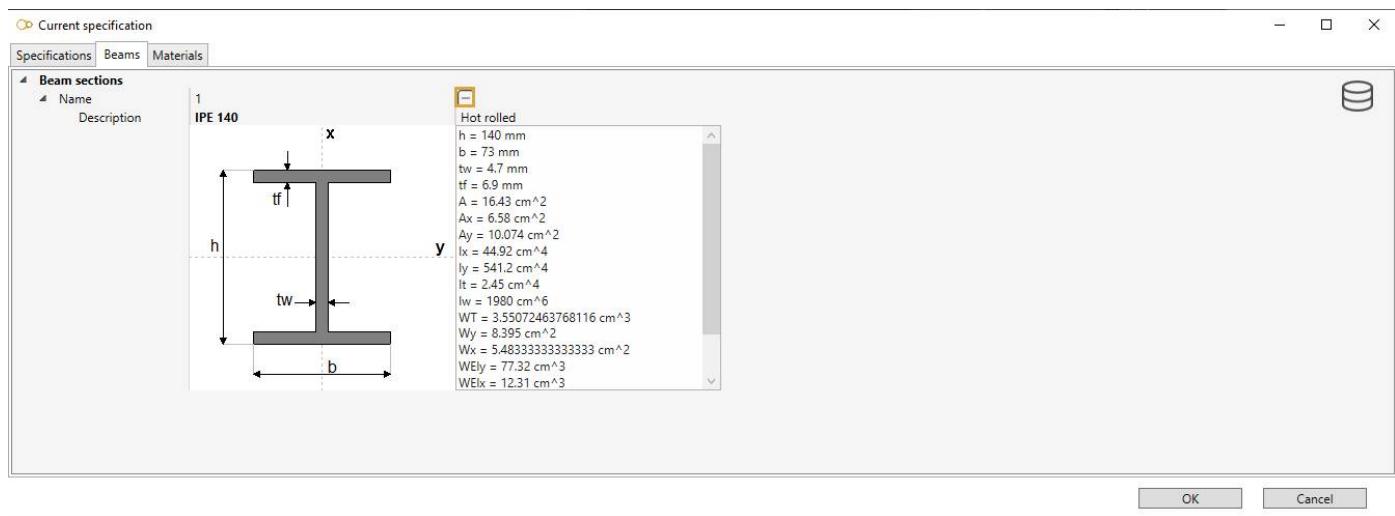
Let's add a new beam section. Click on the **Database** button :



A window with all section appears. Select for example a **IPE 140** and press **OK**.



The **IPE 140** appears in the list with an image and all properties.

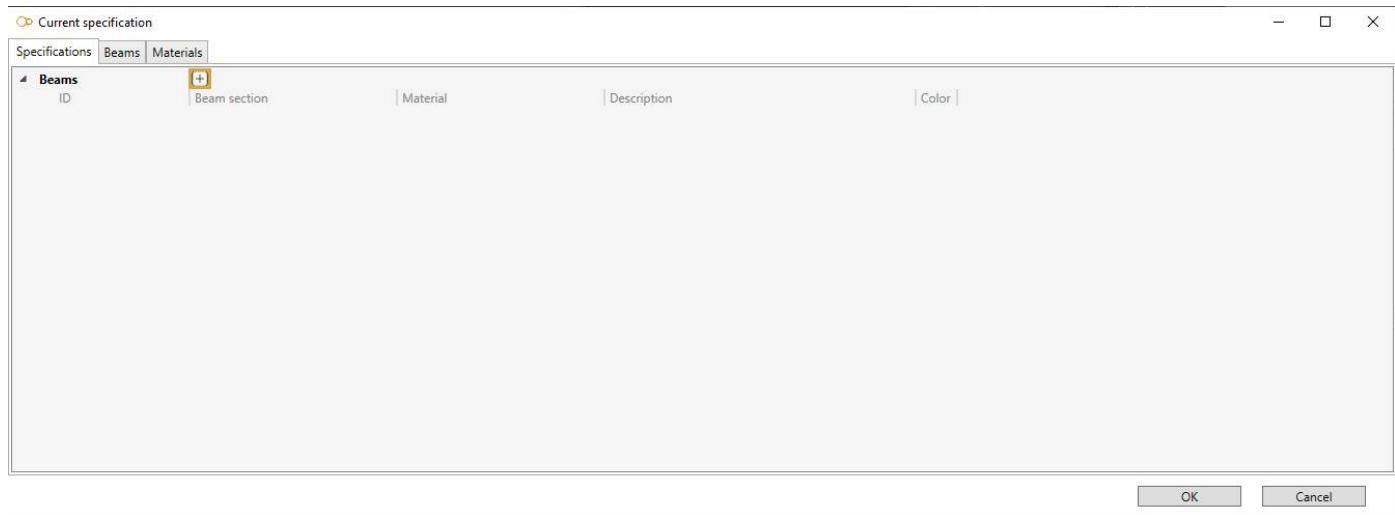


You can remove this section by clicking on the - button.

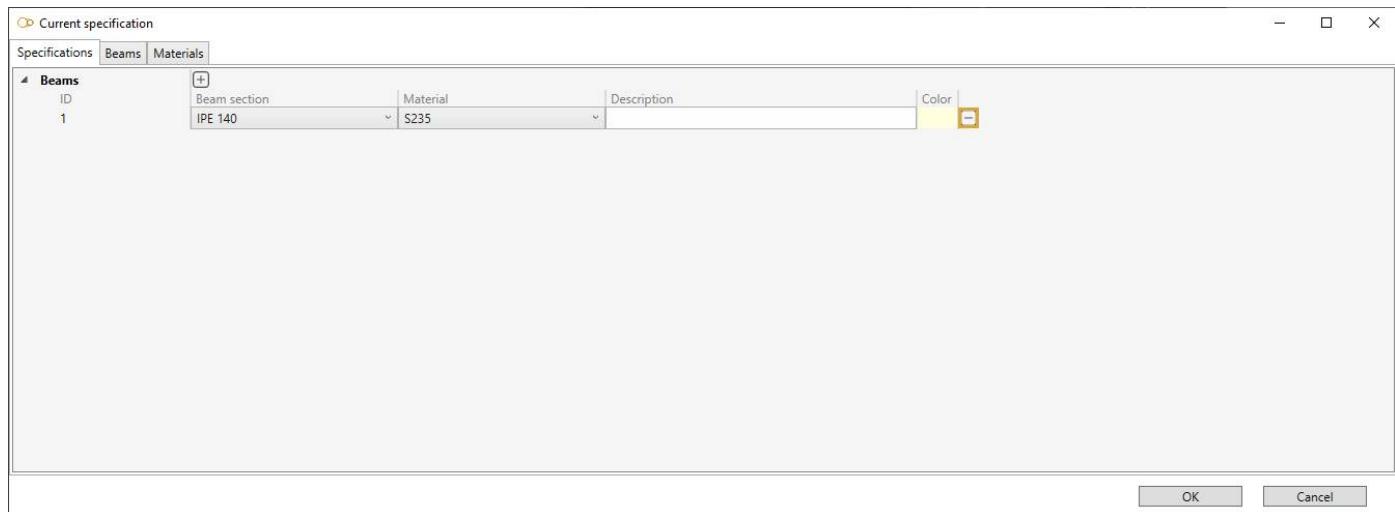
Click [here](#) for more information about creation of beam sections.

1.3 Specification

Finally, you can define the **Specification sections** of the model by defining pairs of Section/Material + description and color.



Click on + button, select the section, define a description (optional) and a color :



You can remove this specification by clicking on the - button.

Click on **OK** to validate the window.

You can now select this specification in the ribbon menu :

1 - IPE 140

100 - S235



Data



Model
Options

Specification

Model options

1. General

Model options

General		Analysis		Modal extraction	
Calculation code :	RCC-M	Anchor code :	EN-1992-4		
Calculation edition :	June 2007	Anchor edition :	2018		
Gravity :	<input type="radio"/> -Y <input checked="" type="radio"/> -Z				
Input units :	<input checked="" type="radio"/> Metric <input type="radio"/> USA <input type="radio"/> USA2				
Output units :	<input checked="" type="radio"/> Metric <input type="radio"/> USA <input type="radio"/> USA2				
Plant :					
Engineer :					
<input type="button" value="OK"/> <input type="button" value="Cancel"/>					

This tab let you define the **Calculation code** and its **Edition**, the direction of the **gravity**, the input and output **Units**, the **Plant** name and the **Engineer** name.

If you need to verify anchors, you must also define the **Anchor code** and **Edition**.

2. Analysis

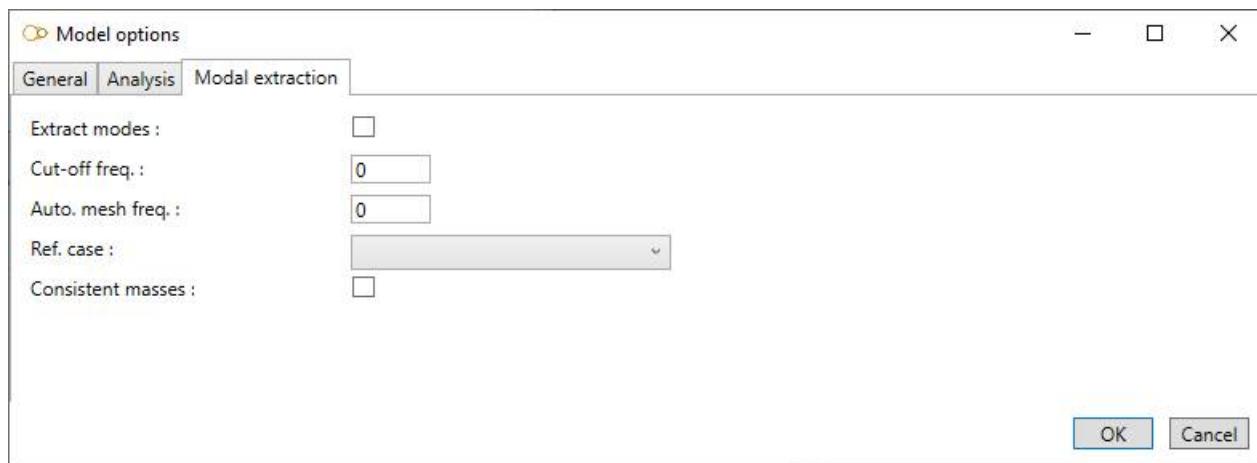
Model options

General		Analysis		Modal extraction	
Def. room T° :	21.1	Ratio steel to concrete :	15	°C	
<input type="button" value="OK"/> <input type="button" value="Cancel"/>					

- **Def. room T°** : default room temperature (used to calculate thermal expansion)
- **Ratio steel to concrete** : ratio of steel to concrete modulus of elasticity (used during anchor verification)

To know the UNIT of a property, just let the mouse over the cell. For example : °C for the room temperature.

3. Modal extraction



- **Extract modes** : check this option to extract the mode shapes
- **Cut-off freq.** : only the mode shapes below to the cut-off frequency are extracted
- **Auto. mesh freq.** : if different from 0, additional mass points will be generated to ensure that the mode shapes up to this frequency are accurate. This frequency should be equal or larger than the cut-off frequency
- **Ref. case** : the Young modulus Eh used for the modal extraction is evaluated at the temperature of the reference case. Room temperature is used if not defined
- **Consistent masses** : if checked, consistent mass matrix is used

Elements and link



1. Elements

MetaStructure proposes several tools to build **structures** :

- Draw structure (beam)
- Add node
- Add rigid
- Add spring
- Add restraint

2. Link

MetaStructure proposes a tool to facilitate the creation of nodes based on piping nodes :

- Add adjacent nodes
- And a tool to make a *connection* between piping nodes and structure nodes :
- Set pipe link

TABLE OF CONTENTS

- [Beam](#)
- [Node](#)
- [Rigid](#)
- [Spring](#)
- [Restraint](#)
- [Adjacent nodes](#)
- [Pipe link](#)

Draw structure

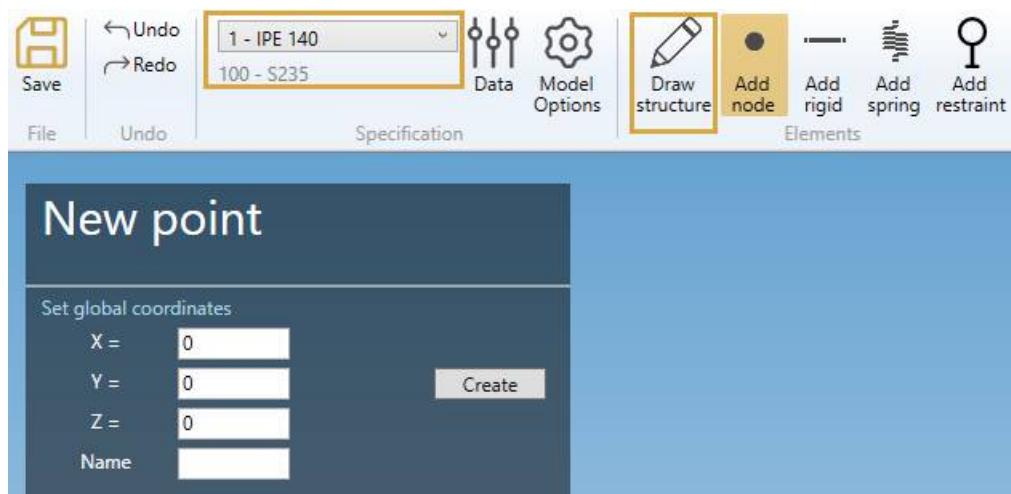
1. New project

Select a current **section**.

Click on the **Draw structure** button :



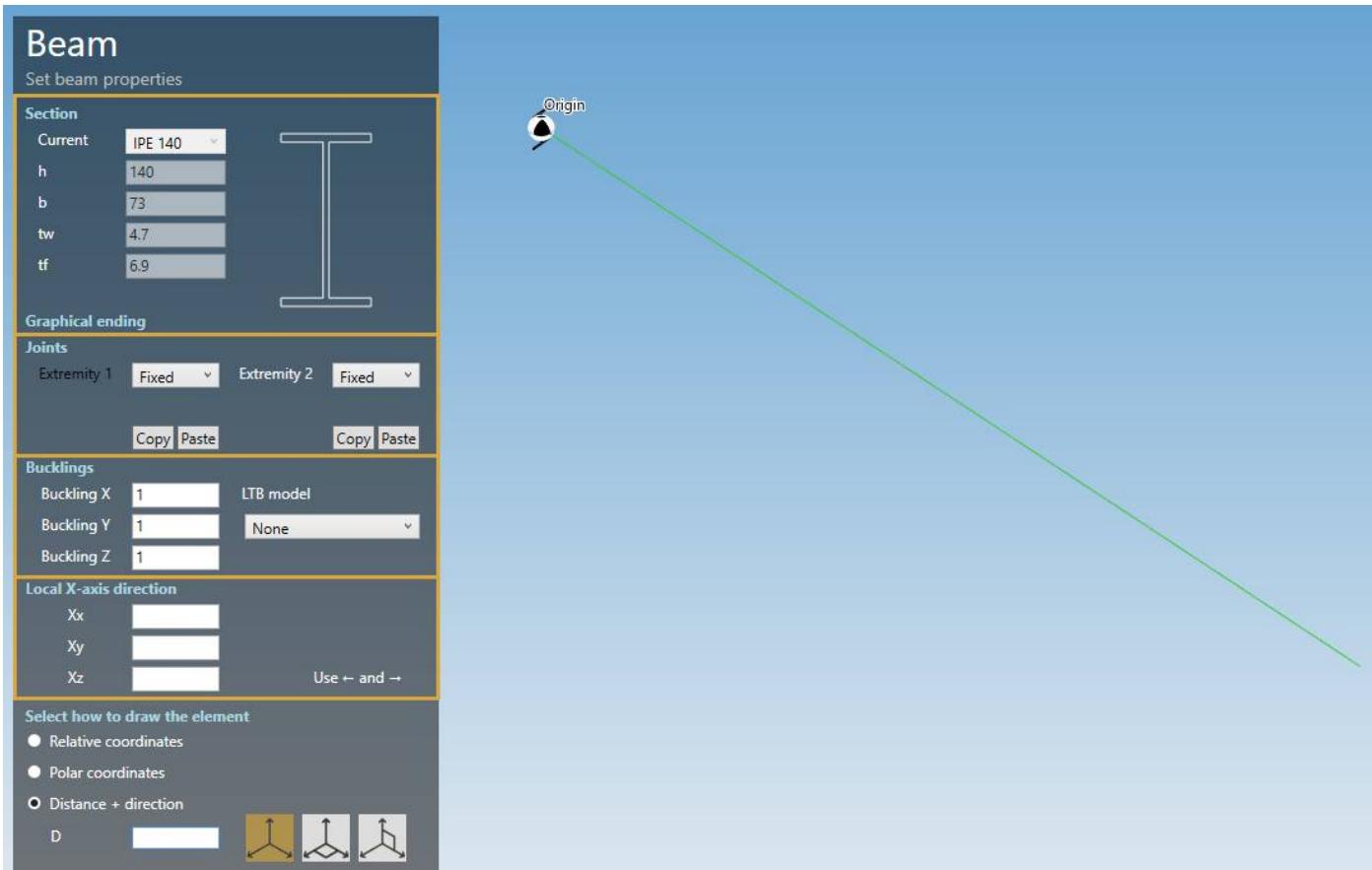
As we start the modeling, we need to create a **First node** :



Click [here](#) for more information about the **First node** creation.

1.2. Draw the first beam

After the first node has been created, you can draw your first beam :



The current **section** is the selected one in the upper **specification** ribbon.

Some properties are shown :

Property	Description	Unit Metric	Unit USA
h	Height	mm	in
b	Width	mm	in
tw	Web thickness	mm	in
tf	Flange thickness	mm	in

To know the UNIT of the value, just let the mouse over the cell.

You can define the joints, the buckling factors and the local-X axis direction. See below.

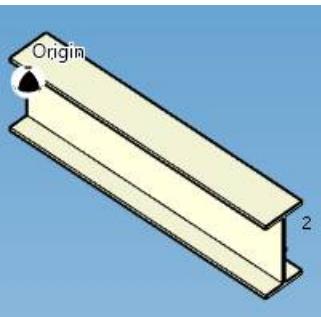
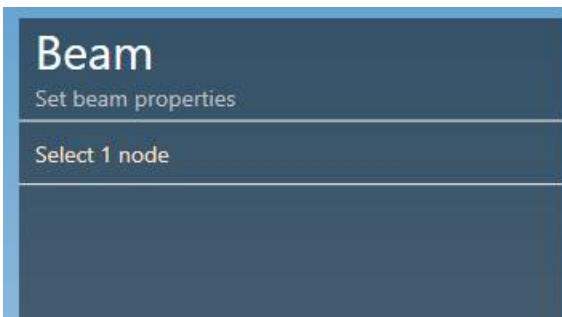
The only thing you have to do is to select how to draw the pipe.

Click [here](#) for more information about the orientation tool.

2. Draw structure

When you click on the **Draw structure** button without selection, the left panel shows a message :

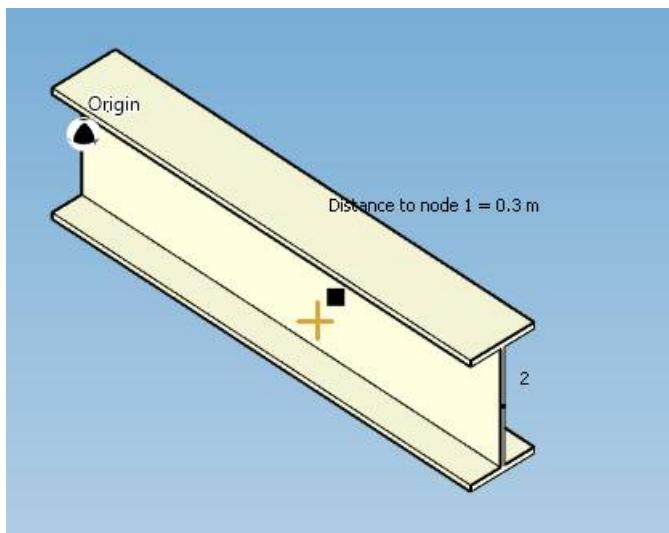
Select a node



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

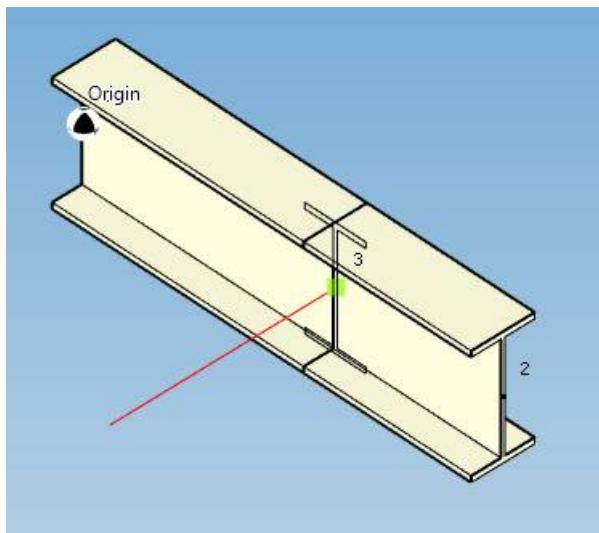
2.1 Intermediate node

You can start a new beam on an intermediate point on the beam under mouse :

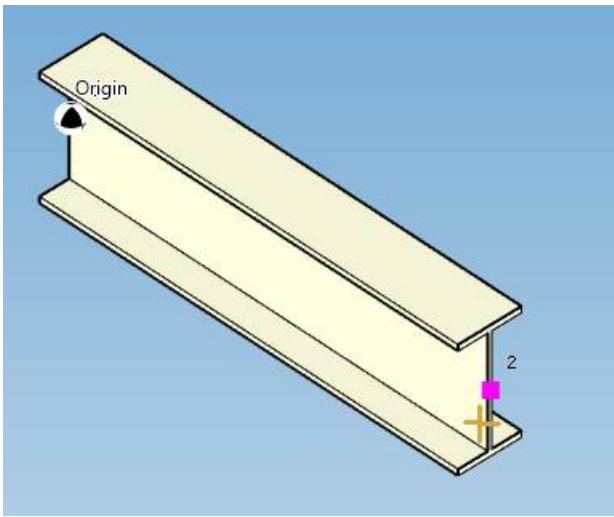


Once the distance from the first node of the beam is correct, **CLICK** on the left mouse button.

MetaStructure will automatically cut the beam and start a new beam from this new node :



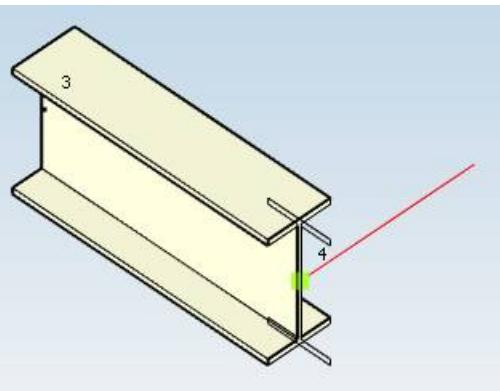
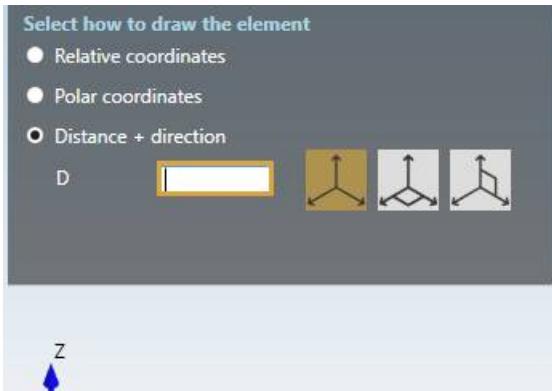
If you want to start from an extremity, be careful to have the extremity node in **MAGENTA** :



2.2 End of beam

You can draw a beam in different way :

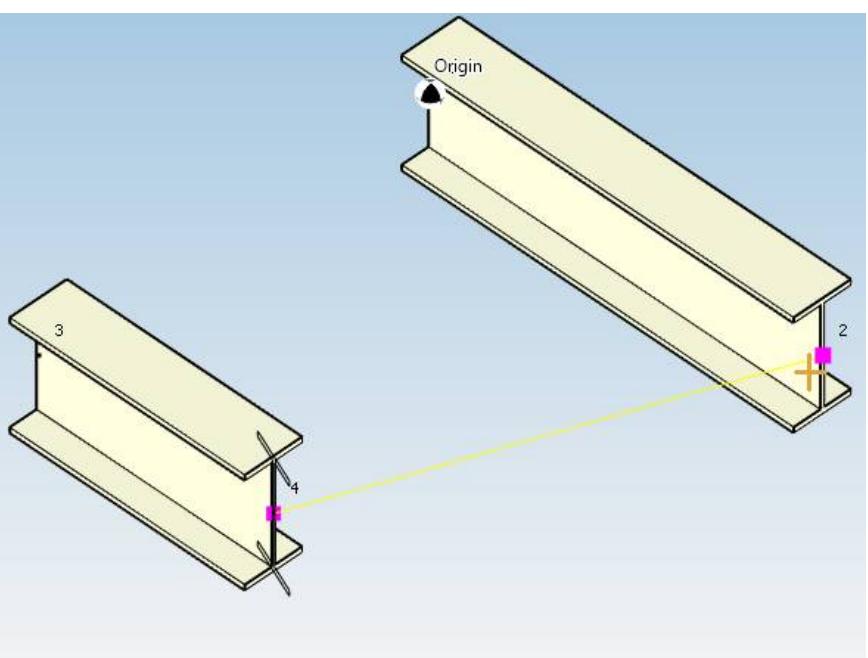
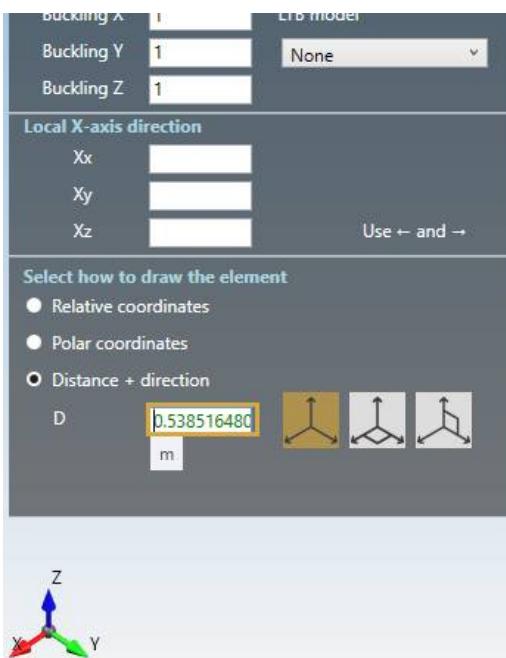
- With the **orientation tool** :



Choose a direction and write the **Length** with the numpad of the keyboard and press **Enter** to create the beam.

Click [here](#) for more information about the orientation tool.

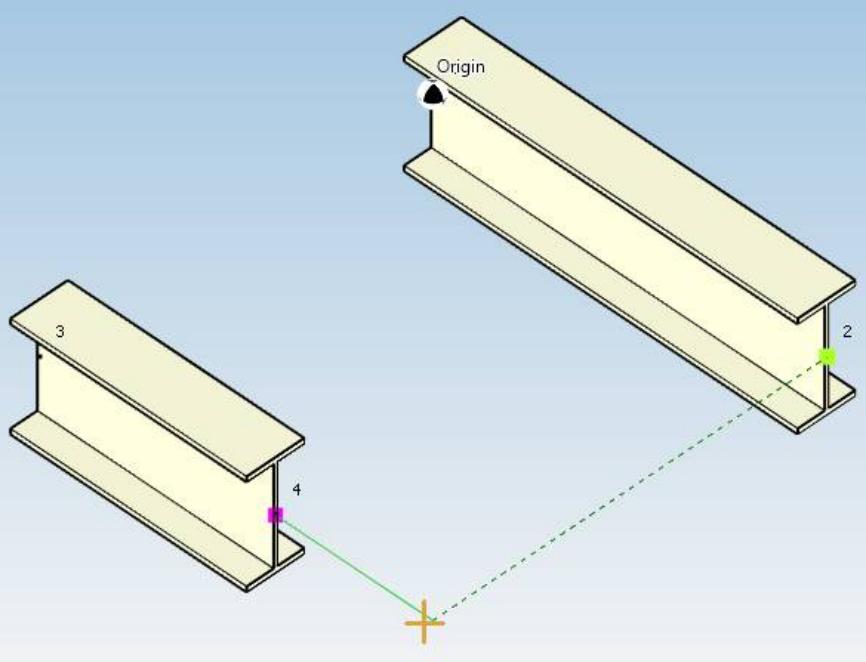
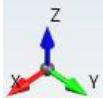
- That ends at an **existing node** :



The node must be in **MAGENTA** color. The distance is automatically set in the cell. Just press **Enter** to create the beam.

- That ends at a projection X/Y/Z of a **Reference node** (in green) :

Buckling X	1	LTB model
Buckling Y	1	None
Buckling Z	1	
Local X-axis direction		
Xx		
Xy		
Xz		Use ← and →
Select how to draw the element		
<input type="radio"/> Relative coordinates		
<input type="radio"/> Polar coordinates		
<input checked="" type="radio"/> Distance + direction		
D	0.2	m

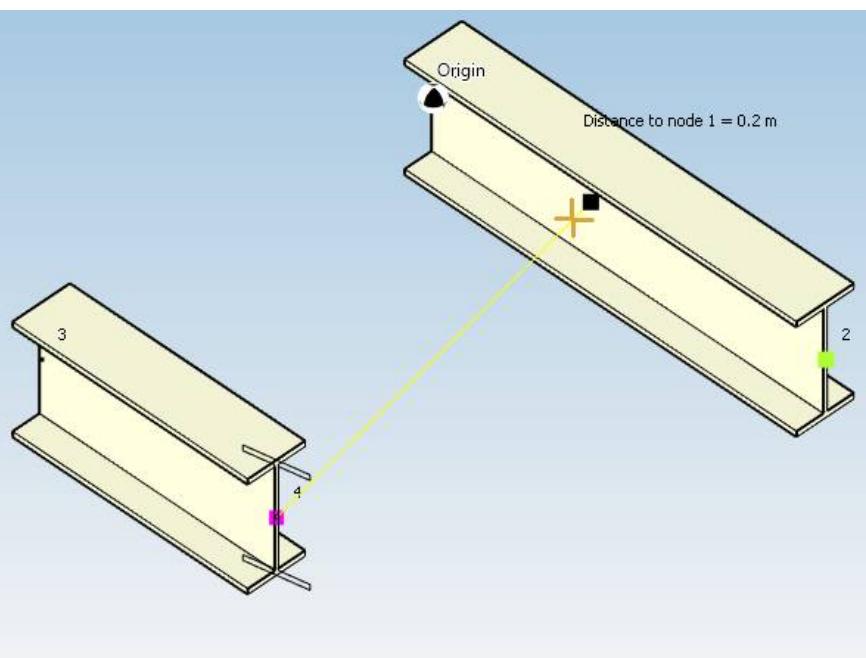
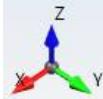


To define a **Reference point**, just move the mouse near an existing node -> it comes green.

The distance is automatically set in the cell. Just press **Enter** to create the beam.

- That ends at an **intermediate node** :

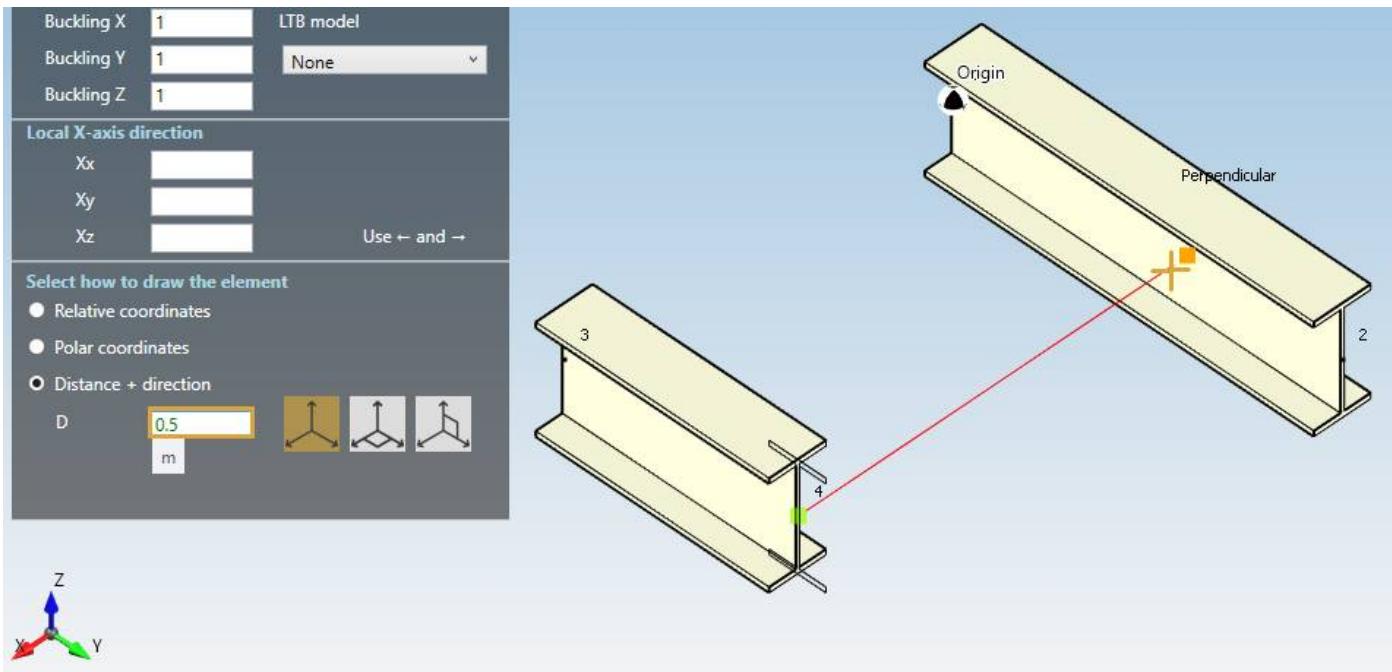
Buckling X	1	LTB model
Buckling Y	1	None
Buckling Z	1	
Local X-axis direction		
Xx		
Xy		
Xz		Use ← and →
Select how to draw the element		
<input type="radio"/> Relative coordinates		
<input type="radio"/> Polar coordinates		
<input checked="" type="radio"/> Distance + direction		
D	0.509901951	m



The distance is automatically set in the cell. Just press **Enter** to create the beam.

MetaStructure will automatically cut the beam

- That ends **perpendicular** to another beam :

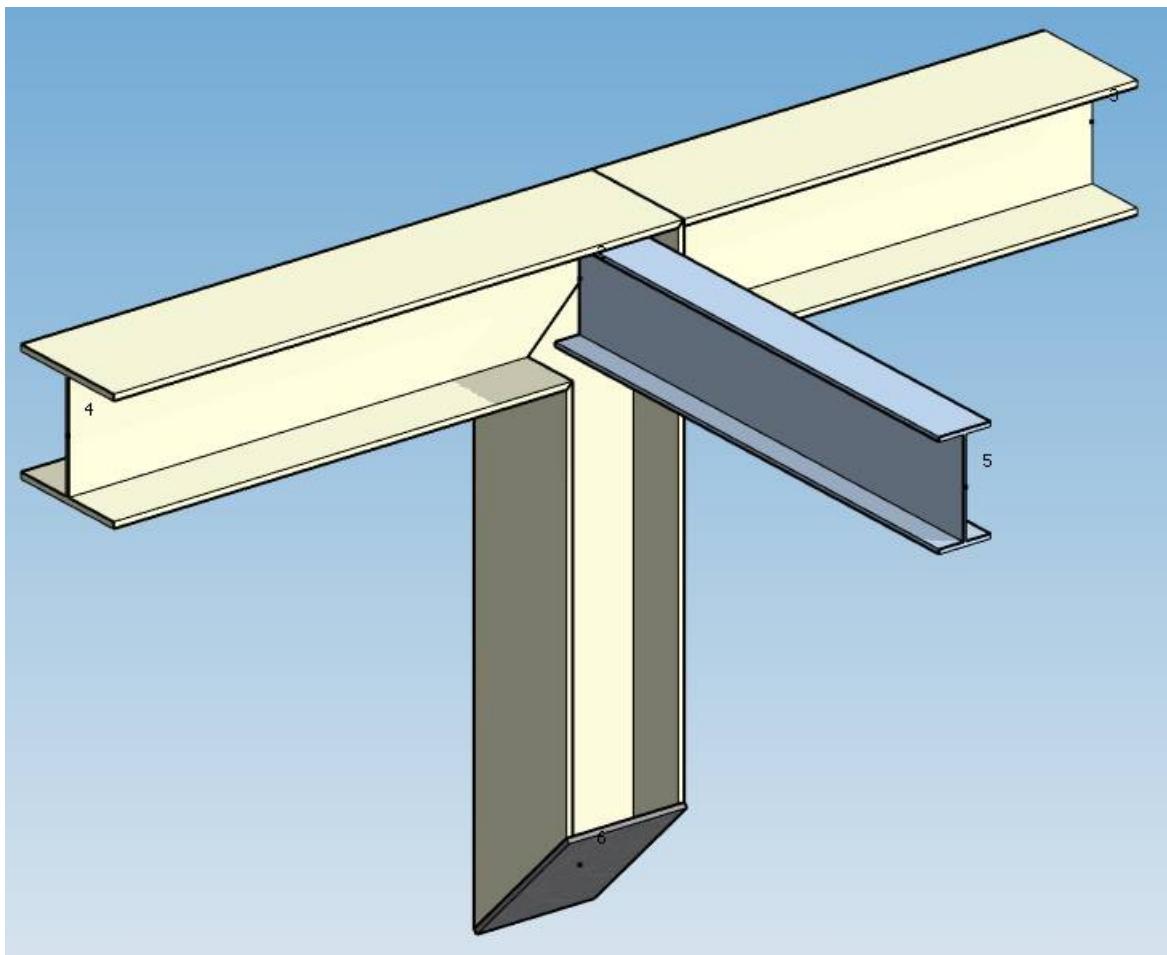


The distance is automatically set in the cell. Just press **Enter** to create the beam.

MetaStructure will automatically cut the beam.

3. Graphical ending

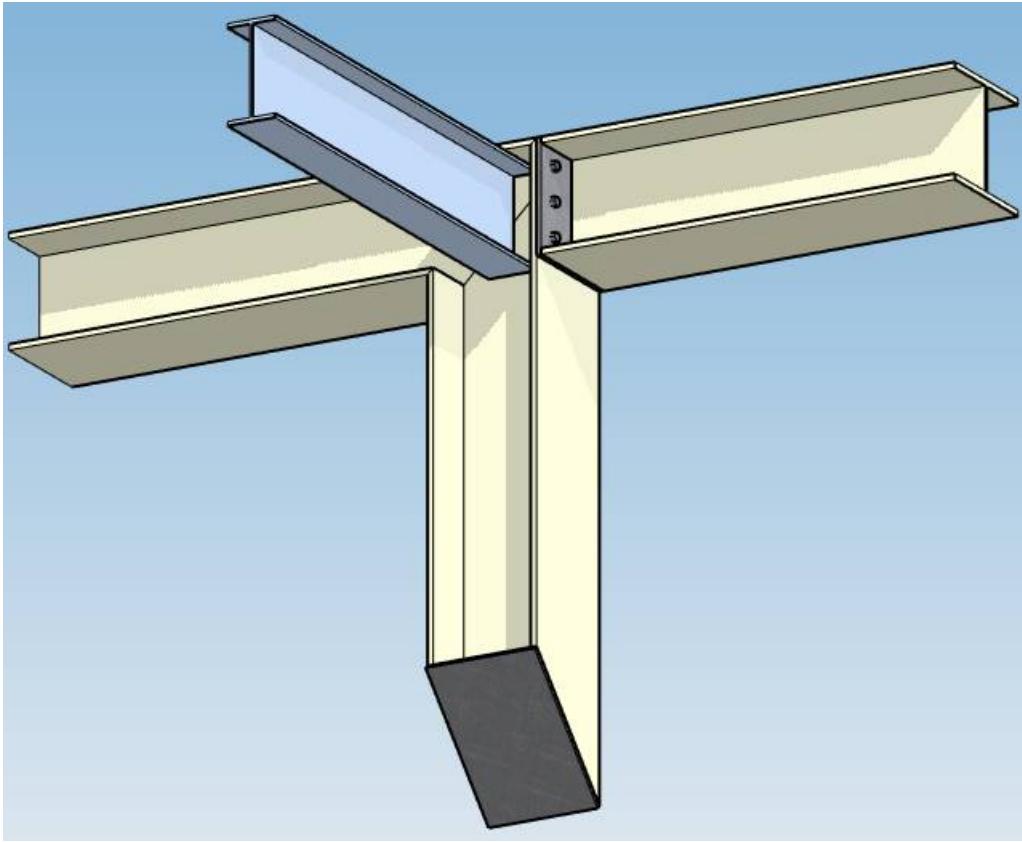
MetaStructure lets you define a graphical ending of every beams:



Click [here](#) for more information about the graphical endings.

4. Joints

MetaStructure proposes several **assemblies** for each extremity of a beam :



Example of a bolting plate

Click [here](#) for more information about assemblies.

5. Bucklings

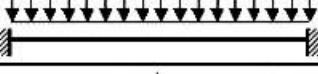
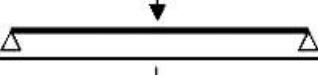
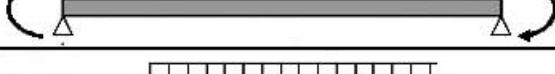
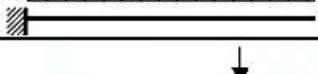
The **buckling** values are factors that multiply the real length of the beam to get the buckling lengths :

Buckles		
Buckling X	1	LTB model
Buckling Y	1	None
Buckling Z	1	

Property	Description	Default
Buckling X	buckling factor in the weak inertia plane	1
Buckling Y	buckling factor in the strong inertia plane	1
Buckling Z	the lateral-torsional buckling factor	1

The **Lateral-Torsional Buckling** model (LTB) must be defined for the calculation of the elastic critical moment according to Eurocode 3.

This critical moment depends on the coefficients C1 and C2 :

Method	C1 and C2	Diagram
Both Hinged Uniform	C1 = 1.13 and C2 = 0.45	
Both Fixed Uniform	C1 = 2.57 and C2 = 1.55	
Both Hinged Nodal	C1 = 1.35 and C2 = 0.59	
Both Fixed Nodal	C1 = 1.69 and C2 = 1.5	
Moments	Cf theoretical manual for C1 (C2 = 0)	
Cantilever Uniform	Cf theoretical manual	
Cantilever Nodal	Cf theoretical manual	

6. Local X-axis direction

The default local-X direction for a beam is :

X-axis	Global vertical axis	Default
Vertical beams	Z	(0, 1, 0)
Non vertical beams	Z	(0, 0, 1)
Vertical beams	Y	(0, 0, 1)
Non vertical beams	Y	(0, 1, 0)

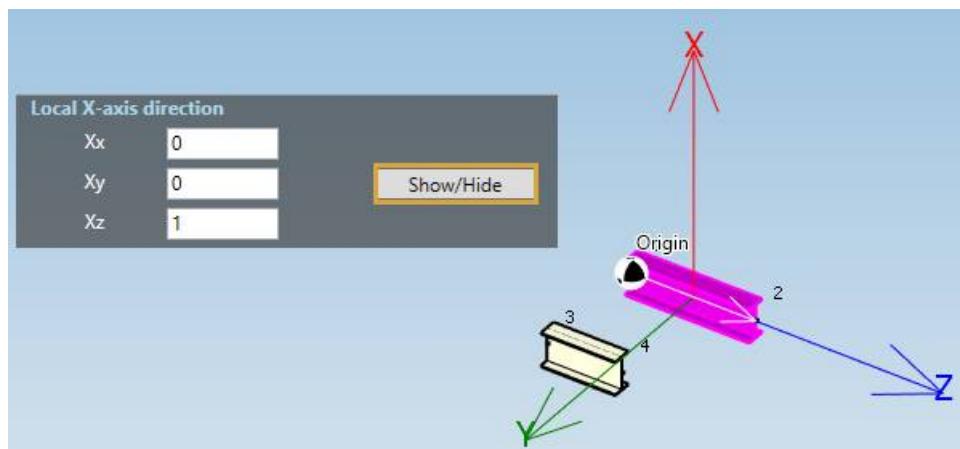
X-axis is always the **weak** axis, Y-axis the **strong** axis and Z-axis the axis of the beam from Node1 to Node2.

During modeling, you can set the values or press \rightarrow or \leftarrow to rotate the section by 90° :



The section is directly drawn as a preview.

When selecting a beam, you can **Show/Hide** the local X-axis direction and modify it :



Changing the local X-axis direction will **turn** the section around its Z-axis.

TABLE OF CONTENTS

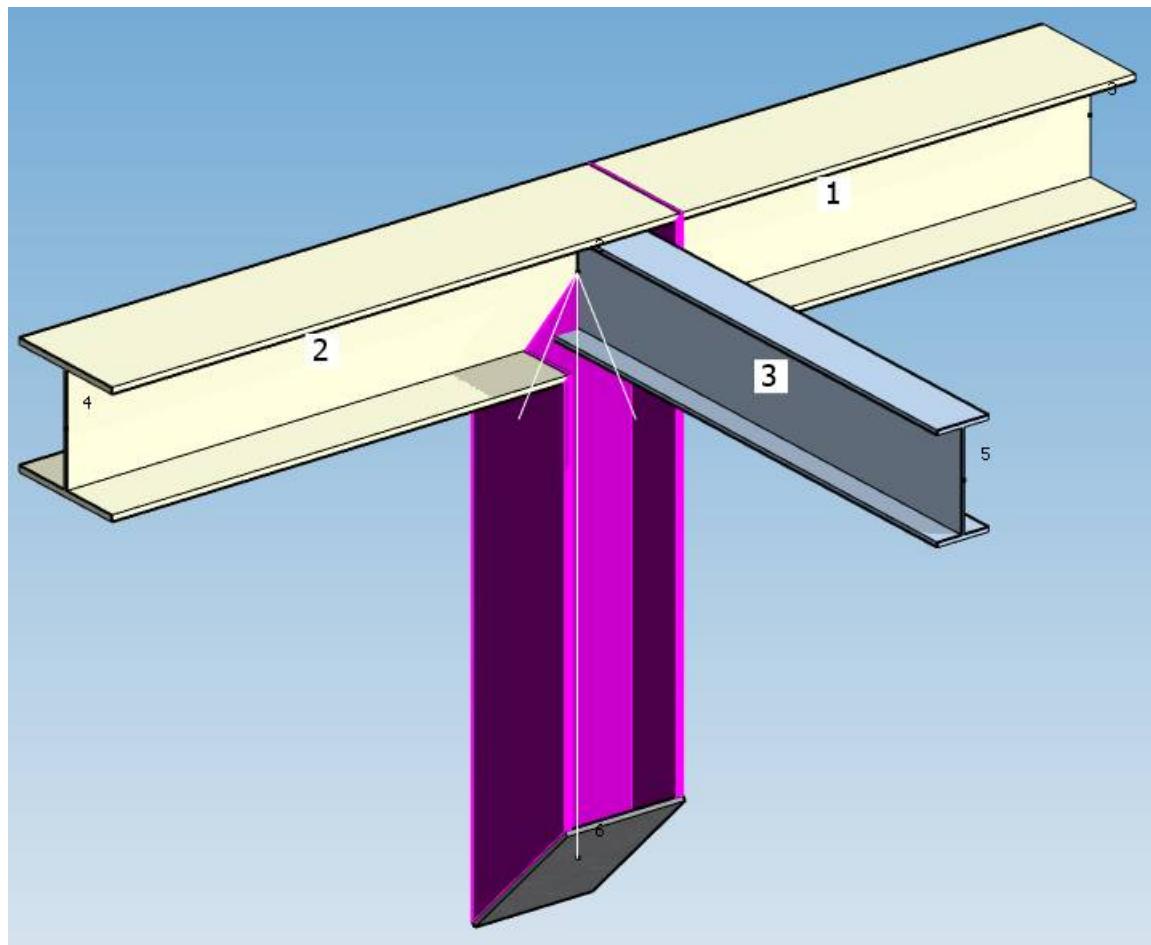
- [Graphical ending](#)
- [Joint](#)

Graphical ending

MetaStructure proposes several endings for each extremity of a beam :

Type	Description
None	Ending plane = plane perpendicular to the current beam axis at the node
Front	Ending plane = nearest plane encountered at the specified beam
Back	Ending plane = furthest plane encountered at the specified beam
Miter	Ending plane = bisector plane between current beam and specified beam
Plate	Ending plane = plane of the anchor plate

1. Definition



The direction of a selected beam is represented by a **white arrow** (Node1 to Node2).

All beams (not parallel to current one) of each extremity are numbered.

At extremity 1 (Node1), the numbers are drawn inside a **black square**.

At extremity 2 (Node2), the numbers are drawn inside a **white square**.

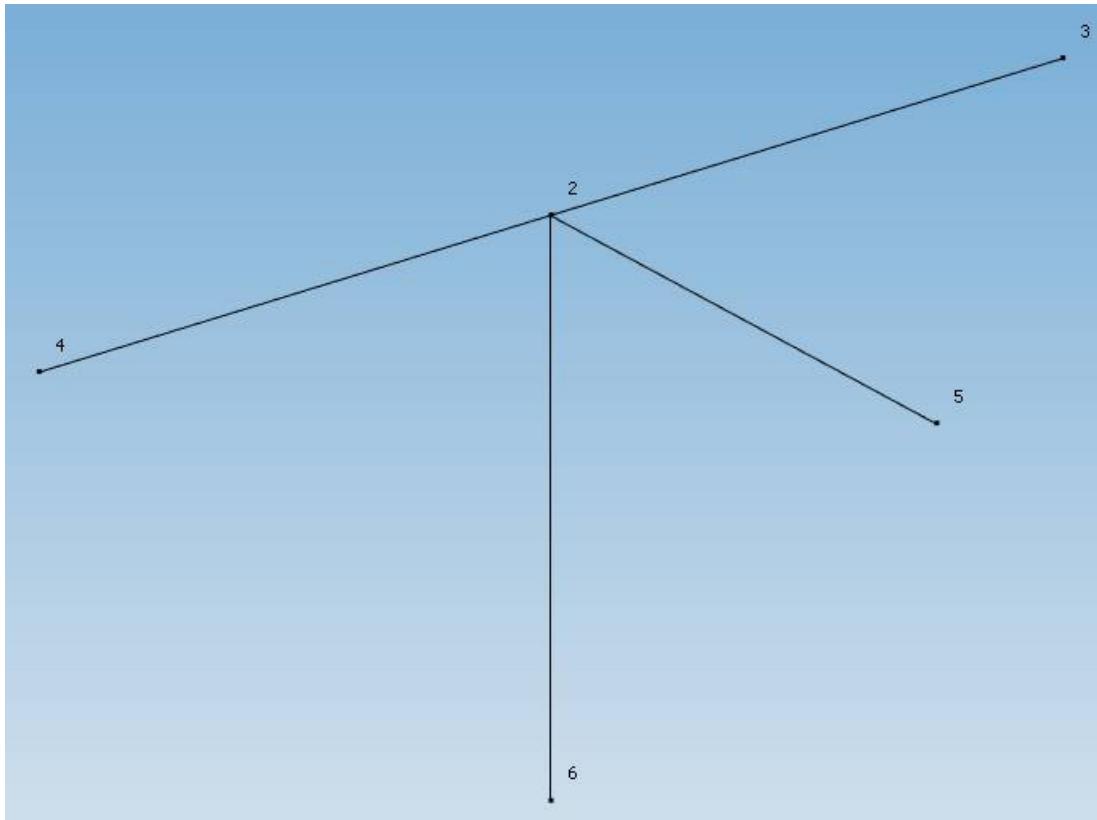
You can specify the beam with which you want a particular ending.

The **Offset** adds (or subtracts if negative value) an offset distance to the ending plane (parallel to the plane).

Property	Unit Metric	Unit USA
Offset	mm	in

WARNING

ATTENTION, the endings are only graphical and have no impact on the calculation. It helps for visualization and for the calculation of plate dimensions. In fact the beams do intersect at the node :



1.1 None ending

Ending plane = plane perpendicular to the current beam at the node :

Graphical ending

Extremity 1	None	Extremity 2	None
Beam 1	1	Beam 2	1
Offset 1	0	Offset 2	0

Joints

Extremity 1	Fixed	Extremity 2	Fixed
-------------	-------	-------------	-------

1.2 Front ending

Ending plane = nearest plane encountered at the specified beam :

Graphical ending

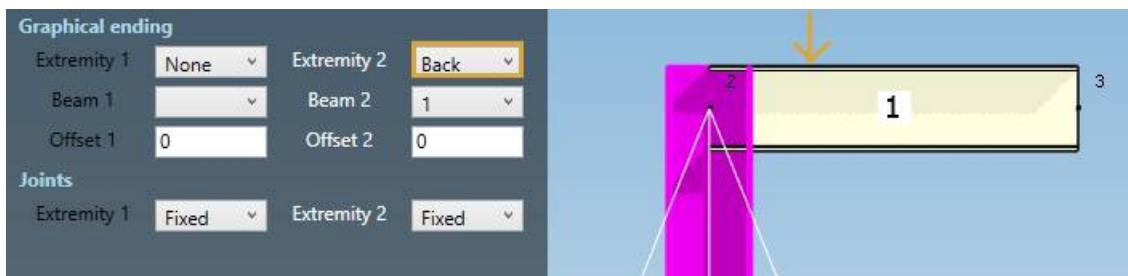
Extremity 1	None	Extremity 2	Front
Beam 1	1	Beam 2	1
Offset 1	0	Offset 2	0

Joints

Extremity 1	Fixed	Extremity 2	Fixed
-------------	-------	-------------	-------

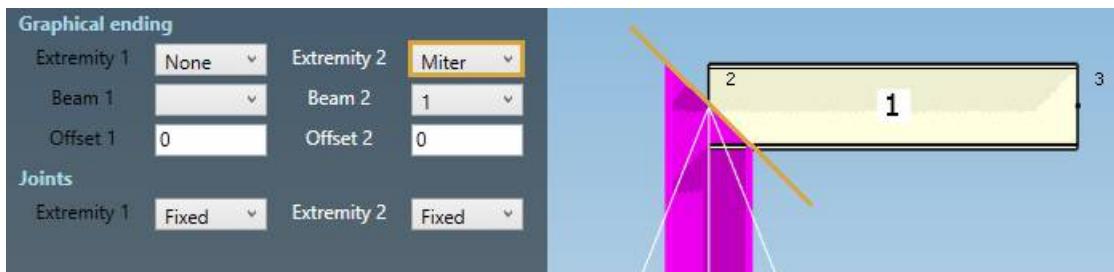
1.3 Back ending

Ending plane = furthest plane encountered at the specified beam :



1.4 Miter ending

Ending plane = bisector plane between current beam and specified beam :



1.5 Plate ending

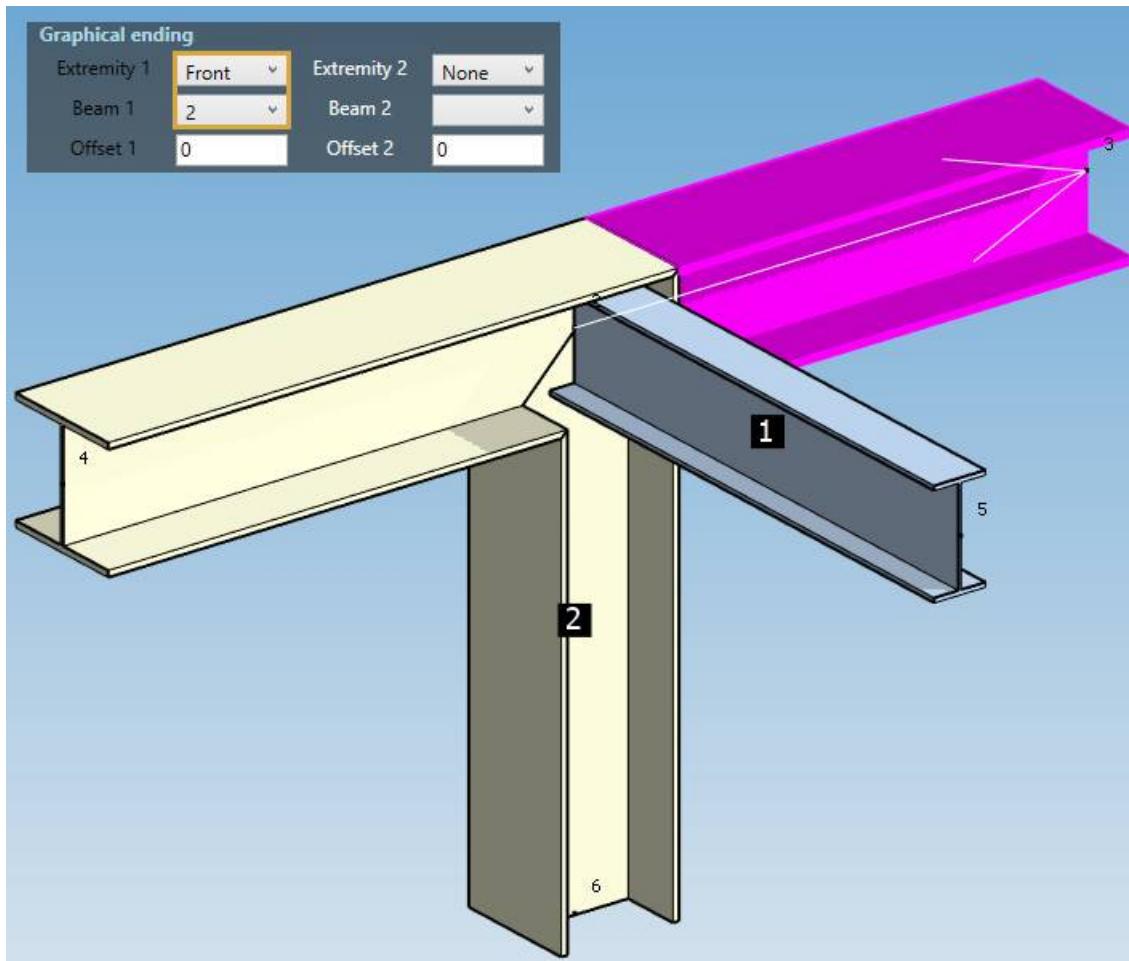
Ending plane = plane of the anchor plate.

An **anchor plate** must exists at the extremity node.

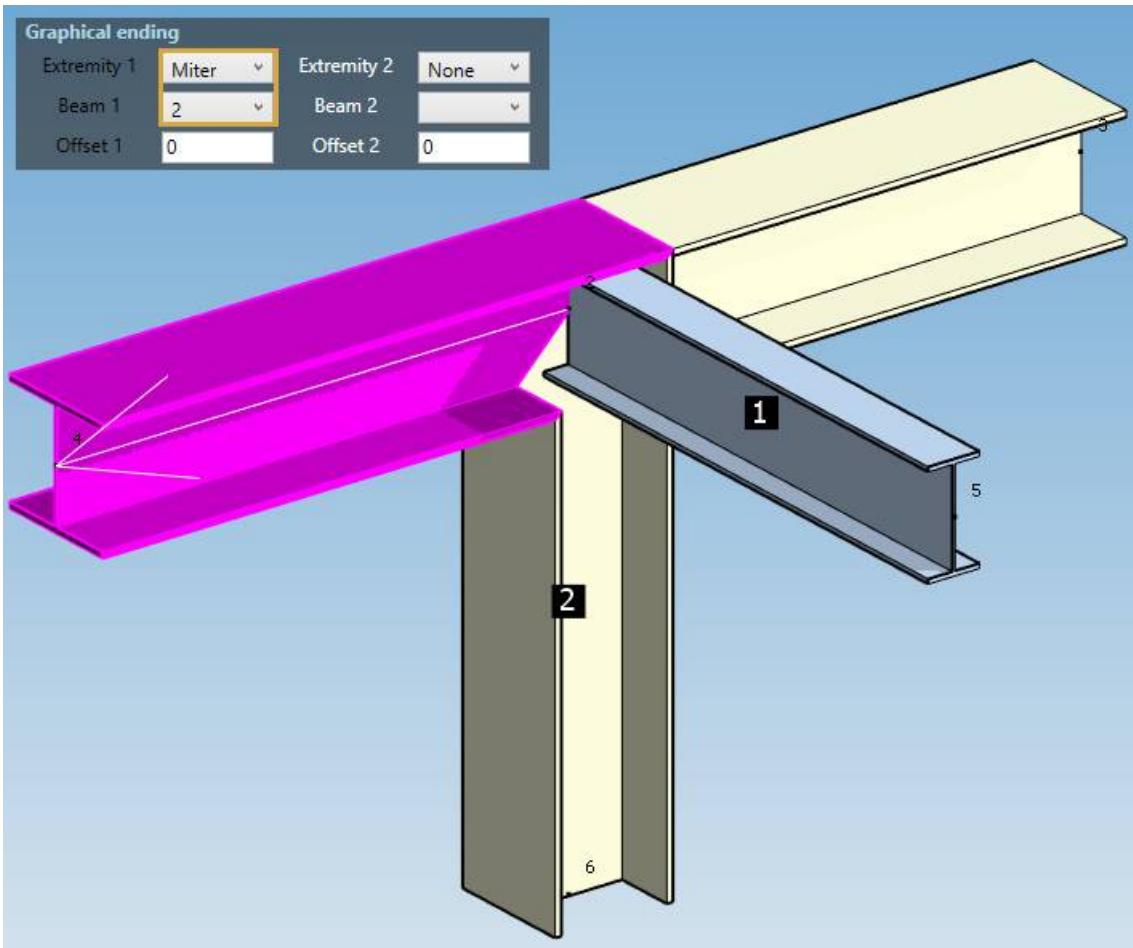
See §2. for an example of plate ending.

2. Examples

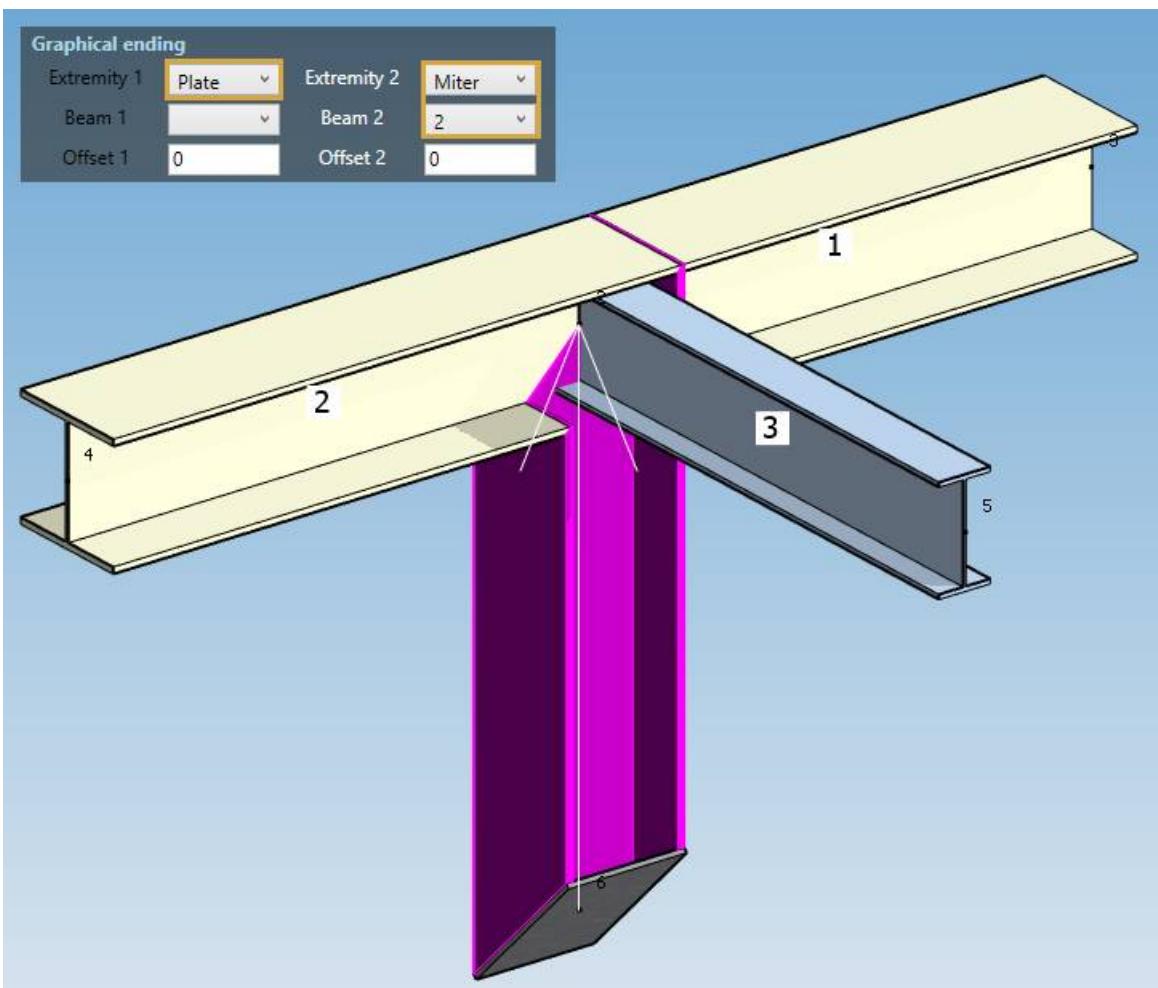
In this example, the selected beam has a **front** ending with beam 2 at its first extremity (black column):



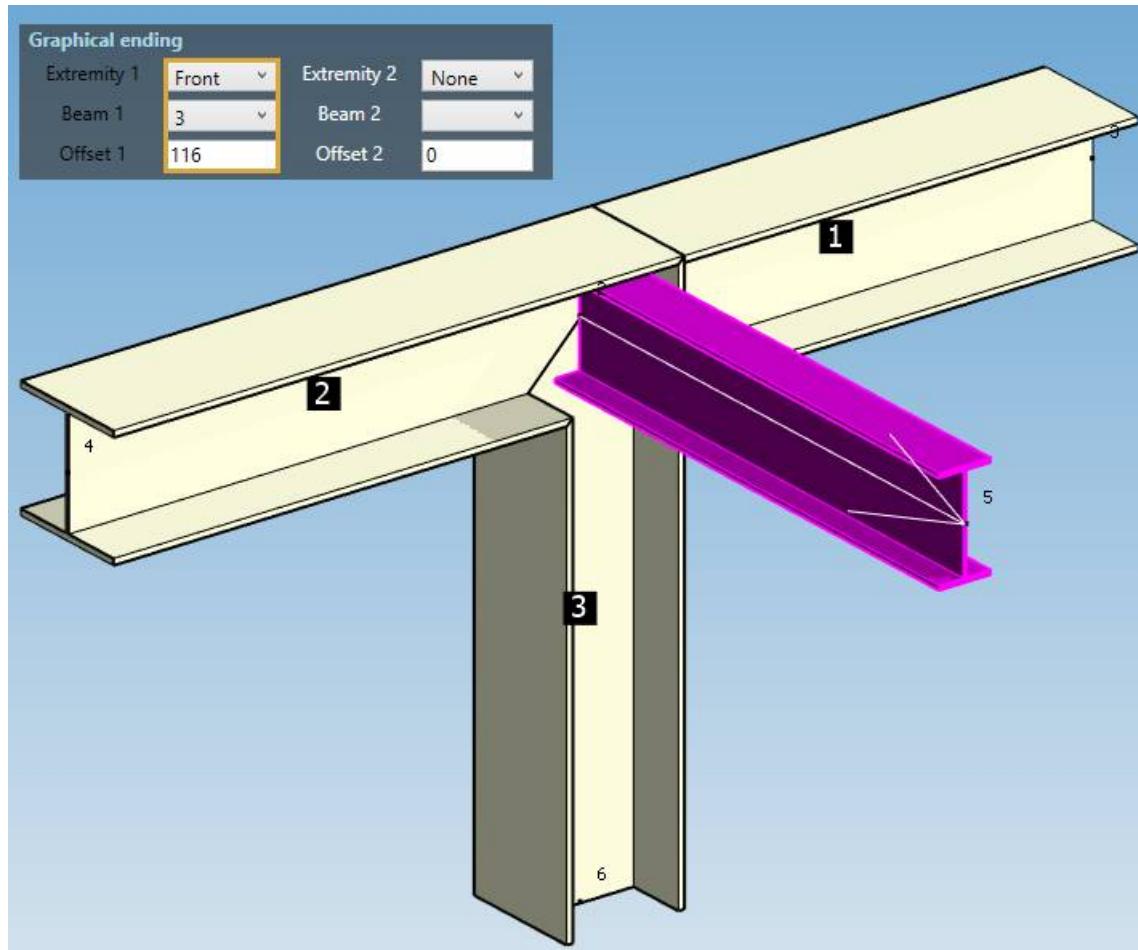
In this example, the selected beam has a **miter** ending with beam 2 at its first extremity (black column) :



In this example, the selected beam has a **plate** ending at its first extremity (black column) and a **miter** ending with beam 2 at its second extremity (white column) :



In this example, the selected beam has a **front** ending with beam 3 with an offset of 116 mm at its first extremity (black column) :



The **offset** corresponds to the distance from the first face encountered to the web = $(240 - 7.5) / 2 = +/- 116$ mm :

Current specification

- Specifications
- Beams
- Materials

Beam sections

Name: HEA 240

Description: Hot rolled

Dimensions:

- h = 230 mm
- b = 240 mm
- tw = 7.5 mm
- tf = 12 mm
- A = 76.84 cm²
- Ax = 17.25 cm²
- Ay = 57.6 cm²
- Ix = 2769 cm⁴
- ly = 7763 cm⁴
- It = 41.55 cm⁴
- lw = 328500 cm⁶
- WT = 34.625 cm³
- Wy = 48 cm²
- Wx = 14.375 cm²
- WEly = 675.1 cm³
- WElx = 230.7 cm³

Click [here](#) to return to **beam** page.



Joint

MetaStructure proposes several **assemblies** for each extremity of a beam :

Type	Description
Fixed	Rigid joint (transmission of moments)
Detailed	User can specify the 3 translation stiffnesses and 3 rotation stiffnesses
Bolted	User can define a bolted joint
Welded	User can define a welded joint

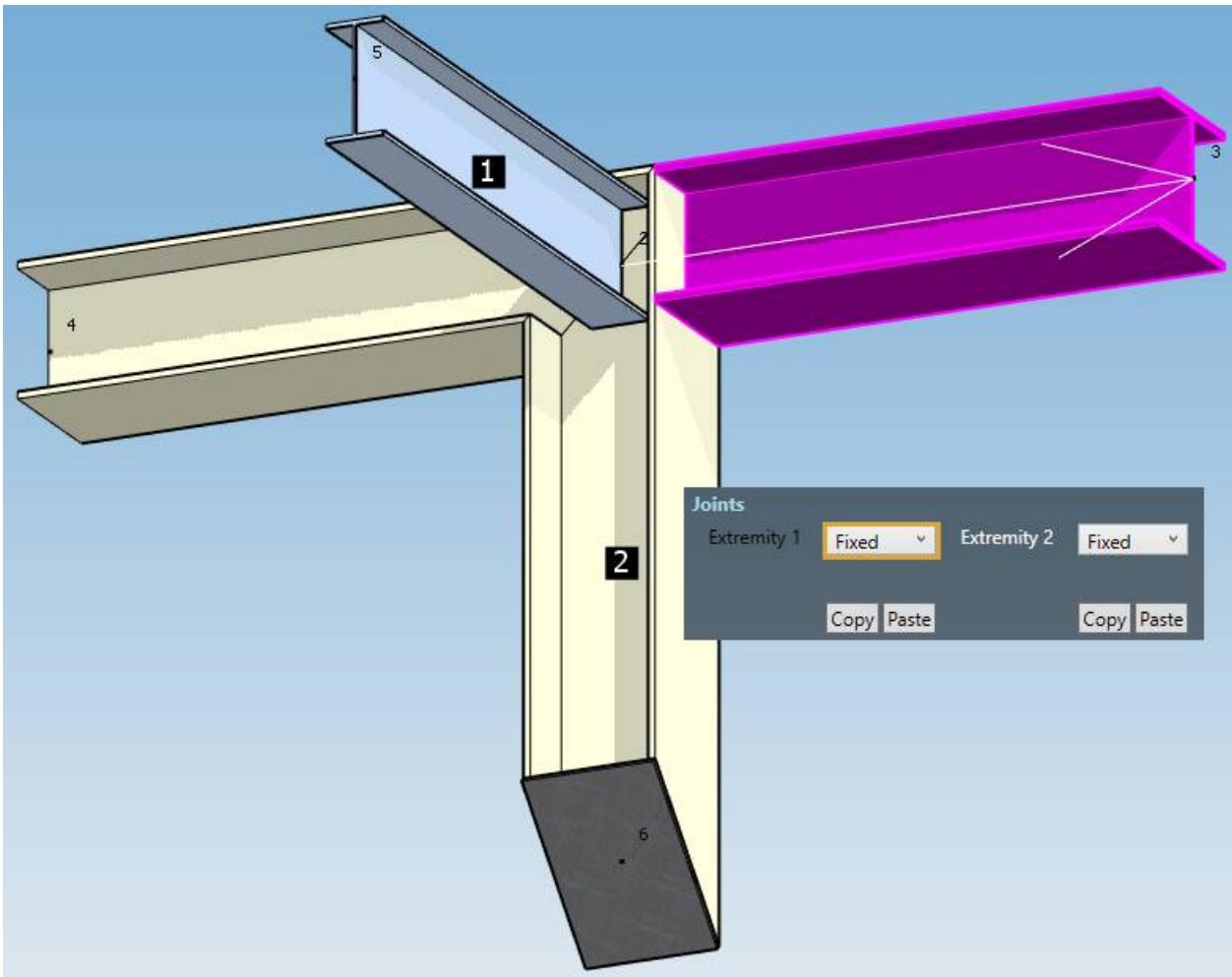
1. Definition

All joint types are at least defined by 6 stiffnesses :

Property	Description	Unit Metric	Unit USA
Kx	Translation stiffness X	kN/mm	kips/in
Ky	Translation stiffness Y	kN/mm	kips/in
Kz	Translation stiffness Z	kN/mm	kips/in
Krx	Rotation stiffness X	kN.m/rad	kips.ft/rad
Kry	Rotation stiffness Y	kN.m/rad	kips.ft/rad
Krz	Rotation stiffness Z	kN.m/rad	kips.ft/rad

1.1 Fixed joint

This is the default rigid connection.



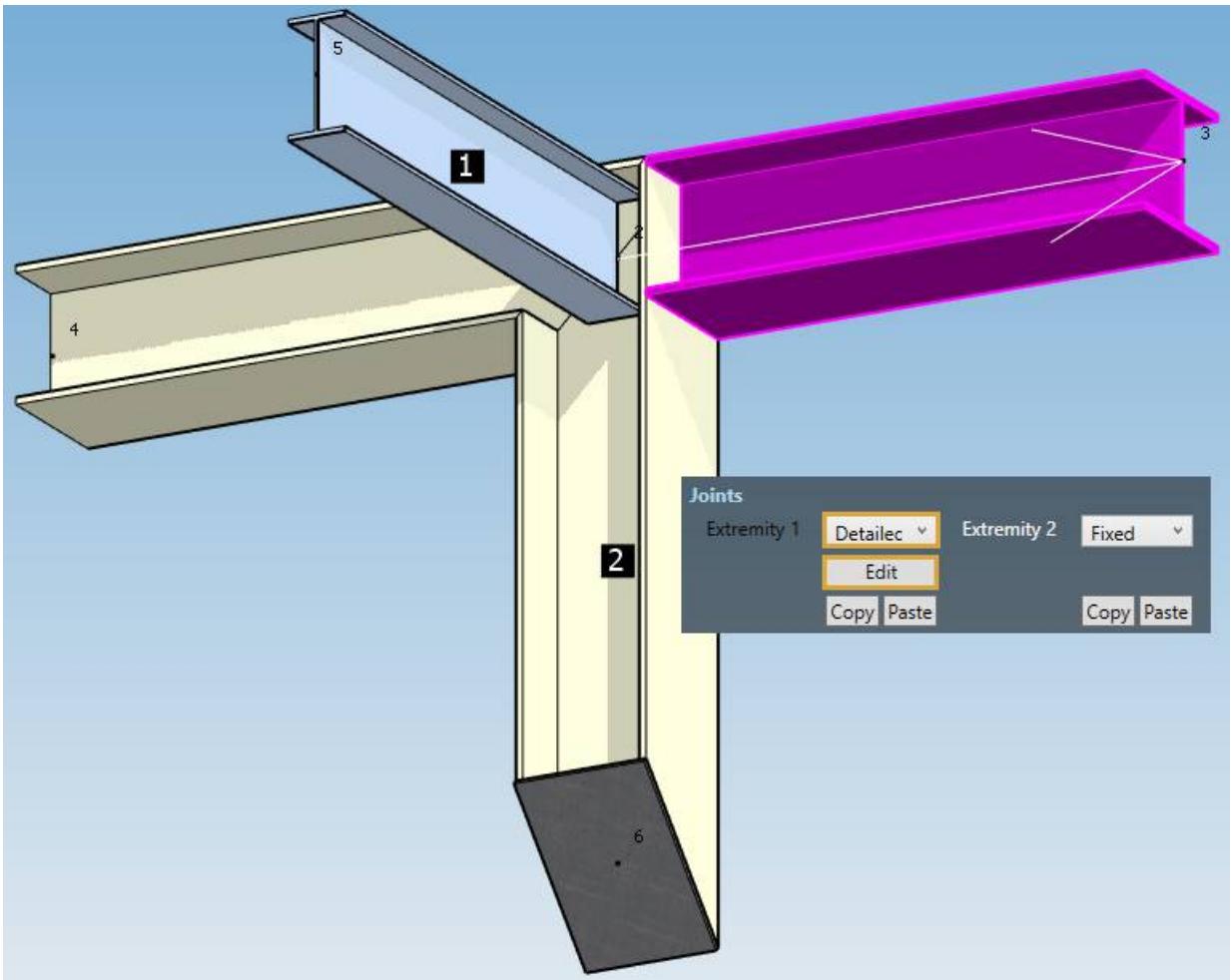
Example of fixed joint on extremity 1 (black side)

Property	Value (MKS units)
Kx, Ky, Kz	1.75e13
Krx, Kry, Krz	1.13e12

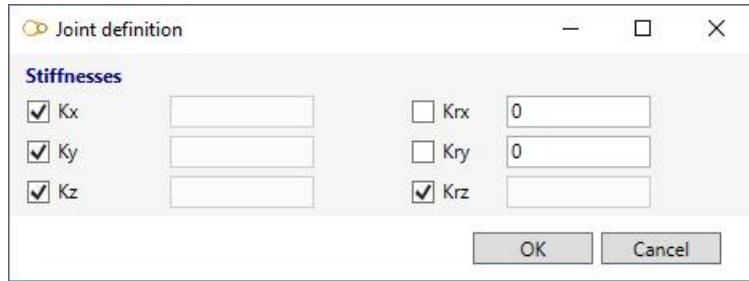
If user wants to define other stiffnesses, he has to set the joint to **detailed**.

1.2 Detailed joint

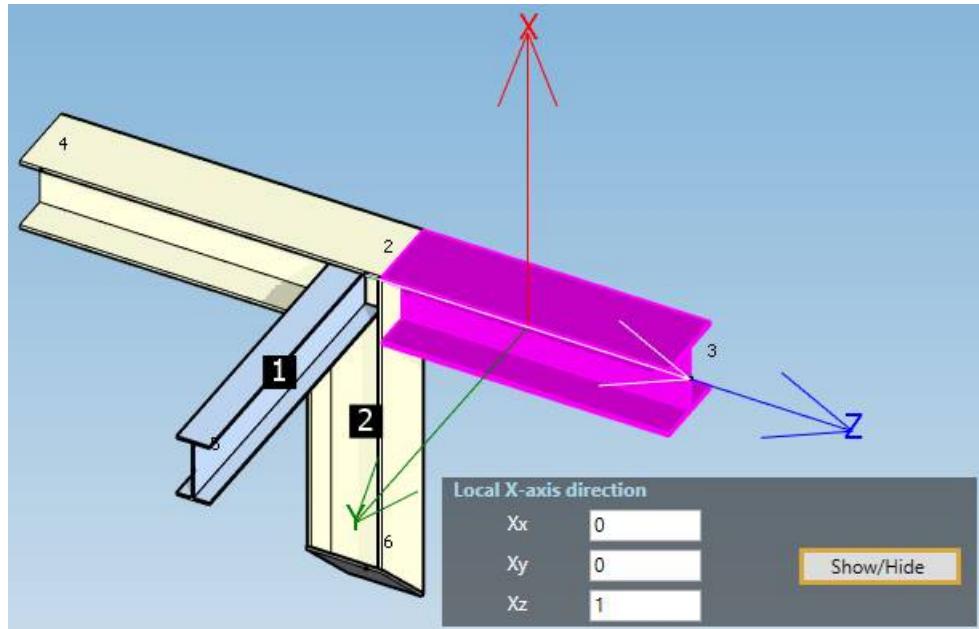
This is the simplest joint where user can define the stiffnesses :



Press the **Edit** button and disable the transmission of the moments (stiffness = 0) around X and Y axis :



To see the local axis, press the **Show/Hide** button of **Local X-axis direction** :

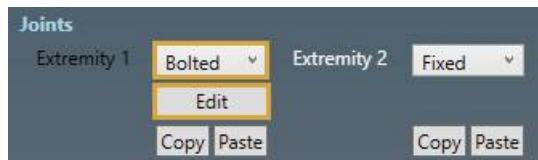


WARNING

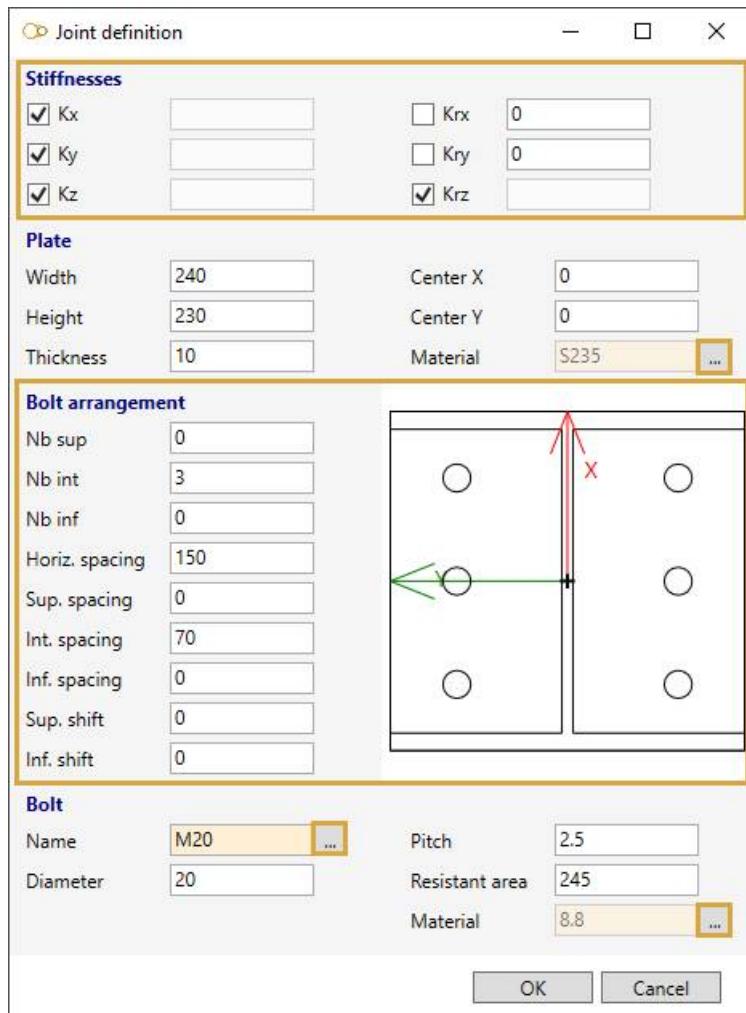
Z-axis is always the axis of the beam in MetaStructure and X-axis is the **weak** axis.

1.3 Bolted joint

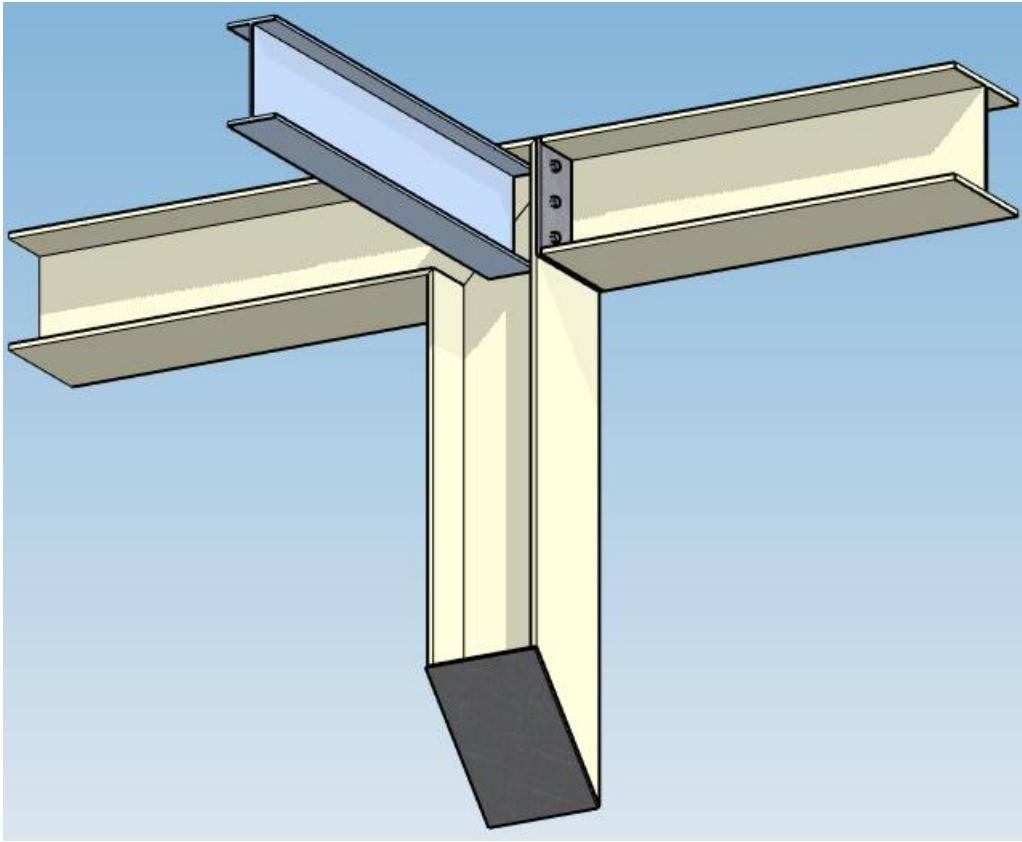
The assembly of a beam to another beam can be defined by a bolted plate :



Press the **Edit** button and set the plate properties :



Press **OK** to validate the bolted plate and press **Modify** to validate the beam :



Stiffnesses :

Property	Description	Unit Metric	Unit USA
Kx	Translation stiffness X	kN/mm	kips/in
Ky	Translation stiffness Y	kN/mm	kips/in
Kz	Translation stiffness Z	kN/mm	kips/in
Krx	Rotation stiffness X	kN.m/rad	kips.ft/rad
Kry	Rotation stiffness Y	kN.m/rad	kips.ft/rad
Krz	Rotation stiffness Z	kN.m/rad	kips.ft/rad

See §1.2 for more information.

Plate :

Explanation of the fields on the picture :

Joint definition

Stiffnesses

<input checked="" type="checkbox"/> Kx	<input type="text"/>	<input type="checkbox"/> Krx	<input type="text"/> 0
<input checked="" type="checkbox"/> Ky	<input type="text"/>	<input type="checkbox"/> Kry	<input type="text"/> 0
<input checked="" type="checkbox"/> Kz	<input type="text"/>	<input checked="" type="checkbox"/> Krz	<input type="text"/>

Plate

Width	260	Center X	30
Height	400	Center Y	0
Thickness	10	Material	S235 <input type="button" value="..."/>

Bolt arrangement

Nb sup	1
Nb int	3
Nb inf	2
Horiz. spacing	150
Sup. spacing	0
Int. spacing	70
Inf. spacing	50
Sup. shift	25
Inf. shift	30

Bolt

Name	M20 <input type="button" value="..."/>	Pitch	2.5
Diameter	20	Resistant area	245
		Material	8.8 <input type="button" value="..."/>

The plate receives the dimensions of the beam when created.

Property	Description	Unit Metric	Unit USA
Width	Width of the plate	mm	in
Height	Height of the plate	mm	in
Thickness	Thickness of the plate	mm	in
Center X	Distance along X between center of plate and center of beam	mm	in
Center Y	Distance along Y between center of plate and center of beam	mm	in

Click on the ... button to select a *Structure* material :

Select material

Library
EN 10025-2

Material
S235

Click [here](#) for more information about database of materials.

Bolt arrangement :

Explanation of the fields on the pictures :

Plate	
Width	260
Height	400
Thickness	10
Center X	30
Center Y	0
Material	S235
Bolt arrangement	
Nb sup	1
Nb int	3
Nb inf	2
Horiz. spacing	150
Sup. spacing	0
Int. spacing	70
Inf. spacing	50
Sup. shift	25
Inf. shift	30
Bolt	
Name	M20
Diameter	20
Pitch	2.5
Resistant area	245
Material	8.8

Bolt arrangement	
Nb sup	1
Nb int	3
Nb inf	2
Horiz. spacing	150
Sup. spacing	0
Int. spacing	70
Inf. spacing	50
Sup. shift	25
Inf. shift	30

Property	Unit Metric	Unit USA
Spacing	mm	in
Shift	mm	in

Bolt :

Bolt	
Name	M20
Diameter	20
Pitch	2.5
Resistant area	245
Material	8.8

Click on the ... button to select a Bolt :

Select bolt

Bolts

ISO

Name	ISO Coarse		
Units	Metric		
Bolts			
Name	Diameter	Pitch	Resistant area
M10	10	1.5	58
M12	12	1.75	84.3
M14	14	2	115
M16	16	2	157
M18	18	2.5	192
M20	20	2.5	245
M22	22	2.5	303
M24	24	3	353
M27	27	3	459
M30	30	3.5	561

OK Cancel

All properties are automatically filled.

Click [here](#) for more information about database of bolts.

Property	Unit Metric	Unit USA
Diameter	mm	in
Pitch	mm	in
Resistant are	mm ²	in ²

Click on the ... button to select a *Bolting* material :

Select material

Library
Bolting

Material
8.8

OK Cancel

Click [here](#) for more information about database of materials.

1.4 Welded joint

The assembly of a beam on another beam can be defined by welding :

Joints

Extremity 1: Welded Extremity 2: Fixed

Edit Copy | Paste Copy | Paste

Press the **Edit** button and set the weld properties :

Joint definition

Stiffnesses

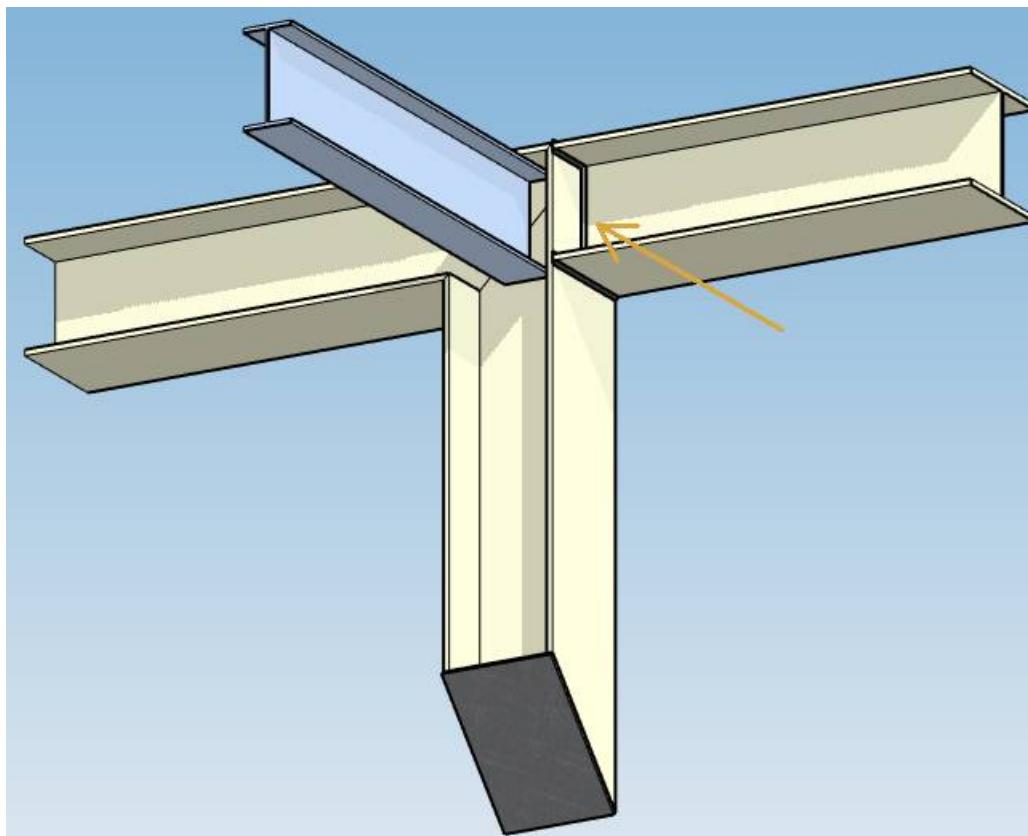
<input checked="" type="checkbox"/> Kx	<input type="text"/>	<input type="checkbox"/> Krx	<input type="text"/> 0
<input checked="" type="checkbox"/> Ky	<input type="text"/>	<input type="checkbox"/> Kry	<input type="text"/> 0
<input checked="" type="checkbox"/> Kz	<input type="text"/>	<input checked="" type="checkbox"/> Krz	<input type="text"/>

Weld

Flange throat	<input type="text"/> 5	Double flange	<input checked="" type="checkbox"/>
Web throat	<input type="text"/> 5	Material	<input type="text"/> Z_SDLX02 <input type="button"/> ...

OK **Cancel**

Press **OK** to validate the welding and press **Modify** to validate the beam :



Stiffnesses :

Property	Description	Unit Metric	Unit USA
Kx	Translation stiffness X	kN/mm	kips/in
Ky	Translation stiffness Y	kN/mm	kips/in
Kz	Translation stiffness Z	kN/mm	kips/in
Krx	Rotation stiffness X	kN.m/rad	kips.ft/rad
Kry	Rotation stiffness Y	kN.m/rad	kips.ft/rad
Krz	Rotation stiffness Z	kN.m/rad	kips.ft/rad

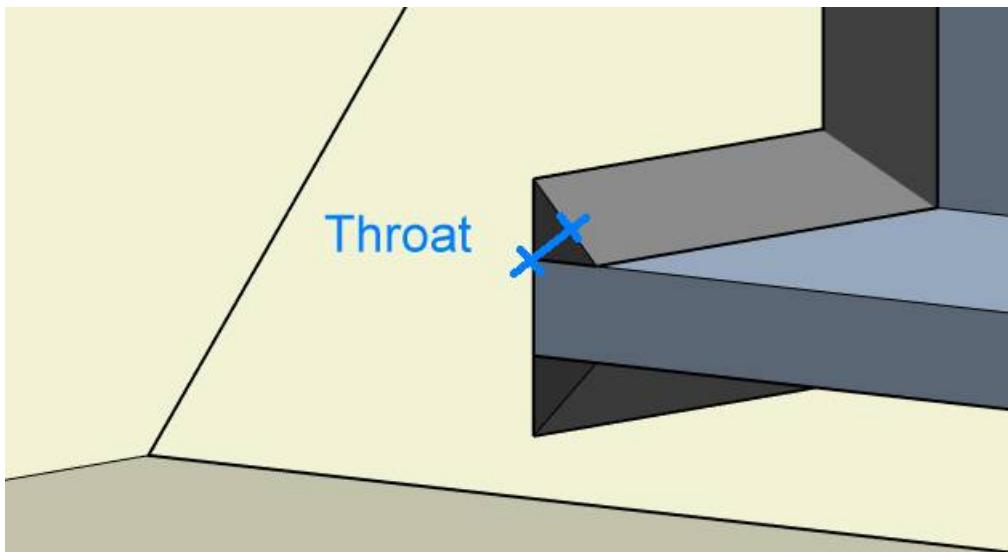
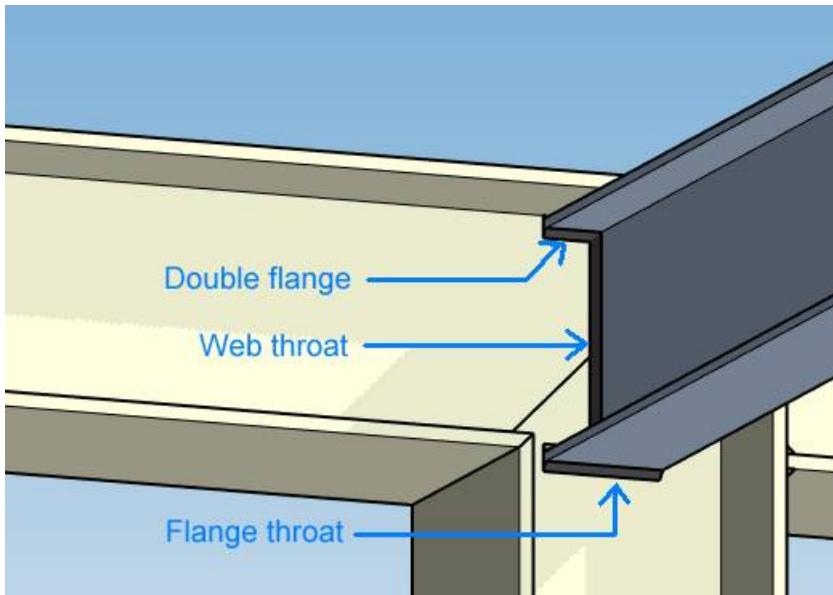
See §1.2 for more information.

Weld :

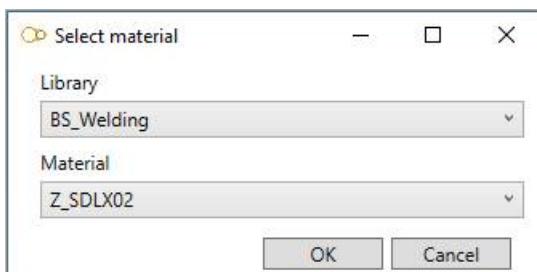
Weld

Flange throat	<input type="text"/> 5	Double flange	<input checked="" type="checkbox"/>
Web throat	<input type="text"/> 5	Material	<input type="text"/> Z_SDLX02 <input type="button"/> ...

Property	Description	Unit Metric	Unit USA
Flange throat	Apothem of the bead	mm	in



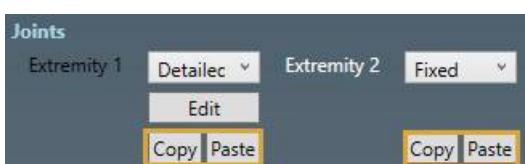
Click on the ... button to select a *Welding* material :



Click [here](#) for more information about database of materials.

1.5 Duplicate joint

The definition of a **joint** (fixed, detailed, bolted or welded) can be copied and pasted to the extremity of any beams :



WARNING

Don't forget to click to the **Modify** button to validate the duplication and see the result in 3D !

Click [here](#) to return to **beam** page.

Create nodes

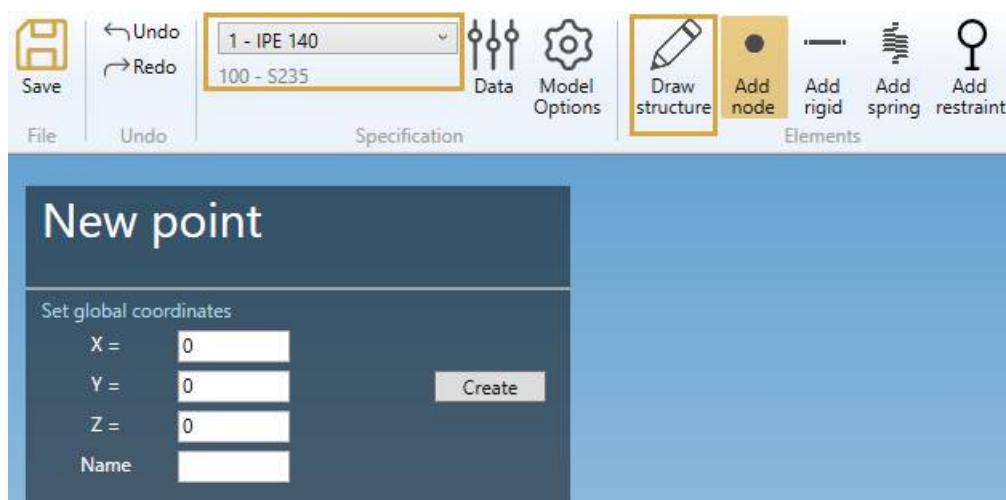
There are 3 possibilities of creation :

- Creation of the **FIRST** node
- Creation of a **GLOBAL** node
- Creation of a **RELATIVE** node

1. First node

The first node panel is called when a first **beam** needs a starting point.

Select a **section** and click on **Draw structure**, the **Add node** button is automatically selected :



Set the **GLOBAL** X, Y, Z coordinates and a name.

See **Global node** just below :

2. Global node

If the selection is **EMPTY**, click on the **Add node** button to create a new global node.

You can **Undo** this command.

Fill the **GLOBAL** X, Y, Z coordinates and a name.

Property	Description	Unit Metric	Unit USA
X	X global coordinate	m	ft
Y	Y global coordinate	m	ft
Z	Z global coordinate	m	ft
Name	Text or number	-	-

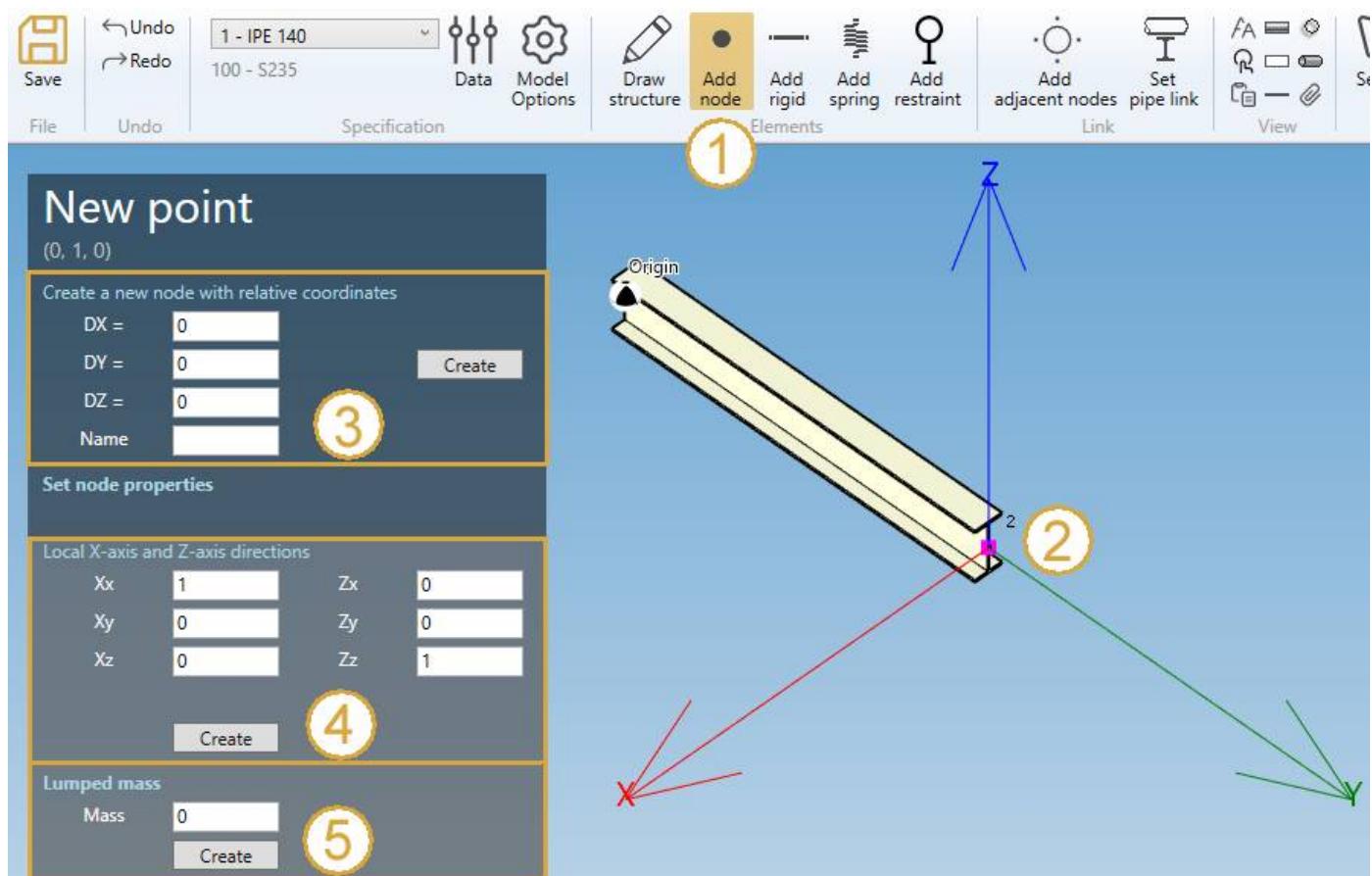
To know the UNIT of the value, just let the mouse over the cell.

If **Name** is blank, the new node will receive an automatic name based on the **Node settings**.

Click [here](#) for more information about node naming.

3. Relative node

If a node is selected, you can create a new node relative to it in **Global directions**:



- 1 Click on **Add node** button
- 2 Select a **node**.
- 3 Define the DX, DY, DZ, name of the new node.
- 4 Click on **Create**

You can **Undo** this command.

Click [here](#) for more information about the selection tool.

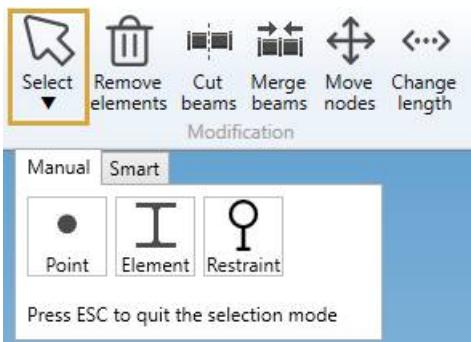
Property	Description	Unit Metric	Unit USA
DX	Relative distance from the selected node in X global direction	m	ft
DY	Relative distance from the selected node in Y global direction	m	ft
DZ	Relative distance from the selected node in Z global direction	m	ft
Name	Text or number	-	-

If **Name** is blank, the new node will receive an automatic name based on the **Node settings**.

Click [here](#) for more information about node naming.

4. Creation of a local coordinate system

You can define a new **local coordinate system** on selected node (4).



1 Click on **Select point** button

2 Select a **node**

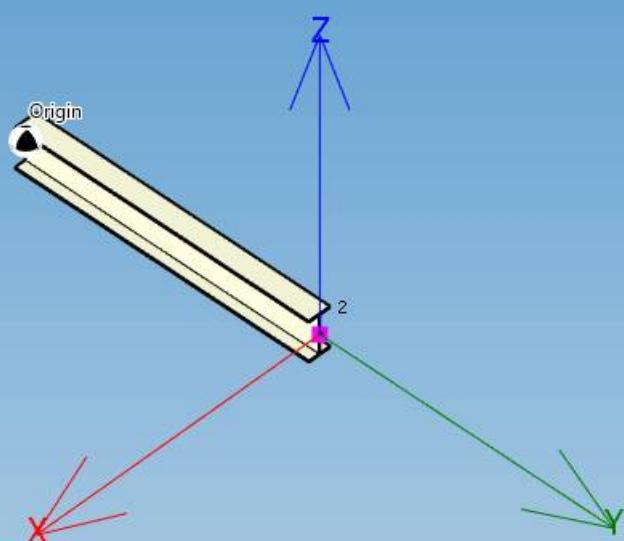
3 Define the local coordinate system

4 Click on **Create**

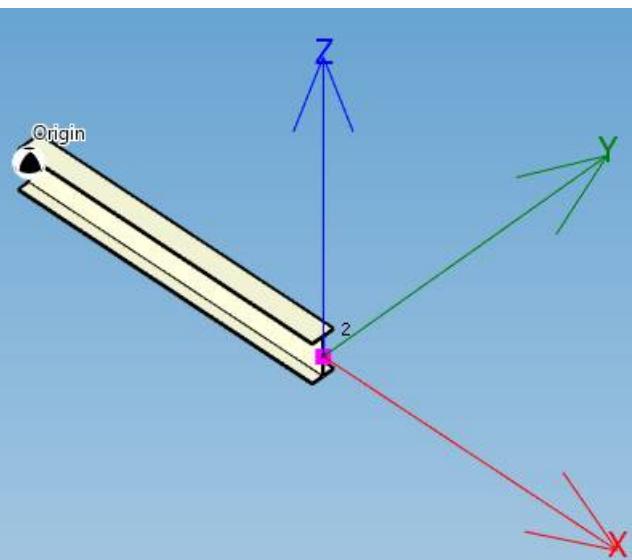
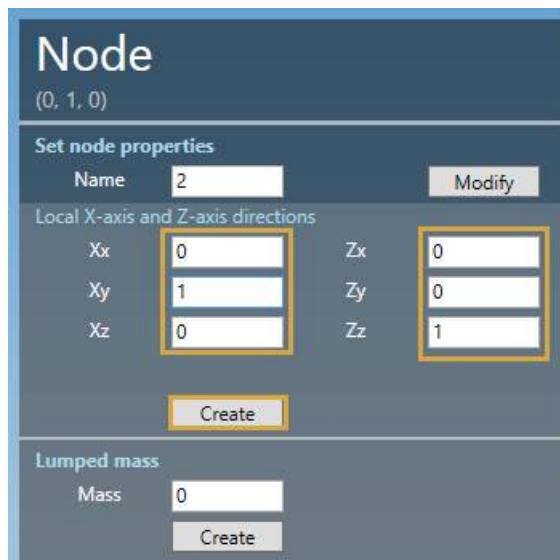
Just define the new **X** axis vector (X_x, X_y, X_z) in global coordinates and the new **Z** axis vector (Z_x, Z_y, Z_z). The **Y** axis will be automatically defined.

By default, the local coordinate system is equal to the global coordinate system.

Example :



Change the X direction = axis of the beam and Z direction unchanged :



Click on **Create** button.

You can **Undo** this command.

5. Creation of a lumped mass

You can define a **lumped mass** on selected node (5) :

Property	Unit Metric	Unit USA
Mass	ton	kips

Click on the **Create** button to add a new lumped mass to the node.

You can later modify or remove this object by selecting this node and click on **Modify/Remove** buttons.

You can **Undo** this command.

6. Modify a node

Change the **Selection mode** to POINT and select a node. Only the **Name** can be changed.

You can **Undo** this command.

Rigid

A rigid is a straight element sufficiently stiff to rigidly transmit all deflections and rotations.



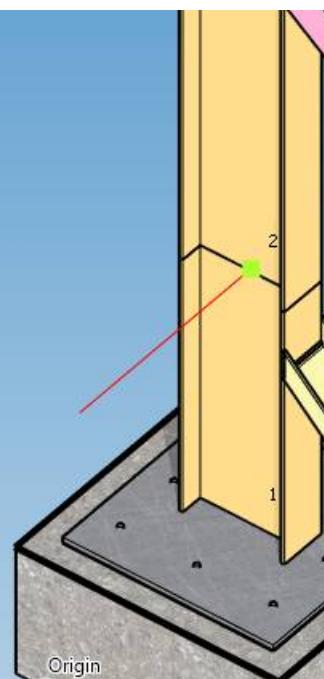
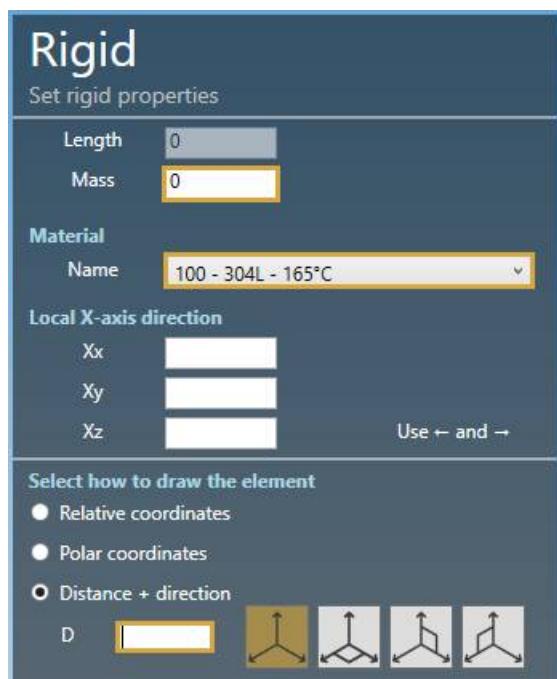
When you click on the **Add rigid** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

1. Create a rigid

- Select a **node**.
- Click the **Add rigid** button.



Property	Unit Metric	Unit USA
Length	m	ft
Mass	ton	kips

The length must be defined by the **orientation tool**.

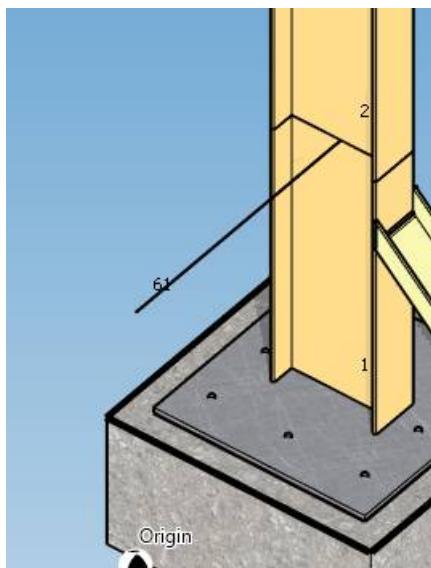
Click [here](#) for more information about the orientation tool.

Set the default material of the rigid.

You can define the **Local X-axis** vector by defining Xx, Xy, Xz in global coordinates.

You can then define the second point of the rigid thanks to the **Orientation tool**.

Create the rigid (represented by a black line) :



You can **undo** this command.

2. Modify/Remove a rigid

A rigid can also be used to simulate an eccentricity between beams.

Rigid
Set rigid properties

Length	0.105
Mass	0.1
Material	
Name	101 - S235JR
Local X-axis direction	
Xx	<input type="button"/>
Xy	<input type="button"/>
Xz	<input type="button"/>

Show/Hide **Modify** **Remove**

A 3D CAD model showing a rigid connection between two vertical yellow structural components. A pink arrow indicates the Local X-axis direction. Dimensions 40 and 45 are shown on the model.

Select a rigid, change some properties and click on the **Modify** button or delete it by clicking the **Remove** button.

You can **undo** this command.

Spring



When you click on the **Add spring** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

1. Create a spring

- Select a **node**.
- Click the **Add spring** button.

Spring

Set spring properties

Length

Material

Name

Local X-axis direction

Xx	<input style="width: 80%; height: 20px; border: 1px solid #ccc; border-radius: 5px; background-color: #f0f0f0;" type="text"/>	Use ← and →
Xy	<input style="width: 80%; height: 20px; border: 1px solid #ccc; border-radius: 5px; background-color: #f0f0f0;" type="text"/>	
Xz	<input style="width: 80%; height: 20px; border: 1px solid #ccc; border-radius: 5px; background-color: #f0f0f0;" type="text"/>	

Translation and rotation stiffness :

Kx	<input style="width: 80%; height: 20px; border: 1px solid #ccc; border-radius: 5px; background-color: #f0f0f0;" type="text"/>	Krx	<input style="width: 80%; height: 20px; border: 1px solid #ccc; border-radius: 5px; background-color: #f0f0f0;" type="text"/>
Ky	<input style="width: 80%; height: 20px; border: 1px solid #ccc; border-radius: 5px; background-color: #f0f0f0;" type="text"/>	Kry	<input style="width: 80%; height: 20px; border: 1px solid #ccc; border-radius: 5px; background-color: #f0f0f0;" type="text"/>
Kz	<input style="width: 80%; height: 20px; border: 1px solid #ccc; border-radius: 5px; background-color: #f0f0f0;" type="text"/>	Krz	<input style="width: 80%; height: 20px; border: 1px solid #ccc; border-radius: 5px; background-color: #f0f0f0;" type="text"/>

Select how to draw the element

Relative coordinates

Polar coordinates

Distance + direction

D

The length must be defined by the **orientation tool**.

Click [here](#) for more information about the orientation tool.

Set the default material of the spring.

You can define the **Local X-axis** vector by defining Xx, Xy, Xz in global coordinates.

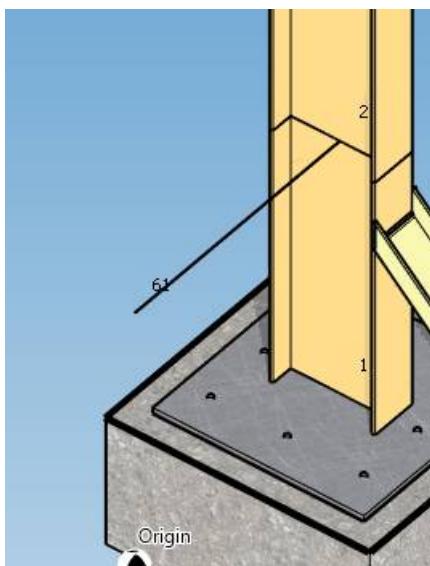
Stiffnesses :

Property	Description	Unit Metric	Unit USA
Length	-	m	ft
Kx	Translation stiffness in X direction	kN/mm	kips/in
Kx	Translation stiffness in Y direction	kN/mm	kips/in
Kx	Translation stiffness in Z direction	kN/mm	kips/in
Krx	Rotation stiffness in X direction	kN.m/rad	kips.ft/rad
Kry	Rotation stiffness in Y direction	kN.m/rad	kips.ft/rad
Krz	Rotation stiffness in Z direction	kN.m/rad	kips.ft/rad

Blank values mean that the corresponding degrees of freedom are blocked.

You can then define the second point of the spring thanks to the **Orientation tool**.

Create the spring (represented by a black line) :



You can **undo** this command.

2. Modify/Remove a spring

Select a spring, change some properties and click on the **Modify** button or delete it by clicking the **Remove** button.

You can **undo** this command.

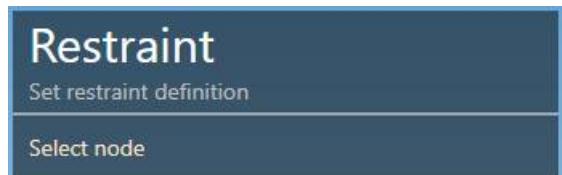
Restraint

Click on the **Add restraint** button :



Without selection, the left panel shows a message :

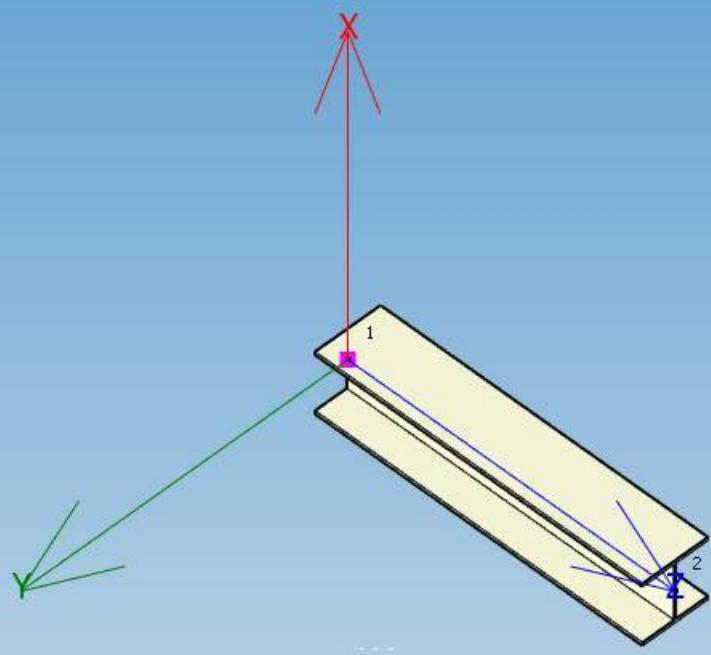
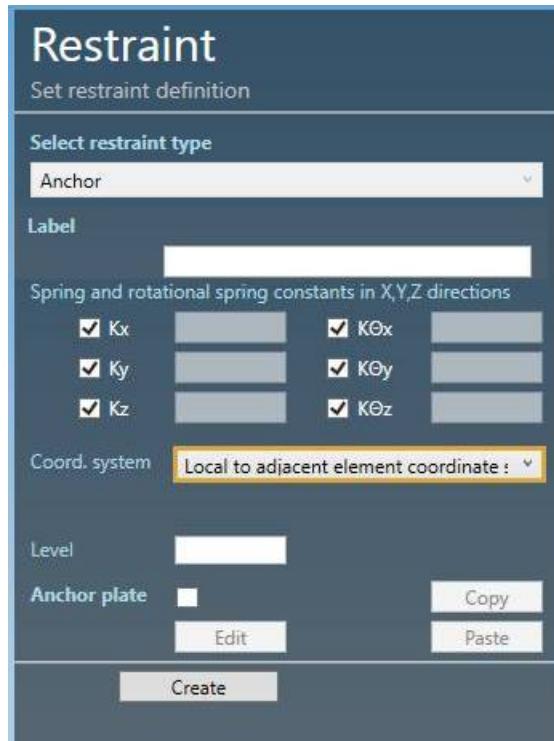
Select a node



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

1. Anchor

MetaStructure lets you only create anchors :



Property	Description	Unit Metric	Unit USA
Label	Label of the restraint	-	-
Kx	Spring Constant acting in X direction	kN/mm	Kips/in
Ky	Spring Constant acting in Y direction	kN/mm	Kips/in
Kz	Spring Constant acting in Z direction	kN/mm	Kips/in
Kθx	Rotational Spring Constant acting in X direction	kN.m/rad	Kips.ft/rad
Kθy	Rotational Spring Constant acting in Y direction	kN.m/rad	Kips.ft/rad
Kθz	Rotational Spring Constant acting in Z direction	kN.m/rad	Kips.ft/rad

To know the UNIT of the value, just move the mouse over the cell.

The labels are shown with the node names view button.

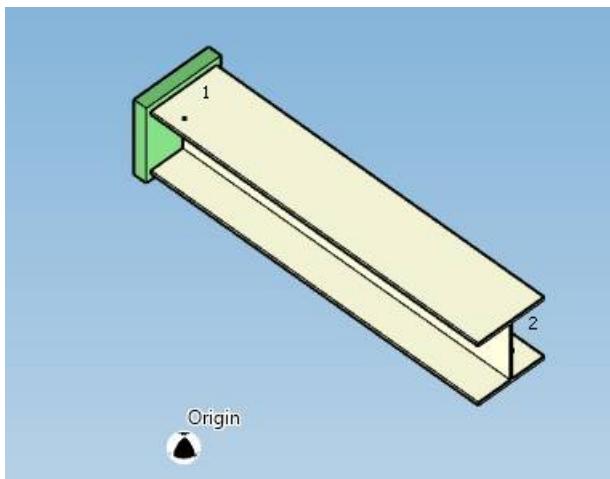
You can change the coordinate system that defines the directions X,Y,Z :

- Global (default)
- Local to adjacent element coordinate system
- Local to the node coordinate system (if exists on node)

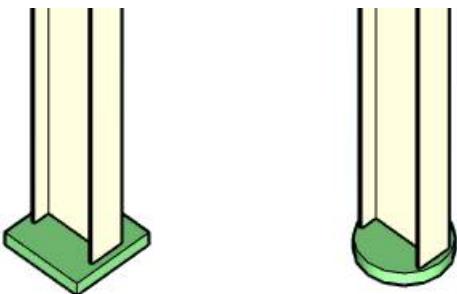
Click [here](#) for more information about new coordinate system on node.

The **Level** is used to identify the spectrum applied to this anchor for multilevel floor response spectrum analysis.

Set the coordinate system to **Local to adjacent element** and click the **Create** button to create an anchor :



If one or more **spring constants** are unchecked, the symbol of the anchor is changed from a box to a disk :

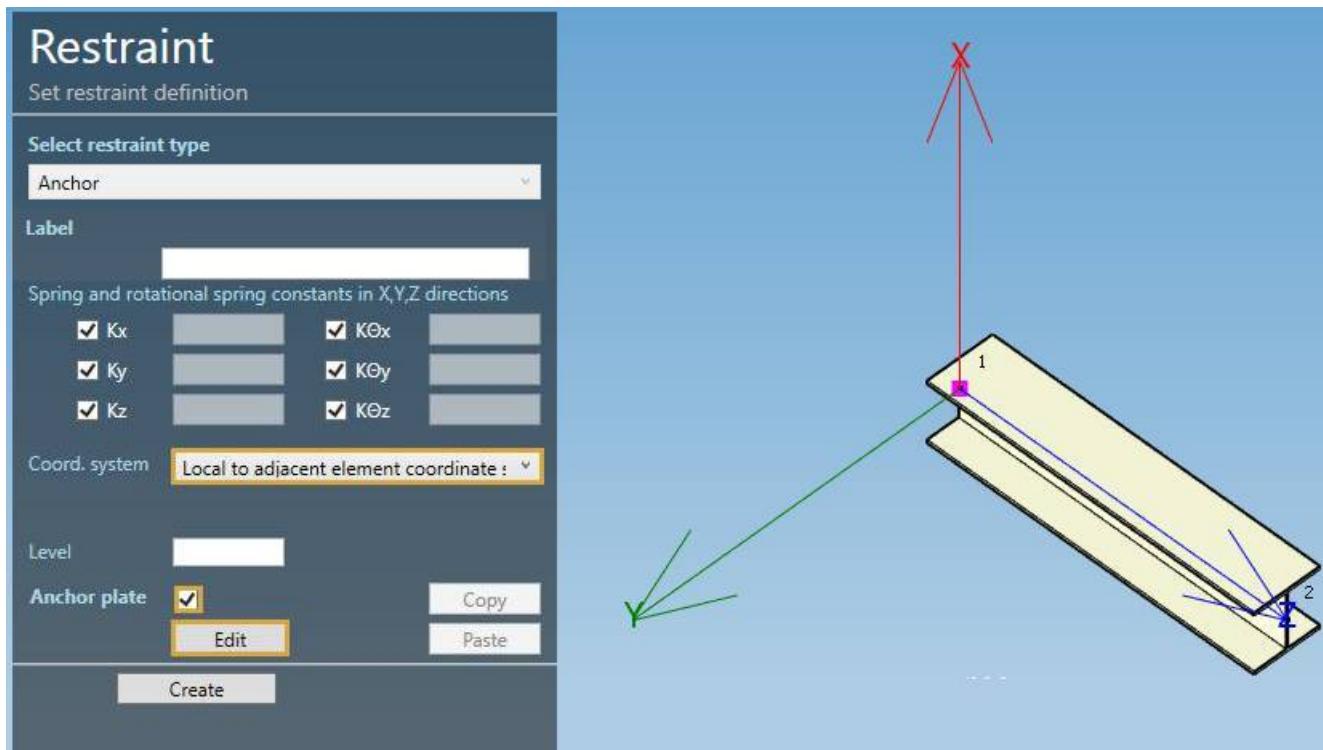


You can **Undo** this command.

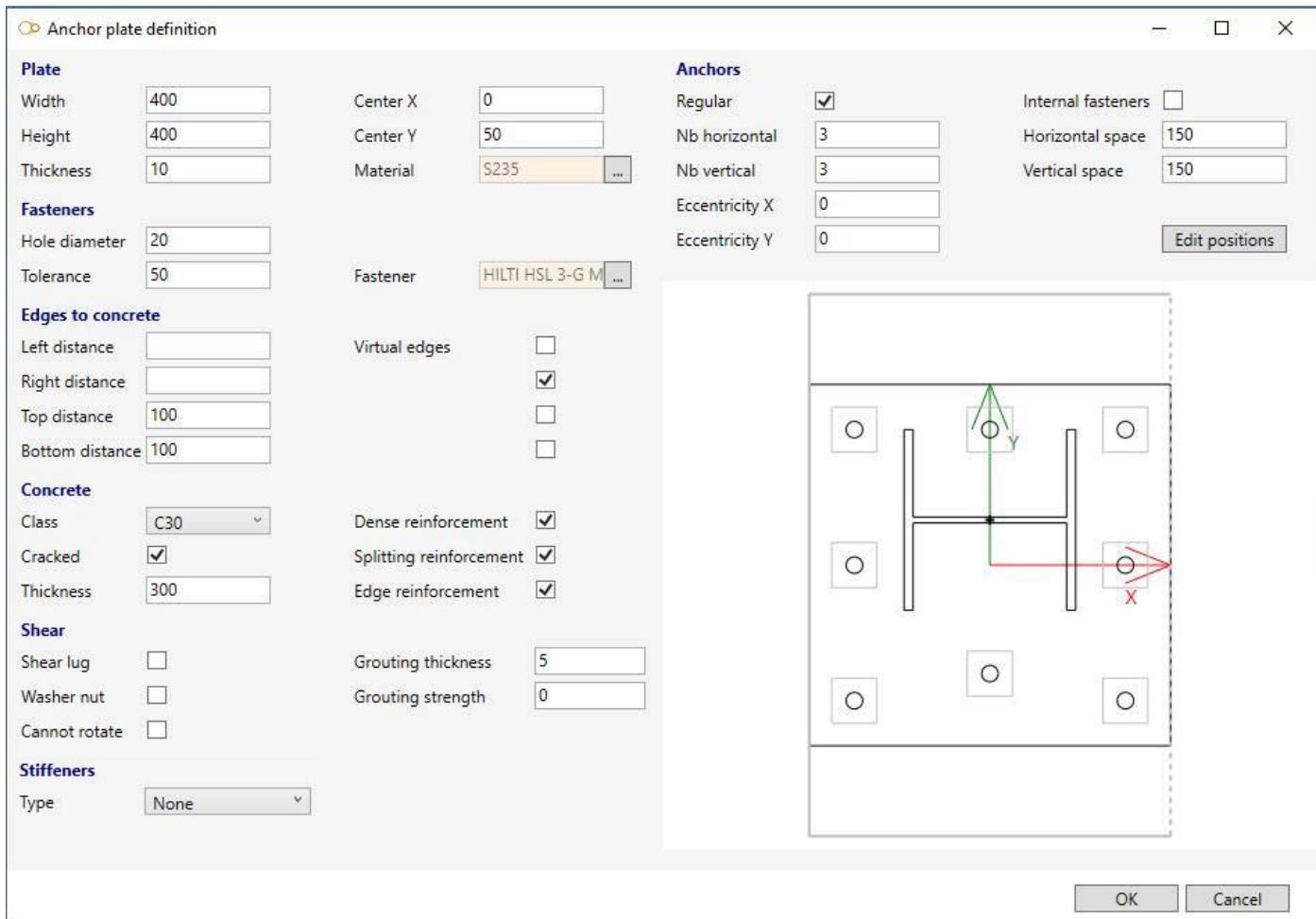
You can later modify or remove this anchor by selecting it and clicking on **Modify/Remove** buttons.

2. Anchor plate

Set the coordinate system to **Local to adjacent element** and check the **Anchor plate** cell :

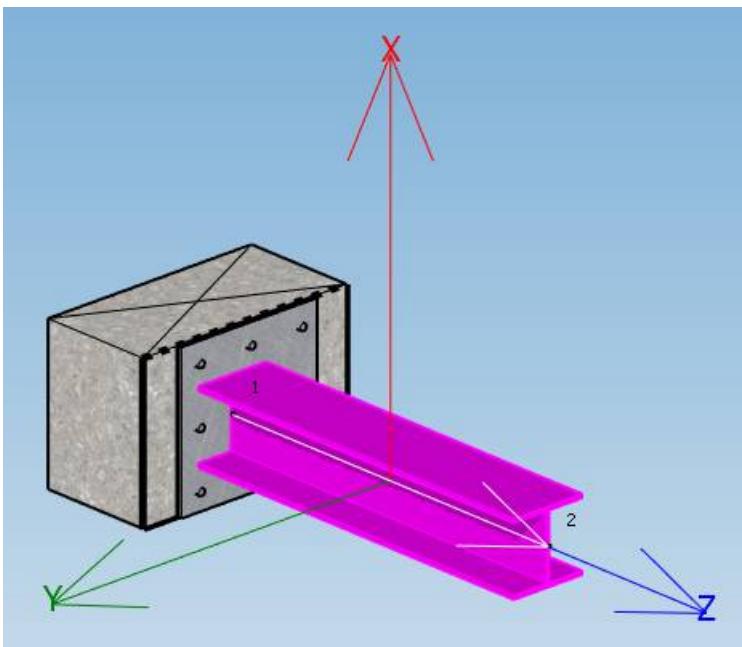


Press the **Edit** button and set the plate properties :

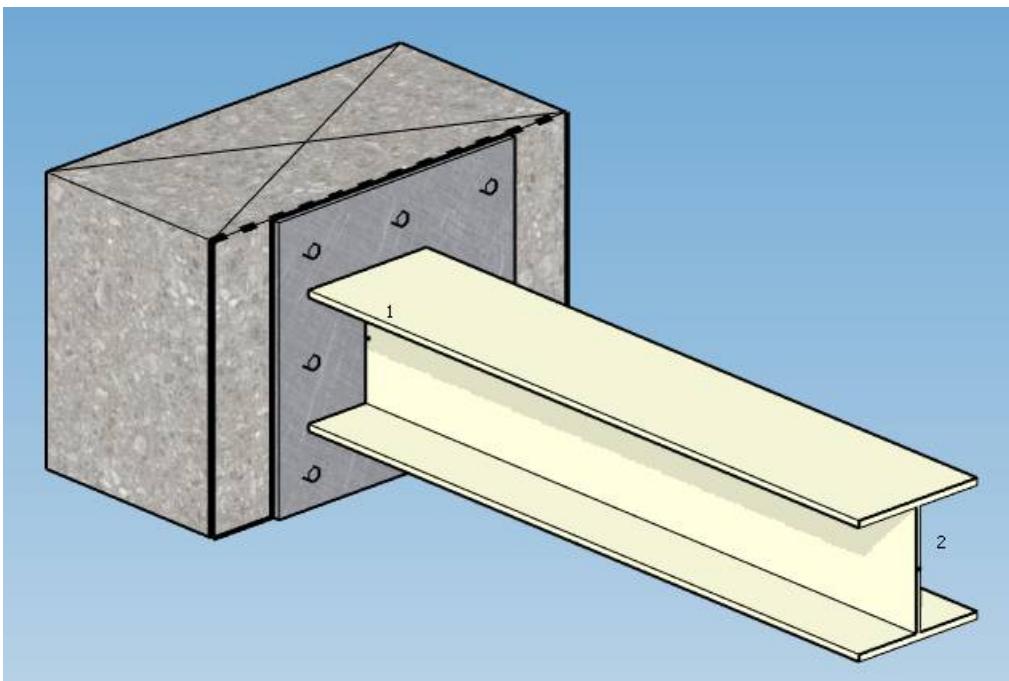


WARNING

Pay close attention to the axes ! The profile is rotated on the preview to match the X-axis of the plate !



Press **OK** to validate the anchor plate and press **Create** to validate the restraint :



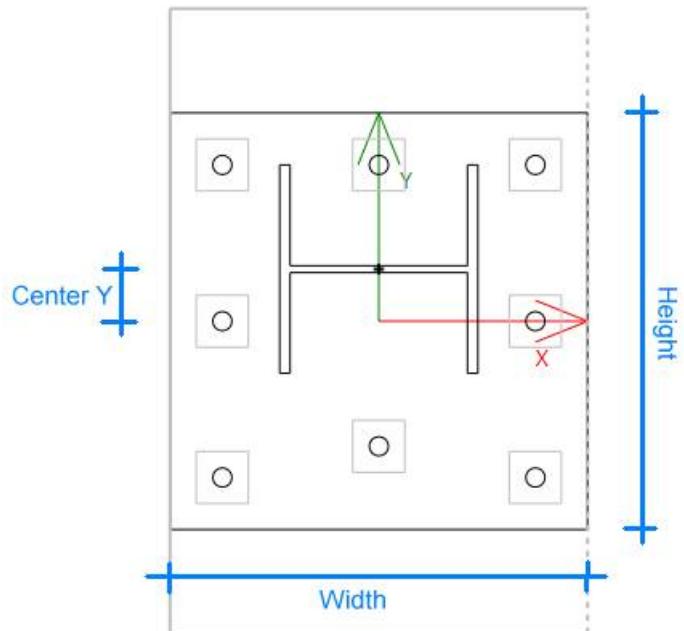
The resulting anchor plate is rotated to correspond to the actual X-axis of the beam ! The plate is perfectly represented with its thickness and bolts. The grouting is represented under the plate with its thickness. The concrete is represented with its dimensions and dashed lines and crosses on virtual edges.

2.1 Plate definition

Define the plate geometry and material :

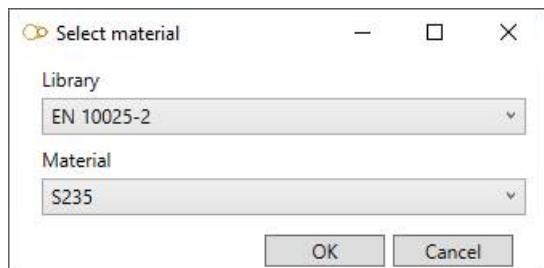
Plate	
Width	<input type="text" value="400"/>
Height	<input type="text" value="400"/>
Thickness	<input type="text" value="10"/>
Center X	<input type="text" value="0"/>
Center Y	<input type="text" value="50"/>
Material	S235 <input type="button" value="..."/>

Explanation of the fields on the picture :



Property	Description	Unit Metric	Unit USA
Width	Width of the plate	mm	in
Height	Height of the plate	mm	in
Thickness	Thickness of the plate	mm	in
Center X	Distance along X between center of plate and center of beam	mm	in
Center Y	Distance along Y between center of plate and center of beam	mm	in

Click on the ... button to select a *Structure* material :



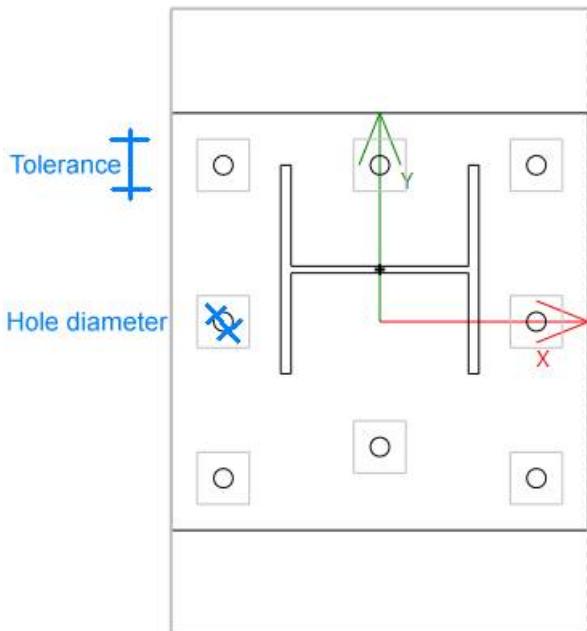
Click [here](#) for more information about database of materials.

2.2 Fastener definition

Define the type of fastener :

Fasteners	
Hole diameter	20
Tolerance	50
Fastener	HILTI HSL 3-G M ...

Explanation of the fields on the picture :



Property	Description	Unit Metric	Unit USA
Hole diameter	Diameter of the hole	mm	in
Tolerance	Anchor drilling area (uncertainty square side)	mm	in

Click on the ... button to select a *Fastener*:

Select fastener

Fasteners

- HILTI**
 - HDA-P M10
 - HDA-P M12
 - HDA-P M16
 - HDA-P M20
 - HDA-T M10
 - HDA-T M16
 - HDA-T M20
 - HILTI HSL 3-G M08
 - HILTI HSL 3-G M10
 - HILTI HSL 3-G M12
 - HILTI HSL 3-G M16**
 - HILTI HSL 3-G M20**
 - HILTI HSL 3-G M24
 - HILTI HST3 M10
 - HILTI HST3 M12
- SPIT**
 - SPIT TRIGA Z M06
 - SPIT TRIGA Z M08
 - SPIT TRIGA Z M10
 - SPIT TRIGA Z M12
 - SPIT TRIGA Z M16
 - SPIT TRIGA Z M20

Name	HILTI HSL 3-G M20																				
ETA reference	Ref. xxx valide jusqu'au XX/XX/XX																				
Type	Mechanical																				
Units	Metric																				
Dimensions	<table border="1"> <tr> <td>Diameter d</td> <td>External diam. dnrom</td> <td>Anchor depth hef</td> <td>Bearing area Ah</td> <td>Instal. factor yinst</td> </tr> <tr> <td>20</td> <td>28</td> <td>125</td> <td>1</td> <td></td> </tr> </table> Cracked concrete <table border="1"> <tr> <td>Min thickness hmin</td> <td>Min spacing smin</td> <td>for dist to edge c >=</td> <td>Min dist to edge cmif for spacing s >=</td> <td></td> </tr> <tr> <td>250</td> <td>125</td> <td>300</td> <td>150</td> <td>300</td> </tr> </table> <input checked="" type="checkbox"/> Same uncracked values	Diameter d	External diam. dnrom	Anchor depth hef	Bearing area Ah	Instal. factor yinst	20	28	125	1		Min thickness hmin	Min spacing smin	for dist to edge c >=	Min dist to edge cmif for spacing s >=		250	125	300	150	300
Diameter d	External diam. dnrom	Anchor depth hef	Bearing area Ah	Instal. factor yinst																	
20	28	125	1																		
Min thickness hmin	Min spacing smin	for dist to edge c >=	Min dist to edge cmif for spacing s >=																		
250	125	300	150	300																	
Steel	Tensile strength fuk: Yield strength fyk: Tension failure: NRks, Normal yms, Accidental yms Shear failure: VRks, Normal yms, Accidental yms Mounting: Without spacing k7, With spacing MRks, 0																				
Concrete cone	Tension failure: Ccr,n, Cracked k1, Uncracked k1, Normal ymc, Accidental ymc																				
Pull-out	Tension failure: Nrkp cracked C20/25, Nrkp uncracked C20/ Normal ymp, Accidental ymp																				

OK Cancel

Click [here](#) for more information about database of fasteners.

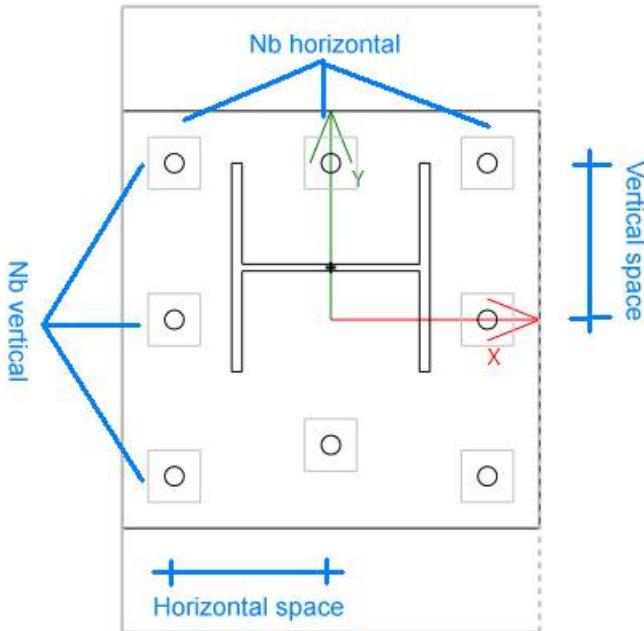
2.3 Anchor definition

Define the fastener arrangement :

Anchors

Regular <input checked="" type="checkbox"/>	Internal fasteners <input type="checkbox"/>
Nb horizontal <input type="text" value="3"/>	Horizontal space <input type="text" value="150"/>
Nb vertical <input type="text" value="3"/>	Vertical space <input type="text" value="150"/>
Eccentricity X <input type="text" value="0"/>	
Eccentricity Y <input type="text" value="0"/>	<input type="button" value="Edit positions"/>

Explanation of the fields on the picture :



Property	Description	Unit Metric	Unit USA
Space	Distance between 2 fasteners	mm	in
Eccentricity	Offset between center of plate and center of fasteners	mm	in

The **Regular** checkbox means that the fasteners are considered to be placed in a regular grid (even if there are small deviations), and are thus subject to regulatory checks.

The **Internal fasteners** checkbox adds internal fasteners inside the grid.

You have also the possibility to edit the position of every fastener and assign a reasonable offset in the case of quasi-regular configurations, subject to regulatory verifications :

Hole positions

ID	X	Y	[+]
1	-150	-150	[-]
2	0	-120	[-]
3	150	-150	[-]
4	-150	0	[-]
5	150	0	[-]
6	-150	150	[-]
7	0	150	[-]
8	150	150	[-]

Select a row and change the X, Y values

To know the UNIT of a property, just let the mouse over the property name.

Property	Unit Metric	Unit USA
X	mm	in
Y	mm	in

2.4 Edges to concrete definition

Define the distance between the plate and the free border of the concrete :

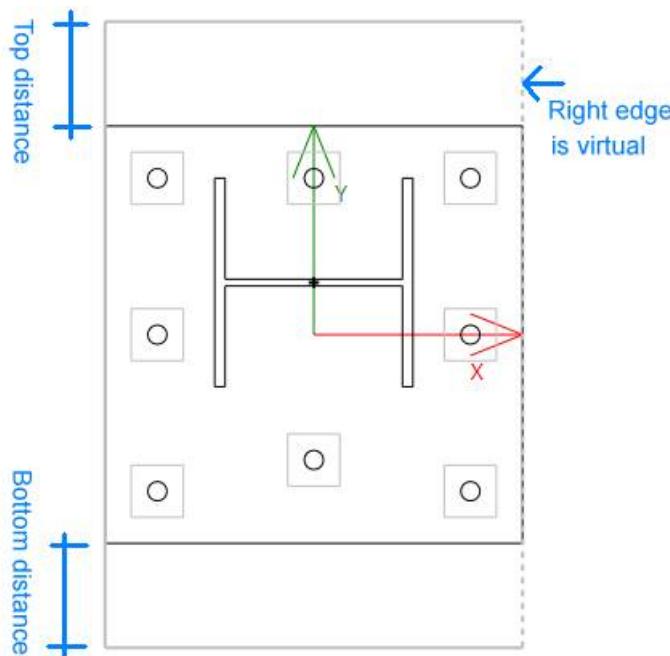
Edges to concrete	
Left distance	<input type="text"/>
Right distance	<input checked="" type="checkbox"/>
Top distance	100
Bottom distance	100
Virtual edges	
<input type="checkbox"/>	<input checked="" type="checkbox"/>
<input type="checkbox"/>	<input type="checkbox"/>
<input type="checkbox"/>	<input type="checkbox"/>

Blank value means that the plate is at an infinite distance from the free border of the concrete.

When a free edge is present, the fasteners are checked with additional failure modes : failure by splitting (tension) and edge failure (shear).

The **Virtual edges** checkboxes let you take account of the interaction between neighboring plates if they are too close to each other.

Explanation of the fields on the picture :



Property	Unit Metric	Unit USA
Distance	mm	in

2.5 Concrete definition

Define the concrete properties :

Concrete	
Class	C30
Cracked	<input checked="" type="checkbox"/>
Thickness	300
Dense reinforcement	<input checked="" type="checkbox"/>
Splitting reinforcement	<input checked="" type="checkbox"/>
Edge reinforcement	<input checked="" type="checkbox"/>

Choose between C20, C25, C30, C35, C40, C45, C50, C55, C60 classes, cracked or not, with or without reinforcements (dense, splitting, edge).

Property	Unit Metric	Unit USA
Thickness	mm	in

2.6 Shear and grouting definition

Shear

Shear lug	<input type="checkbox"/>	Grouting thickness	5
Washer nut	<input type="checkbox"/>	Grouting strength	0
Cannot rotate	<input type="checkbox"/>		N/mm ²

Property	Unit Metric	Unit USA
Thickness	mm	in
Strength	N/mm ²	ksi

Grouting: enter the thickness and strength of the mortar if leveling is planned. To model an anchor with spacing but no mortar, it suffices to define the thickness of the mortar equal to the spacing and the resistance equal to 0.

The **Shear lug** checkbox means that all the shear forces are assumed to be taken up by the lug. Only the bending moments and the normal force acting on the plate are considered for the calculation of the fasteners.

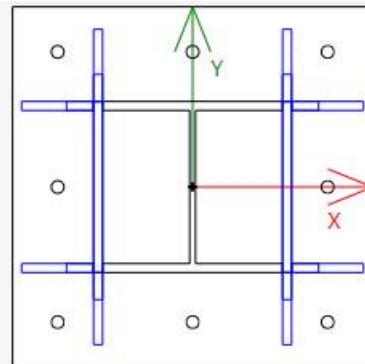
The **Washer nut** checkbox means that the bending moment in the anchor induced by shear can be reduced if a washer and a nut are directly attached to the concrete surface (in case of non-null grouting thickness).

The **Cannot rotate** checkbox means that the anchor plate is assumed to be free to rotate.

2.7 Stiffeners

Define stiffeners to reinforce the plate :

Stiffeners	
Type	Complete
Web thickness	10
Web +X length	80
Web +X top length	30
Web -X length	80
Web -X top length	30
Web bottom height	150
Height	300
Flange thickness	10
Flange +Y length	80
Flange +Y top length	30
Flange -Y length	80
Flange -Y top length	30
Flange bottom height	150



Types :

- None
- Web
- Flange
- WebFlange
- Special (only for channel and I)
- Complete (only for channel and I)

You can define the thickness, heights and the lengths of the stiffeners in both sides.

Property	Unit Metric	Unit USA
Length, height and thickness	mm	in

The axis are depending on the defined coordinate system :

- Global (default)
- Local to adjacent element coordinate system
- Local to the node coordinate system (if exists on node)

WARNING

The definition of stiffeners is only useful for **finite element analysis** !

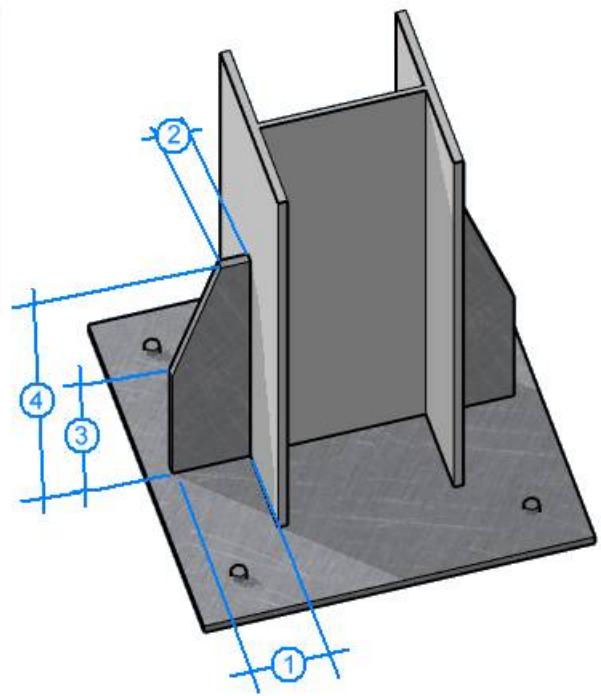
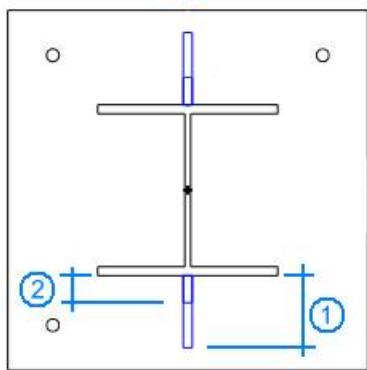
Click [here](#) for more information about *Finite Element Analysis* of anchor plates.

2.7.1 Web

The stiffeners are aligned with the web of the profile :

Stiffeners

Type	Web	Height	(4) 300
Web thickness	10		
Web +X length	80		
Web +X top length	30		
Web -X length	80	①	
Web -X top length	30	②	
Web bottom height	150	③	



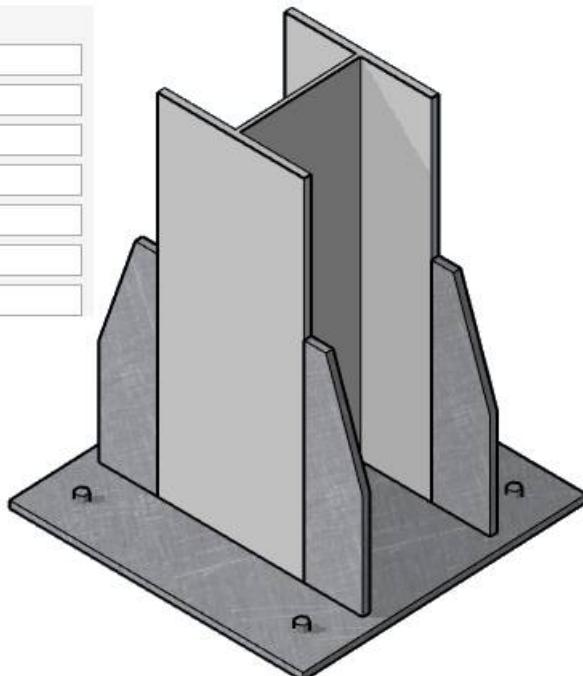
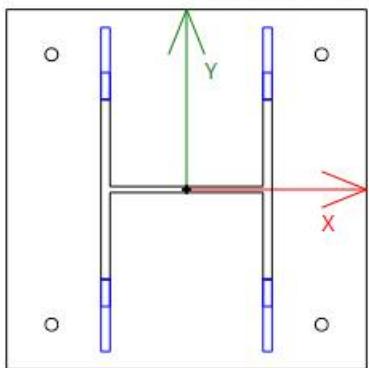
X,Y axis based on local to adjacent element coordinate system in the picture

2.7.2 Flange

The stiffeners are aligned with the flanges of the profile :

Stiffeners

Type	Flange	Height	300
Flange thickness	10		
Flange +Y length	80		
Flange +Y top length	30		
Flange -Y length	80		
Flange -Y top length	30		
Flange bottom height	150		



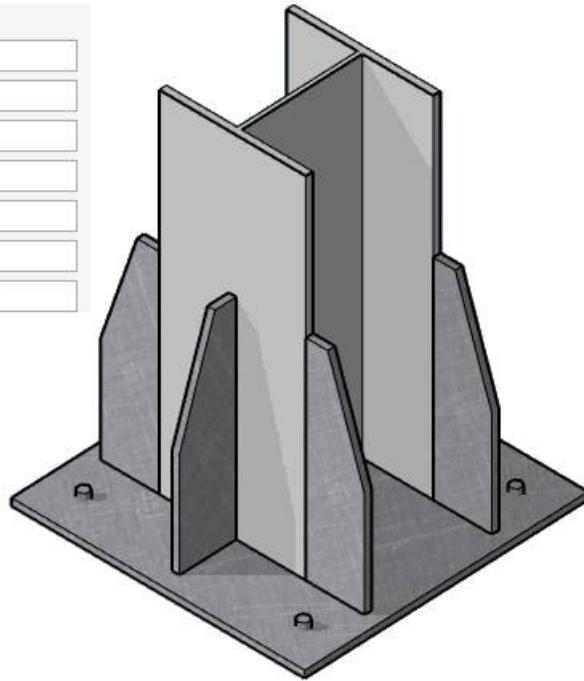
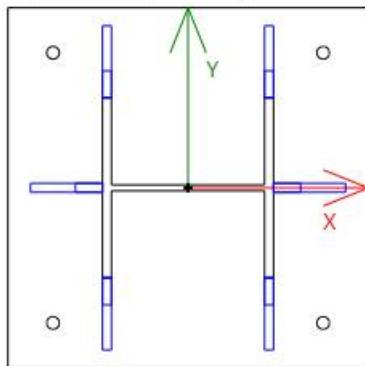
X,Y axis based on local to adjacent element coordinate system in the picture

2.7.3 Web + Flange

The stiffeners are aligned with the web and the flanges of the profile :

Stiffeners

Type	WebFlange	Height	300
Web thickness	10	Flange thickness	10
Web +X length	80	Flange +Y length	80
Web +X top length	30	Flange +Y top length	30
Web -X length	80	Flange -Y length	80
Web -X top length	30	Flange -Y top length	30
Web bottom height	150	Flange bottom height	150



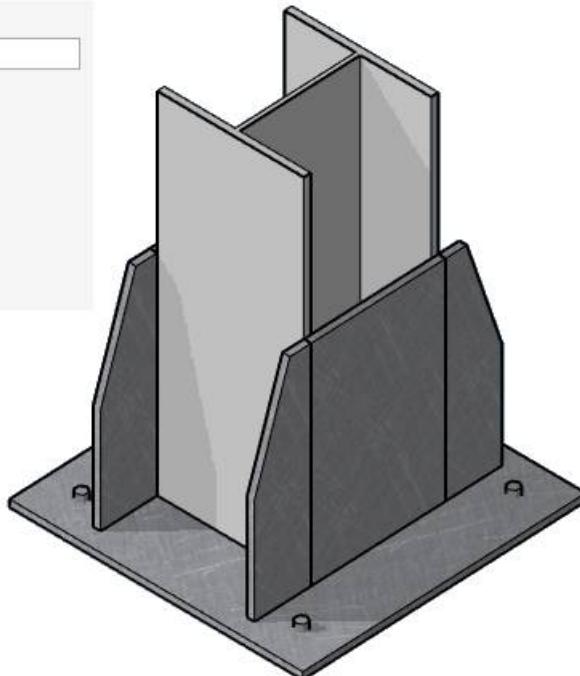
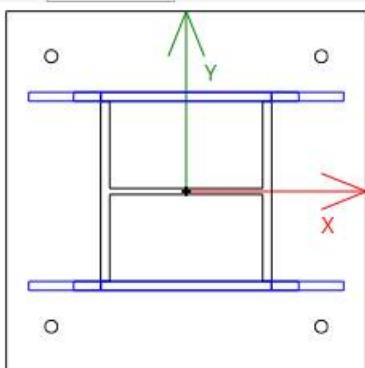
X,Y axis based on local to adjacent element coordinate system in the picture

2.7.4 Special (only for channel and I profile)

The stiffeners close the profile with a plate :

Stiffeners

Type	Special	Height	300
Web thickness	10		
Web +X length	80		
Web +X top length	30		
Web -X length	80		
Web -X top length	30		
Web bottom height	150		



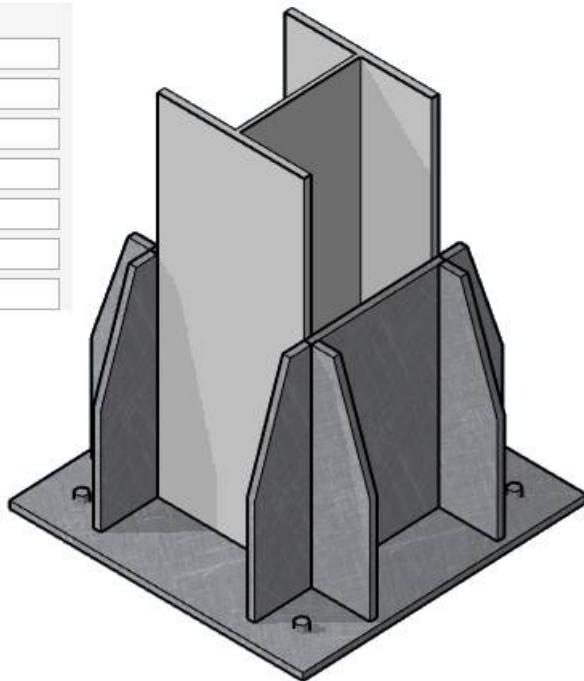
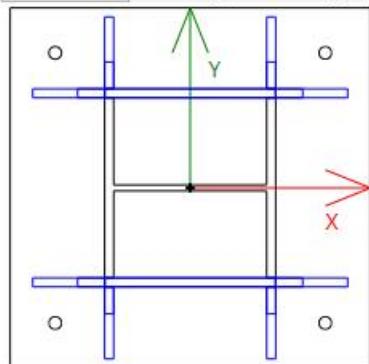
X,Y axis based on local to adjacent element coordinate system in the picture

2.7.5 Complete (only for channel and I profile)

A combination of Special and Flange stiffeners :

Stiffeners

Type	Complete	Height	300
Web thickness	10	Flange thickness	10
Web +X length	80	Flange +Y length	80
Web +X top length	30	Flange +Y top length	30
Web -X length	80	Flange -Y length	80
Web -X top length	30	Flange -Y top length	30
Web bottom height	150	Flange bottom height	150



X,Y axis based on local to adjacent element coordinate system in the picture

2.8 Anchor plate duplication

The definition of an **anchor plate** can be copied and pasted to another restraint :

Restraint

Set restraint definition

Select restraint type : **Anchor**

Spring and rotational spring constants in X,Y,Z directions :

<input checked="" type="checkbox"/> Kx	<input checked="" type="checkbox"/> KOx
<input checked="" type="checkbox"/> Ky	<input checked="" type="checkbox"/> KOy
<input checked="" type="checkbox"/> Kz	<input checked="" type="checkbox"/> KOz

Coord. system : Local to adjacent element coordinate :

Level :

Anchor plate : Copy Paste

Edit Modify Remove

Copy the anchor plate from the selected restraint

Restraint

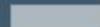
Set restraint definition

Select restraint type

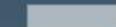
Anchor

Spring and rotational spring constants in X,Y,Z directions

Kx



Kθx



Ky



Kθy



Kz



Kθz

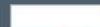


Coord. system

Local to adjacent element coordinate :



Level



Anchor plate



3

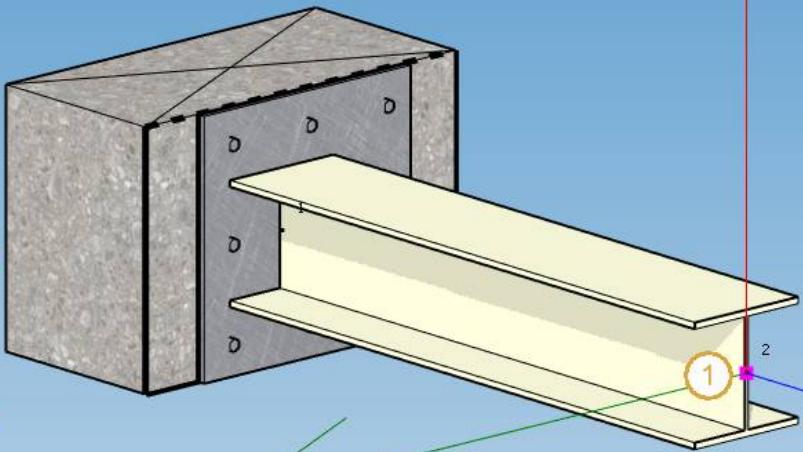
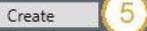
Edit

Copy

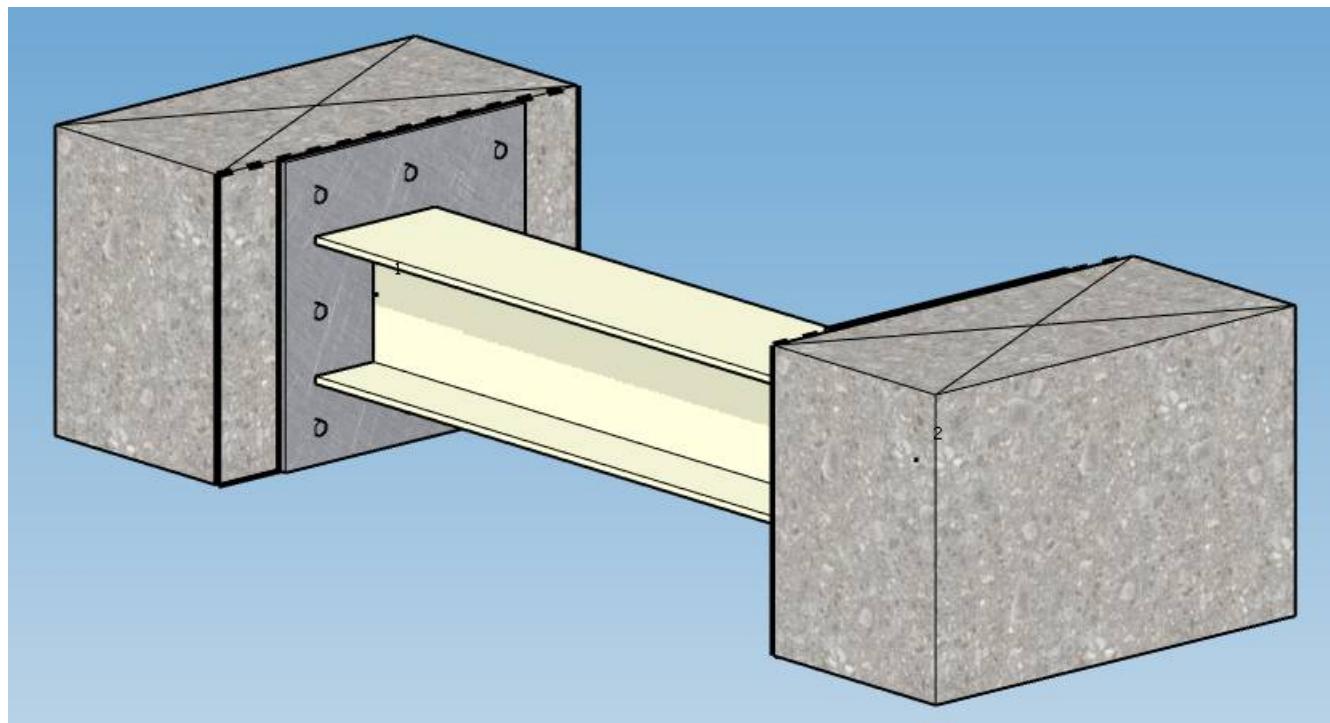
Paste

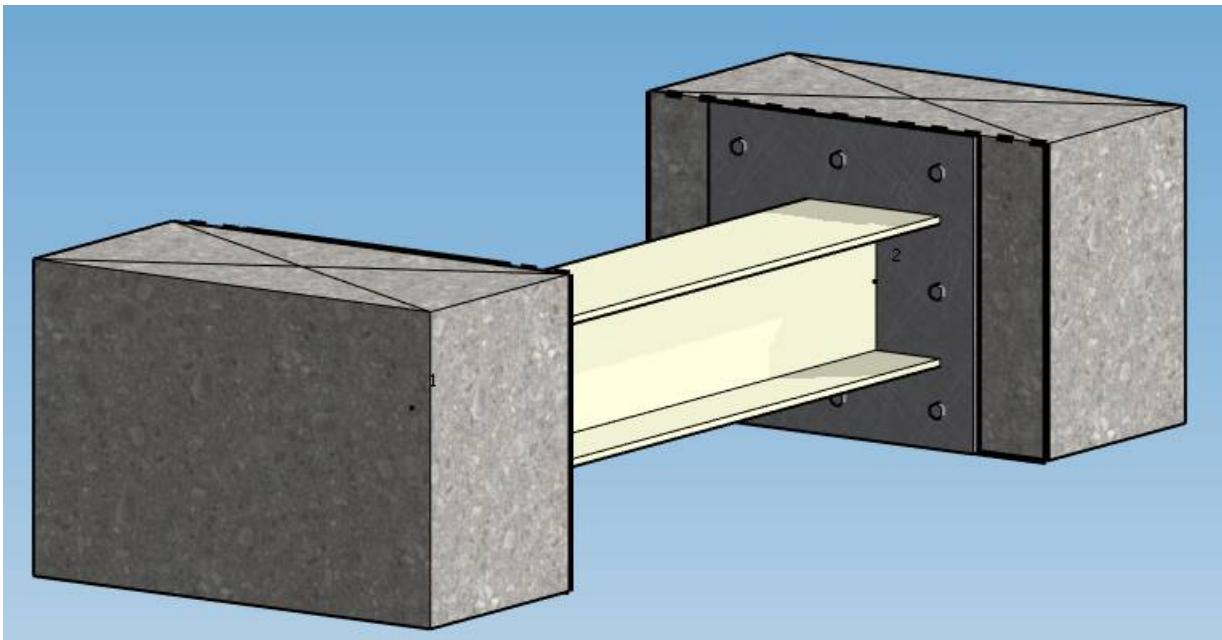


Create



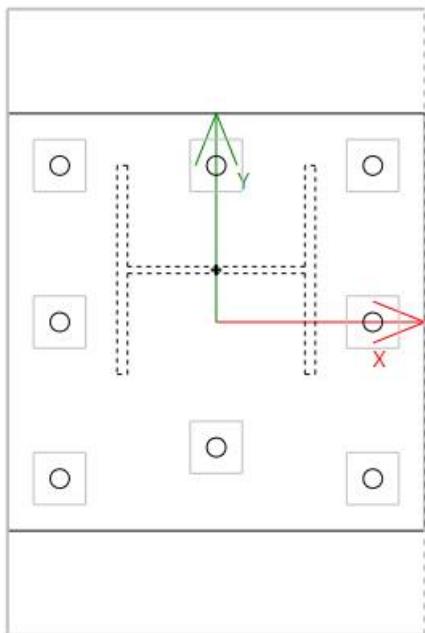
Press Add restraint, select a node with a beam (1), change coordinate system to "Local to adjacent element" (2), check "Anchor plate" (3), click on "Paste" (4) and finally click on "Create" :





WARNING

ATTENTION, unlike the previous preview of the plate, this new one show the beam in dashed lines. This means that the preview is looking in opposite direction (from outside -> concrete -> plate -> beam) :

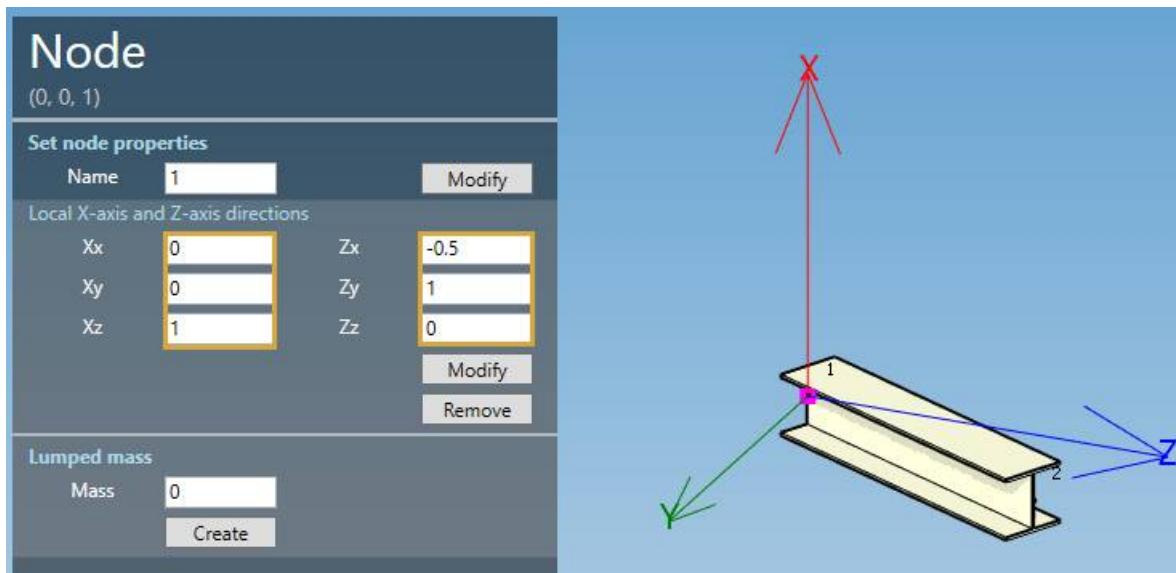


2.9 User defined local coordinate system

It is also possible to create an anchor plate that follows user defined local coordinate system.

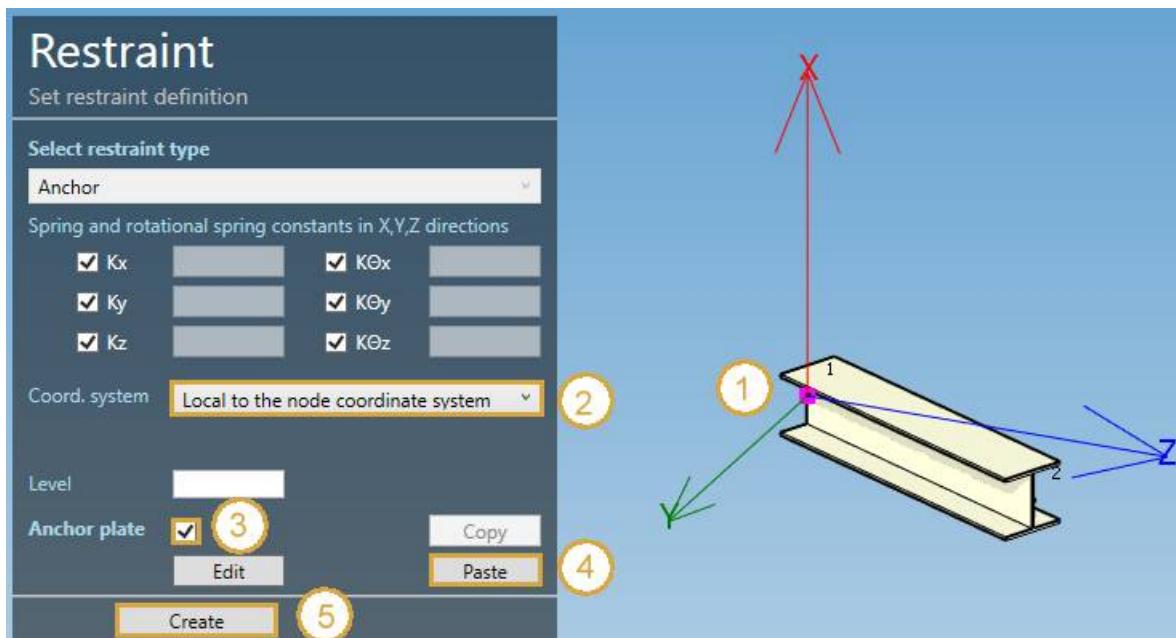
Example :

Select the node where the anchor plate will be placed and create a local coordinate system :

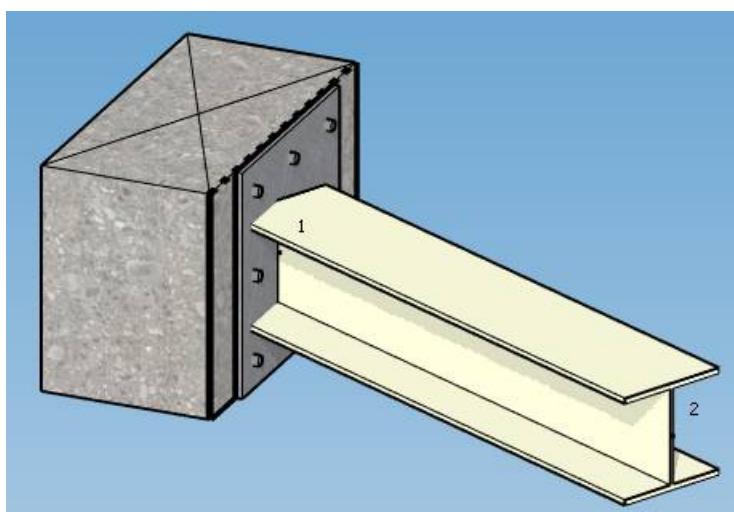


WARNING

Set the X-axis equal to the beam X-axis (vertical in this example)



Press Add restraint, select a node with a beam (1), change coordinate system to "Local to node coordinate system" (2), check "Anchor plate" (3), click on "Paste" (4) and finally click on "Create" :



For better visual result, select the beam and change the first graphical ending to **Plate** :

Beam

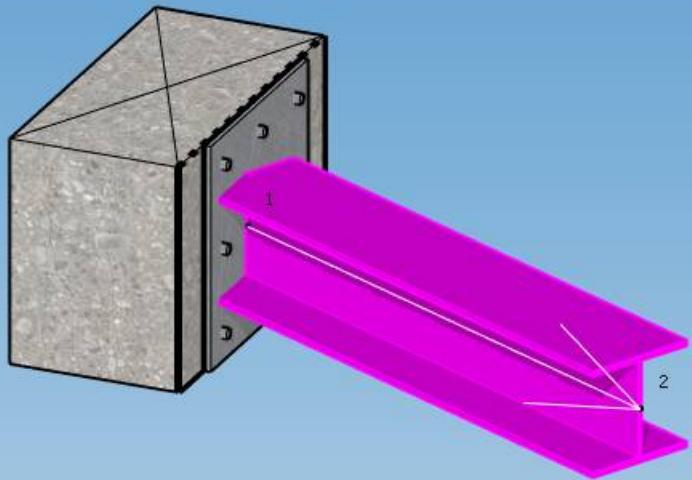
Set beam properties

Section

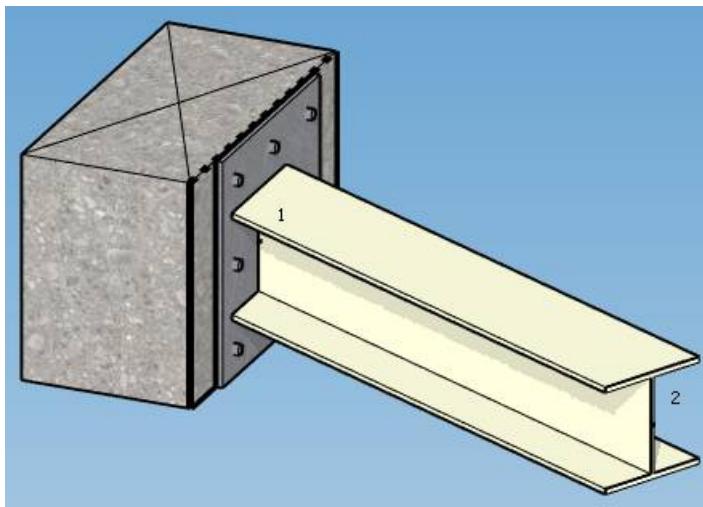
Current	HEA 200	
h	190	
b	200	
tw	6.5	
tf	10	

Graphical ending

Extremity 1	Plate	Extremity 2	None
Beam 1		Beam 2	
Offset 1	0	Offset 2	0

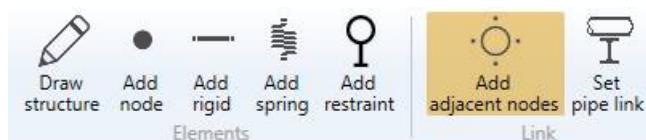


Click on the **Modify** button :



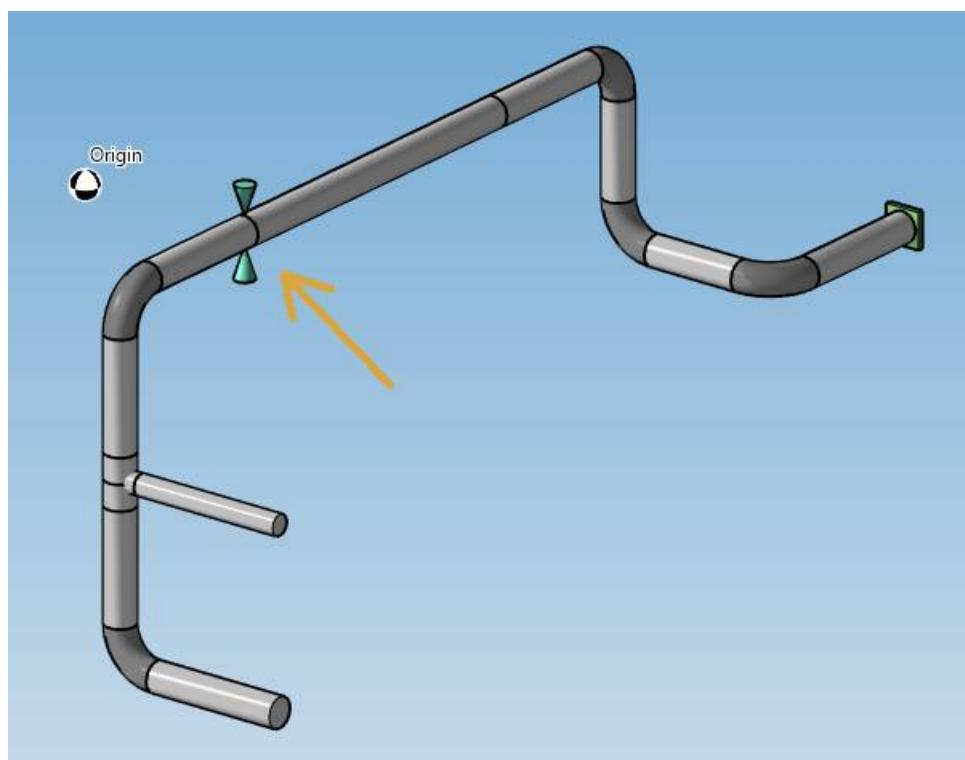
Click [here](#) for more information about the graphical endings.

Adjacent nodes



This tool is useful to construct *structure* nodes around *piping* nodes.

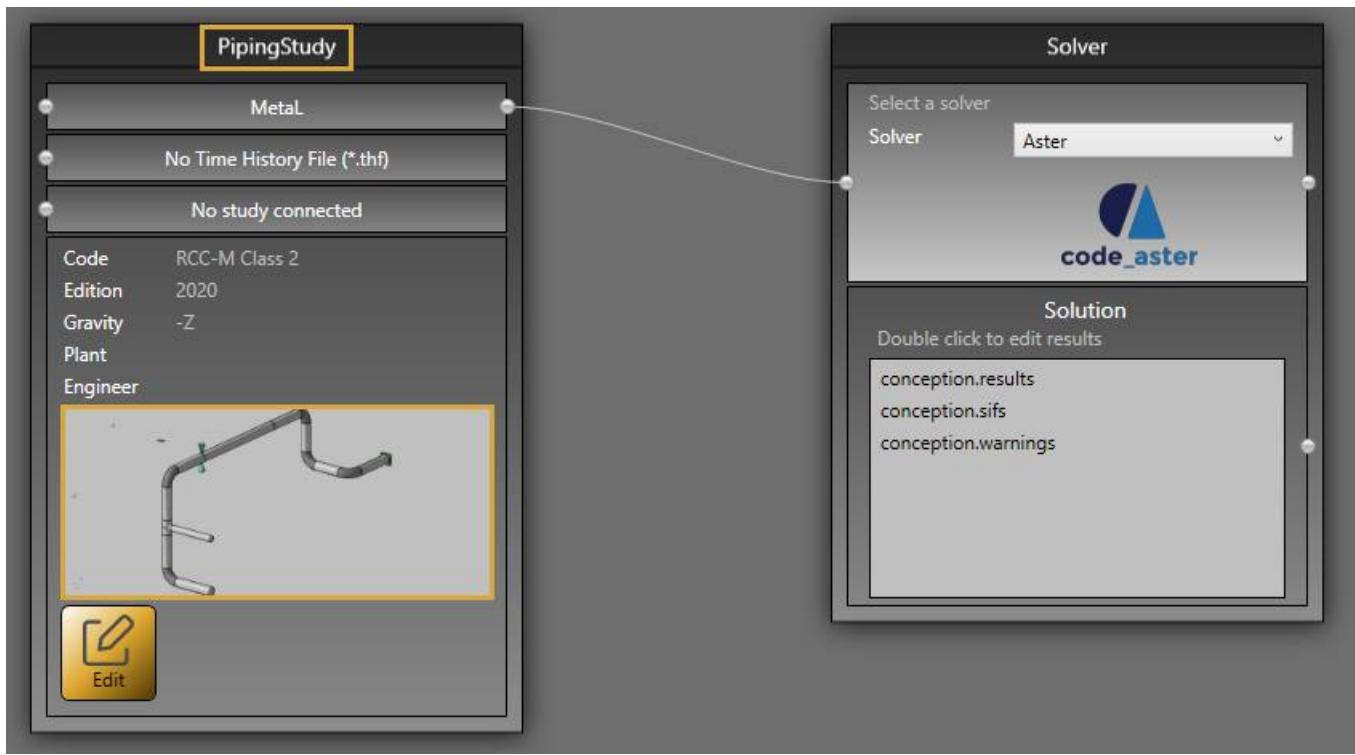
Imagine a small piping system with a **restraint** at a node :



We want to build a **support** at this node.

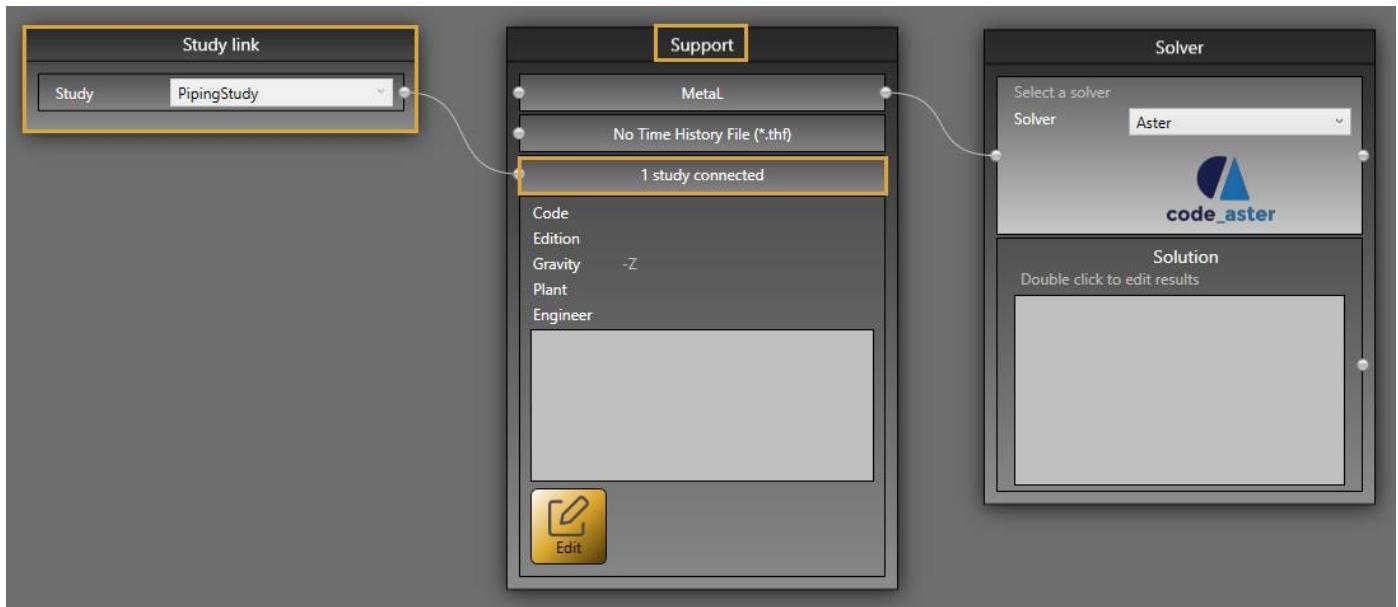
1. Piping

Create a **piping** study and draw the model :



2. Support

Create a **structure** study and add a **link** to *PipingStudy*:

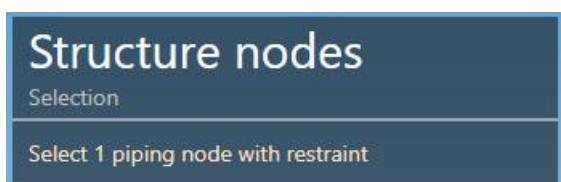


Set the calculation code, add materials and beams.

3. Structure nodes

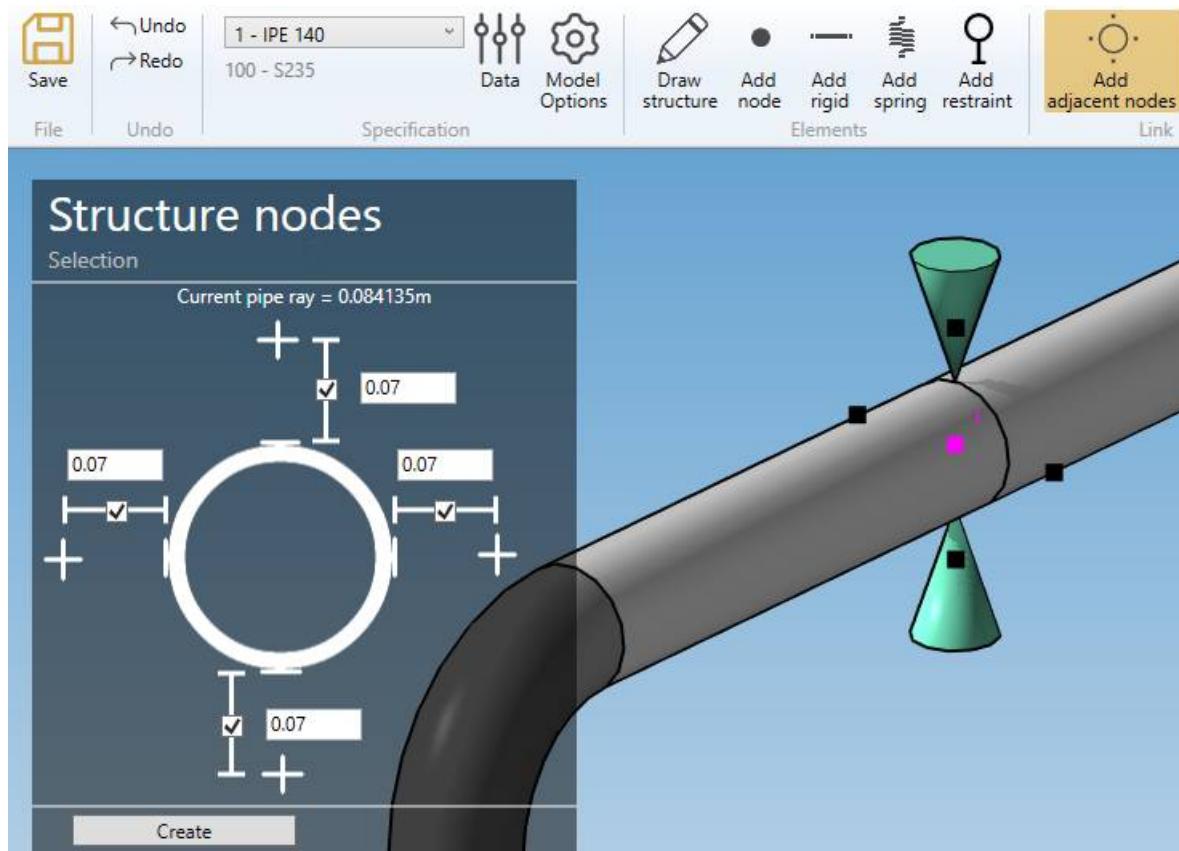


When you click on the **Add adjacent nodes** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

Select a piping node with a restraint :

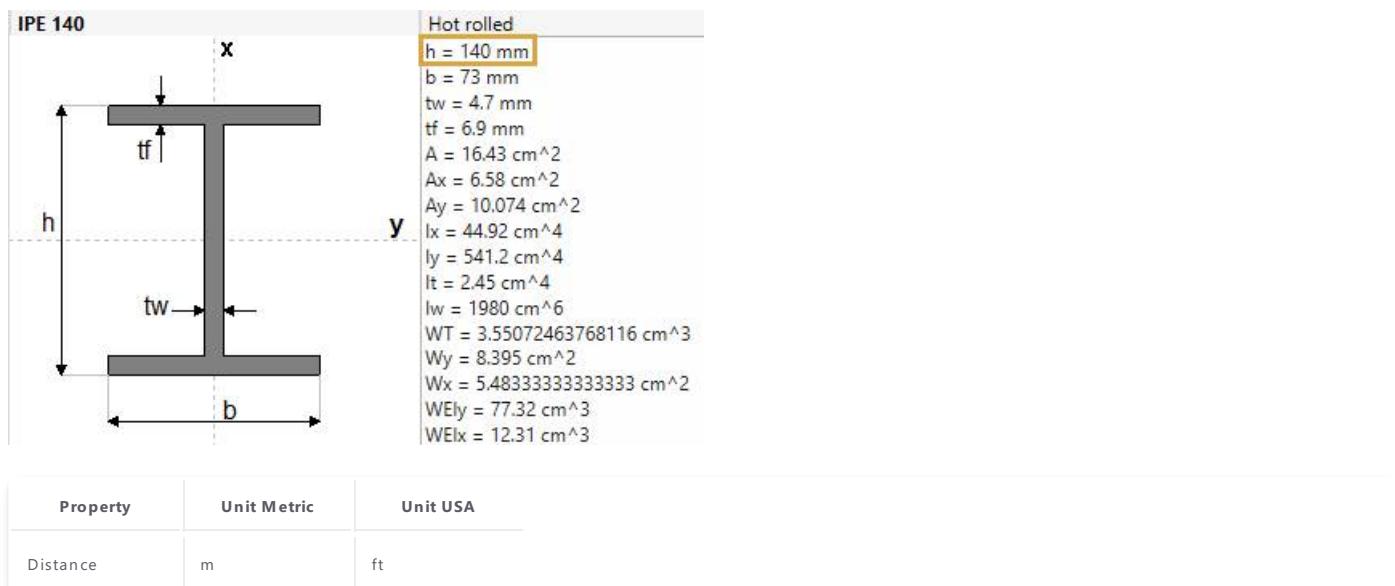


MetaStructure proposes to create up to 4 nodes perpendicular to the pipe direction.

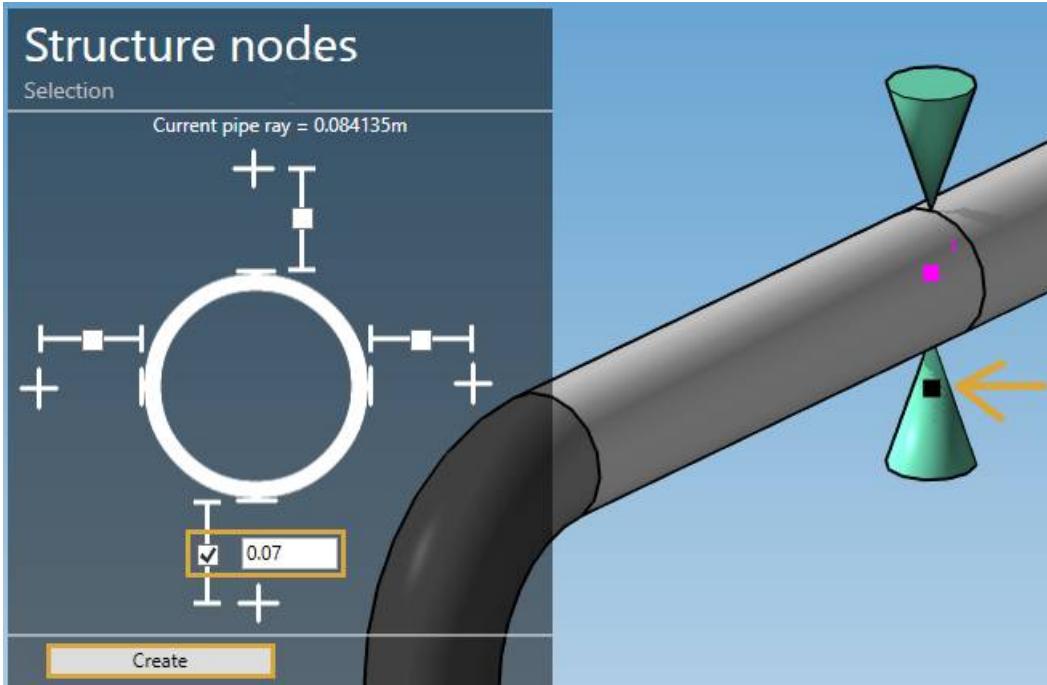
Checkboxes let you select the desired nodes.

The left panel shows the pipe radius at the selected node and for the 4 directions, the distance of the new nodes.

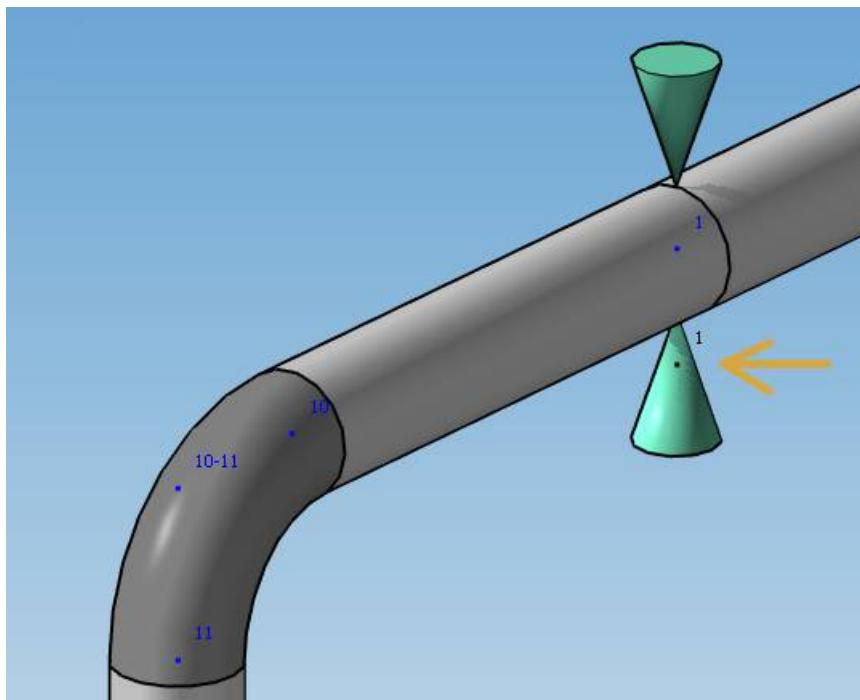
The default value (0.07 in this example) corresponds to half the height of the current beam (IPE 140) :



Select only the bottom node and click on the **Create** button :

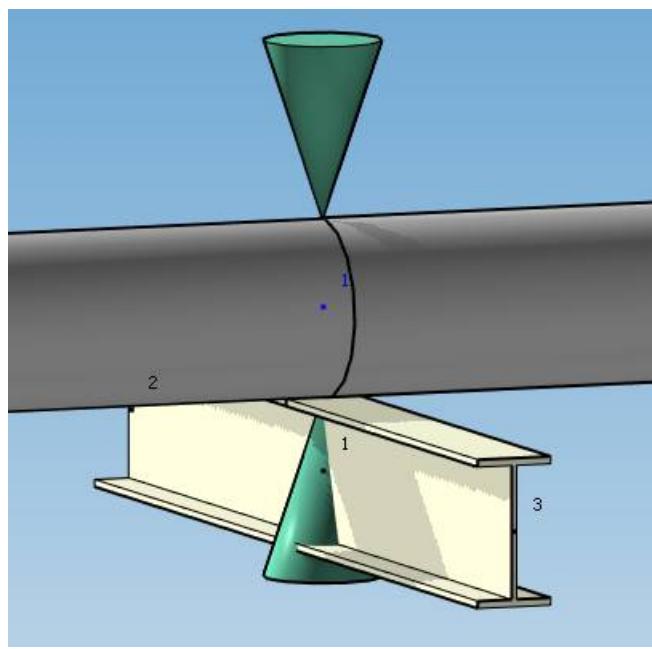


The **structure** node 1 is created :



The piping nodes are numbered in blue

When you later draw beams at this new node, you will see that the profile is perfectly adjacent to the pipe :



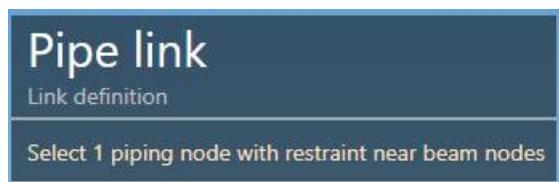
Pipe link



This tool creates a **Link** between one **piping** node and multiple **structure** nodes.

This link is necessary for load definition (External load) and during analysis.

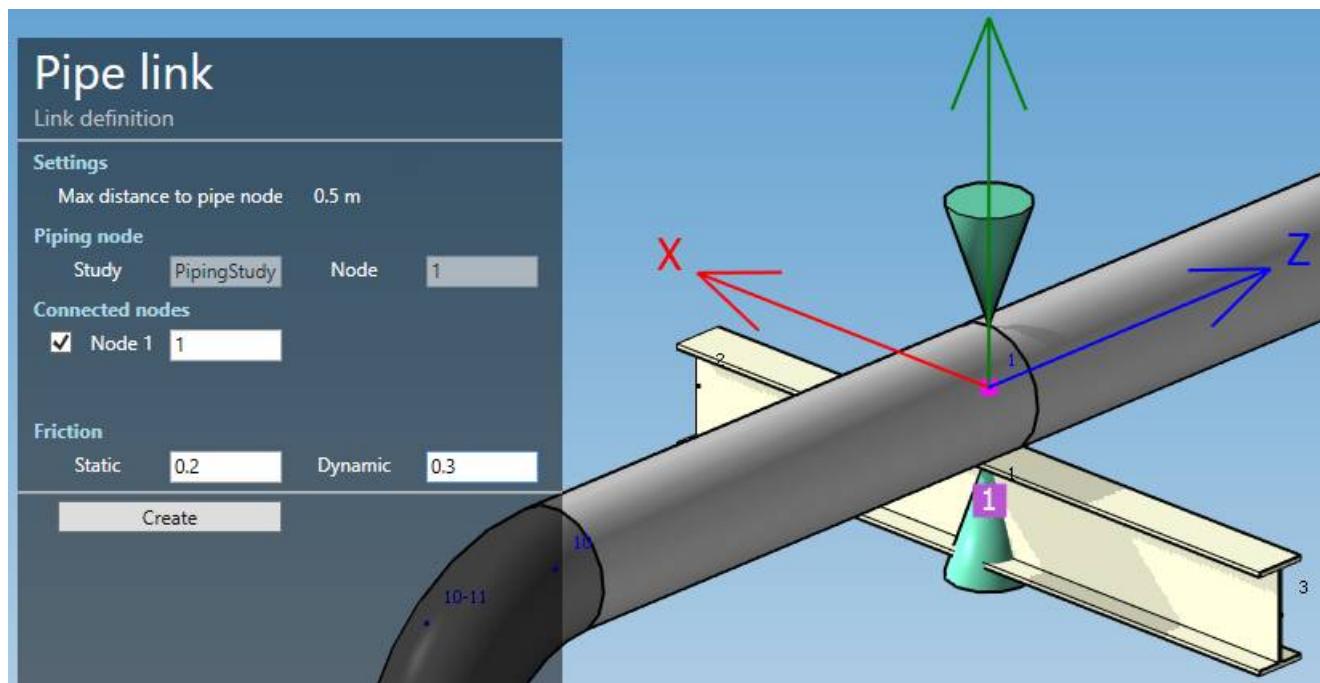
When you click on the **Set pipe link** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select a node.

1. Create a link

Select a piping node with a restraint and beam nodes around :



MetaStructure looks for all beam nodes around the selected pipe node at a distance smaller than a default one (0.5 m in this example) and perpendicular to the pipe direction.

Click [here](#) for more information about the settings.

It shows also the linked study name (PipingStudy) and the piping node name (1).

In this example, MetaStructure detects only one connected node :

```
Node 1 = 1
```

You can uncheck some connected nodes or change the node name.

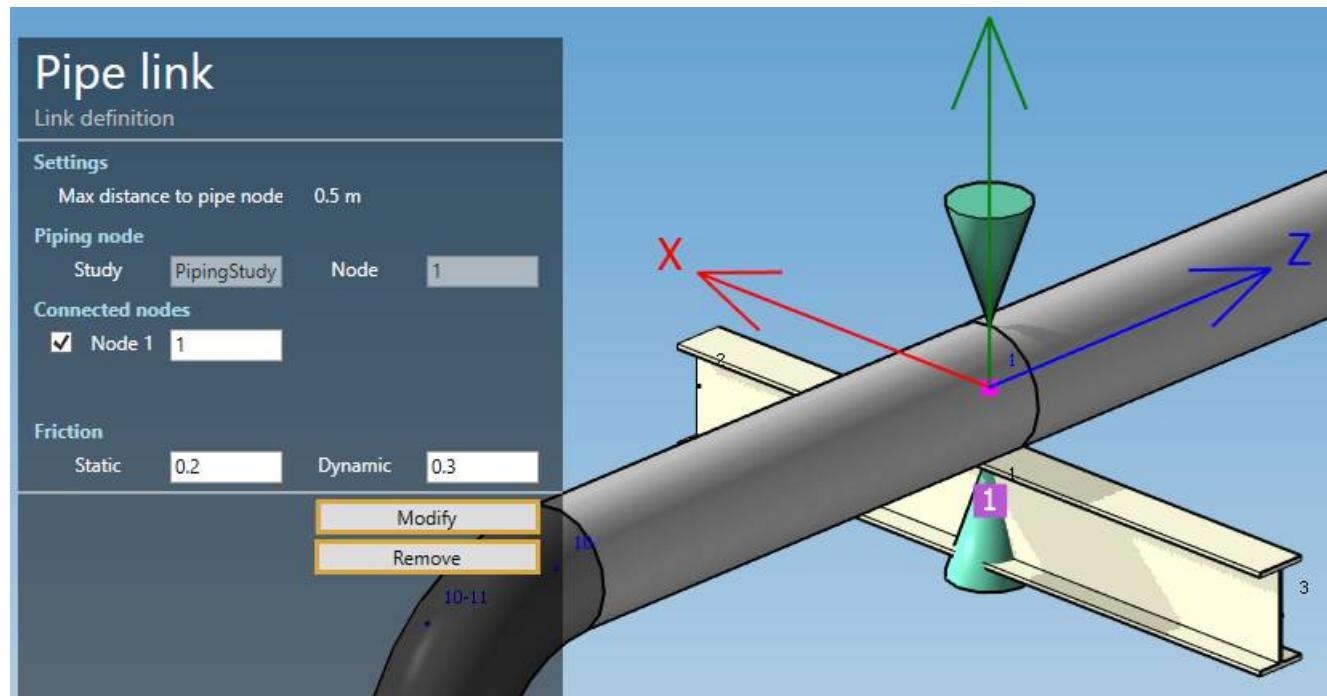
Finally, you can set a static and dynamic friction factor.

Click on the **Create button** to create a new link.

You can **undo** this command.

2. Modify/remove a link

Select a piping node with a link :



Change some properties and click on the **Modify** button or delete it by clicking the **Remove** button.

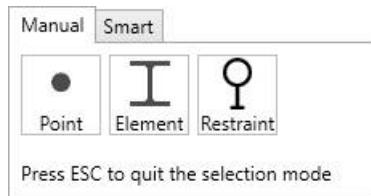
You can **undo** this command.

Modification



1. Selection

Click on **Select** button to open the selection window :



- Click on **Point** to select nodes
- Click on **Element** to select elements (beam, rigid and spring)
- Click on **Restraint** to select restraints
- Press **ESC** on keyboard to hide the selection window

You can now select objects of the corresponding type.

The selection is done by pressing the left mouse button above the desired object

Click [here](#) to have more information about the selection.

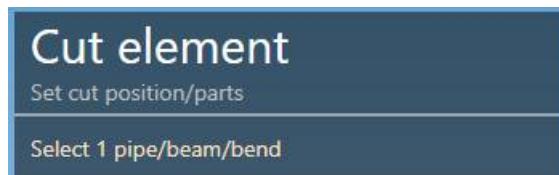
2. Remove elements

Select elements and click the **Remove elements** button to delete these elements.

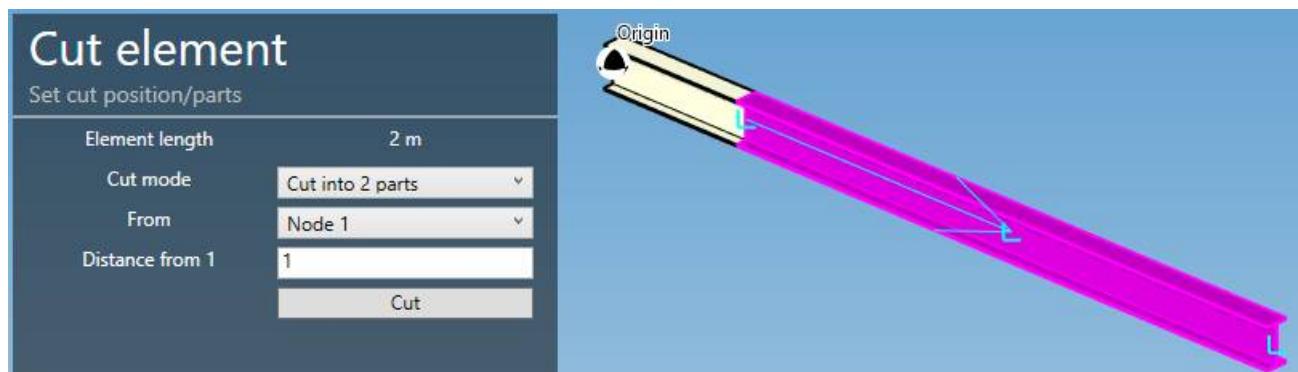
You can **Undo** this command.

3. Cut beams

When you click on the **Cut beams** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **ELEMENT**. You can so directly select a beam.



Cut mode can be :

- Cut into 2 parts
- Cut into N parts

The current length of the beam is shown on top.

By default, a distance from the first node of the beam is calculated, which is the half length of the selected beam (middle point).

You can choose from which node to start and the distance.

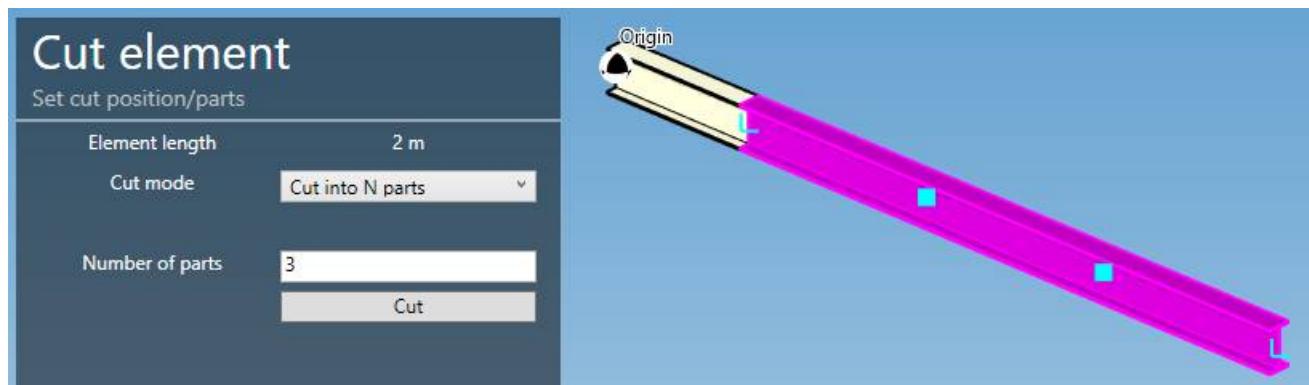
Property	Unit Metric	Unit USA
Length	m	ft
Distance	m	ft

Click on **Cut** to execute the command.

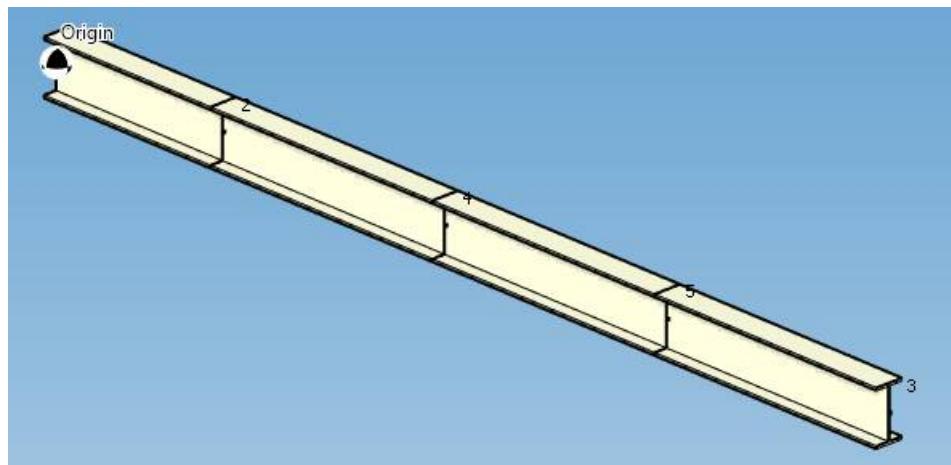
To know the UNIT of the value, just let the mouse over the cell.

You can **Undo** this command.

The second possibility is to cut in N parts with preview :



Click on the **Cut** button :



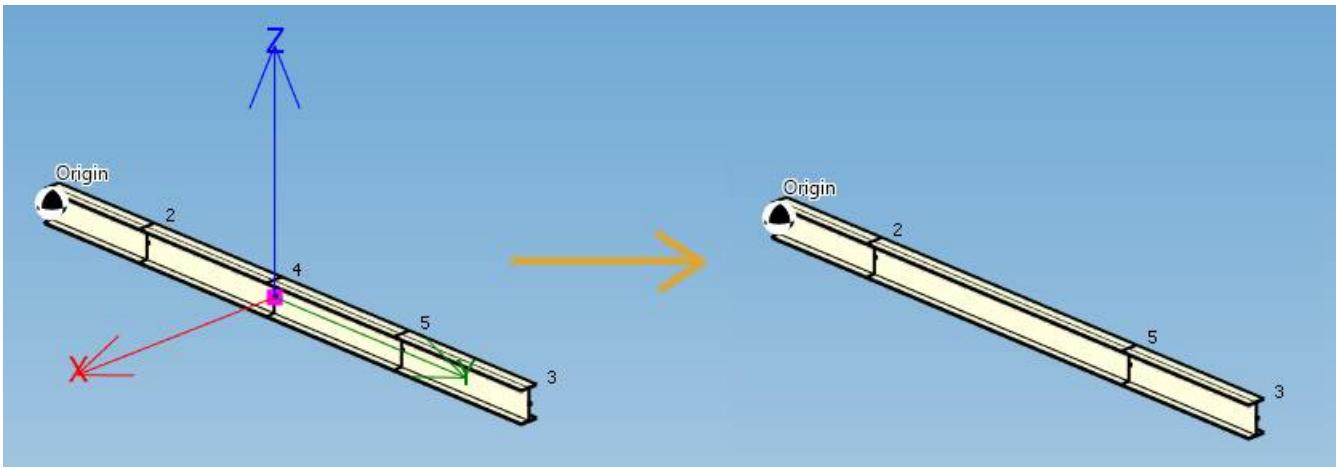
You can **Undo** this command.

4. Merge beams

Select a **Node** between 2 similar colinear beams and click the **Merge** button.

The 2 beams must have the same **Material** and the same **Section**.

The 2 beams will be replaced by one beam.



WARNING

Attention, the force, restraint, lumped mass or local coordinates defined on the node will be destroyed.

You can **Undo** this command.

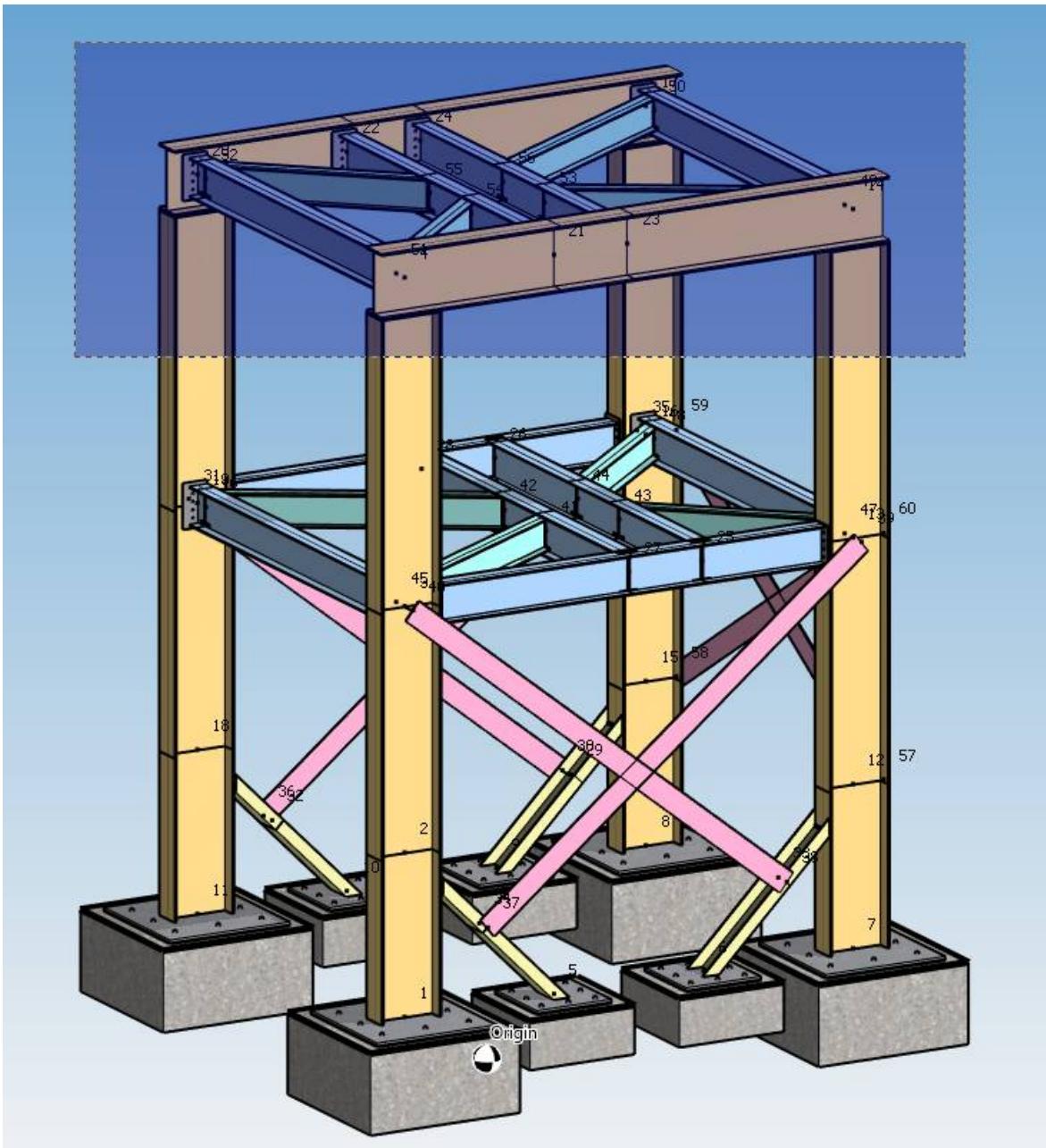
5. Move nodes

When you click on the **Move nodes** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **POINT**. You can so directly select nodes.

Select the nodes one by one (with CTRL) or by a selection rectangle :



Set the offset to apply to the selected nodes :

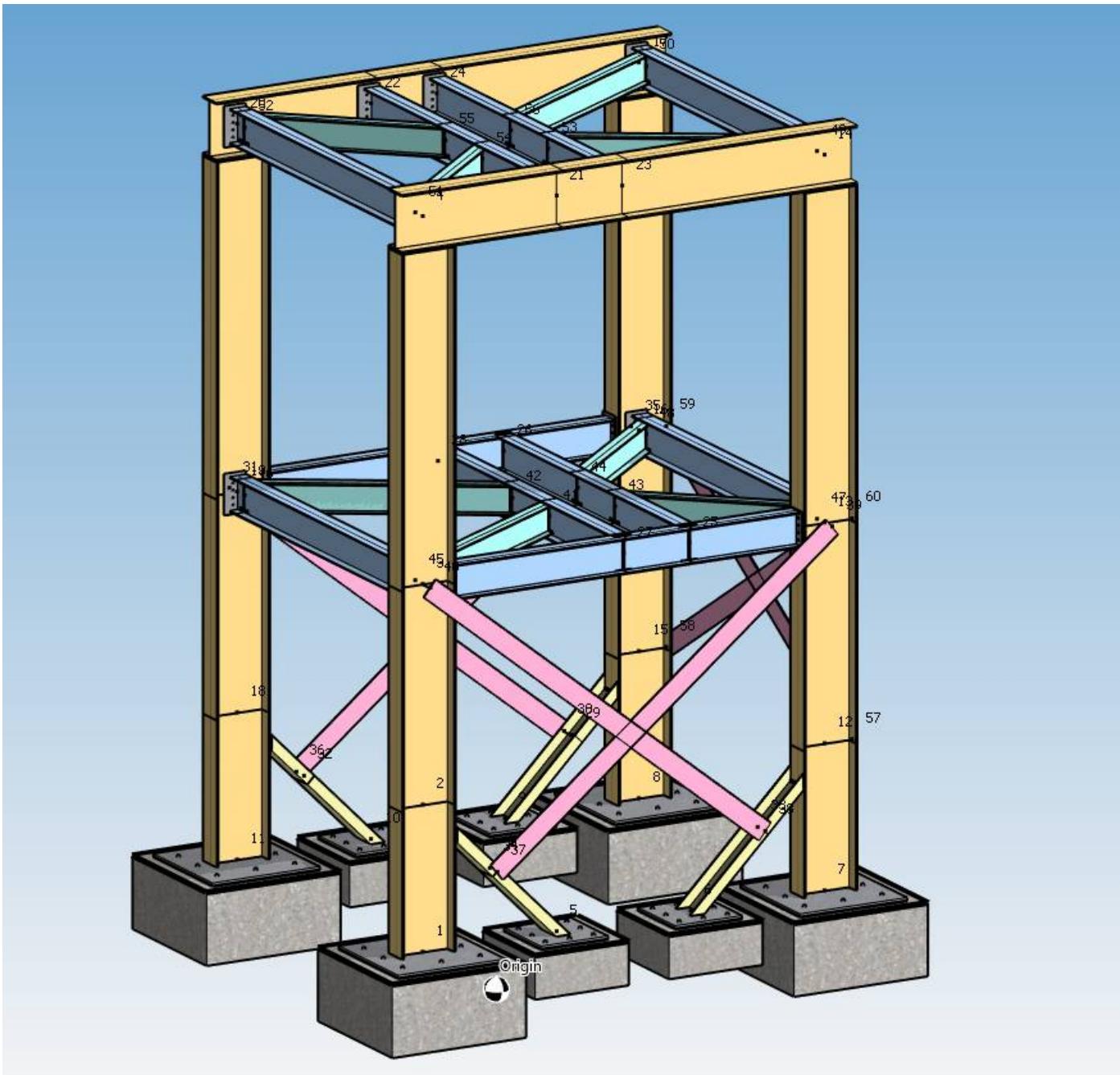
Move nodes

Set the translation values

DX	<input type="text" value="0"/>
DY	<input type="text" value="0"/>
DZ	<input type="text" value="0.5"/>

Property	Unit Metric	Unit USA
DX	m	ft
DY	m	ft
DZ	m	ft

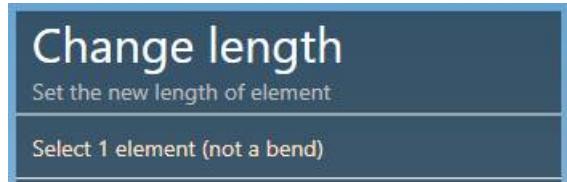
Validate the command by clicking the **Move** button :



You can **Undo** this command.

6. Change length/direction

When you click on the **Change length** button without selection, the left panel shows a message :



The **selection mode** is automatically set to **ELEMENT**. You can so directly select an element (beam, rigid or spring).

Change mode :

- Change length
- Change direction

You can select only one element.

6.1 Change length

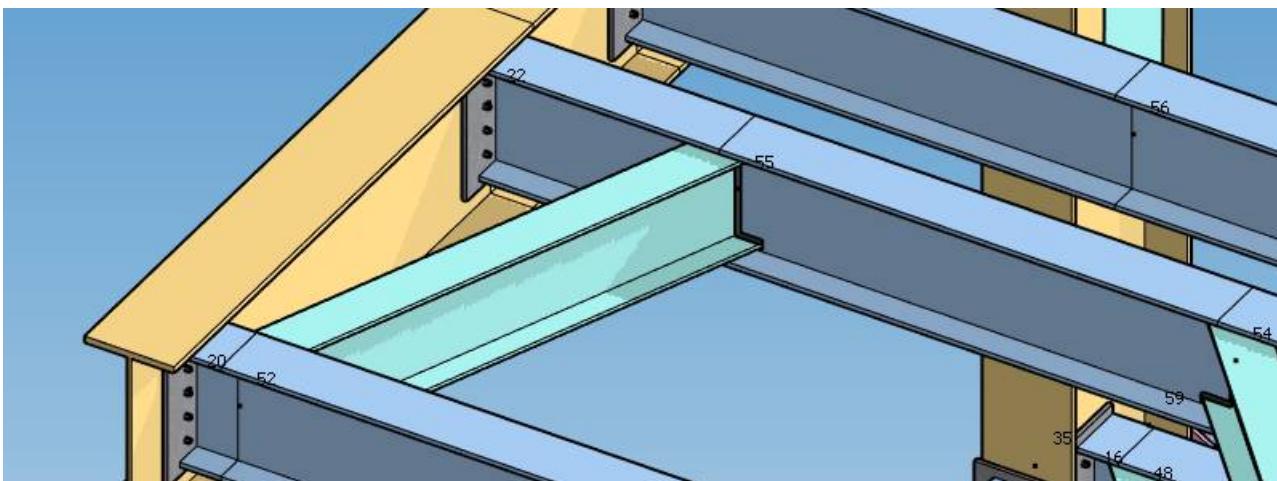
Select an element and set the new length :



Property	Unit Metric	Unit USA
Length	m	ft

Set the new length to 1 and uncheck the **Shift** cell.

Click on the **Modify** button :



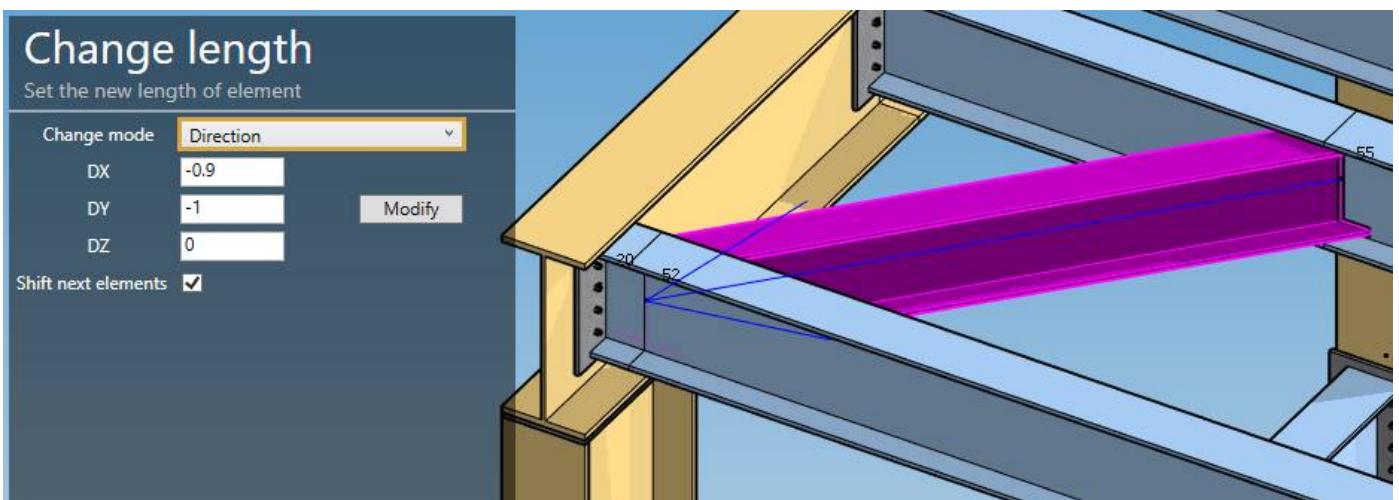
The point **55** moved.

The graphical ending of the "green" beam and the welding have been automatically recalculated !

You can **Undo** this command.

6.2 Change direction

Select an element and set the mode **Direction** :



The panel shows the actual DX, DY, DZ

Set the new direction (DX, DY, DZ) and uncheck the **Shift** cell :

Change length

Set the new length of element

Change mode **Direction**

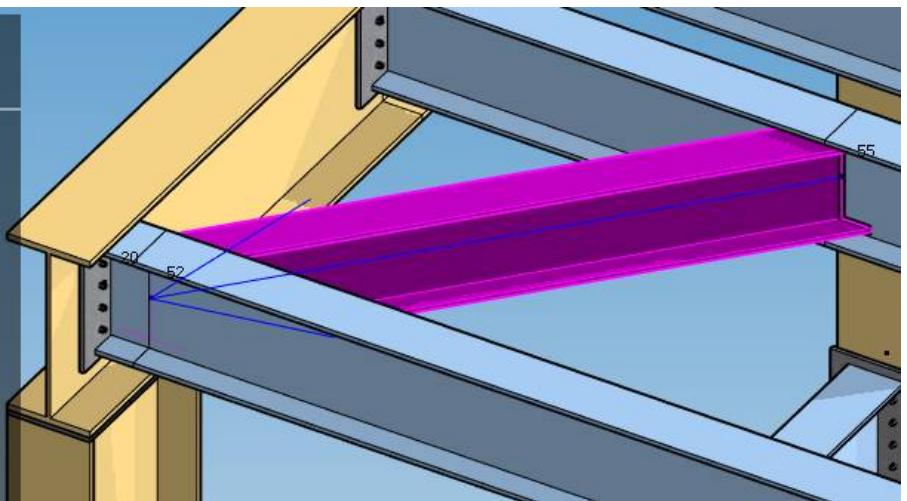
DX **-0.4**

DY **-1**

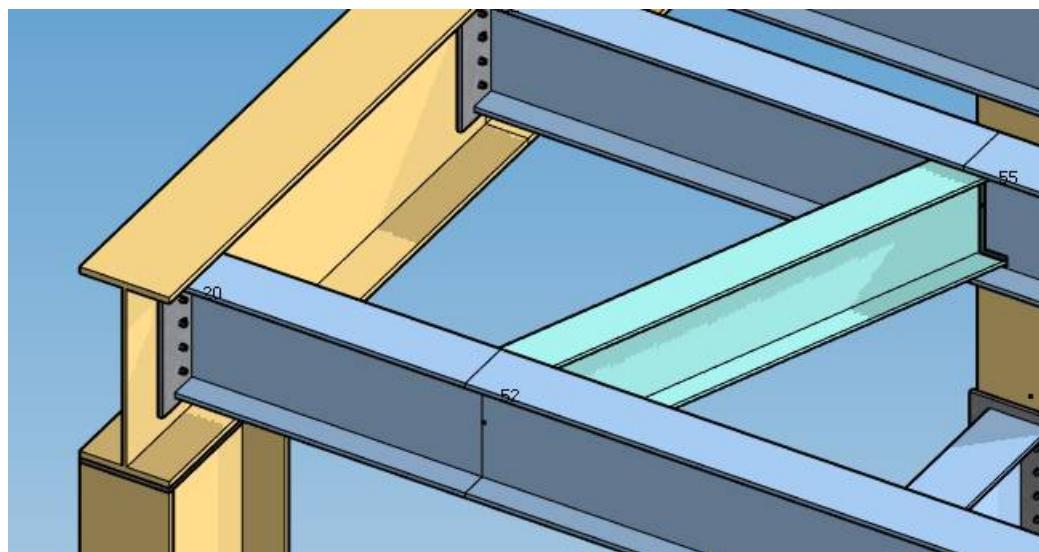
DZ **0**

Modify

Shift next elements



Click on the **Modify** button :



The point **52** moved.

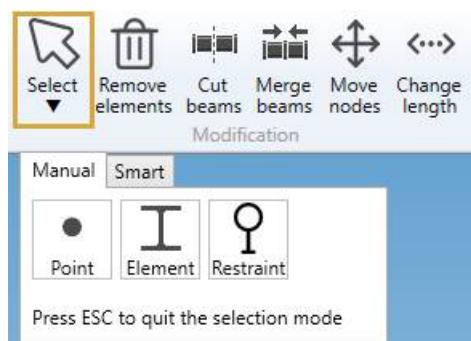
The graphical ending of the "green" beam and the welding have been automatically recalculated !

You can **Undo** this command.

Selection

MetaStructure lets you select one or more objects of the same type.

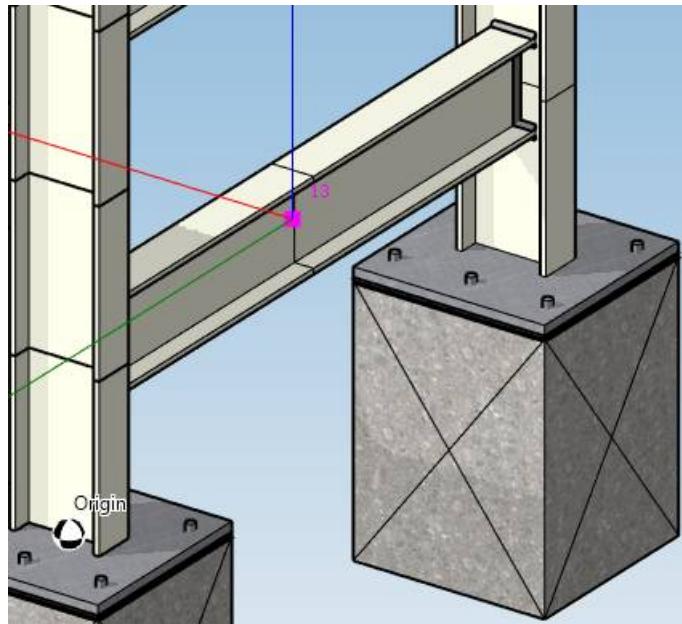
Click on the **Select** button to open the **Selection tool** :



1. Manual selection

POINT :

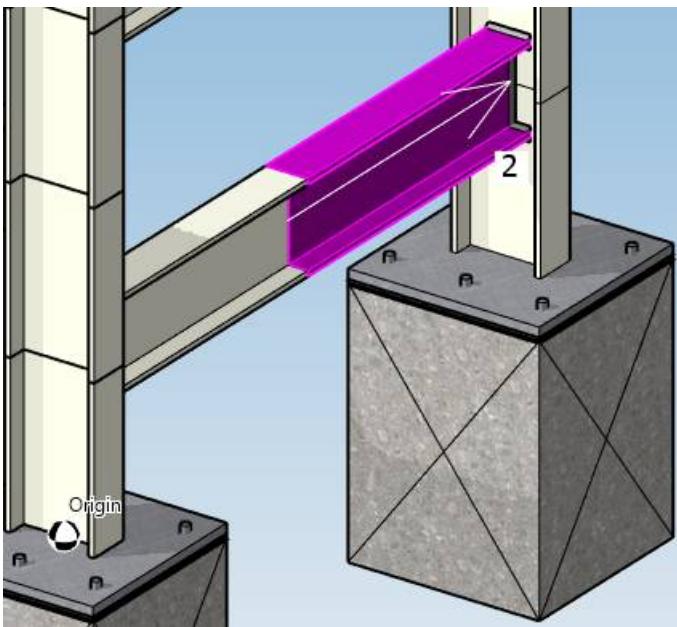
Click on the **Point** button to select **Nodes** or **element extremities**.



The node is highlighted and its **Name** appears in magenta.

ELEMENT :

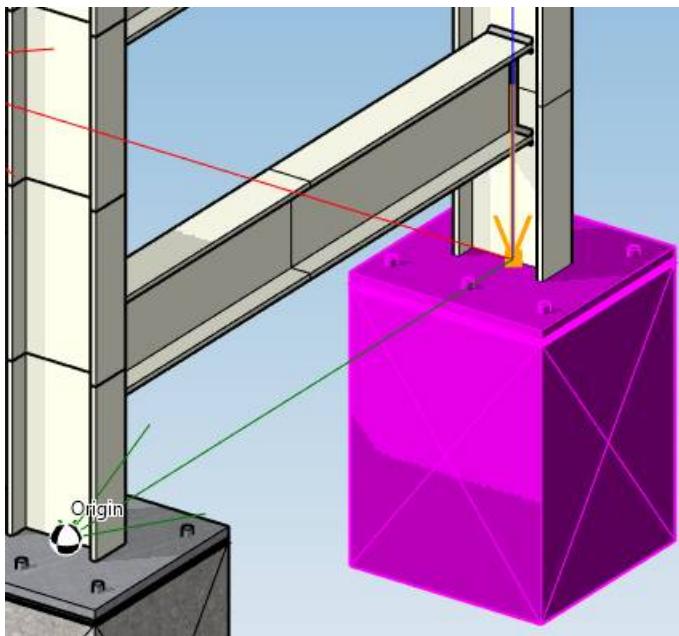
Click on the **Element** button to select an **element**.



The element is highlighted in magenta.

RESTRAINT :

Click on the **Restraint** button to select a **restraint**.



The restraint is highlighted in magenta.

CLEAR SELECTION :

Clicking in the 3D view where no objects are present without **CTRL key** pressed will clear the selection.

CANCEL SELECTION :

Just press **ESC** on keyboard to hide the **selection tool**.

2. Smart selection

Not yet implemented

3. Selection type

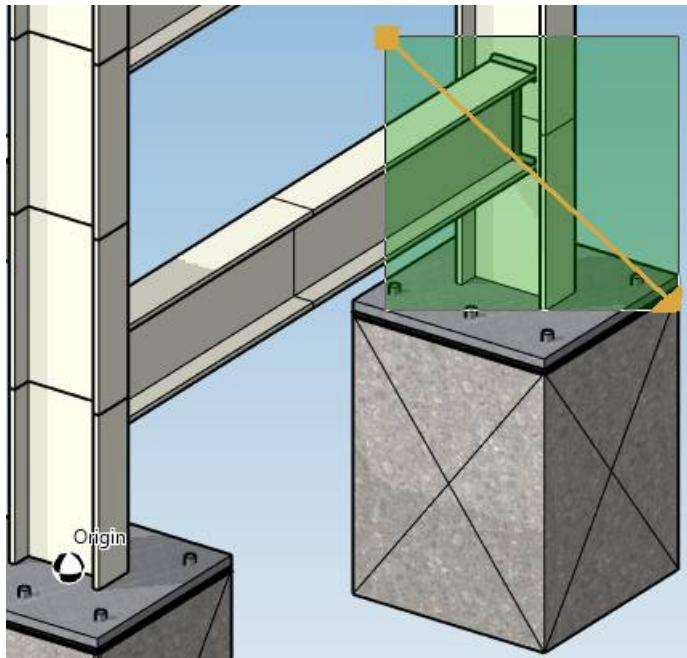
MOUSE CLICK :

To select **ONE** object, just make a click with the **left mouse button**:



RECTANGLE SELECTION :

It is also possible to make a rectangular selection :



- 1 Press the **left mouse button**
- 2 Move the mouse without releasing the left mouse button
- 3 Release the left mouse button

Every object that intersects the **rectangle** will be selected (in magenta).

MULTIPLE SELECTION :

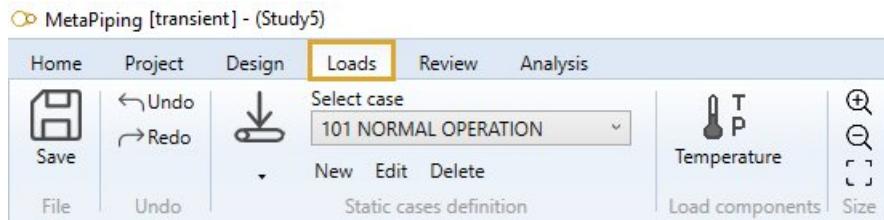
You can make multiple selection by pressing the **CTRL** key on keyboard.



Loads

The **LOADS** ribbon tab lets you create static, dynamic, thermal transient and combined load cases.

1. Ribbon menu



1.1 File

During the **load case** definition, every command is automatically saved in a temporary file (conception.~metaL).

You can decide to save the last modifications or cancel it and return to the study screen.

A floppy disk on the left side of the ribbon indicates the state of the file on disk :



Save means that everything has been saved.



Save means that something has changed and the MetaL need to be saved.

WARNING

If the application crashes, MetaPiping will prompt to reopen the last modifications.

1.2 Undo/Redo



Every command in MetaPiping is stored in a **command list**.

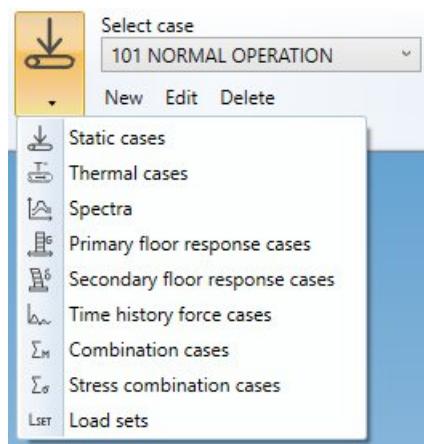
You can *navigate* through this list by pressing the **Undo/Redo** buttons.

TABLE OF CONTENTS

- [Piping](#)
- [Structure](#)

Piping loads

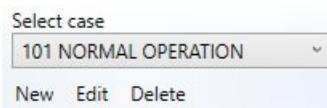
Click on the **LOAD** button to have access to the **load case types**:



After selection, the choice becomes the **current load case type**.

1. Edition

For all case types, you can **create**, **modify** or **remove** a selected case :



- **New** : open an empty window of **current load case type** : press OK to create the load
- **Edit** : open a window of **current load case type** with the selected case : press OK to modify the load case
- **Delete** : remove the selected case
- **Selection** : list of all cases of the **current load case type**

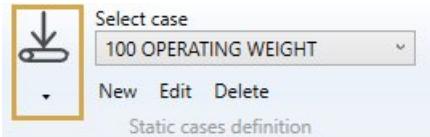
TABLE OF CONTENTS

- [Static cases](#)
- [Thermal cases](#)
- [Spectra](#)
- [Primary floor response cases](#)
- [Secondary floor response cases](#)
- [Time history cases](#)
- [Combination cases](#)
- [Stress combination cases](#)
- [Load sets](#)

Static cases

Static cases consist of internal and/or external static loadings on the piping system. MetaPiping will calculate corresponding displacements, reactions and stresses.

All existing static cases are listed in the combobox with their **Case number + Title** :



When editing or creating a static case, the definition window shows up. There are three tabs : General, Options and Applied loads.

1. General

Static case definition	
General	Options
Case number :	100
Title :	OPERATING WEIGHT
Category :	Sustained - Operating
Level :	(dropdown)
Equation :	2. Eq. 9 (design)
<input type="button" value="OK"/> <input type="button" value="Cancel"/>	

Enter a **Case number** and a **Title**.

Category :

- Sustained - Operating
- Sustained - Test
- Sustained - Empty
- Sustained - Design
- Occasional - Wind
- Occasional - Snow
- Occasional - Distributed
- Occasional - Acceleration
- Occasional - SAM
- Thermal expansion
- Settlement

- Cold spring
- Dummy (for oper. cond.)
- User defined

Depending on the chosen category, some options are pre-selected in tab **Options** and the loads that may be applied are listed in tab **Applied loads**.

Level :

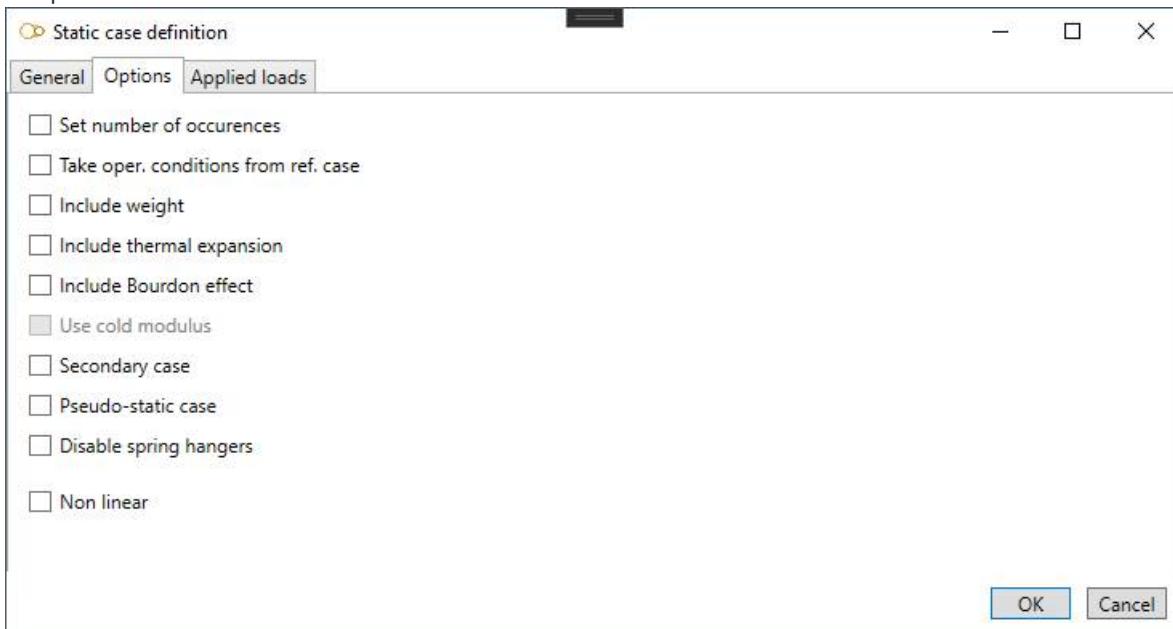
This field is intended for nuclear piping codes only. The possible values are : A (normal conditions), B (upset), C (emergency), D (faulted) and T (test).

Equation :

The equations to be verified are code-dependent. For example, here are the equations for code RCC-M Class 2:

- 1- None (no stress calculation)
- 2- Equation 6 (sustained)
- 3- Equation 10 B (occasional)
- 4- Equation 10 C (occasional)
- 5- Equation 10 D (occasional)
- 6- Equation 10 T (test)
- 7- Equation 7 (thermal expansion)
- 8- Equation 8 (thermal expansion + sustained)
- S- Equation 9 (settlement)

2. Options

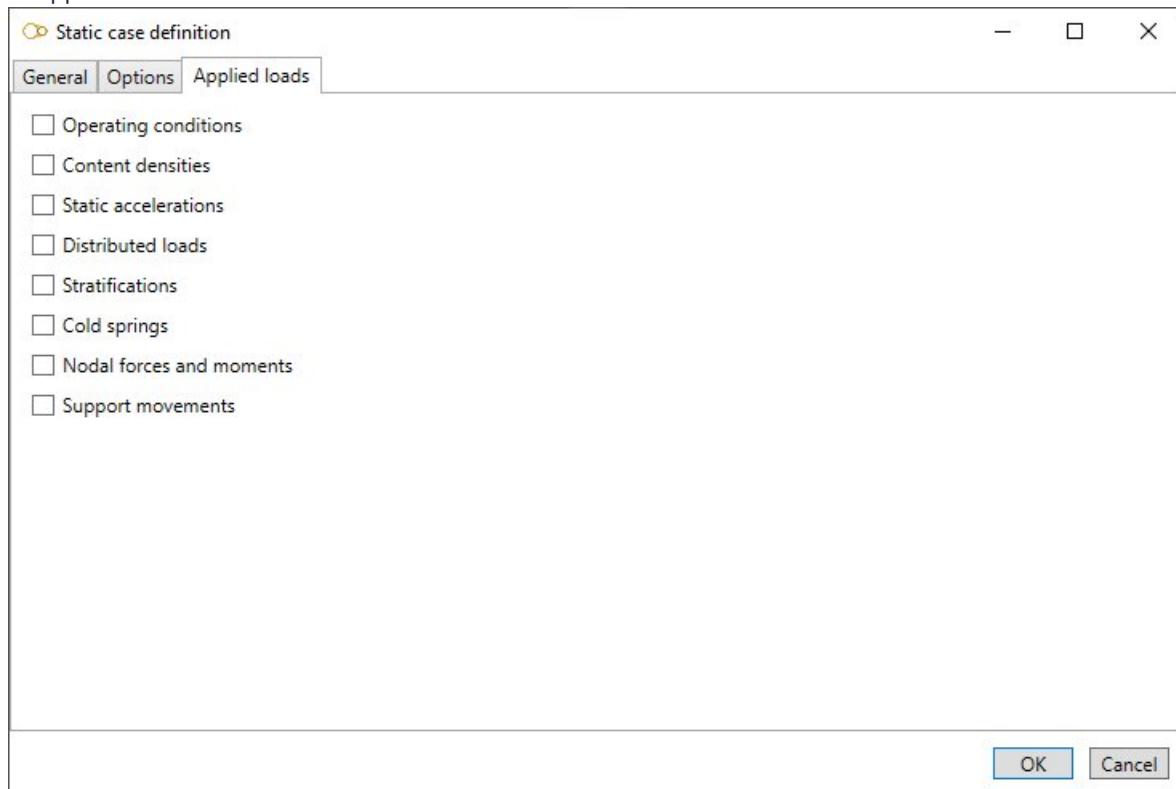


- **Set number of occurrences** : enter the number of cycles (thermal expansion cases only)
- **Take oper. conditions from ref. case** : select the load case which pressure and temperature will be used for thermal expansion, Young modulus, Bourdon effect and bend stiffening. If no reference case is selected and no operating conditions are defined for the load case, room temperature is used and no pressure is considered
- **Include weight** : if checked, the weight is considered in the calculation. Four options are available: operating weight, test weight, empty weight and design weight. Design weight is equivalent to operating weight, except that spring hangers are pinned. Design weight can be useful for determining the installation force to be exerted on the spring hangers
- **Include thermal expansion** : if checked, thermal expansion is considered in the calculation
- **Include Bourdon effect** : if checked, expansion due to internal pressure is considered in the calculation
- **Use cold modulus** : if checked, Young modulus at room temperature is used for this case (only if **Hot modulus** is set in **Model options**)
- **Secondary case** : to be checked for displacement-type cases like thermal expansion. Depending on the piping code (e.g. RCC-M Class 2), the moments may be multiplied by Ec/Eh when the **Hot modulus** is used
- **Pseudo-static cases** : to be checked for dynamic cases that are calculated as static cases (e.g. earthquake calculated with a pseudo-static

acceleration)

- **Disable spring hangers** : for some cases like settlement and cold springing, it is common practice to readjust the spring hangers to their design parameters. This is equivalent to disabling the spring hangers
- **Non linear** : if checked, non linear restraint is considered in the calculation

3. Applied loads



Select the loads that will be applied to the load case. For some loads like temperature and pressure, it is possible to set default values that will apply to all elements. The user can of course supersede the default values for some elements by selecting them and specifying different values.

3.1 Operating conditions

<input checked="" type="checkbox"/> Operating conditions	<input checked="" type="checkbox"/> Set default	Temperature :	200
		Pressure :	0.1

You can define operating conditions on elements of the model.

The temperature is used for multiple purposes :

- the Young modulus Eh is evaluated at this temperature if the **Hot modulus** option is used. The Young modulus at room temperature Ec is used otherwise if no reference case is specified
- thermal expansion is applied if **Include thermal expansion** is checked
- the allowable stress Sh for non-Class 1 piping is evaluated at this temperature if the **Hot allowable** option is used. The design temperature is used otherwise

The pressure is used for multiple purposes :

- the longitudinal stress due to pressure thrust
- the stiffening effect on elbows and miter bends
- the SIF reducing effect on elbows and miter bends

You can define **DEFAULT** operating conditions that will be applied on all elements of the model.

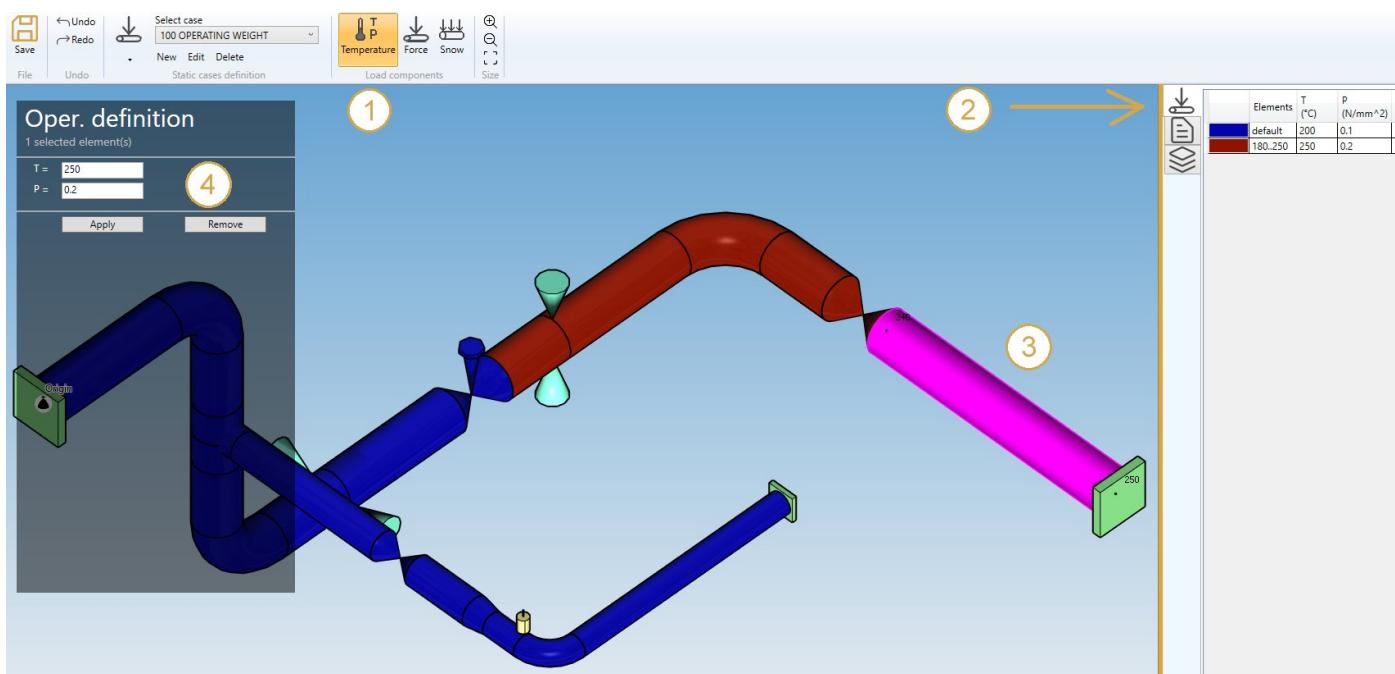
Define the default **temperature** and **pressure** of the load :

Property	Unit Metric	Unit USA
Temperature	°C	°F
Pressure	N/m ²	lb/ft ²

If you check this option, you will see this **button** in the ribbon:



- 1 Select the **Temperature** button
- 2 Open the right **Data panel**
- 3 Select one or more elements
- 4 Define the new **temperature** and **pressure** for these elements
- 5 Apply



You can **undo** this command.

Click on the **Remove** button to delete the operating condition load on selected elements.

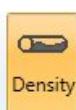
3.2 Content densities

<input checked="" type="checkbox"/> Content densities	<input checked="" type="checkbox"/> Set default	Density :	<input type="text" value="1"/>
---	---	-----------	--------------------------------

By checking **Content densities** in **Applied loads** you can define fluid densities on elements of the model.

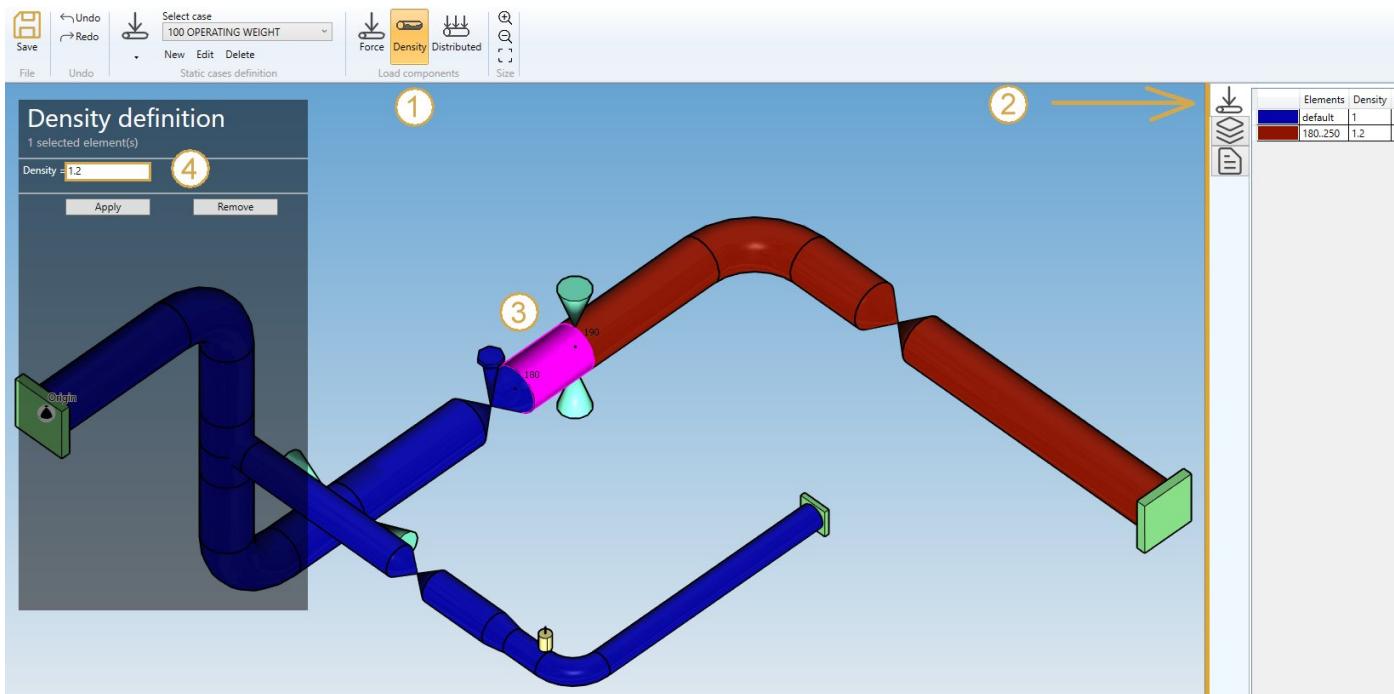
You can also define the **DEFAULT** fluid density so that you only have to define the elements that doesn't follow the default density. If no default density is set, and no density is specified on some elements, the default density provided in **Model options** or directly in **Data** will be used.

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Density** button
- 2 Open the right **Data panel**

- 3 Select one or more elements
- 4 Define the new **density** for these elements
- 5 Apply



You can **undo** this command.

Click on the **Remove** button to delete the density on selected elements.

3.3 Static accelerations

<input checked="" type="checkbox"/> Static accelerations	<input checked="" type="checkbox"/> Set default	GX : 0.5
		GY : 0
		GZ : 0

By checking **Static acceleration** in **Applied loads** you can define acceleration loads on elements of the model.

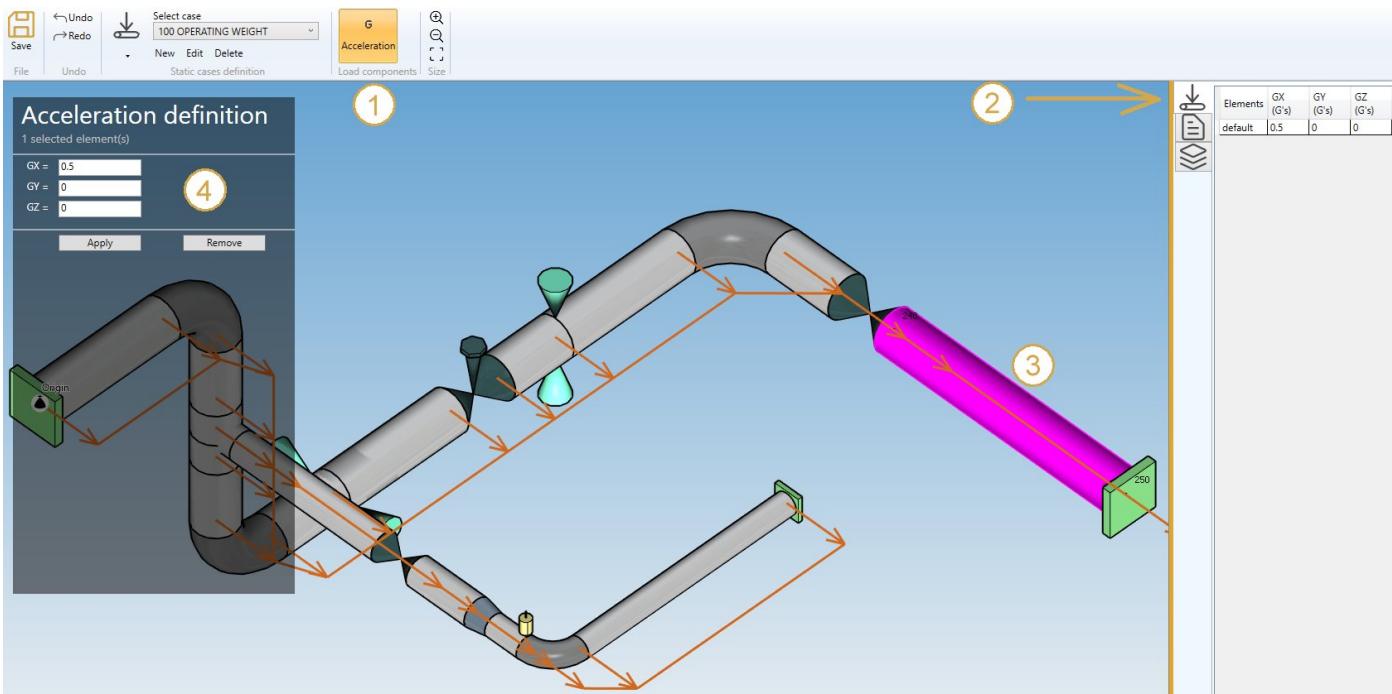
You can also define **DEFAULT** accelerations that will be applied to all elements of the model :

Property	Unit Metric	Unit USA
GX	fraction of G	fraction of G
GY	fraction of G	fraction of G
GZ	fraction of G	fraction of G

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Acceleration** button
- 2 Open the **Data panel** on the right
- 3 Select one or more elements
- 4 Define the new **vector acceleration** for these elements
- 5 Apply



You can **undo** this command.

Click on the **Remove** button to delete the load on selected elements.

You can also change the size of the drawn accelerations by clicking on these buttons :



3.4 Distributed loads

By checking **Distributed loads** in **Applied loads** you can define distributed loads on elements of the model. There are three types of distributed loads: simple distributed load (force per unit length), wind and snow.

Simple distributed loads may be applied for weight and occasional cases other than wind and snow cases. Wind loads may only be applied for wind cases and snow loads for snow cases.

3.4.1 SIMPLE DISTRIBUTED LOAD (DISTRIBUTED)

<input checked="" type="checkbox"/> Distributed loads	Distributed	<input checked="" type="checkbox"/> Set default	FX :	0
			FY :	-2
			FZ :	0

You can define **DEFAULT** distributed loads that will be applied to all elements of the model :

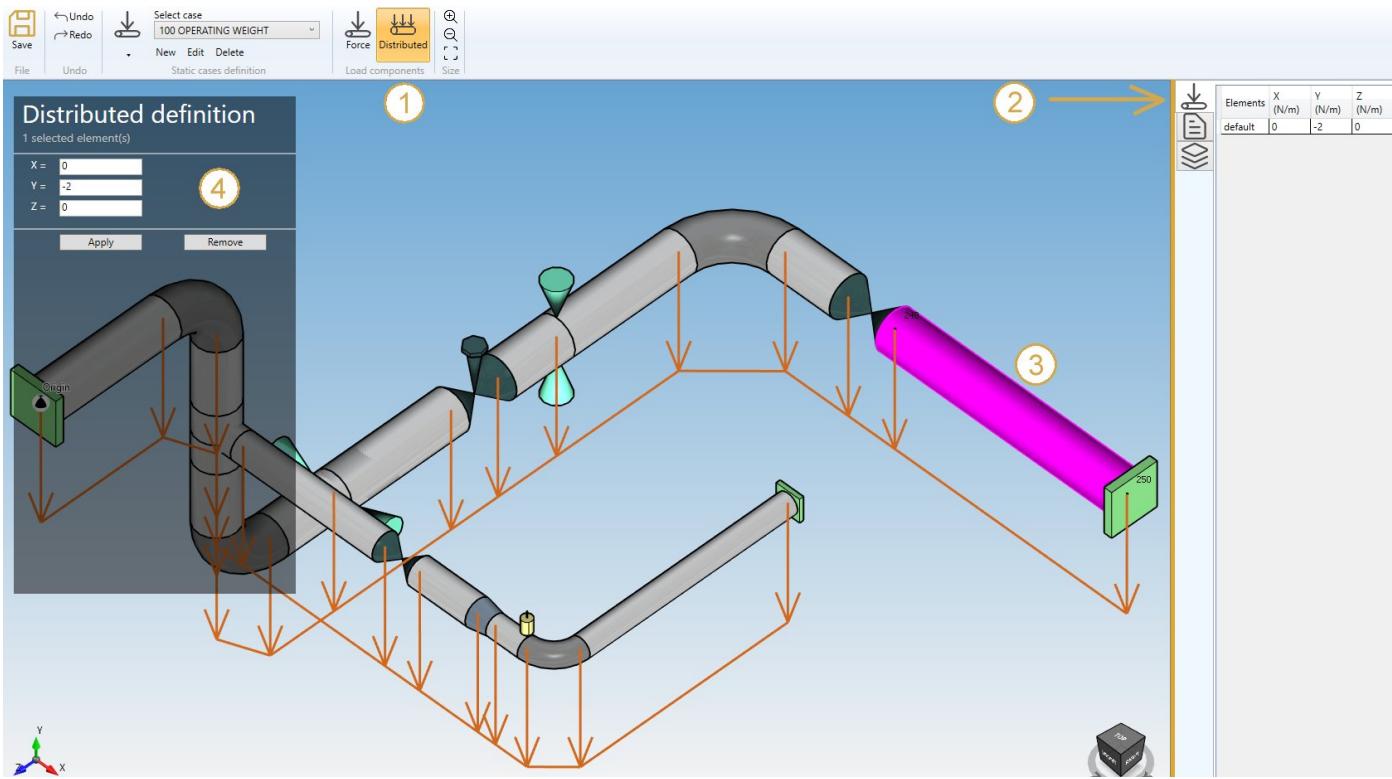
Property	Unit Metric	Unit USA
FX	N/m	lb/ft
FY	N/m	lb/ft
FZ	N/m	lb/ft

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Distributed** button

- 2 Open the right **Data panel**
- 3 Select one or more elements
- 4 Define the new **vector force** for these elements
- 5 Apply



You can **undo** this command.

Click on the **Remove** button to delete the load on selected elements.

You can **undo** this command.

You can also change the size of the drawn forces by clicking on these buttons :



3.4.2 WIND LOAD

<input checked="" type="checkbox"/> Distributed loads	Wind	<input checked="" type="checkbox"/> Set default	Pressure :	0.1
			Dir. /X :	1
			Dir. /Y :	0
			Dir. /Z :	0

You can define **DEFAULT** wind loads that will be applied to all elements of the model.

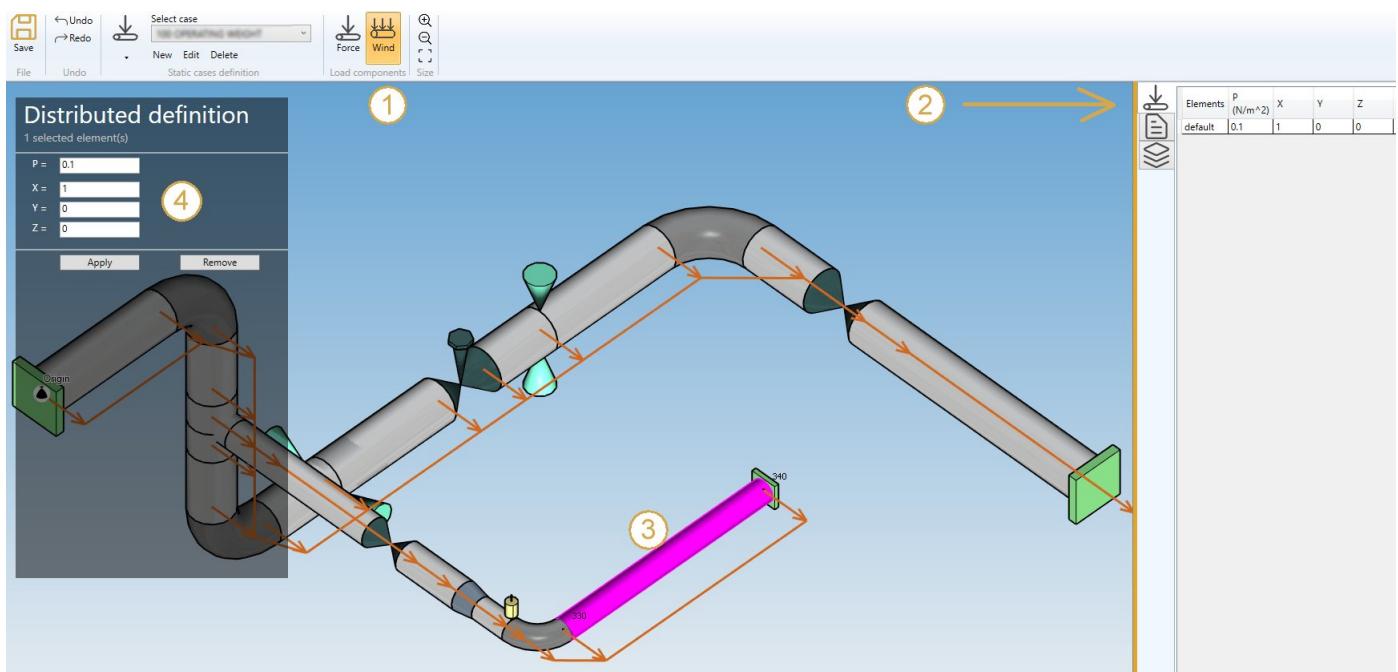
Define the wind **pressure** and **direction** :

Property	Unit Metric	Unit USA
Pressure	N/m ²	lb/ft ²
Dir. /X	-	-
Dir. /Y	-	-
Dir. /Z	-	-

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Wind** button
- 2 Open the right **Data panel**
- 3 Select one or more elements
- 4 Define the new **pressure** and **direction** for these elements
- 5 Apply



You can **undo** this command.

Click on the **Remove** button to delete the wind load on selected elements.

You can **undo** this command.

You can also change the size of the drawn forces by clicking on these buttons :



3.4.3 SNOW LOAD

<input checked="" type="checkbox"/> Distributed loads	Snow	<input checked="" type="checkbox"/> Set default	Pressure : 0.2
---	------	---	----------------

You can define **DEFAULT** snow loads that will be applied to all elements of the model.

Define the snow **pressure** (direction is vertical downwards) :

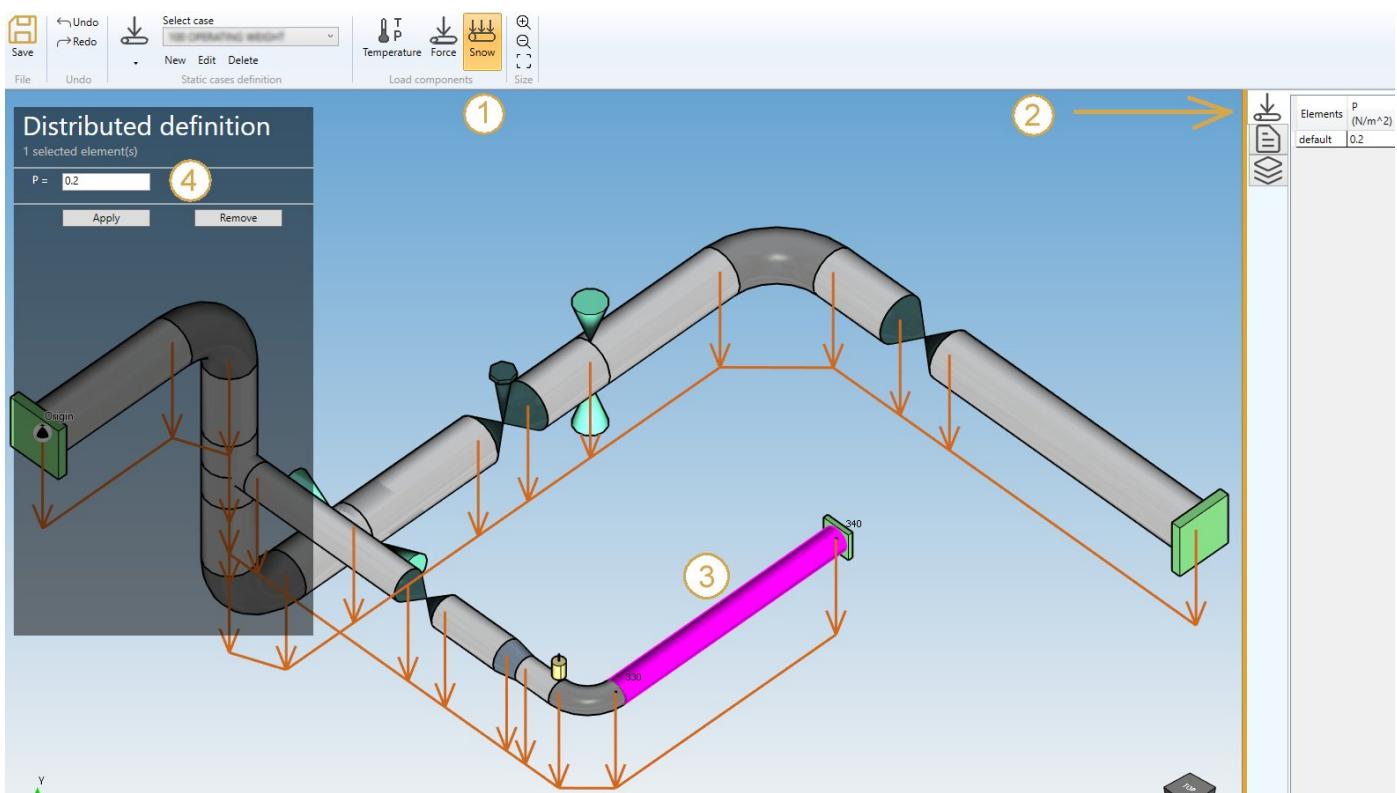
Property	Unit Metric	Unit USA
Pressure	N/m ²	lb/ft ²

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Snow** button

- 2 Open the right **Data panel**
- 3 Select one or more elements
- 4 Define the new **pressure** for these elements
- 5 Apply

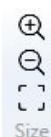


You can **undo** this command.

Click on the **Remove** button to delete the snow load on selected elements.

You can **undo** this command.

You can also change the size of the drawn forces by clicking on these buttons :



3.5 Stratifications

Stratifications

You can define stratifications on several elements of the model.

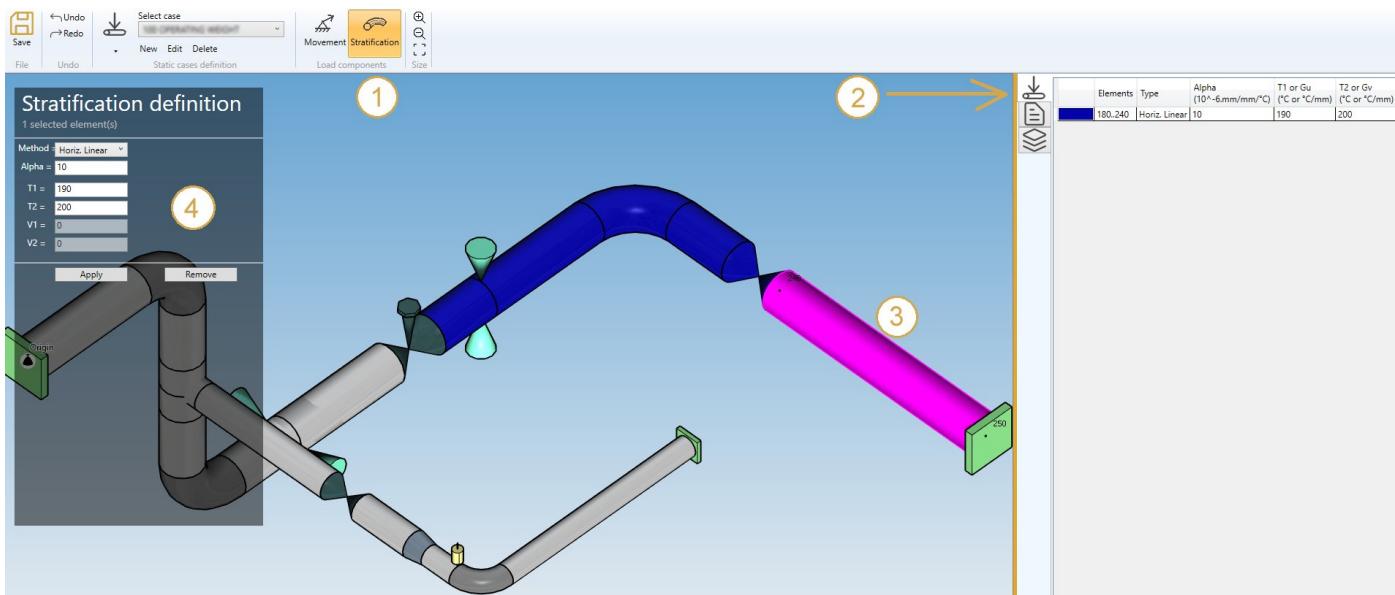
Thermal stratification occurs when there is a non-constant, time independent temperature distribution in the pipe wall. Typically, this situati

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Stratification** button
- 2 Open the right **Data panel**
- 3 Select one or more elements
- 4 Define the new **stratification** for these elements

5 Apply



Method :

- Horiz. Linear
- Horiz. Step
- Horiz. Laminar
- Horiz. Gu
- Skewed

Properties :

Depending on the selected method, the properties are :

Horiz. Linear :

Property	Definition	Unit Metric	Unit USA
Alpha	Mean coef. of thermal expansion of pipe	$10^{-6}\text{mm/mm}^{\circ}\text{C}$	$10^{-6}\text{in/in}^{\circ}\text{F}$
T1	Temperature at bottom of the pipe	$^{\circ}\text{C}$	$^{\circ}\text{F}$
T2	Temperature at top of the pipe	$^{\circ}\text{C}$	$^{\circ}\text{F}$

Horiz. Step :

Property	Definition	Unit Metric	Unit USA
Alpha	Mean coef. of thermal expansion of pipe	$10^{-6}\text{mm/mm}^{\circ}\text{C}$	$10^{-6}\text{in/in}^{\circ}\text{F}$
T1	Temperature at bottom of the pipe	$^{\circ}\text{C}$	$^{\circ}\text{F}$
T2	Temperature at top of the pipe	$^{\circ}\text{C}$	$^{\circ}\text{F}$
V1	Vertical coordinate of the discontinuity	mm	in

Horiz. Laminar :

Property	Definition	Unit Metric	Unit USA
Alpha	Mean coef. of thermal expansion of pipe	$10^{-6}\text{mm/mm/}^{\circ}\text{C}$	$10^{-6}\text{in/in/}^{\circ}\text{F}$
T1	Temperature at bottom of the pipe	$^{\circ}\text{C}$	$^{\circ}\text{F}$
T2	Temperature at top of the pipe	$^{\circ}\text{C}$	$^{\circ}\text{F}$
V1	Vertical coordinate of the laminar flow (down)	mm	in
V2	Vertical coordinate of the laminar flow (up)	mm	in

Horiz. Gu :

Property	Definition	Unit Metric	Unit USA
Alpha	Mean coef. of thermal expansion of pipe	$10^{-6}\text{mm/mm/}^{\circ}\text{C}$	$10^{-6}\text{in/in/}^{\circ}\text{F}$
Gu	stratification value inside the pipe	$^{\circ}\text{C/mm}$	$^{\circ}\text{F/in}$

Skewed :

Stratification definition in **LOCAL COORDINATES** (U, V, W) :

Property	Definition	Unit Metric	Unit USA
Alpha	Mean coef. of thermal expansion of pipe	$10^{-6}\text{mm/mm/}^{\circ}\text{C}$	$10^{-6}\text{in/in/}^{\circ}\text{F}$
Gu	stratification value inside the pipe in U direction	$^{\circ}\text{C/mm}$	$^{\circ}\text{F/in}$
Gv	stratification value inside the pipe in V direction	$^{\circ}\text{C/mm}$	$^{\circ}\text{F/in}$
VX	X component of the local V vector	-	-
VY	Y component of the local V vector	-	-
VZ	Z component of the local V vector	-	-

You can **undo** this command.

Click on the **Remove** button to delete the load on selected restraints.

3.6 Cold springs

Cold springs

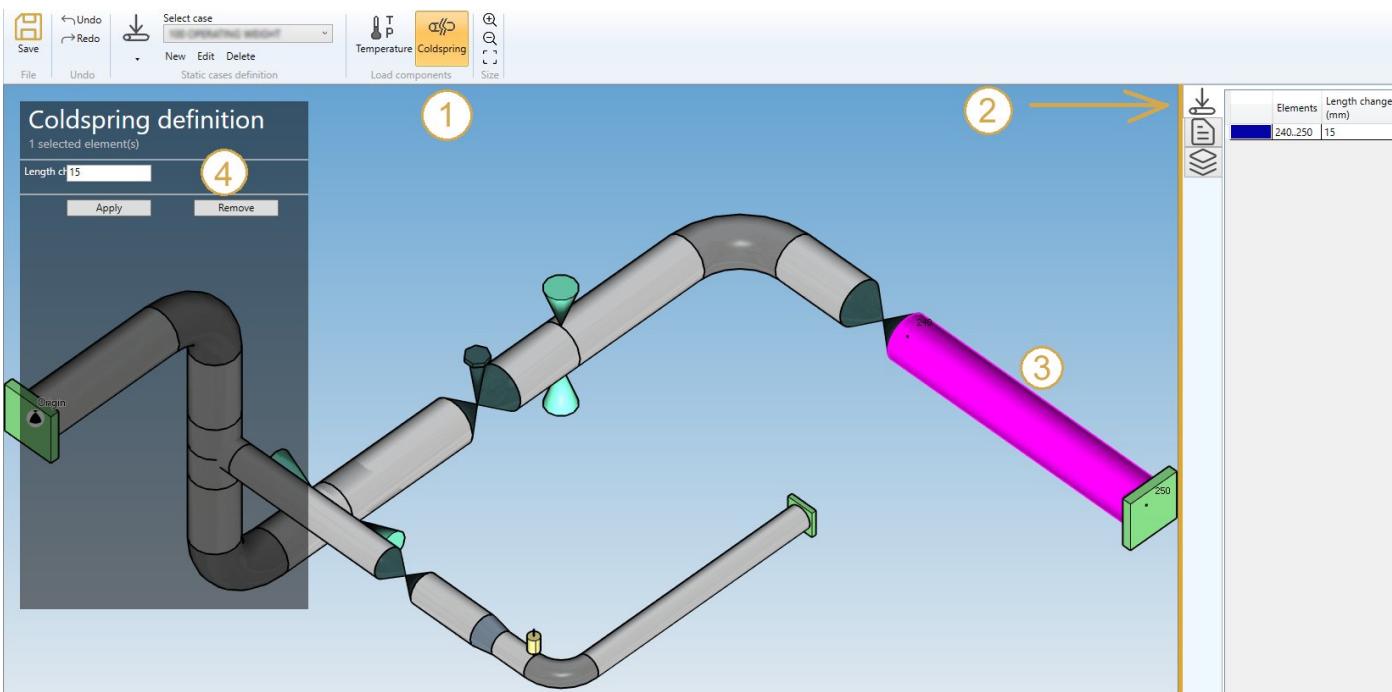
By checking **Cold springs** in **Applied loads** you can define cold spring (pre-stress) on several elements of the model.

The length of elements that will be altered by an explicit amount will cause forces, moments and stresses in the piping system.

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Coldspring** button
- 2 Open the right **Data panel**
- 3 Select one or more elements
- 4 Define the new **extension** for these elements
- 5 Apply



Property	Unit Metric	Unit USA	Remark
Length change	mm	in	Positive value for extension

You can **undo** this command.

Click on the **Remove** button to delete the load on selected restraints.

3.7 Nodal forces and moments

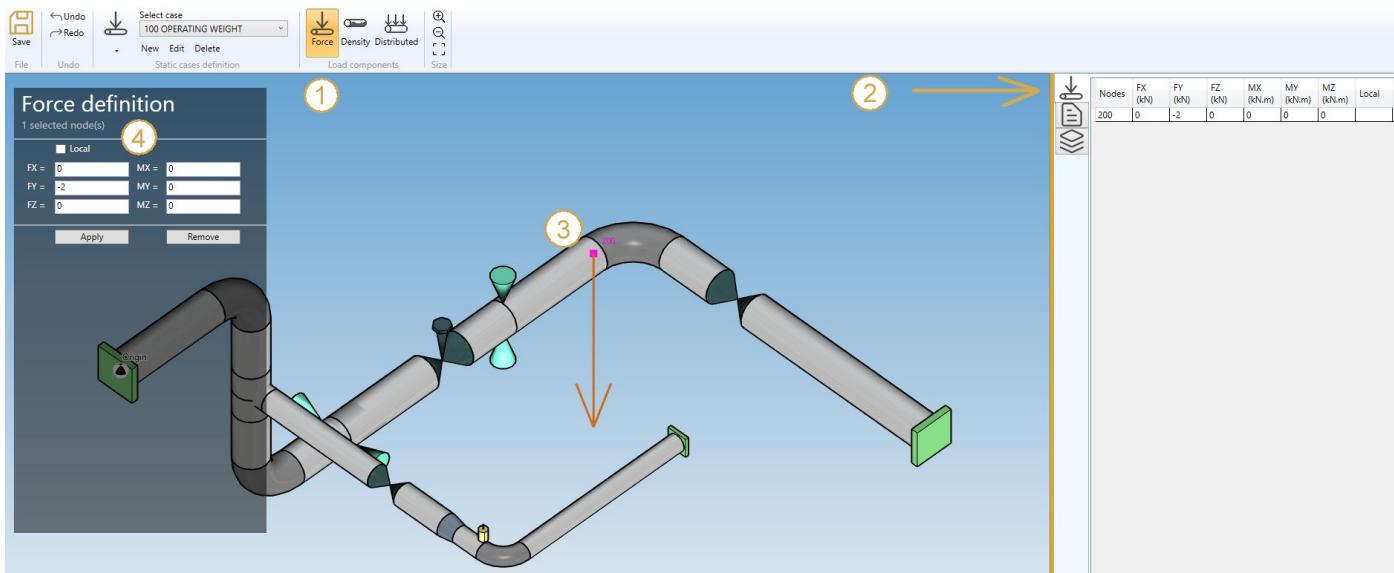
Nodal forces and moments

You can define nodal forces and moments on nodes of the model.

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Force** button
- 2 Open the right **Data panel**
- 3 Select one or more nodes
- 4 Define the new **force and moment** on these nodes
- 5 Apply



Property	Unit Metric	Unit USA
FX	kN	kips
FY	kN	kips
FZ	kN	kips
MX	kN.m	kips.ft
MY	kN.m	kips.ft
MZ	kN.m	kips.ft

You can **undo** this command.

Click on the **Remove** button to delete the force/moment on selected nodes.

You can **undo** this command.

You can also change the size of the drawn forces by clicking on these buttons :



3.8 Support movements

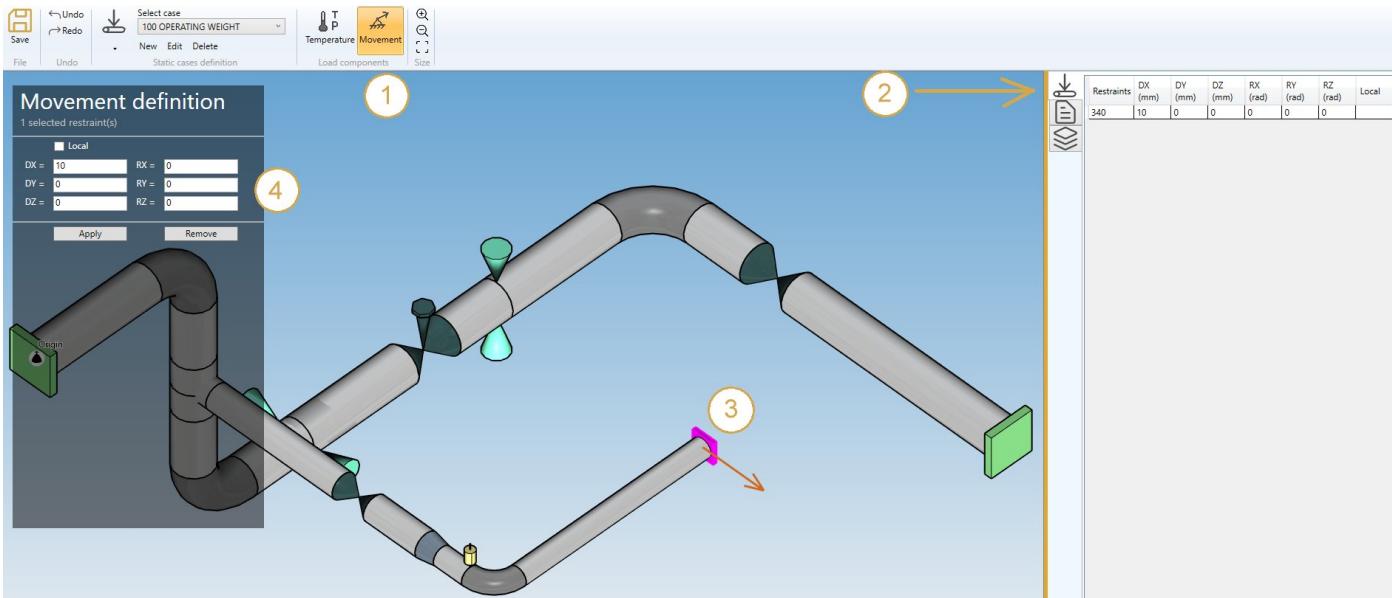
Support movements

You can define movements on supports of the model.

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Movement** button
- 2 Open the right **Data panel**
- 3 Select one or more restraints
- 4 Define the new **vector displacement** and **vector rotation** for these restraints
- 5 Apply



Property	Unit Metric	Unit USA
DX	mm	in
DY	mm	in
DZ	mm	in
RX	rad	rad
RY	rad	rad
RZ	rad	rad

The values can be set in **local** axis.

You can **undo** this command.

Click on the **Remove** button to delete the load on selected restraints.

You can **undo** this command.

You can also change the size of the drawn movements by clicking on these buttons :



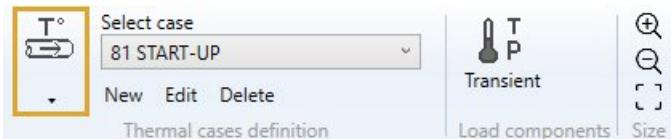


Thermal cases (Class 1 only)

This case consists of the definition of TRANSIENTS that describes the evolution of the fluid temperature (*) during time inside the pipes. Additional stresses will result from the temperature gradients.

(*) also absolute pressure, flow rate, state or film coefficient.

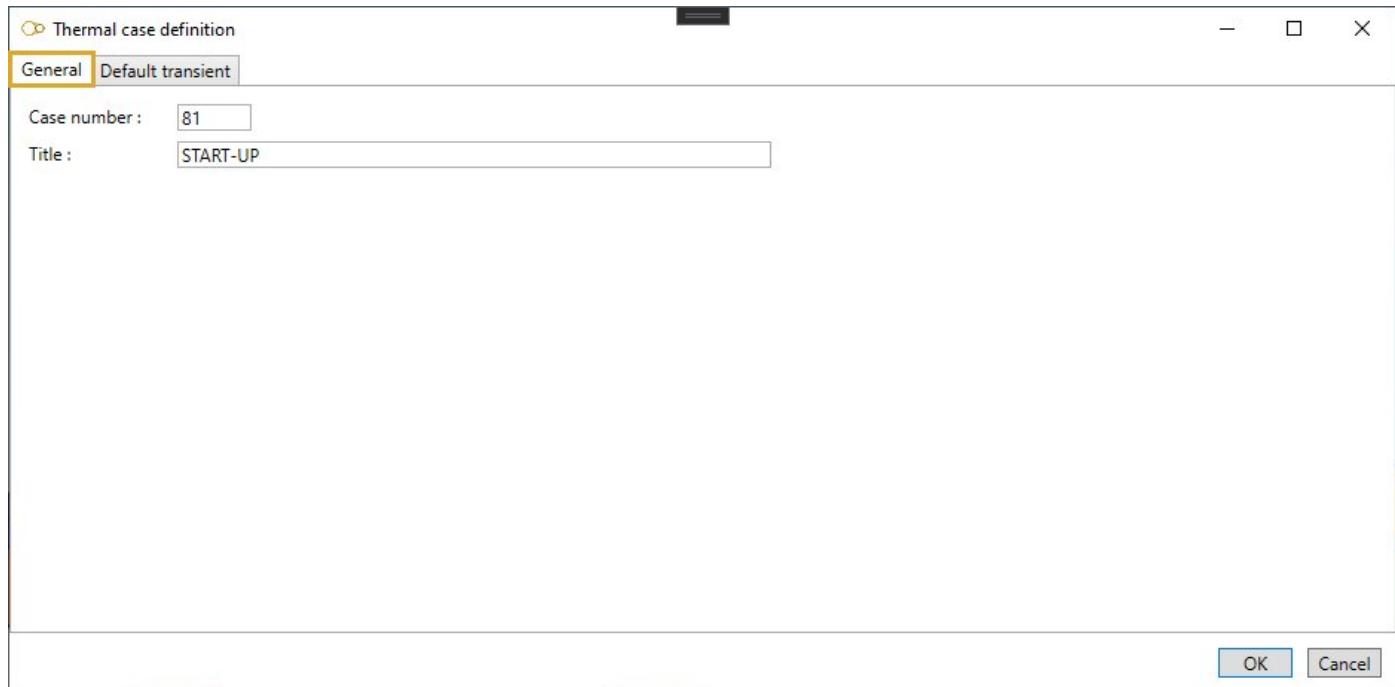
When selecting **Thermal cases**, all existing thermal cases are listed in the combobox :



The cases appear with their **Case number + Title**.

When editing, the definition window shows up :

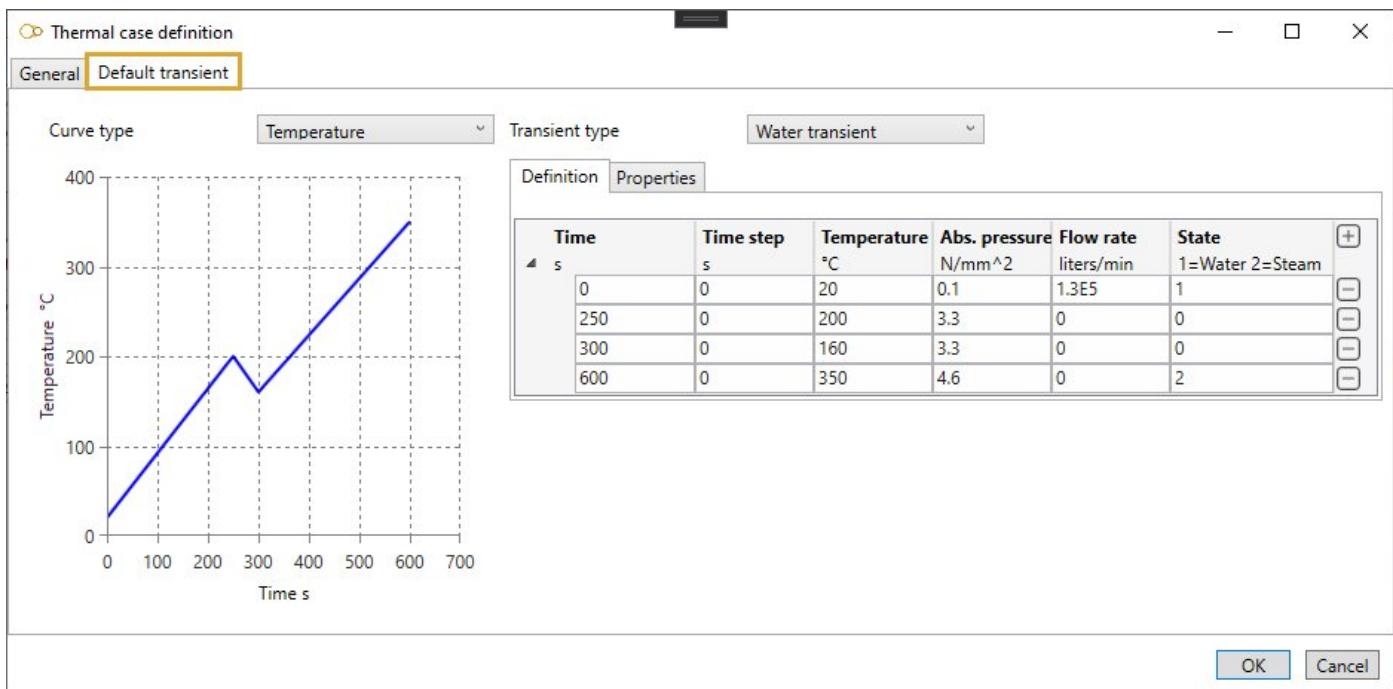
1. General



Enter a **Case number** and a **Title**.

2. Default transient

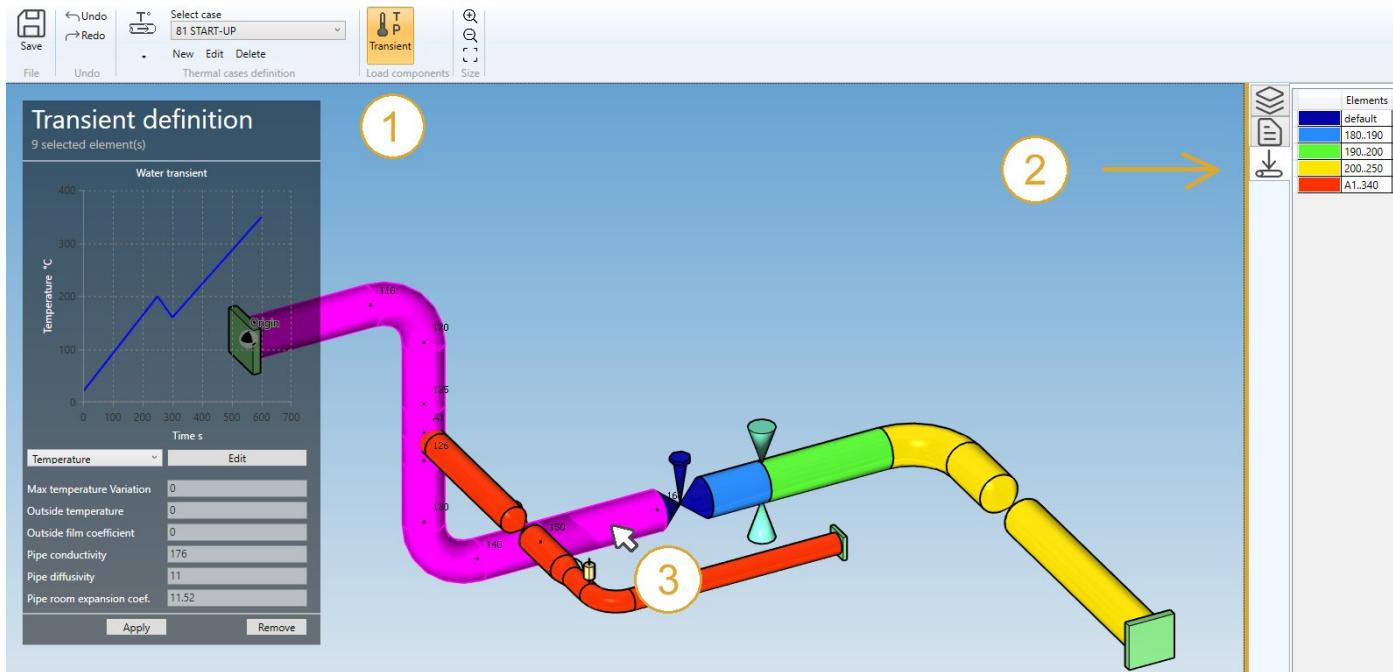
If defined, all elements of the model will use the same **default transient** :



See §5. for more information about edition of a **transient**.

3. Data panel

Click on the **Transient** button (1) to define transient to elements :



A right panel can be opened by clicking on the top button (2). It presents the elements with the same transient (1 color for each transient).

Click on the same button to hide the panel.

Select one or more element (3) to define a **Transient** (instead of the default one).

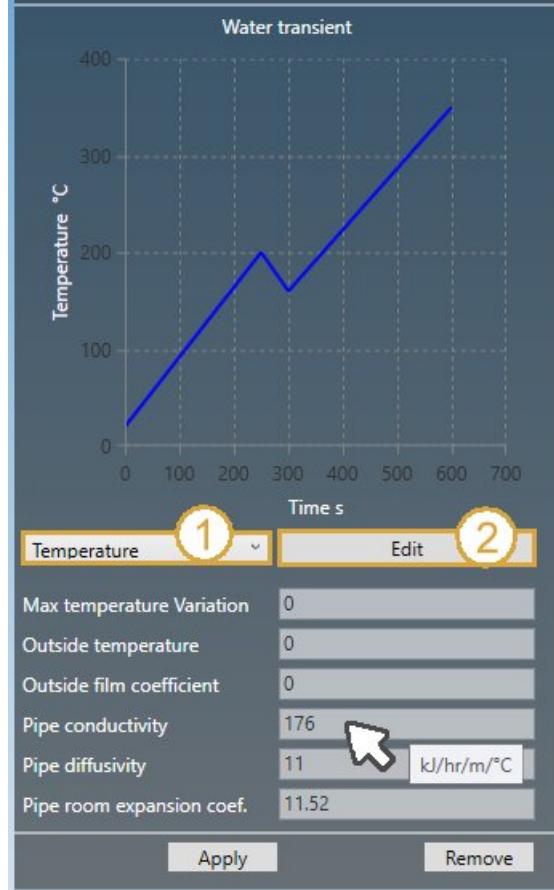
Click [here](#) for more information about the selection tool.

4. Left panel

The main properties of the selected transient (or the default one of the thermal case) are shown on the left panel :

Transient definition

9 selected element(s)



Select the graph type (1) :

- Evolution of the **temperature of the fluid** during time
- Evolution of the **absolute pressure of the fluid** during time
- Evolution of the **flow rate of the fluid** during time
- Evolution of the **state of the fluid** during time

Click on the **Edit** button (2) to modify the current transient.

See §5. for more information about edition of a **transient**.

Other properties (not editable) :

Property	Unit Metric	Unit USA	Remark
Max temperature variation	°C	°F	If blank, MetaPiping use the default value
Outside temperature	°C	°F	Temperature outside pipe
Outside film coefficient	kJ/hr/m ² /°C	btu/hr/ft ² /°F	External heat exchange coefficient
Pipe conductivity	kJ/hr/m/°C	btu/hr/ft/°F	If blank, MetaPiping uses the value entered in the material definition
Pipe diffusivity	mm ² /s	ft ² /hr	If blank, MetaPiping uses the value entered in the material definition
Pipe room expansion coefficient	10 ⁻⁶ .mm/mm/°C	10 ⁻⁶ .in/in/°F	Pipe thermal expansion coefficient at 21°C. If blank, MetaPiping uses the value entered in the material definition

To know the **UNIT** of a property, just move the mouse over the cells.

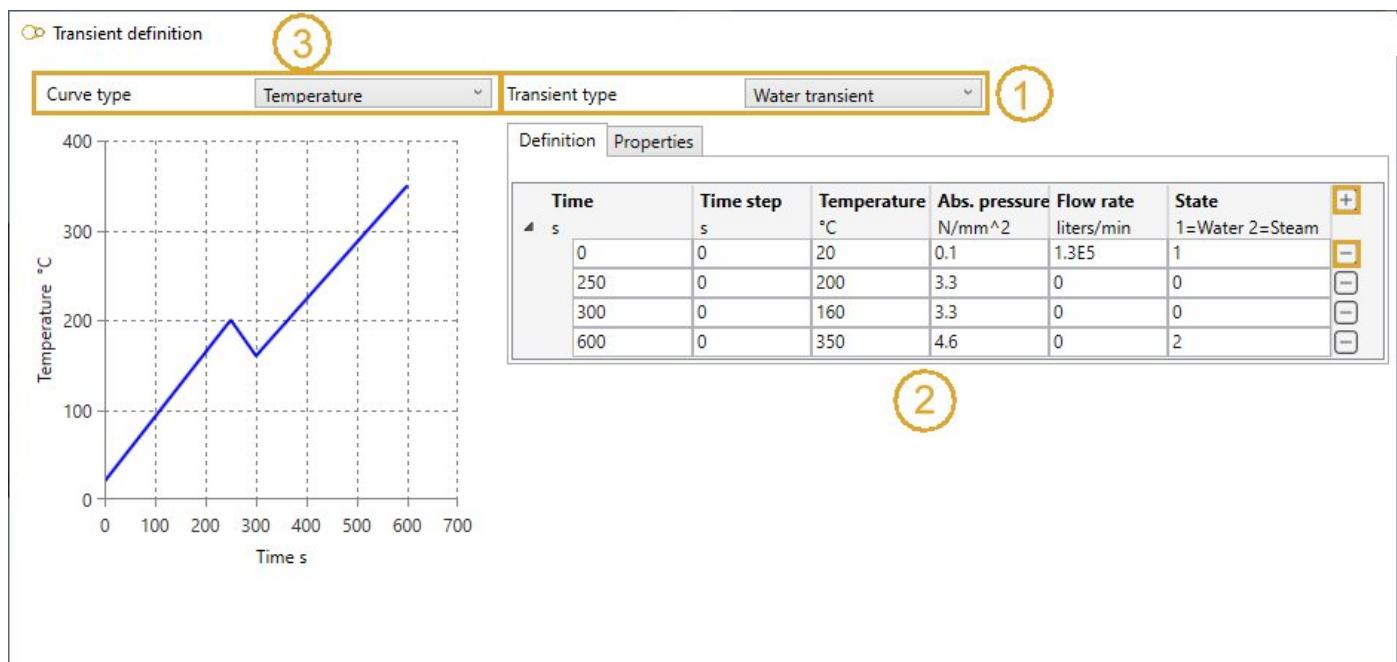
Click on the **Apply** button to define the **current transient** to selected elements.

Click on the **Remove** button to remove the **current transient** and reaffect the **Default transient** to selected elements.

5. Transient definition

To define a **Transient**, first select the type (1) :

- Water transient
- Fluid film coefficient



5.1 Definition

Enter the values of the graphs (2). The first time step must be 0.

Click on the + button to add a row. Click on the - button to remove a row.

For each row, define the properties :

Water transient :

Property	Unit Metric	Unit USA	Remark
Time	s	s	
Time step	s	s	Informative
Temperature	°C	°F	
Absolute pressure	N/mm²	psi	
Flow rate	liters/min	gallons/min	
State	-	-	1 for Water, 2 for Steam

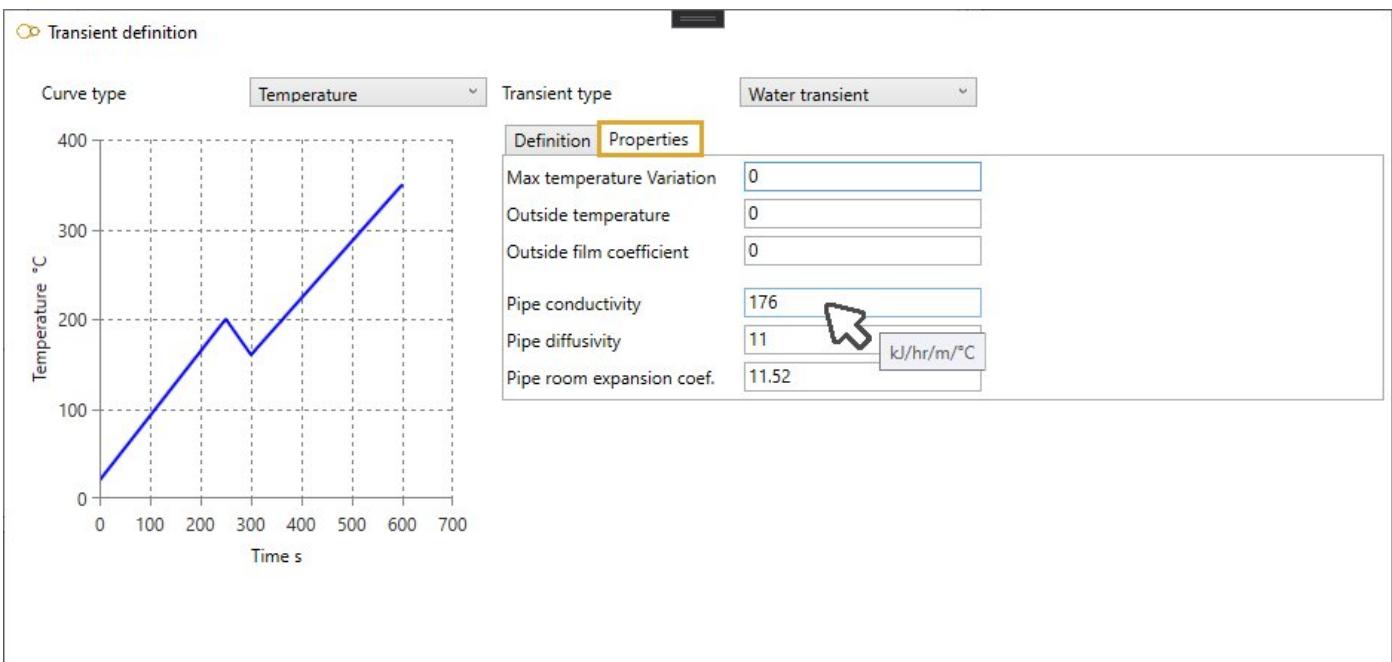
Fluid film coefficient :

Property	Unit Metric	Unit USA	Remark
Time	s	s	
Temperature	°C	°F	
Film coefficient	kJ/hr/m²/°C	btu/hr/ft²/°F	

Tip : for blank values, MetaPiping will make linear interpolations.

5.2 Properties

Enter other properties :

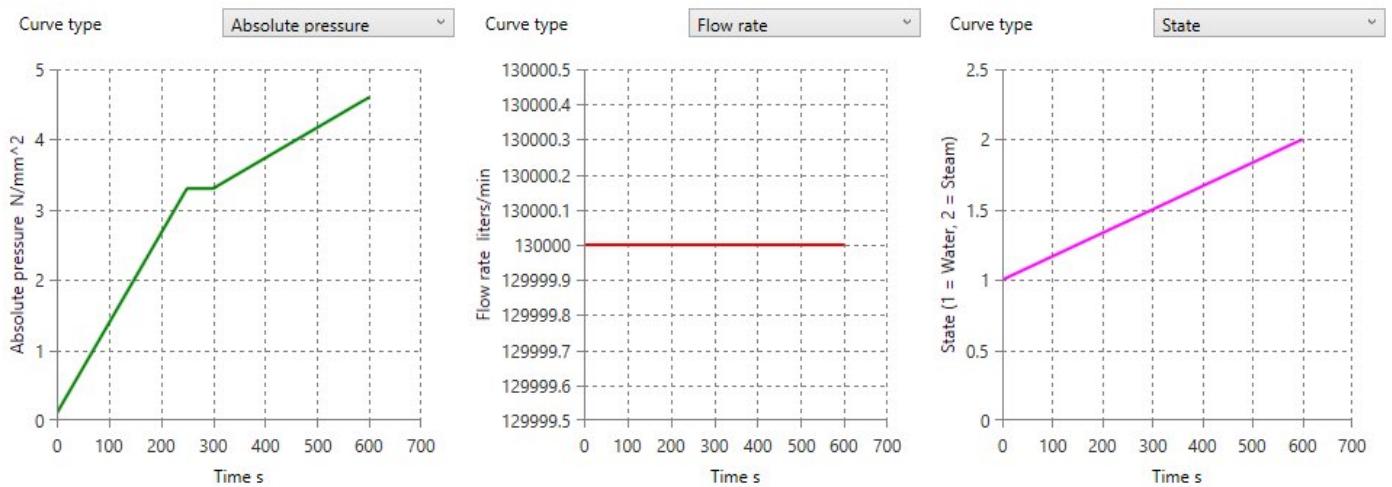


Property	Unit Metric	Unit USA	Remark
Max temperature variation	°C	°F	If blank, MetaPiping use the default value
Outside temperature	°C	°F	Temperature outside pipe
Outside film coefficient	kJ/hr/m ² /°C	btu/hr/ft ² /°F	External heat exchange coefficient
Pipe conductivity	kJ/hr/m/°C	btu/hr/ft/°F	If blank, MetaPiping uses the material property
Pipe diffusivity	mm ² /s	ft ² /hr	If blank, MetaPiping uses the material property
Pipe room expansion coefficient	10 ⁻⁶ .in/in/°F	10 ⁻⁶ .in/in/°C	Pipe thermal expansion coefficient at 21°C. If blank, MetaPiping use the material property

To know the UNIT of a property, just move the mouse over the cells.

5.3 Curve type

Select the **curve type** in the list (3) :





Spectra

This case consists of the definition of series of SPECTRA that describe the support movement loading for primary and secondary floor response cases.

When selecting **Spectra**, all existing spectra are listed in the combobox :



The spectra appear with their **Event number + Title**.

When editing, the definition window shows up :

1. General

Spectra definition

General **Level definition** **Spectra viewer**

Event number :	<input type="text" value="2"/>
Name :	<input type="text"/>
Title :	<input type="text" value="SPECTRE SMS"/>
X axis :	<input type="text" value="Frequency"/>
Interpolation :	<input type="text" value="Linear / Linear accel."/>
Shift :	<input type="text" value="0"/>
Percent damping :	<input type="text" value="0"/>
Duration :	<input type="text" value="0"/>

OK **Cancel**

Enter an **Event number**, a **Name** and a **Title**.

The primary and secondary floor response cases with the same **EVENT** will use the spectra defined herein.

X axis :

- Spectrum is **frequency** vs. acceleration
- Spectrum is **period** vs. acceleration

Interpolation :

- Linear / Linear acceleration
- Linear frequency / Linear acceleration
- Linear period / Linear acceleration
- Log frequency / Linear acceleration
- Log frequency / Log acceleration

Shift :

The shift (in %) determines an interval about each natural frequency. MetaPiping will use the maximum acceleration in each such interval, as determined by interpolating the spectrum data specified.

This method is used for peak broadening. 0 or blank = no shift.

Percent damping :

The damping is used to determine the intermodal correlation factors for the double sum, Rosenblueth and Der Kiureghian methods.

Duration :

Duration of earthquake (in seconds).

0 or blank = infinite (10^6 s).

2. Level definition

In this tab, you can create/remove **LEVELS** for the same **EVENT** :

The screenshot shows the 'Spectra definition' dialog box with the 'Level definition' tab selected. The 'General' tab is also visible. There are buttons for 'Add Level' and 'Remove Level'. A list of 11 levels (Level 1 to Level 11) is shown. Below the list are input fields for 'Level no.' (set to 1), 'DX', 'DY', 'DZ', and 'Phase' (set to 0). To the right of these fields is a database icon, which is highlighted with a yellow box. Below the input fields are three sections: 'Dir X', 'Dir Y', and 'Dir Z', each with a table of frequency and acceleration values. At the bottom are buttons for 'OK' and 'Cancel'.

The **LEVEL** number relates the spectra to the supports.

Click [here](#) to have more information about supports.

DX, DY, DZ :

Maximum expected X, Y, Z displacements (mm, in).

Phase :

The response for levels with the same phase number > 0 are assumed to be in phase and will be added algebraically.

Spectrum in direction X, Y, Z :

Enter **acceleration** for each **frequency/period**.

Attention, the frequency/period must be entered in ascending order.

The possible cases are :

- X + Y + Z = a spectrum for each direction
- X + YZ = a spectrum for X and a common one for Y and Z
- Y + XZ = a spectrum for Y and a common one for X and Z
- Z + XY = a spectrum for Z and a common one for X and Y
- XYZ = the same spectrum for all directions

F1 and F2 :

Low frequency f1 and high frequency f2 for Gupta and Lindley-Yow methods.

2.1 Database

The **Database** button on upper right corner gives an access to the spectra libraries :

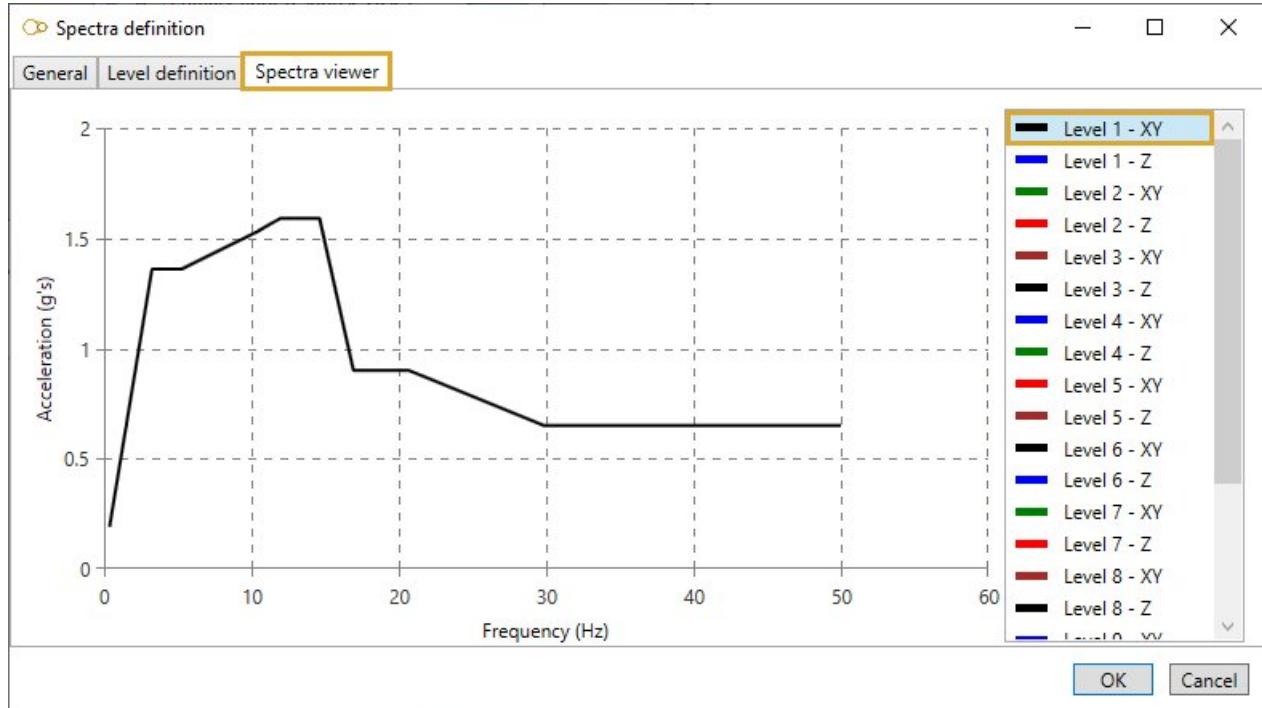


Select a library on the left treeview, then select a row that represents a particular damping for a predefined spectrum. Some values are just informative. Click **OK**. The properties are copied to the spectrum screen.

Click [here](#) to have more information about spectrum database.

3. Spectra viewer

Select a level on the right list to see the spectrum graph :



4. Restraints

You can now select **restraints** or create one on a selected node and affect the **Level** property (set the level number) :

Restraint

Set restraint definition

Select restraint type

Multiple restraint

Label

Fixed rigid restraint in X,Y,Z directions

Dx Rx

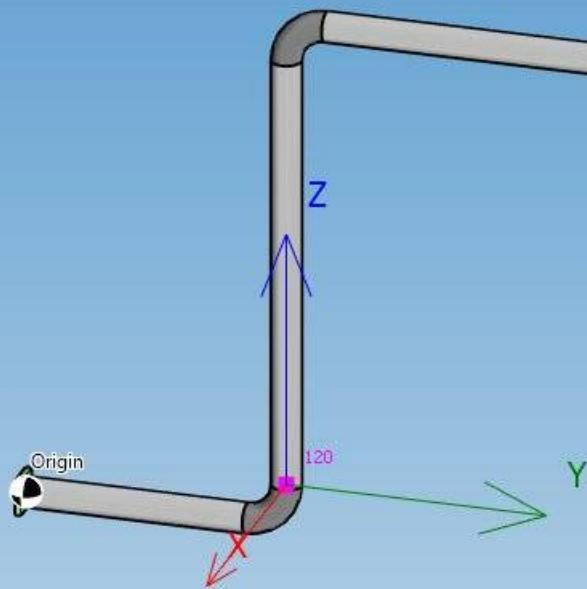
Dy Ry

Dz Rz

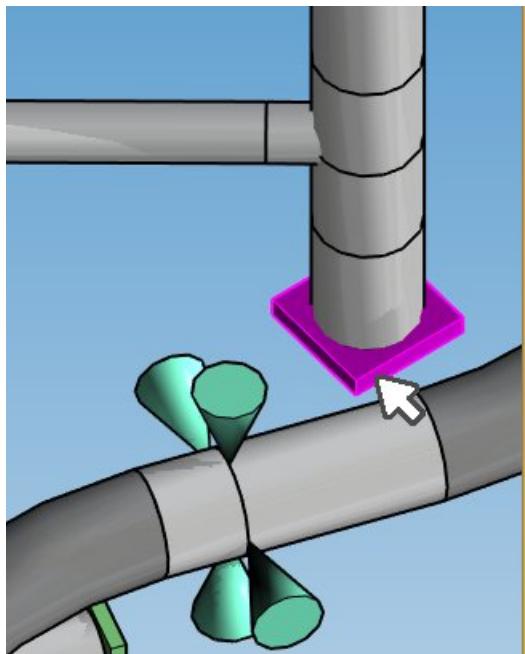
Coord. system Global

Level

Create



You can also check the property on **Review** mode (type Restraints):



Data lists

Node	Restraint type	Level	
P295	Anchor	2	
P272	Anchor	2	
P253	LinearRestraint	8	
P253	LinearRestraint	8	
P238	Anchor	6	
P148	Anchor	2	
P112	Anchor	2	
P111	Anchor	2	
P081	Anchor	10	
307C	LinearRestraint	2	
307C	LinearRestraint	2	
301C	LinearRestraint	2	
301C	LinearRestraint	2	
289C	LinearRestraint	2	
289C	LinearRestraint	2	
271C	LinearRestraint	2	
271C	LinearRestraint	2	
260C	LinearRestraint	2	



Primary floor response cases

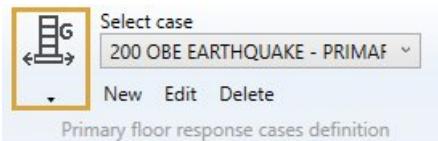
This load describes the floor response spectra for each support level. This analysis is used to obtain bound solutions for simple multilevel cases (including single level cases) for which the dynamic loading is entirely due to movements at the support points.
The "levels" are groups of supports which are moving in parallel and are entered using the level number field on supports.

WARNING

Attention, to obtain the complete solution for multilevel problems, it is necessary to calculate the primary and secondary parts of the solution separately and then to combine them.

MetaPiping will calculate the bounds for the **PRIMARY** part of the solution $u(t)$ for displacements, rotations, forces, moments and stresses due to the dynamic support movement loading.

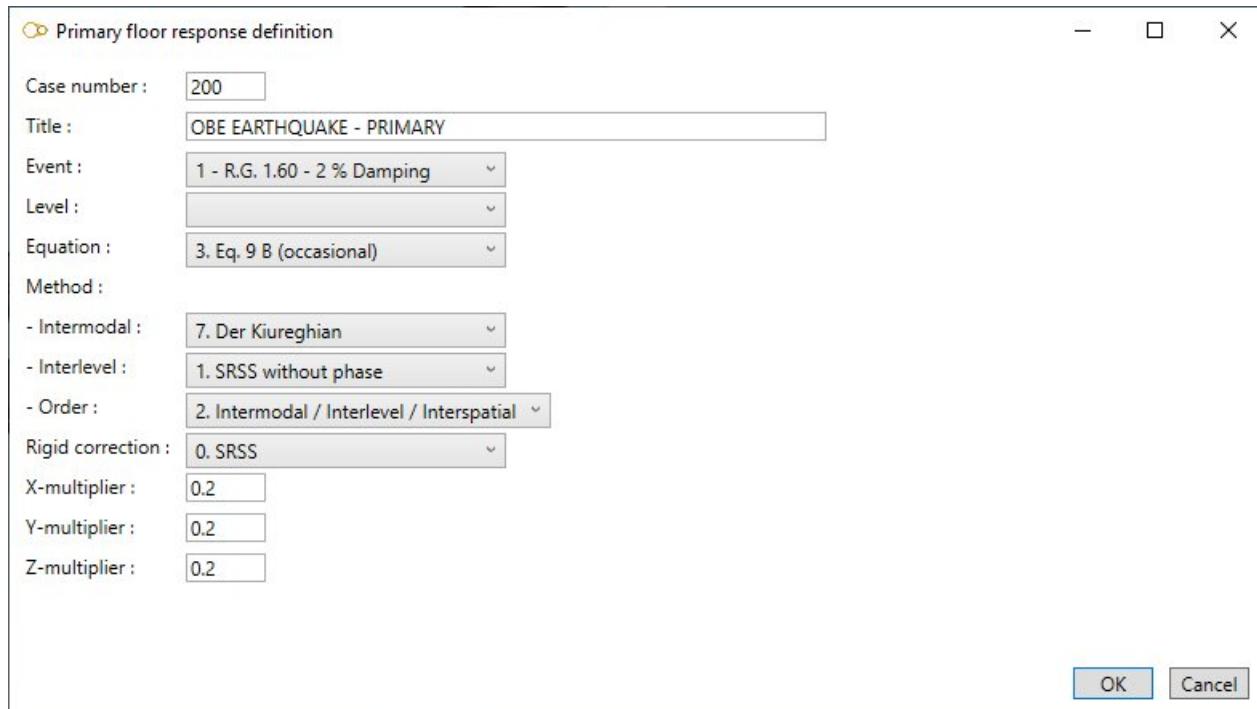
When selecting **Primary floor response cases**, all existing primary floor response cases are listed in the combobox :



The cases appear with their **Case number + Title**.

1. General

When editing, the definition window shows up :



Enter a **Case number** and a **Title**.

Event :

Select a **Spectra** : *R.G. 1.60 - 2% Damping* in this example.

See §2.

Level :

This field is intended for nuclear piping codes only. The possible values are : A (normal conditions), B (upset), C (emergency), D (faulted) and T (test).

Equation :

The equations are code-dependent.

Method :

For intermodal :

- 1- Grouping
- 2- Ten percent
- 3- Double sum
- 4- SRSS
- 5- All coupling
- 6- Rosenblueth
- 7- Der Kiureghian

For interlevel :

- 0- Absolute without phase
- 1- SRSS without phase
- 2- Algebraic
- 3- Absolute with phase
- 4- SRSS with phase
- 5- Envelope
- 6- SRSS with counterphase

For order :

- 0- Interlevel / Intermodal / Interspatial
- 1- Interlevel / Interspatial / Intermodal
- 2- Intermodal / Interlevel / Interspatial

Rigid correction :

- 0- SRSS
- 1- None
- 2- Absolute
- A- SRSS with modal
- B- Gupta
- C- Lindley-Yow

X, Y, Z multipliers :

This factor multiplies the X, Y, Z component of the loading.

If a factor is zero or blank, then the corresponding loading component will be zero.

2. Spectra definition

In this example, we use this **Spectra loading** :

Spectra definition

General Level definition Spectra viewer

Event number : 1

Name : RG1.60

Title : R.G. 1.60 - 2 % Damping

X axis : Frequency

Interpolation : Log freq. / Linear accel.

Shift : 0

Percent damping : 0

Duration : 0

It contains 2 **LEVELS** with corresponding spectra :

Spectra definition

General Level definition Spectra viewer

Add Level Remove Level

Level 1 Level 2

Level no. : 1 DX : 5 DY : 0 DZ : 7 Phase : 0

Dir X

	Freq. (Hz)	Acc. (g's)
1	0.25	0.58
2	2.5	4.25
3	9	3.54
4	33	1
5		

Dir Y Y Y=X

	Freq. (Hz)	Acc. (g's)
1	0.25	0.38
2	3.5	4.05
3	9	3.54
4	33	1
5		

Dir Z Z Z=X Z=Y

	Freq. (Hz)	Acc. (g's)

F1 (Hz) : 0 F2 (Hz) : 0 F1 (Hz) : 0 F2 (Hz) : 0 F1 (Hz) : F2 (Hz) :

OK Cancel

Click [here](#) for more information about the spectra definition.

3. Restraint definition

In this example, all restraints are assigned a **LEVEL** :

MetaPiping [transient] - (Study5)

Home Project Design Loads Review Analysis

Select Measure Node coordinates Elements Sections Materials Restraints Tees End conditions Lumped masses Local coordinates Specifications SIF Stress On/Off

Data lists

Node	Restraint type	Level
150	LinearRestraint	1
190	LinearRestraint	1
250	Anchor	1
100	Anchor	2
320	SpringHanger	2
340	Anchor	2

Level 1 for the main line.

Level 2 for the secondary line.

4. Secondary floor response cases

Click [here](#) for more information about the secondary floor response cases.



Secondary floor response cases

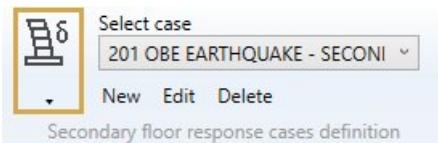
This load describes the floor response spectra for each support level. This analysis is used to obtain bound solutions for simple multilevel cases (including single level cases) for which the dynamic loading is entirely due to movements at the support points.
The "levels" are groups of supports which are moving in parallel and are entered using the level number field on supports.

WARNING

Attention, to obtain the complete solution for multilevel problems, it is necessary to calculate the primary and secondary parts of the solution separately and then to combine them.

MetaPiping will calculate the bounds for the **SECONDARY** part of the solution $v(t)$ for displacements, rotations, forces, moments and stresses due to the dynamic support movement loading.

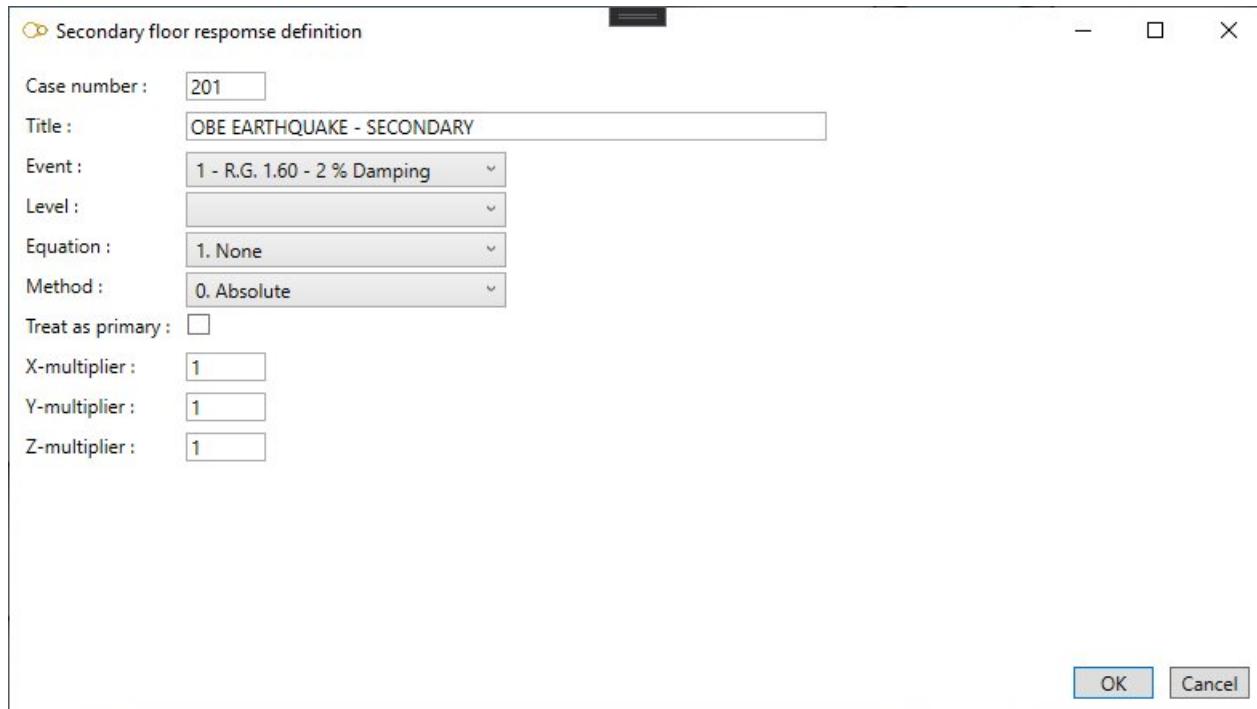
When selecting **Secondary floor response cases**, all existing secondary floor response cases are listed in the combobox :



The cases appear with their **Case number + Title**.

1. General

When editing, the definition window shows up :



Enter a **Case number** and a **Title**.

Event :

Select a **Spectra** : *R.G. 1.60 - 2% Damping* in this example.

See §2.

Level :

This field is intended for nuclear piping codes only. The possible values are : A (normal conditions), B (upset), C (emergency), D (faulted) and T (test).

Equation :

The equations are code-dependent.

Method :

The solutions for each group of supports are combined using one of the following method in order to obtain the total secondary response :

- 0- Absolute
- 1- SRSS

Treat as primary :

Secondary floor response cases are treated as secondary by default. Depending on the piping code, the moments may then be multiplied by Ec/Eh (like thermal expansion) when the **Hot modulus** is used. If **Treat as primary** is checked, the moments are not multiplied by Ec/Eh. This is recommended when the total seismic moments (primary + secondary) are used in combination with the dead weight moments in the equation for occasional stresses.

X, Y, Z multipliers :

This factor multiplies the X, Y, Z component of the loading.

If a factor is zero or blank, then the corresponding loading component will be zero.

2. Spectra definition

In this example, we use this **Spectra loading** :

Spectra definition	
General Level definition Spectra viewer	
Event number :	1
Name :	RG1.60
Title :	R.G. 1.60 - 2 % Damping
X axis :	Frequency
Interpolation :	Log freq. / Linear accel.
Shift :	0
Percent damping :	0
Duration :	0

It contains 2 **LEVELS** with corresponding spectra :

Spectra definition

General Level definition Spectra viewer

Add Level Remove Level

Level 1 Level 2

Level no.: 1 DX: 5 DY: 0 DZ: 7 Phase: 0 

Dir X

	Freq. (Hz)	Acc. (g's)
1	0.25	0.58
2	2.5	4.25
3	9	3.54
4	33	1
5		

F1 (Hz): 0 F2 (Hz): 0

Dir Y Y Y=X

	Freq. (Hz)	Acc. (g's)
1	0.25	0.38
2	3.5	4.05
3	9	3.54
4	33	1
5		

F1 (Hz): 0 F2 (Hz): 0

Dir Z Z Z=X Z=Y

	Freq. (Hz)	Acc. (g's)
1		
2		
3		
4		
5		

F1 (Hz): F2 (Hz):

OK **Cancel**

Click [here](#) for more information about the spectra definition.

3. Primary floor response cases

Click [here](#) for more information about the primary floor response cases.



Time history cases

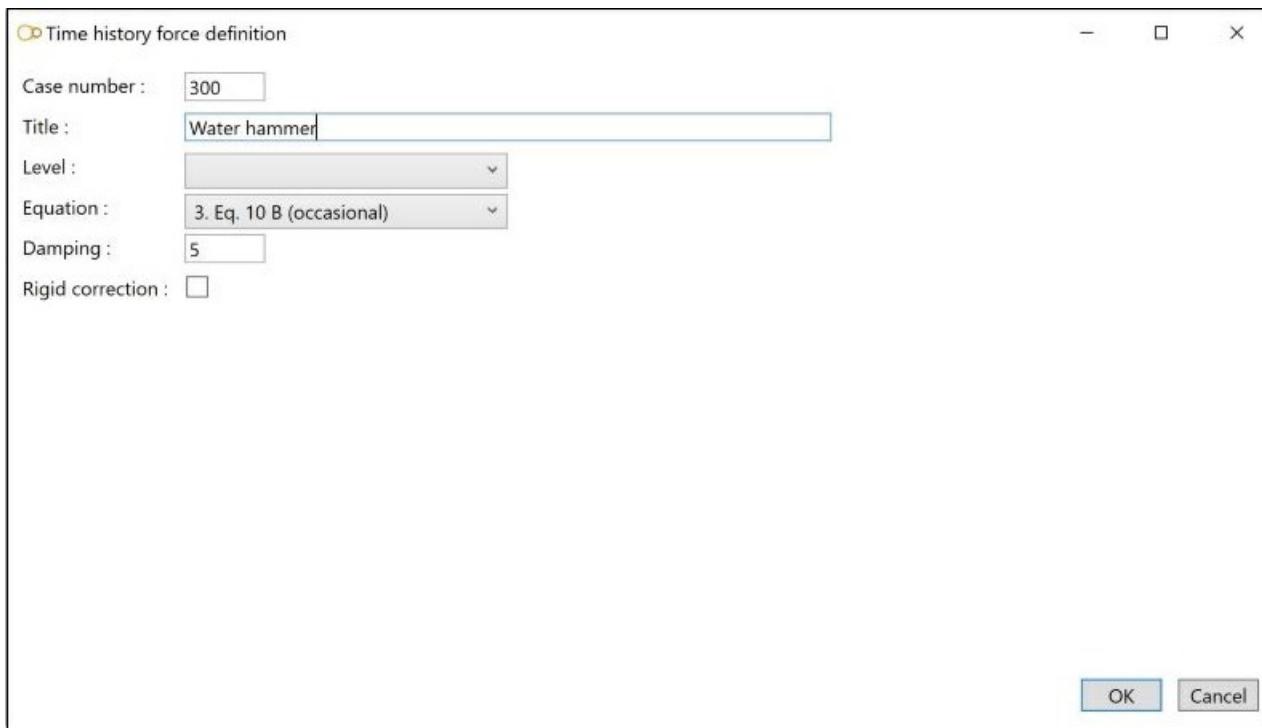
This case consists of dynamic forces and moments applied to some nodes of the model. The solution is obtained using the modal superposition method. Time histories are entered in an external THF file connected to the study beforehand.

When selecting **Time history cases**, all existing time history cases are listed in the combobox :



The cases appear with their **Case number + Title**.

When editing, the definition window shows up :



Enter a **Case number** and a **Title**.

Level :

This field is intended for nuclear piping codes only. The possible values are : A (normal conditions), B (upset), C (emergency), D (faulted) and T (test).

Equation :

The equations are code-dependent.

Damping :

Enter the damping value (in percentage).

Rigid correction :

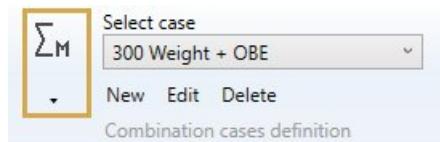
If checked, include the effect of the higher "rigid" modes (i.e. modes above the cut-off frequency).



Combination cases

This case consists of combining the results of load cases, dynamic cases or previously calculated combination cases to form a new case.

When selecting **Combination cases**, all existing combinations are listed in the combobox :



The cases appear with their **Case number + Title**.

1. General

When editing, the definition window shows up :

Case	Factor
1	200
2	100
3	
4	
5	
6	
7	

Case	Type	Title
100	Static	OPERATING WEIGHT
101	Static	NORMAL OPERATION
102	Static	50% POWER
200	Primary floor resp	OBE EARTHQUAKE - PRIMARY
201	Secondary floor r	OBE EARTHQUAKE - SECONDARY

Enter a **Case number** and a **Title**.

Ref. case :

Operating pressures (and allowable stresses) of the reference case will be used for this case. The calculation of the allowable stresses depends on the current **calculation code**.

Click [here](#) for more information about all possible codes.

Level :

This field is intended for nuclear piping codes only. The possible values are : A (normal conditions), B (upset), C (emergency), D (faulted) and T (test).

Equation :

The equations are code-dependent.

Method :

- 0- Algebraic addition
- 1- Absolute addition
- 2- SRSS
- 3- Seismic
- 4- Maximum absolute
- 6- Algebraic maximum
- 7- Algebraic minimum
- 8- Range
- 9- Max resultant moment range
- S- Max thermal stress range

Duration :

In hours.

Dynamic load flag :

- For ASME Class 1 : the flag is a number from 1 to 9 that identifies the dynamic moments to be used in Equation 13
- For RCC-M Class 1 : the flag with number 1 identifies the operational earthquake

Number of cycles :

- For ASME Class 1 : number of sub-cycles predicted for each occurrence of the dynamic event (default = 1)
- For RCC-M Class 1 : number of occurrences of the seismic event if the analysis type is 0, or number of seismic sub-cycles if the analysis type is 1

Analysis type :

- For RCC-M Class 1 only : enter 0 (or blank) for the total earthquake moments and 1 for the primary (inertial) earthquake moments

2. Combined cases

Enter the **number** of the constituent cases and the corresponding **factor** (1) :

Combined cases		Filter	All cases	
Case	Factor	Case	Type	Title
1 200	1	100	Static	OPERATING WEIGHT
2 100	1	101	Static	NORMAL OPERATION
3		102	Static	50% POWER
4	1	200	Primary floor resp	OBE EARTHQUAKE - PRIMARY
5		201	Secondary floor n	OBE EARTHQUAKE - SECONDARY
6				
7				

On right side, as a reminder, the list of all cases (2). You can filter them by selecting the type :

- All
- Static
- Dynamic
- Combination

Stress combination cases

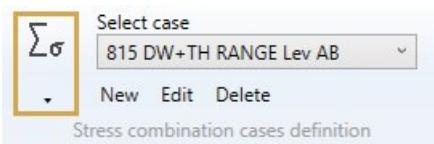
This case consists of combining the results of load cases, dynamic cases or previously calculated combination cases to form a new case and of calculating additive stresses which are based on the resultant moments of the constituent cases.

WARNING

ATTENTION, this case may be used for all piping codes except Class 1 and ASME HDPE Code.

Click [here](#) for more information about the piping codes.

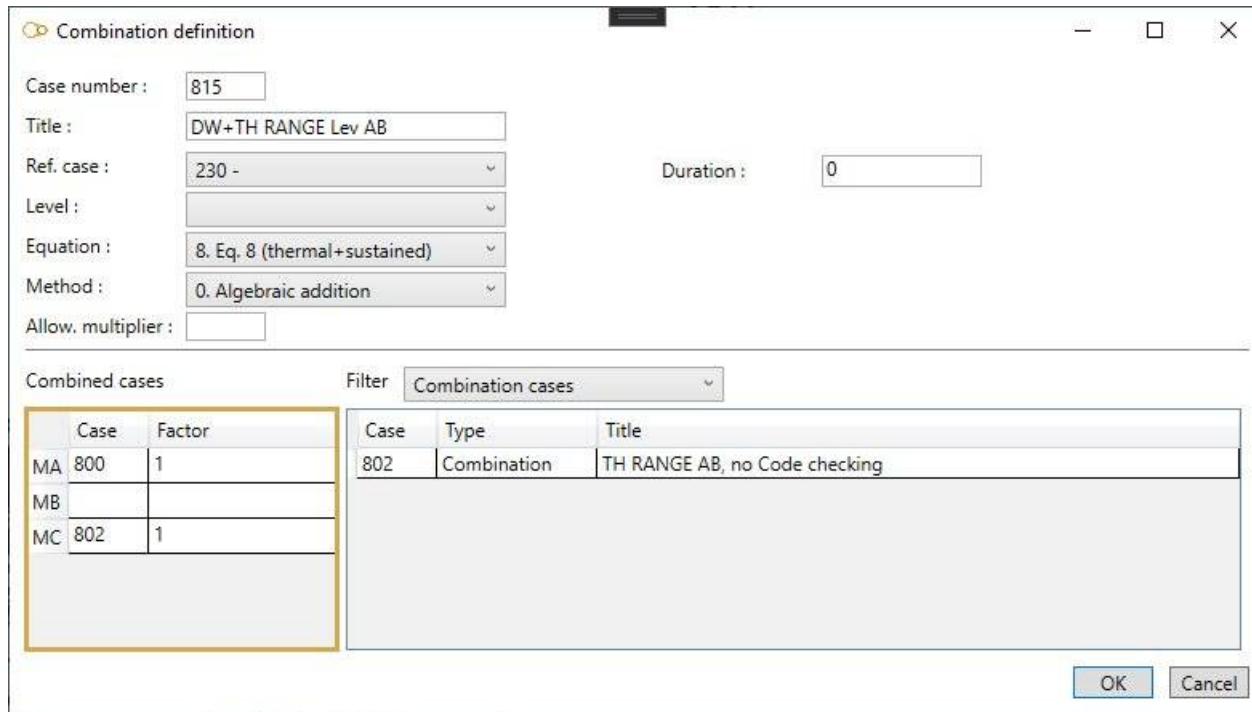
When selecting **Stress combination cases**, all existing stress combinations are listed in the combobox :



The loads appear with their **Case number + Title**.

1. General

When editing, the definition window shows up :



Enter a **Case number** and a **Title**.

Ref. case :

Operating pressures (and allowable stresses) of the reference case will be used for this case. The calculation of allowable stresses depends on the current **calculation code**.

Click [here](#) for more information about all possible codes.

Level :

This field is intended for nuclear piping codes only. The possible values are : A (normal conditions), B (upset), C (emergency), D (faulted) and T

(test).

Equation :

The equations are code-dependent.

Method :

- 0- Algebraic addition
- 1- Absolute addition
- 2- SRSS
- 3- Seismic
- 4- Maximum absolute
- 5- Maximum resultant
- 6- Algebraic maximum
- 7- Algebraic minimum
- 8- Range
- 9- Max resultant moment range
- S- Max thermal stress range

Allowable multiplier f :

If this field is not blank and different from 0, then the allowable stress for this case will be $f * Sh$.

Duration :

In hours.

2. Combined cases

Enter the **number** of the constituent cases and the corresponding **factor**.

MA = Sustained load case

MB = Occasional load case

MC = Thermal load case

Combined cases		Filter	Combination cases	
Case	Factor	Case	Type	Title
MA 800	1	802	Combination	TH RANGE AB, no Code checking
MB				
MC 802	1			

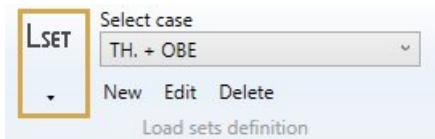
On right side, as a reminder, the list of all cases. You can filter them by selecting the type :

- All
- Static
- Dynamic
- Combination

Load sets (Class 1 only)

A load set is a specific combination of load cases assumed to act simultaneously on a piping system at a given instant. Pairs of load sets are employed in pipe stress analysis to determine physical changes in the system from one load set instant to another. Typical load set parameters, which may occur in various combinations, are : pressure, temperature, moment and thermal transient effects.

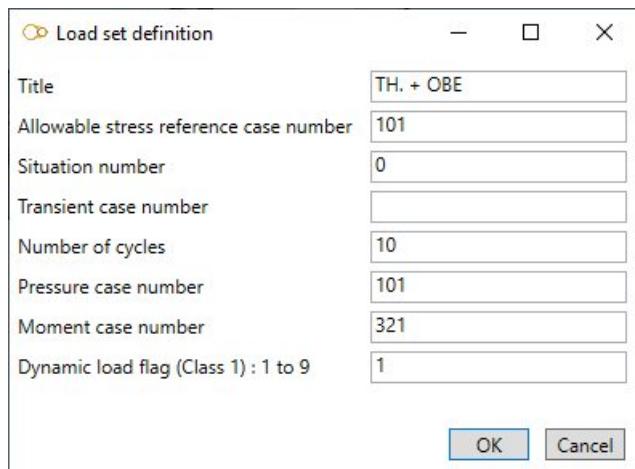
When selecting **Load sets**, all existing load sets are listed in the combobox :



The load sets appear with their **Title**. There is no **Case number** for that kind of case.

1. General

When editing, the definition window shows up :



Enter a **Title**.

Allowable stress reference case number :

The temperature of the reference case is used to determine the allowable stress intensity Sm.

Situation number :

For RCC-M Class 1 code only : the two load sets, which form a situation, must be defined with the same situation number. Moreover, the two load sets must have the same number of cycles. The situations are used to determine the maximum value of the penalty factor Ke.

Transient case number :

Number of the thermal case associated with the load set.

Use positive (+) sign for heat up transients
Use negative (-) sign for cool down transients

Number of cycles :

Number of expected occurrences.

Pressure case number :

The pressure associated with the load set is taken from the pressure case.

If the moment case for this load set is a dynamic case, then moments used in evaluating equation 12 will be the moments for the pressure case multiplied by Ec/Eh.

Moment case number :

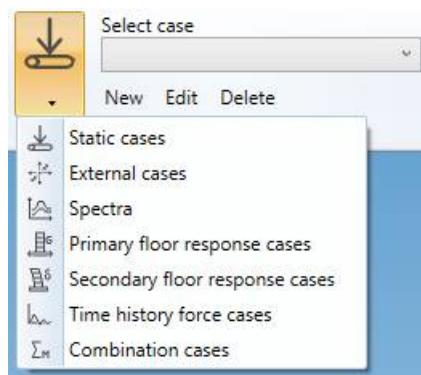
The moments associated with the load set are taken from the moment case.

Dynamic load flag :

For ASME Class 1 only : if the moment case for this load set is a dynamic case, then the moments used in the evaluation of equation 13 are taken from the combination case with the same flag number.

Structure loads

Click on the **LOAD** button to have access to the **load case types**:

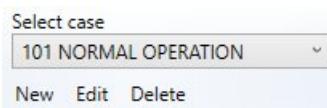


After selection, the choice becomes the **current load case type**.

External cases is visible only if there is at least one linked study and one or more links between piping nodes and structure nodes.

1. Edition

For all case types, you can **create**, **modify** or **remove** a selected case :



- **New** : open an empty window of **current load case type** : press OK to create the load
- **Edit** : open a window of **current load case type** with the selected case : press OK to modify the load case
- **Delete** : remove the selected case
- **Selection** : list of all cases of the **current load case type**

TABLE OF CONTENTS

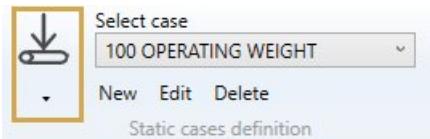
- [Static cases](#)
- [External cases](#)
- [Spectra](#)
- [Primary floor response cases](#)
- [Secondary floor response cases](#)
- [Time history cases](#)
- [Combination cases](#)



Static cases

Static cases consist of internal and/or external static loadings on the piping system.
MetaStructure will calculate corresponding displacements, rotations, forces, moments and stresses.

All existing static cases are listed in the combobox with their **Case number + Title** :



When editing or creating a static case, the definition window shows up. There are three tabs : General, Options and Applied loads.

1. General

Static case definition

General Options Applied loads

Case number :	100
Title :	OPERATING WEIGHT
Level :	(dropdown menu)
Equation :	2. All
Check fasteners :	<input type="checkbox"/>

OK **Cancel**

Enter a **Case number** and a **Title**.

Level :

This field is intended for nuclear codes only. The possible values are : A (normal conditions), B (upset), C (emergency), D (faulted) and T (test).

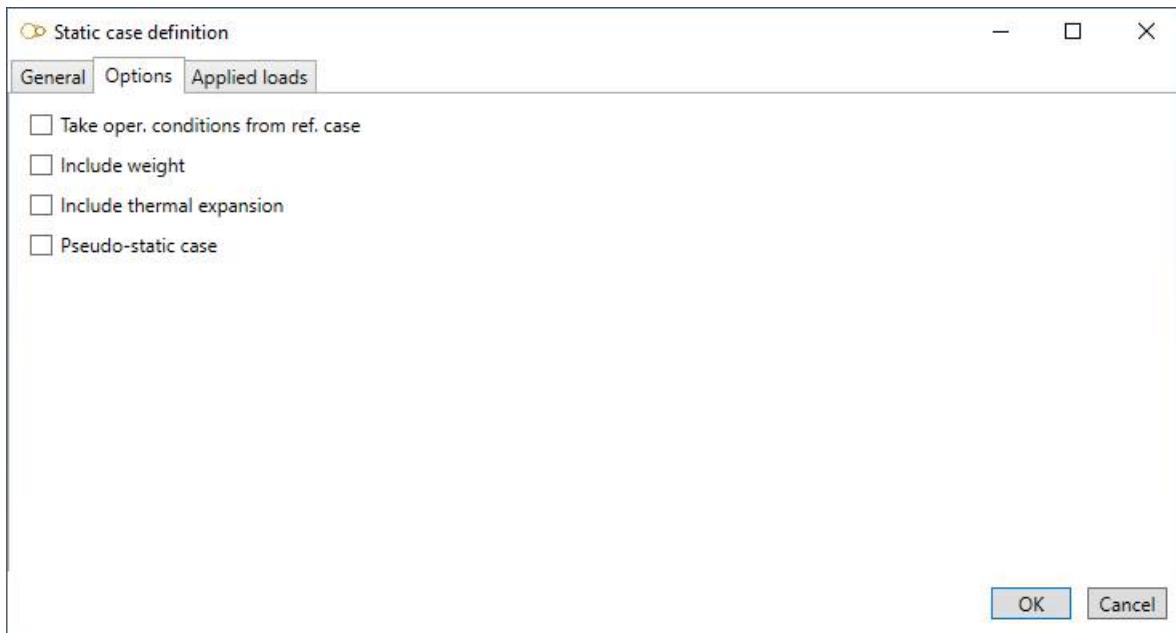
Equation :

- 1- None (no stress calculation)
- 2- All

Check fasteners :

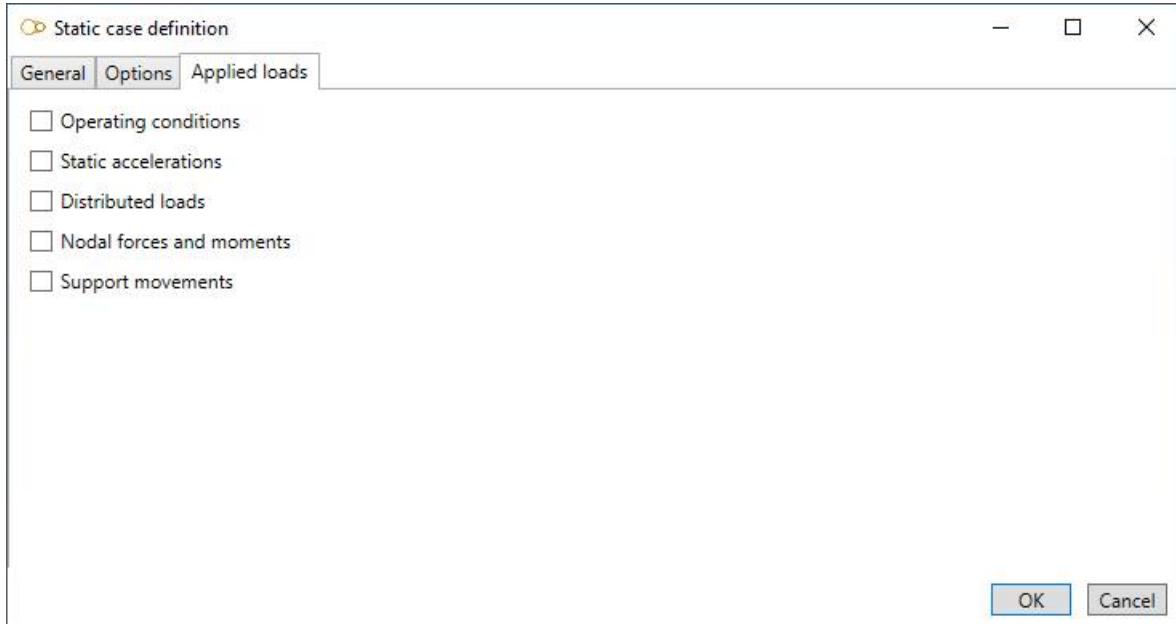
If checked, the fasteners will be verified.

2. Options



- **Take oper. conditions from ref. case** : select the load case which temperature will be used for thermal expansion and Young modulus. If no reference case is selected and no operating conditions are defined for the load case, room temperature is used.
- **Include weight** : if checked, the weight is considered in the calculation.
- **Include thermal expansion** : if checked, thermal expansion is considered in the calculation
- **Pseudo-static cases** : to be checked for dynamic cases that are calculated as static cases (e.g. earthquake calculated with a pseudo-static acceleration)

3. Applied loads



Select the loads that will be applied to the load case. For some loads like temperature, it is possible to set default values that will apply to all elements. The user can of course supersede the default values for some elements by selecting them and specifying different values.

3.1 Operating conditions

<input checked="" type="checkbox"/> Operating conditions	<input checked="" type="checkbox"/> Set default	Temperature :	50
--	---	---------------	----

You can define operating conditions on elements of the model.

The temperature is used for Young modulus, allowable stress and also for thermal expansion if **Include thermal expansion** is checked.

You can define **DEFAULT** operating conditions that will be applied on all elements of the model.

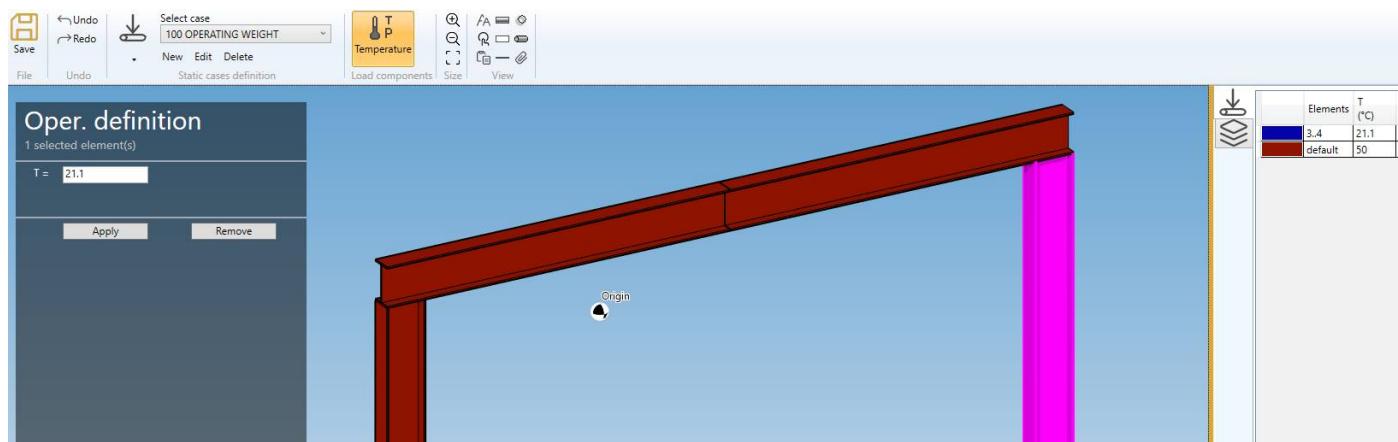
Define the default **temperature** of the load :

Property	Unit Metric	Unit USA
Temperature	°C	°F

If you check this option, you will see this **button** in the ribbon:



- 1 Select the **Temperature** button
- 2 Open the right **Data panel**
- 3 Select one or more elements
- 4 Define the new **temperature** for these elements
- 5 Apply



You can **undo** this command.

Click on the **Remove** button to delete the operating load on selected elements.

3.2 Static accelerations

<input checked="" type="checkbox"/> Static accelerations	<input checked="" type="checkbox"/> Set default	GX : 0.5
		GY : 0
		GZ : 0

By checking **Static acceleration** in **Applied loads** you can define acceleration loads on elements of the model.

You can also define **DEFAULT** accelerations that will be applied to all elements of the model :

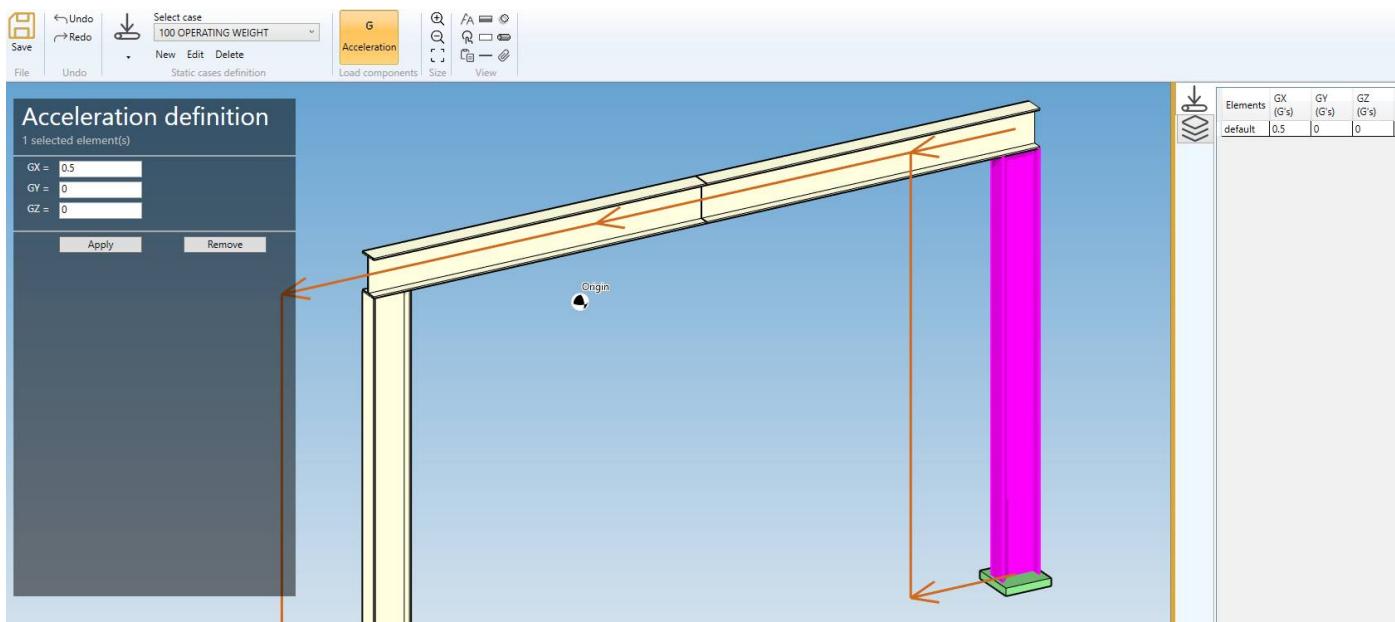
Property	Unit Metric	Unit USA
GX	fraction of G	fraction of G
GY	fraction of G	fraction of G
GZ	fraction of G	fraction of G

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Acceleration** button

- 2 Open the **Data panel** on the right
- 3 Select one or more elements
- 4 Define the new **vector acceleration** for these elements
- 5 Apply



You can **undo** this command.

Click on the **Remove** button to delete the load on selected elements.

You can also change the size of the drawn accelerations by clicking on these buttons :



3.3 Distributed loads

By checking **Distributed loads** in **Applied loads** you can define distributed loads on elements of the model. There are three types of distributed loads: simple distributed load (force per unit length), wind and snow.

Simple distributed loads may be applied for weight and occasional cases other than wind and snow cases. Wind loads may only be applied for wind cases and snow loads for snow cases.

3.3.1 SIMPLE DISTRIBUTED LOAD (DISTRIBUTED)

<input checked="" type="checkbox"/> Distributed loads	Distributed	<input checked="" type="checkbox"/> Set default	FX :	0
			FY :	-2
			FZ :	0 N/m

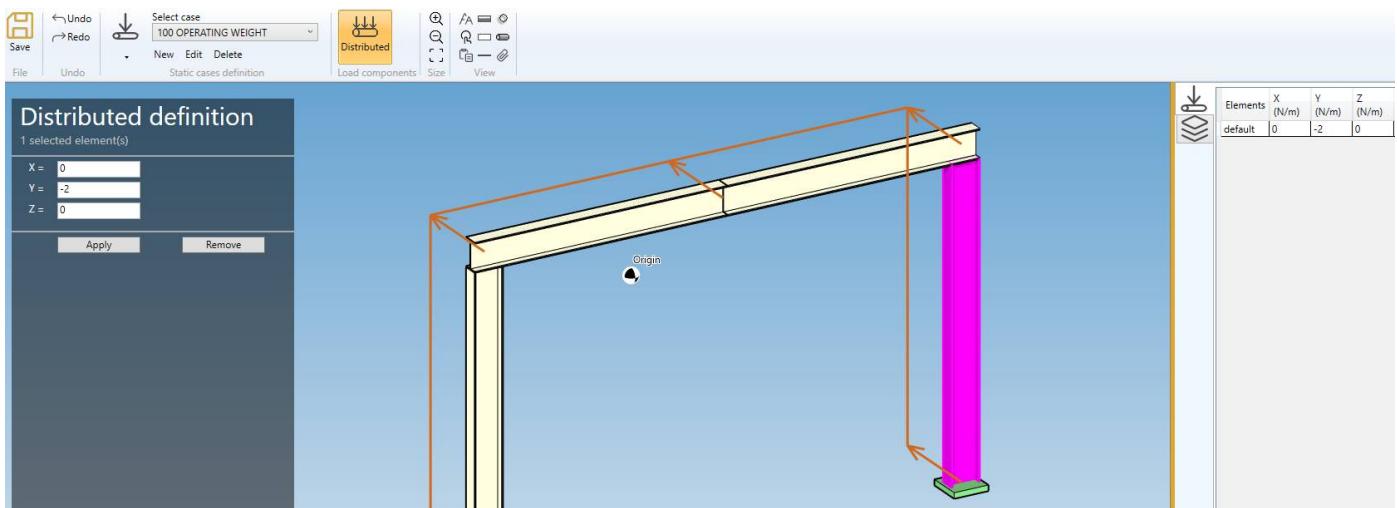
You can define **DEFAULT** distributed loads that will be applied to all elements of the model :

Property	Unit Metric	Unit USA
FX	N/m	lb/ft
FY	N/m	lb/ft
FZ	N/m	lb/ft

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Distributed** button
- 2 Open the right **Data panel**
- 3 Select one or more elements
- 4 Define the new **vector force** for these elements
- 5 Apply



You can **undo** this command.

Click on the **Remove** button to delete the load on selected elements.

You can also change the size of the drawn forces by clicking on these buttons :



3.3.2 WIND LOAD

<input checked="" type="checkbox"/> Distributed loads	Wind	<input checked="" type="checkbox"/> Set default	Pressure :	0.1
			Dir. /X :	0
			Dir. /Y :	1 N/m ²
			Dir. /Z :	0

You can define **DEFAULT** wind loads that will be applied to all elements of the model.

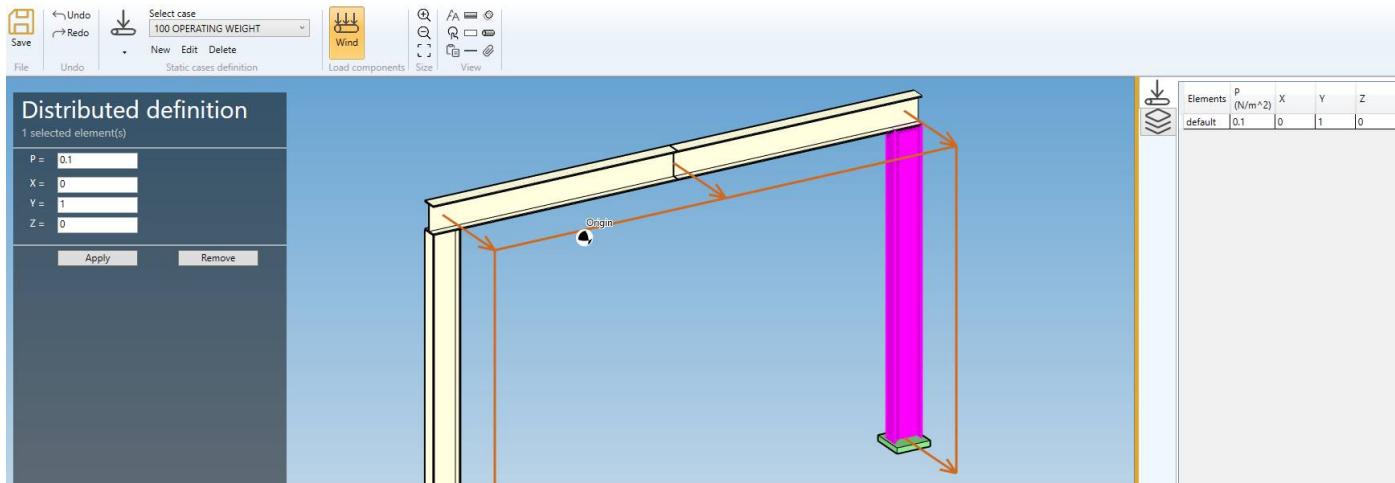
Define the wind **pressure** and **direction** :

Property	Unit Metric	Unit USA
Pressure	N/m ²	lb/ft ²
Dir. /X	-	-
Dir. /Y	-	-
Dir. /Z	-	-

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Wind** button
- 2 Open the right **Data panel**
- 3 Select one or more elements
- 4 Define the new **pressure** and **direction** for these elements
- 5 Apply



You can **undo** this command.

Click on the **Remove** button to delete the wind load on selected elements.

You can also change the size of the drawn forces by clicking on these buttons :



3.4.3 SNOW LOAD

<input checked="" type="checkbox"/> Distributed loads	Snow	<input checked="" type="checkbox"/> Set default	Pressure : 0.2
---	------	---	----------------

You can define **DEFAULT** snow loads that will be applied to all elements of the model.

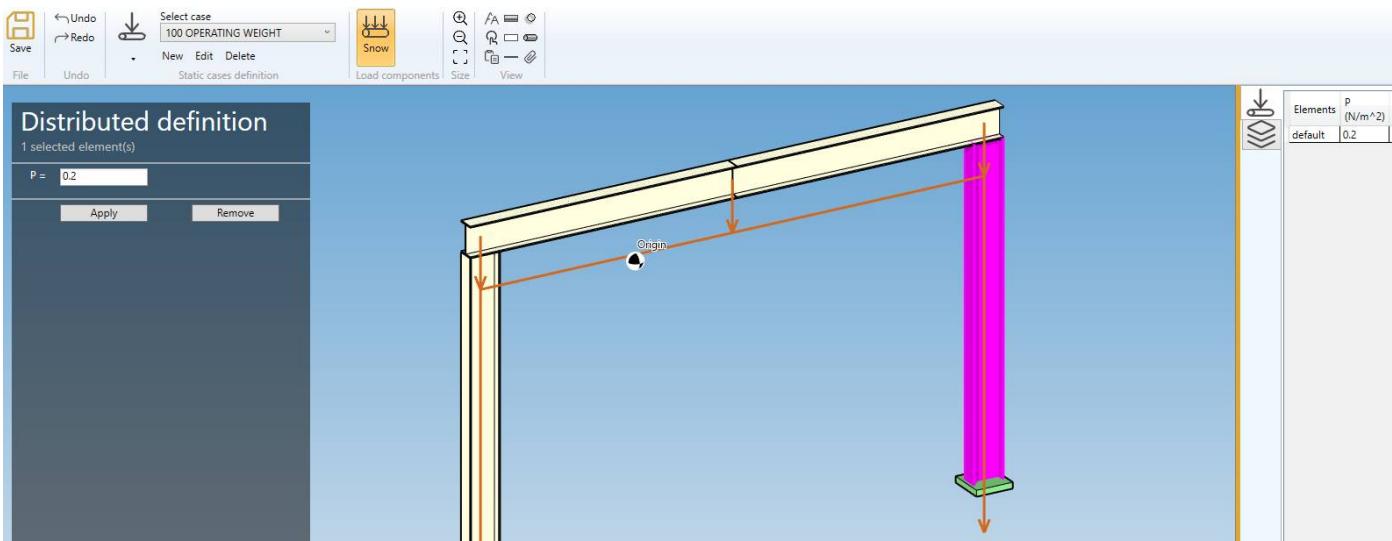
Define the snow **pressure** (direction is vertical downwards) :

Property	Unit Metric	Unit USA
Pressure	N/m²	lb/ft²

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Snow** button
- 2 Open the right **Data panel**
- 3 Select one or more elements
- 4 Define the new **pressure** for these elements
- 5 Apply



You can **undo** this command.

Click on the **Remove** button to delete the snow load on selected elements.

You can also change the size of the drawn forces by clicking on these buttons :



3.5 Nodal forces and moments

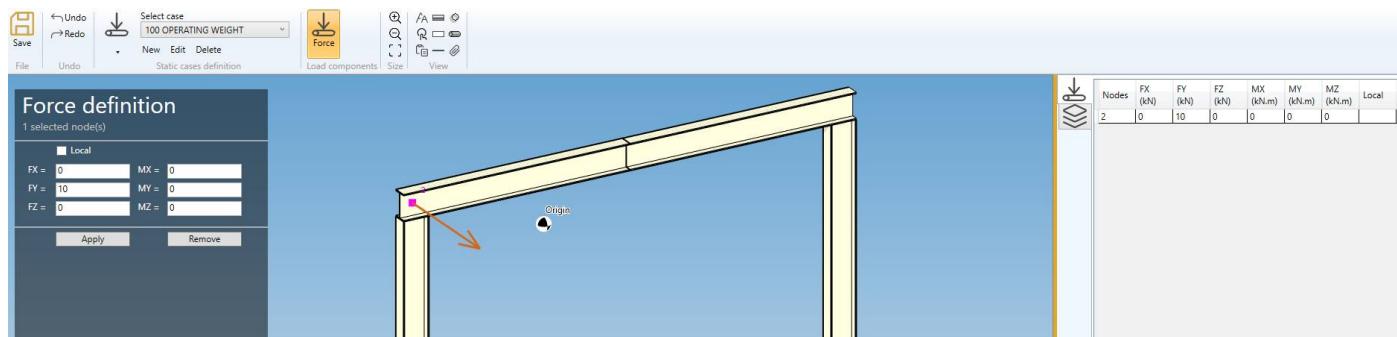
Nodal forces and moments

You can define nodal forces and moments on nodes of the model.

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Force** button
- 2 Open the right **Data panel**
- 3 Select one or more nodes
- 4 Define the new **force and moment** on these nodes
- 5 Apply



Property	Unit Metric	Unit USA
FX	kN	kips
FY	kN	kips
FZ	kN	kips
MX	kN.m	kips.ft
MY	kN.m	kips.ft
MZ	kN.m	kips.ft

You can **undo** this command.

Click on the **Remove** button to delete the force/moment on selected nodes.

You can also change the size of the drawn forces by clicking on these buttons :



3.6 Support movements

Support movements

You can define movements on supports of the model.

If you check this option, you will see this **button** in the ribbon :



- 1 Select the **Movement** button
- 2 Open the right **Data panel**
- 3 Select one or more restraints
- 4 Define the new **vector displacement** and **vector rotation** for these restraints
- 5 Apply

The screenshot shows the software's interface for defining support movements. On the left, the 'Movement definition' dialog is open, showing a list of 1 selected restraint(s). It contains fields for vector displacement (DX = 0, DY = 10, DZ = 0) and vector rotation (RX = 0, RY = 0, RZ = 0), with an 'Apply' button at the bottom. The main workspace displays a 3D model of a frame structure. On the right, the 'Data panel' shows a table of restraints:

Restraints	DX (mm)	DY (mm)	DZ (mm)	RX (rad)	RY (rad)	RZ (rad)	Local
5	0	10	0	0	0	0	

Property	Unit Metric	Unit USA
DX	mm	in
DY	mm	in
DZ	mm	in
RX	rad	rad
RY	rad	rad
RZ	rad	rad

The values can be set in **local** axis.

You can **undo** this command.

Click on the **Remove** button to delete the load on selected restraints.

You can also change the size of the drawn movements by clicking on these buttons :



External cases

External loads can be applied to the model via **links** between piping nodes and structure nodes.

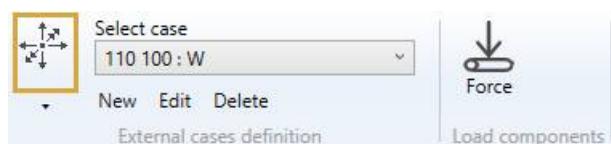
One or more **piping** studies must be linked to the current **structure** study.

Click [here](#) for more information about study link.

One or more **link** must be created between piping nodes and structure nodes.

Click [here](#) for more information about link.

When selecting **External cases**, all existing external cases are listed in the combobox :



The cases appear with their **Case number + Title**.

When editing, the definition window shows up :

1. General

Field	Value
Case number:	110
Title :	100 : W
External study :	Study2
External case :	100 : W

Enter a **Case number** and a **Title**.

Select an external study in the list.

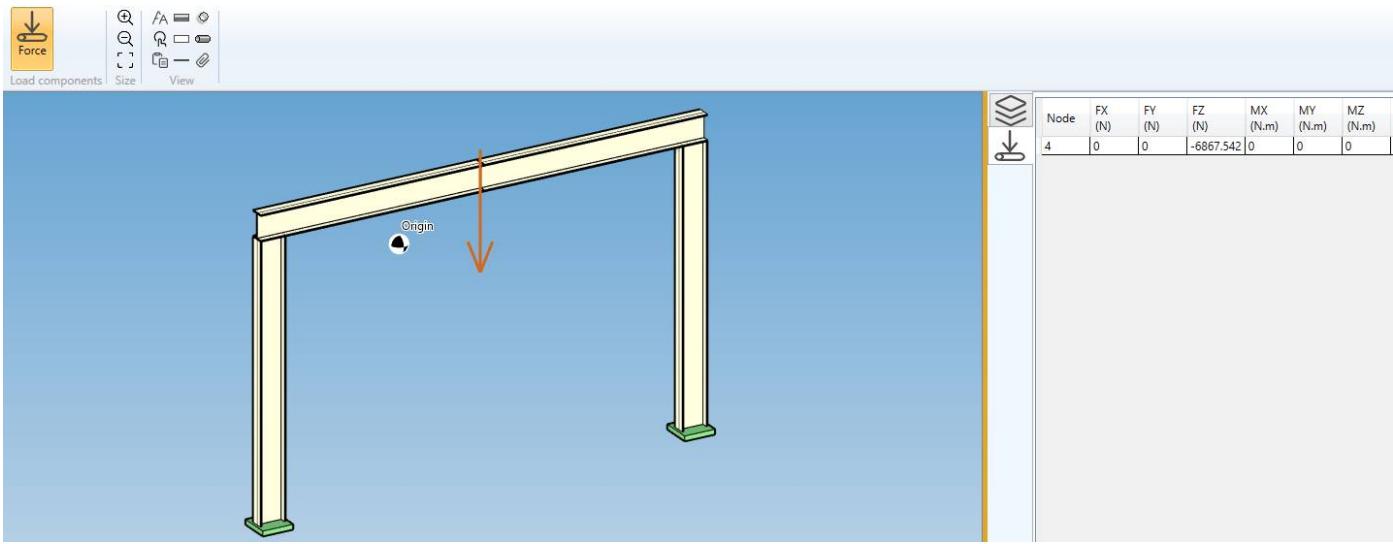
Select then an external case.

2. Forces

You will see this **button** in the ribbon :



You can only see that the **reactions** on the supports in the piping study become **forces/moment**s on several nodes of the current structure (thanks to links).



This screen is not interactable

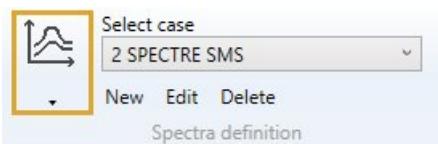
Property	Unit Metric	Unit USA
FX	N	lb
FY	N	lb
FZ	N	lb
MX	N.m	lb.ft
MY	N.m	lb.ft
MZ	N.m	lb.ft



Spectra

This case consists of the definition of series of SPECTRA that describe the support movement loading for primary and secondary floor response cases.

When selecting **Spectra**, all existing spectra are listed in the combobox :



The spectra appear with their **Event number + Title**.

When editing, the definition window shows up :

1. General

Spectra definition

General **Level definition** **Spectra viewer**

Event number :	<input type="text" value="2"/>
Name :	<input type="text"/>
Title :	<input type="text" value="SPECTRE SMS"/>
X axis :	<input type="text" value="Frequency"/>
Interpolation :	<input type="text" value="Linear / Linear accel."/>
Shift :	<input type="text" value="0"/>
Percent damping :	<input type="text" value="0"/>
Duration :	<input type="text" value="0"/>

OK **Cancel**

Enter an **Event number**, a **Name** and a **Title**.

The primary and secondary floor response cases with the same **EVENT** will use the spectra defined herein.

X axis :

- Spectrum is **frequency** vs. acceleration
- Spectrum is **period** vs. acceleration

Interpolation :

- Linear / Linear acceleration
- Linear frequency / Linear acceleration
- Linear period / Linear acceleration
- Log frequency / Linear acceleration
- Log frequency / Log acceleration

Shift :

The shift (in %) determines an interval about each natural frequency. MetaPiping will use the maximum acceleration in each such interval, as determined by interpolating the spectrum data specified.

This method is used for peak broadening. 0 or blank = no shift.

Percent damping :

The damping is used to determine the intermodal correlation factors for the double sum, Rosenblueth and Der Kiureghian methods.

Duration :

Duration of earthquake (in seconds).

0 or blank = infinite (10^6 s).

2. Level definition

In this tab, you can create/remove **LEVELS** for the same **EVENT** :

The screenshot shows the 'Spectra definition' dialog box with the 'Level definition' tab selected. The 'General' tab is also visible. There are buttons for 'Add Level' and 'Remove Level'. A list of 11 levels (Level 1 to Level 11) is shown. Below the list are input fields for 'Level no.' (set to 1), 'DX', 'DY', 'DZ', and 'Phase' (set to 0). To the right of these fields is a database icon, which is highlighted with a yellow box. Below the input fields are three sections: 'Dir X', 'Dir Y', and 'Dir Z', each with a table of frequency and acceleration values. At the bottom are buttons for 'OK' and 'Cancel'.

The **LEVEL** number relates the spectra to the supports.

Click [here](#) to have more information about supports.

DX, DY, DZ :

Maximum expected X, Y, Z displacements (mm, in).

Phase :

The response for levels with the same phase number > 0 are assumed to be in phase and will be added algebraically.

Spectrum in direction X, Y, Z :

Enter **acceleration** for each **frequency/period**.

Attention, the frequency/period must be entered in ascending order.

The possible cases are :

- X + Y + Z = a spectrum for each direction
- X + YZ = a spectrum for X and a common one for Y and Z
- Y + XZ = a spectrum for Y and a common one for X and Z
- Z + XY = a spectrum for Z and a common one for X and Y
- XYZ = the same spectrum for all directions

F1 and F2 :

Low frequency f1 and high frequency f2 for Gupta and Lindley-Yow methods.

2.1 Database

The **Database** button on upper right corner gives an access to the spectra libraries :

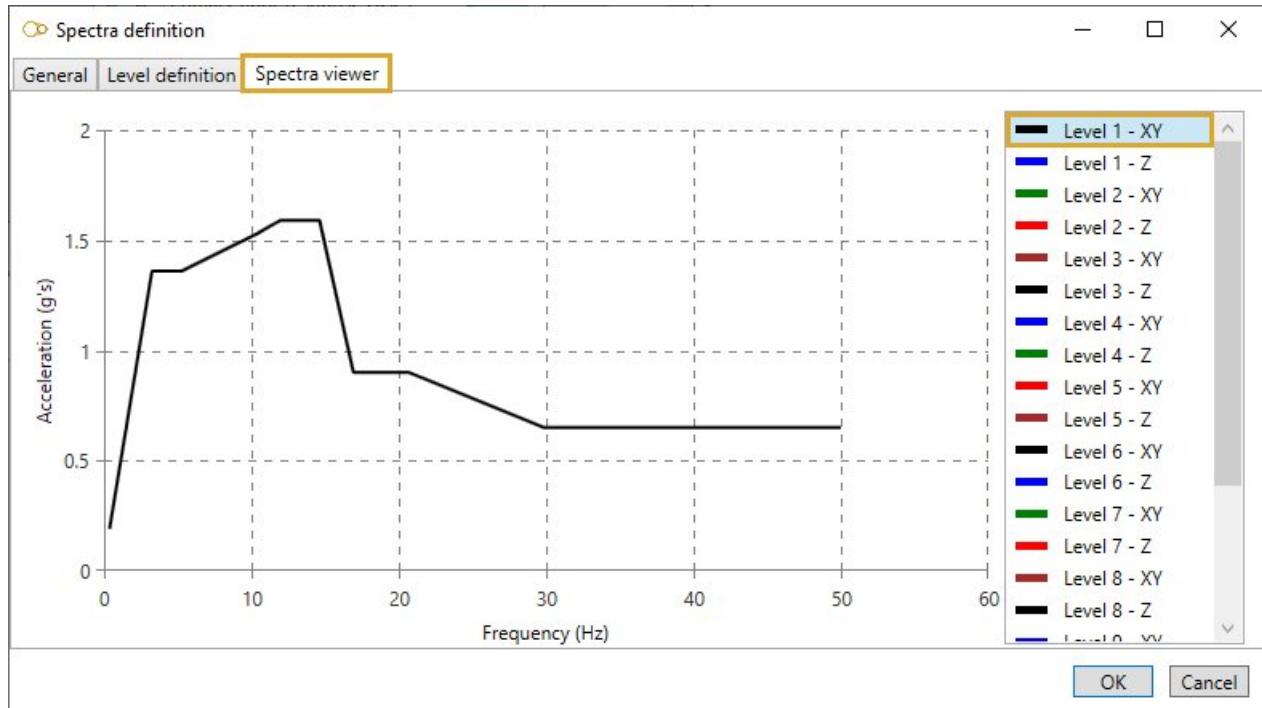


Select a library on the left treeview, then select a row that represents a particular damping for a predefined spectrum. Some values are just informative. Click **OK**. The properties are copied to the spectrum screen.

Click [here](#) to have more information about spectrum database.

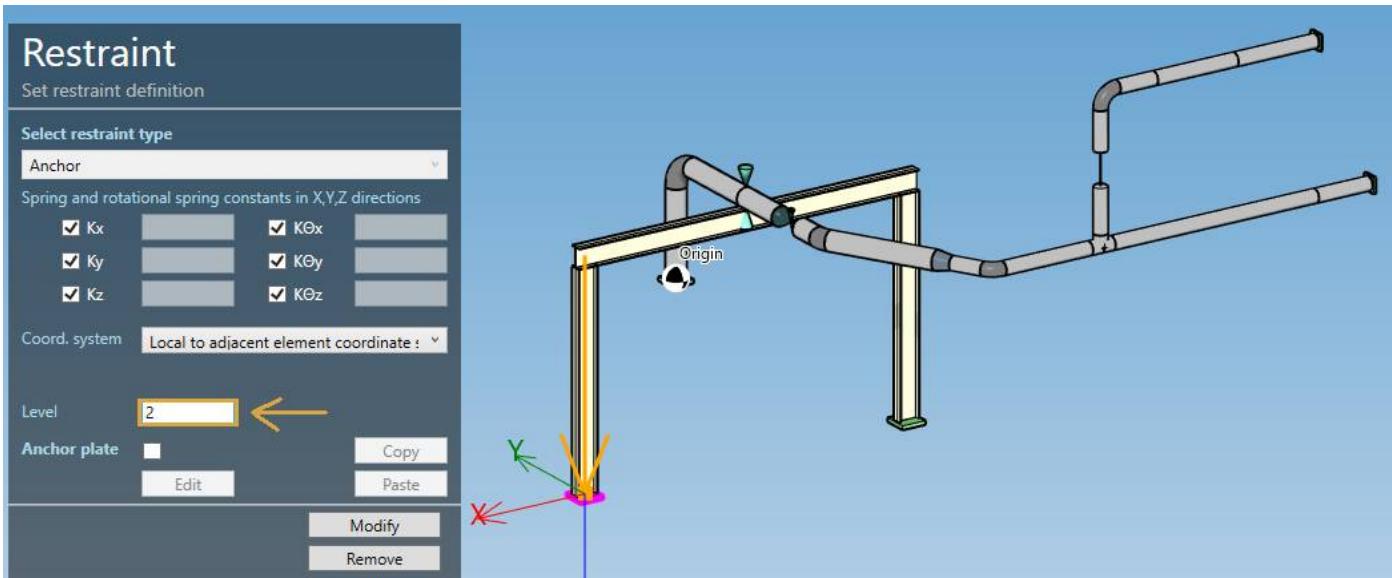
3. Spectra viewer

Select a level on the right list to see the spectrum graph :

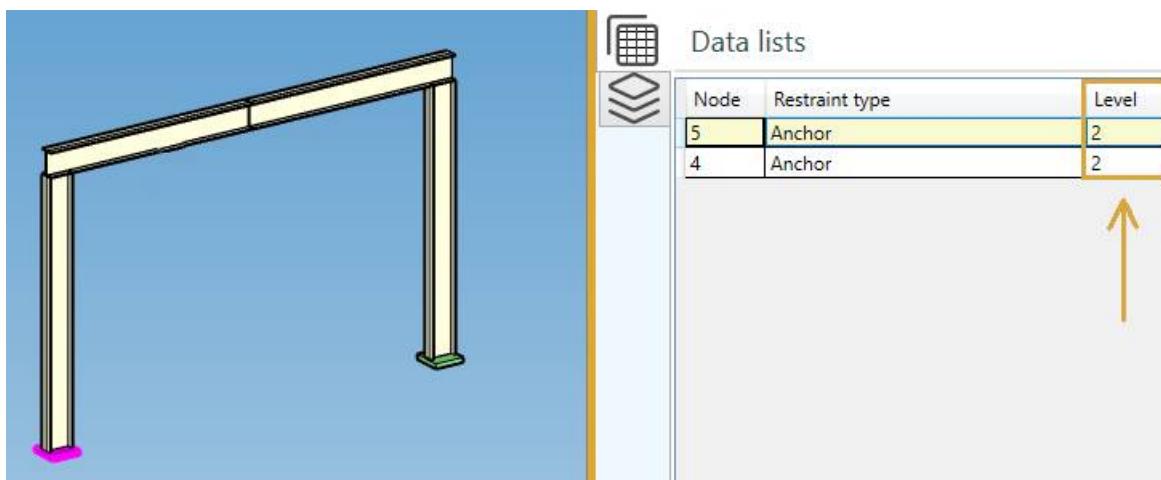


4. Restraints

You can now select **restraints** or create one on a selected node and affect the **Level** property (set the level number) :



You can also check the property on **Review** mode (type Restraints):





Primary floor response cases

This load describes the floor response spectra for each support level. This analysis is used to obtain bound solutions for simple multilevel cases (including single level cases) for which the dynamic loading is entirely due to movements at the support points.
The "levels" are groups of supports which are moving in parallel and are entered using the level number field on supports.

WARNING

Attention, to obtain the complete solution for multilevel problems, it is necessary to calculate the primary and secondary parts of the solution separately and then to combine them.

MetaStructure will calculate the bounds for the **PRIMARY** part of the solution $u(t)$ for displacements, rotations, forces, moments and stresses due to the dynamic support movement loading.

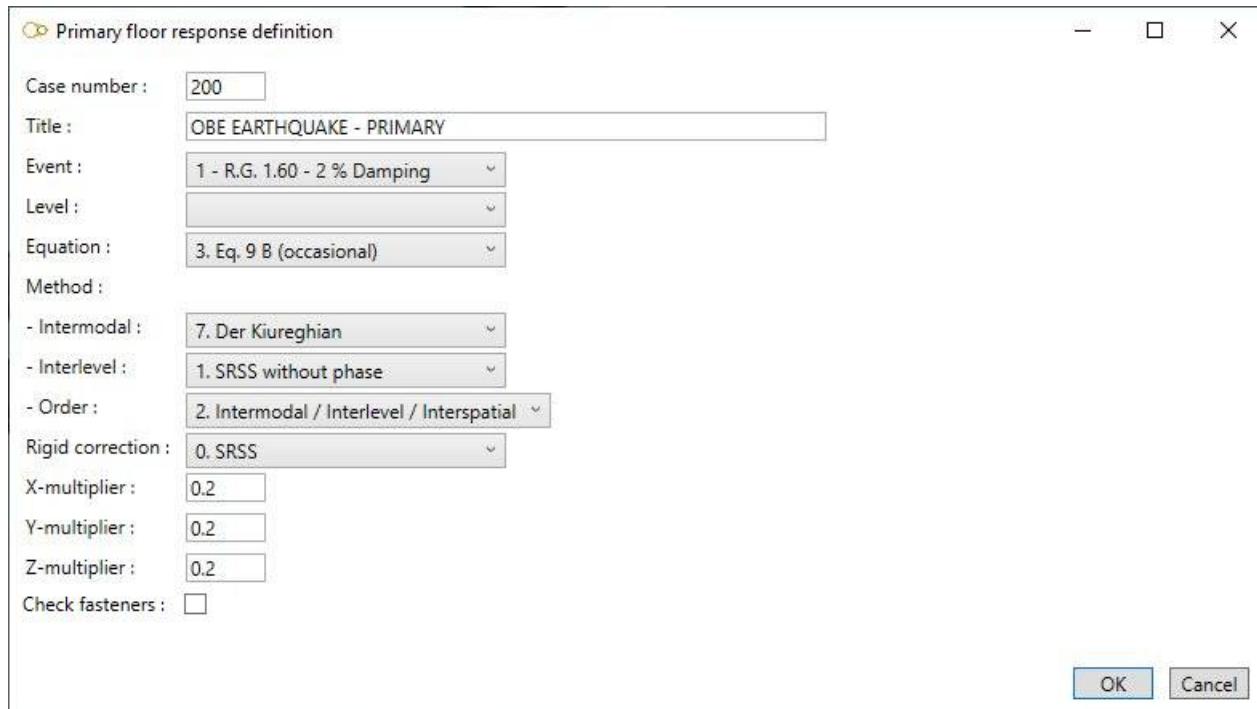
When selecting **Primary floor response cases**, all existing primary floor response cases are listed in the combobox :



The cases appear with their **Case number + Title**.

1. General

When editing, the definition window shows up :



Enter a **Case number** and a **Title**.

Event :

Select a **Spectra** : R.G. 1.60 - 2% Damping in this example.

See §2.

Level :

This field is intended for nuclear piping codes only. The possible values are : A (normal conditions), B (upset), C (emergency), D (faulted) and T (test).

Equation :

The equations are code-dependent.

Method :

For intermodal :

- 1- Grouping
- 2- Ten percent
- 3- Double sum
- 4- SRSS
- 5- All coupling
- 6- Rosenblueth
- 7- Der Kiureghian

For interlevel :

- 0- Absolute without phase
- 1- SRSS without phase
- 2- Algebraic
- 3- Absolute with phase
- 4- SRSS with phase
- 5- Envelope
- 6- SRSS with counterphase

For order :

- 0- Interlevel / Intermodal / Interspatial
- 1- Interlevel / Interspatial / Intermodal
- 2- Intermodal / Interlevel / Interspatial

Rigid correction :

- 0- SRSS
- 1- None
- 2- Absolute
- A- SRSS with modal
- B- Gupta
- C- Lindley-Yow

X, Y, Z multipliers :

This factor multiplies the X, Y, Z component of the loading.

If a factor is zero or blank, then the corresponding loading component will be zero.

Check fasteners :

If checked, the fasteners will be verified.

2. Spectra definition

In this example, we use this **Spectra loading** :

Spectra definition

General Level definition Spectra viewer

Event number : 1

Name : RG1.60

Title : R.G. 1.60 - 2 % Damping

X axis : Frequency

Interpolation : Log freq. / Linear accel.

Shift : 0

Percent damping : 0

Duration : 0

It contains 2 **LEVELS** with corresponding spectra :

Spectra definition

General Level definition Spectra viewer

Add Level Remove Level

Level 1 Level 2

Level no. : 1 DX : 5 DY : 0 DZ : 7 Phase : 0

Dir X

	Freq. (Hz)	Acc. (g's)
1	0.25	0.58
2	2.5	4.25
3	9	3.54
4	33	1
5		

Dir Y Y Y=X

	Freq. (Hz)	Acc. (g's)
1	0.25	0.38
2	3.5	4.05
3	9	3.54
4	33	1
5		

Dir Z Z Z=X Z=Y

	Freq. (Hz)	Acc. (g's)

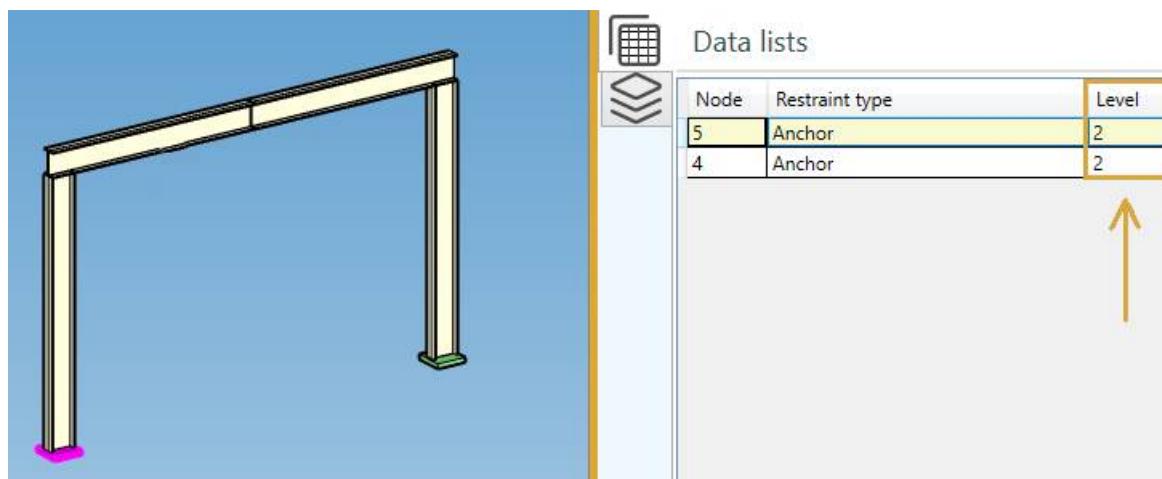
F1 (Hz) : 0 F2 (Hz) : 0 F1 (Hz) : 0 F2 (Hz) : 0 F1 (Hz) : 0 F2 (Hz) : 0

OK Cancel

Click [here](#) for more information about the spectra definition.

3. Restraint definition

In this example, all restraints are assigned a **LEVEL** :



4. Secondary floor response cases

Click [here](#) for more information about the secondary floor response cases.



Secondary floor response cases

This load describes the floor response spectra for each support level. This analysis is used to obtain bound solutions for simple multilevel cases (including single level cases) for which the dynamic loading is entirely due to movements at the support points.
The "levels" are groups of supports which are moving in parallel and are entered using the level number field on supports.

WARNING

Attention, to obtain the complete solution for multilevel problems, it is necessary to calculate the primary and secondary parts of the solution separately and then to combine them.

MetaStructure will calculate the bounds for the **SECONDARY** part of the solution $v(t)$ for displacements, rotations, forces, moments and stresses due to the dynamic support movement loading.

When selecting **Secondary floor response cases**, all existing secondary floor response cases are listed in the combobox :



The cases appear with their **Case number + Title**.

1. General

When editing, the definition window shows up :



Enter a **Case number** and a **Title**.

Event :

Select a **Spectra** : R.G. 1.60 - 2% Damping in this example.

See §2.

Level :

This field is intended for nuclear piping codes only. The possible values are : A (normal conditions), B (upset), C (emergency), D (faulted) and T (test).

Equation :

The equations are code-dependent.

Method :

The solutions for each group of supports are combined using one of the following method in order to obtain the total secondary response :

- 0- Absolute
- 1- SRSS

Treat as primary :

Secondary floor response cases are treated as secondary by default. Depending on the code, the moments may then be multiplied by Ec/Eh (like thermal expansion). If **Treat as primary** is checked, the moments are not multiplied by Ec/Eh. This is recommended when the total seismic moments (primary + secondary) are used in combination with the dead weight moments in the equation for occasional stresses.

X, Y, Z multipliers :

This factor multiplies the X, Y, Z component of the loading.

If a factor is zero or blank, then the corresponding loading component will be zero.

Check fasteners :

If checked, the fasteners will be verified.

2. Spectra definition

In this example, we use this **Spectra loading** :

Spectra definition	
General Level definition Spectra viewer	
Event number :	1
Name :	RG1.60
Title :	R.G. 1.60 - 2 % Damping
X axis :	Frequency
Interpolation :	Log freq. / Linear accel.
Shift :	0
Percent damping :	0
Duration :	0

It contains 2 **LEVELS** with corresponding spectra :

Spectra definition

General Level definition Spectra viewer

Add Level Remove Level

Level 1 Level 2

Level no.: 1 DX: 5 DY: 0 DZ: 7 Phase: 0 

Dir X

	Freq. (Hz)	Acc. (g's)
1	0.25	0.58
2	2.5	4.25
3	9	3.54
4	33	1
5		

F1 (Hz): 0 F2 (Hz): 0

Dir Y Y Y=X

	Freq. (Hz)	Acc. (g's)
1	0.25	0.38
2	3.5	4.05
3	9	3.54
4	33	1
5		

F1 (Hz): 0 F2 (Hz): 0

Dir Z Z Z=X Z=Y

	Freq. (Hz)	Acc. (g's)
1		
2		
3		
4		
5		

F1 (Hz): F2 (Hz):

OK **Cancel**

Click [here](#) for more information about the spectra definition.

3. Primary floor response cases

Click [here](#) for more information about the primary floor response cases.



Time history cases

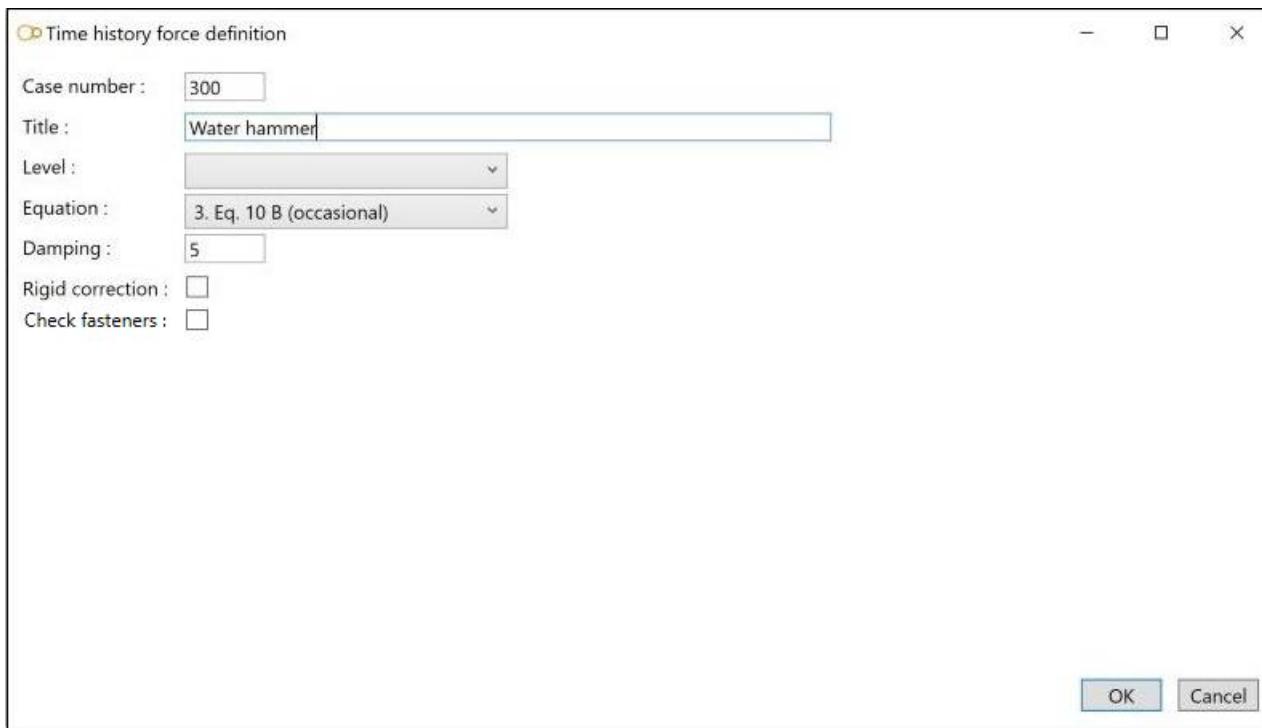
This case consists of dynamic forces and moments applied to some nodes of the model. The solution is obtained using the modal superposition method. Time histories are entered in an external THF file connected to the study beforehand.

When selecting **Time history cases**, all existing time history cases are listed in the combobox :



The cases appear with their **Case number + Title**.

When editing, the definition window shows up :



Enter a **Case number** and a **Title**.

Level :

This field is intended for nuclear piping codes only. The possible values are : A (normal conditions), B (upset), C (emergency), D (faulted) and T (test).

Equation :

The equations are code-dependent.

Damping :

Enter the damping value (in percentage).

Rigid correction :

If checked, include the effect of the higher "rigid" modes (i.e. modes above the cut-off frequency).

Check fasteners :

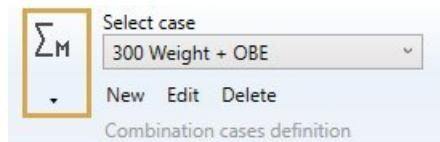
If checked, the fasteners will be verified.



Combination cases

This case consists of combining the results of load cases, dynamic cases or previously calculated combination cases to form a new case.

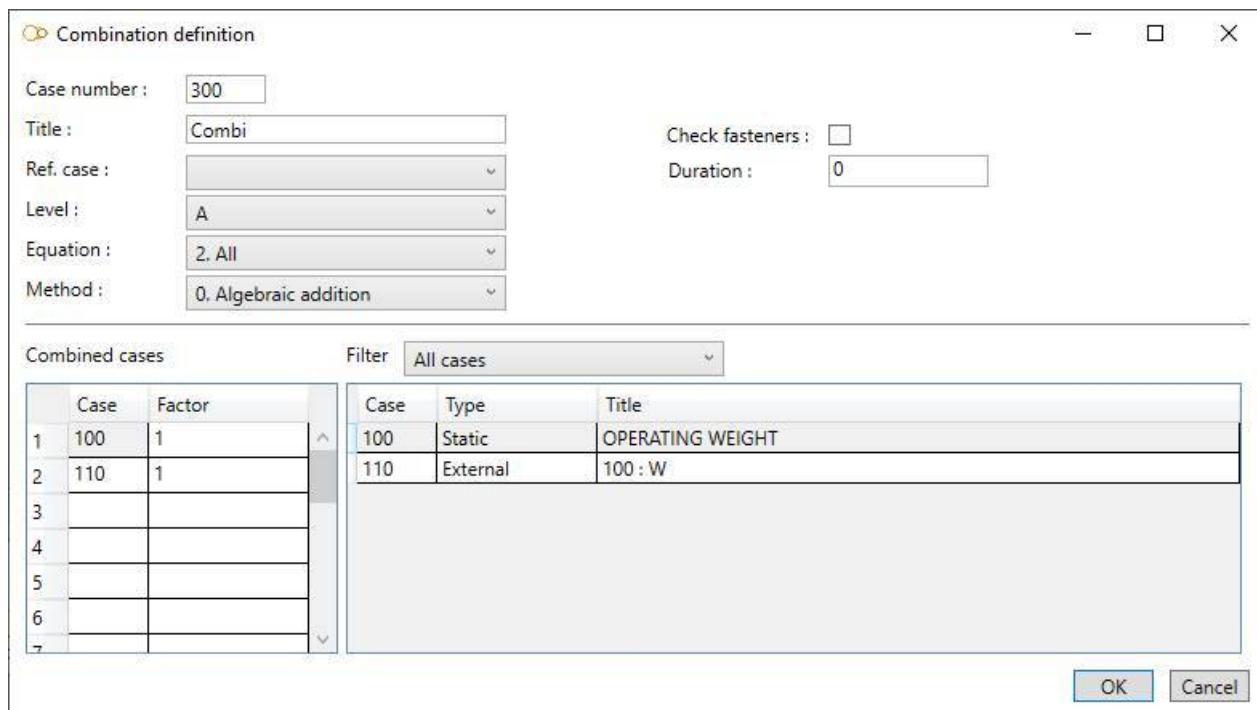
When selecting **Combination cases**, all existing combinations are listed in the combobox :



The cases appear with their **Case number + Title**.

1. General

When editing, the definition window shows up :



Enter a **Case number** and a **Title**.

Ref. case :

The allowable stress is evaluated at the temperature of the refence case (room temperature if not specified).

Level :

This field is intended for nuclear codes only. The possible values are : A (normal conditions), B (upset), C (emergency), D (faulted) and T (test).

Equation :

- 1- None
- 2- All

Method :

- 0- Algebraic addition
- 1- Absolute addition

- 2- SRSS
- 3- Seismic
- 4- Maximum absolute
- 6- Algebraic maximum
- 7- Algebraic minimum
- 8- Range
- 9- Max resultant moment range

Check fasteners :

If checked, the fasteners will be verified.

Duration :

In hours.

2. Combined cases

Enter the **number** of the constituent cases and the corresponding **factor** (1) :

Combined cases		Filter	All cases	
Case	Factor	Case	Type	Title
1	100	1	Static	OPERATING WEIGHT
2	110	1	External	100 : W
3				
4				
5				
6				
7				

On right side, as a reminder, the list of all cases (2). You can filter them by selecting the type :

- All
- Static
- Dynamic
- Combination
- External cases



Review

The **REVIEW** ribbon tab lets you verify all data of the model.

TABLE OF CONTENTS

- [Piping](#)
- [Structure](#)



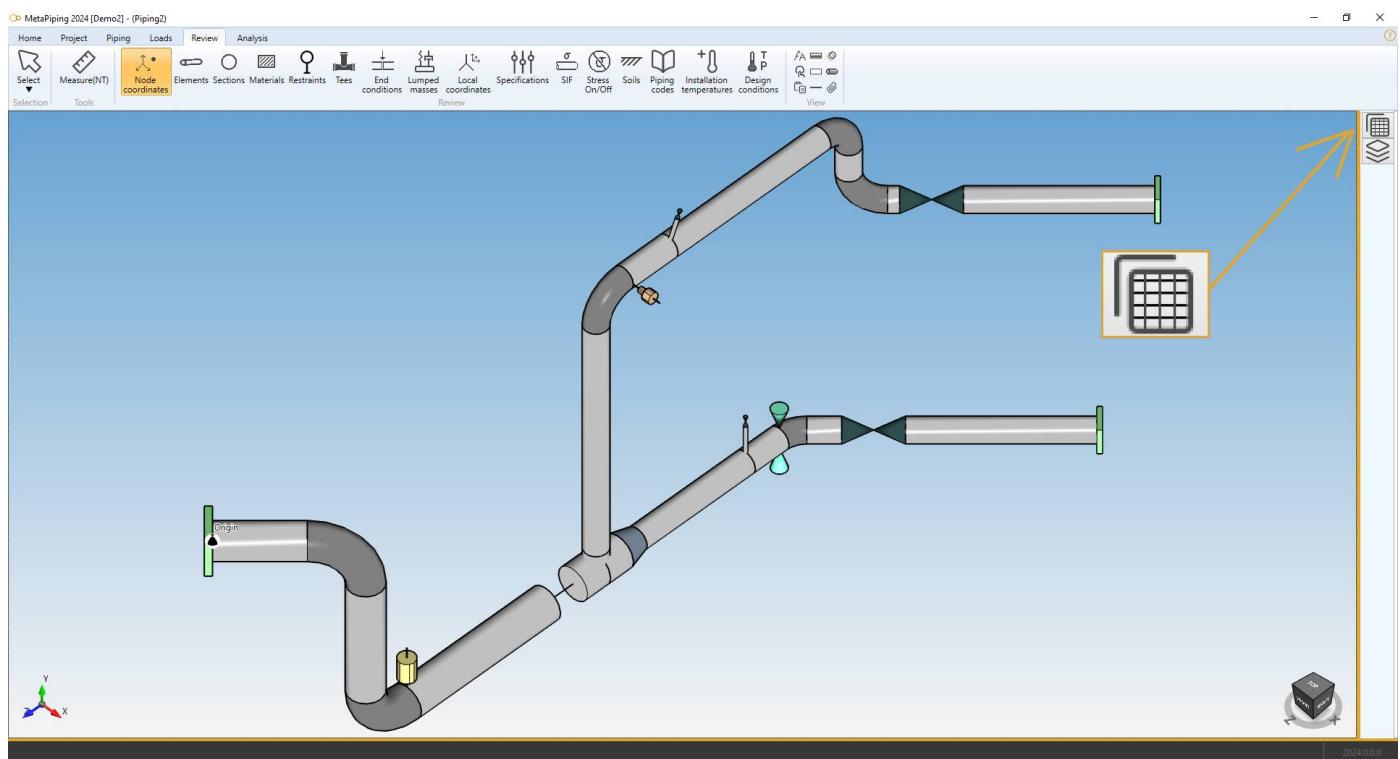
Piping review

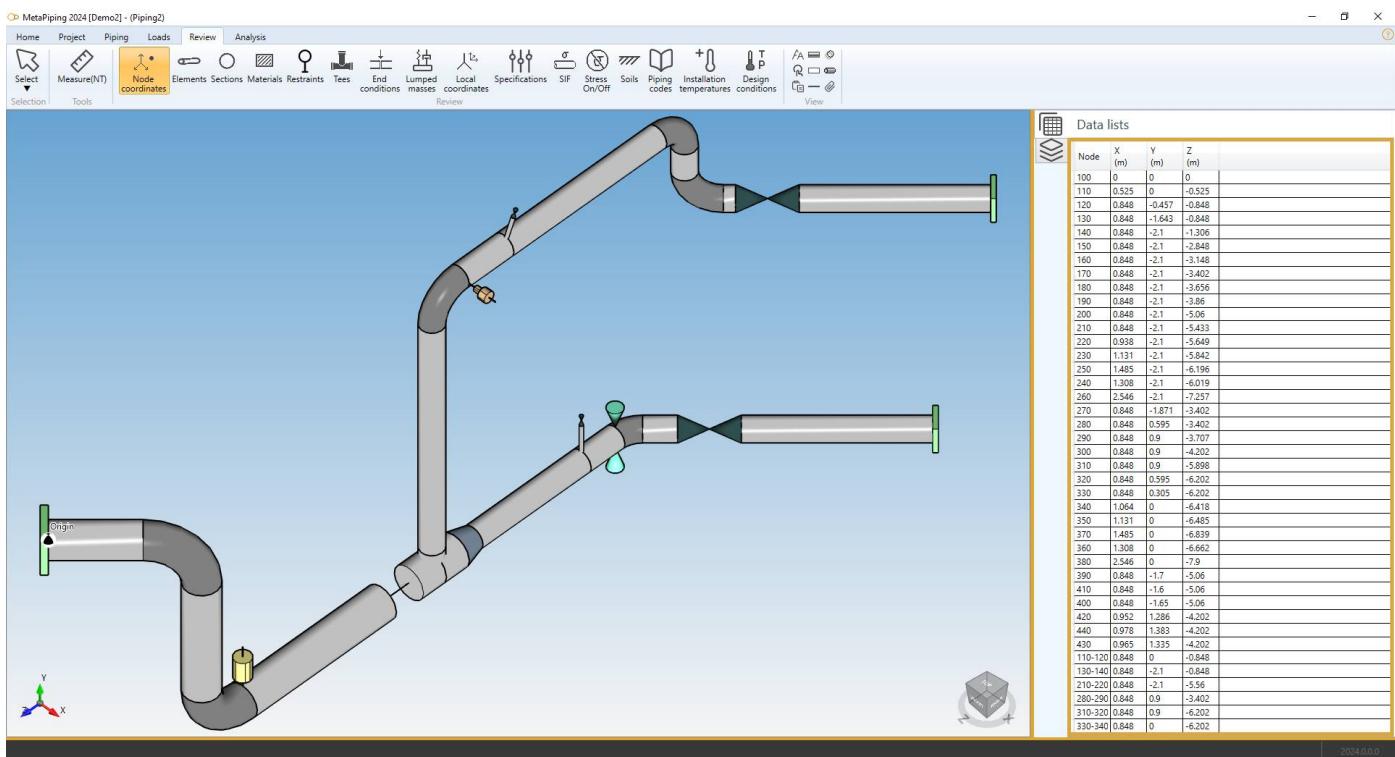
1. Ribbon menu



2. Data panel

A right panel can be opened by clicking on the top button. It presents the selected data :



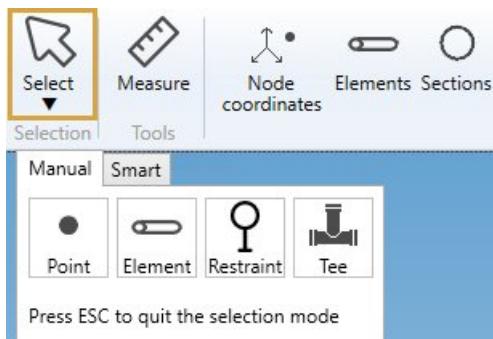


Click on the same button to hide the panel. Shortcut = F2.

TIP : you can copy the data by CTRL+C and paste in other software.

3. Select tool

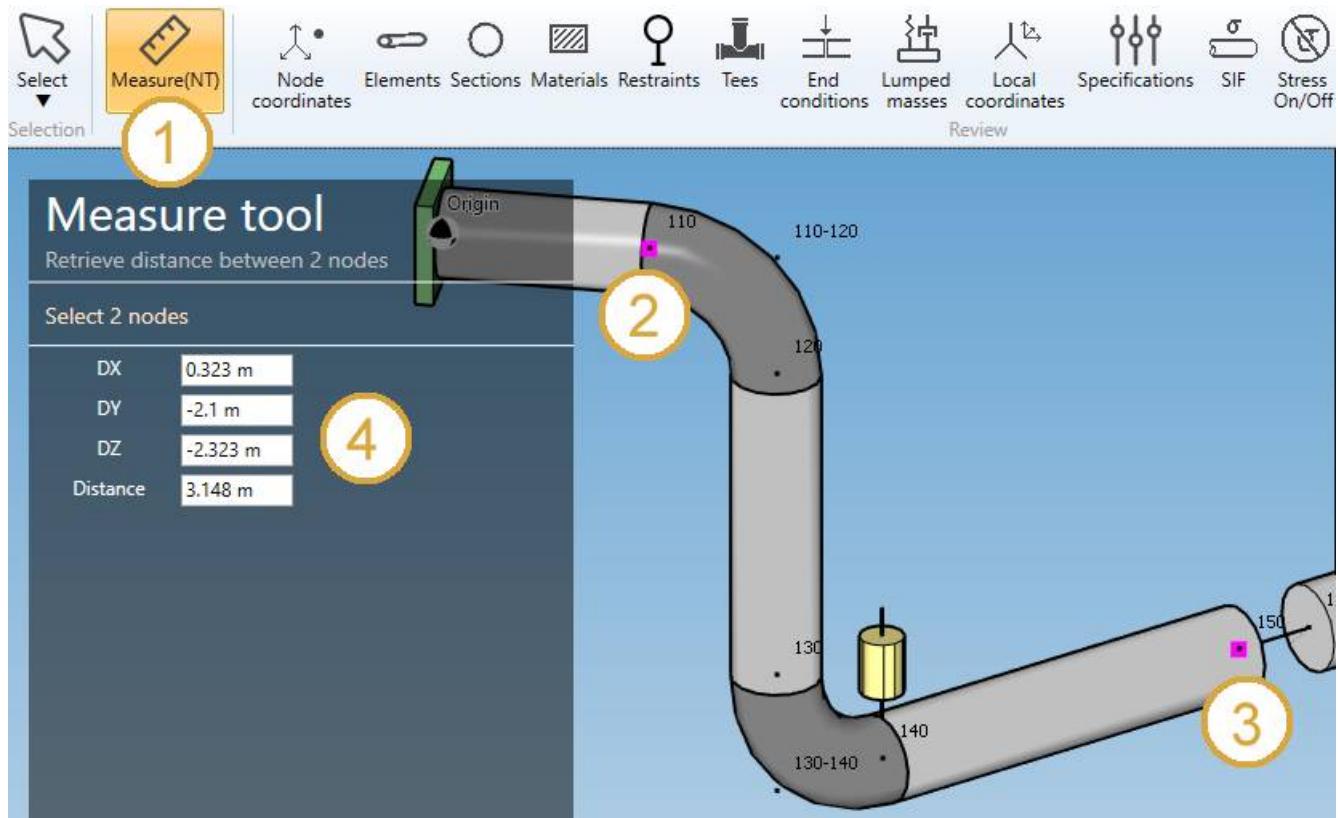
Click on the **Select** button to open the **Selection tool** :



Click [here](#) for more information about the selection tool.

4. Measure tool

MetaPiping lets you measure the **distance** between 2 nodes :



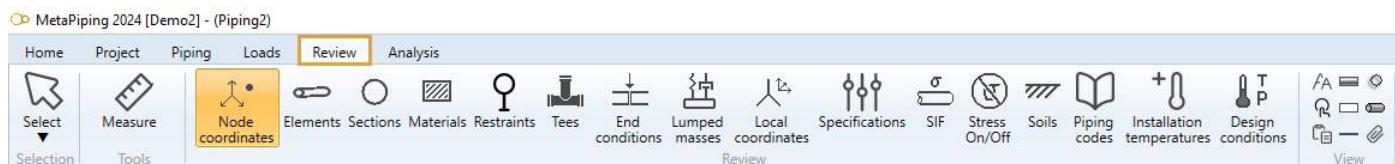
- 1 Select the **Measure** tool
- 2 Select the first node
- 3 Select the second node with CTRL pressed
- 4 The distances appear on the left panel

You can continue to select other nodes (with CTRL pressed), only the 2 last ones will be taken in account.

Property	Description	Unit Metric	Unit USA
DX	Distance on global X	m	ft
DY	Distance on global Y	m	ft
DZ	Distance on global Z	m	ft
Distance	Distance between the 2 nodes	m	ft

The **Selection mode** is automatically set to POINT when clicking the **Measure** button.

5. Node coordinates



Click on the **Node coordinates** button :

Node	X (m)	Y (m)	Z (m)
100	0	0	0
110	0.525	0	-0.525
120	0.848	-0.457	-0.848
130	0.848	-1.643	-0.848
140	0.848	-2.1	-1.306
150	0.848	-2.1	-2.848
160	0.848	-2.1	-3.148
170	0.848	-2.1	-3.402
180	0.848	-2.1	-3.656
190	0.848	-2.1	-3.86
200	0.848	-2.1	-5.06
210	0.848	-2.1	-5.433
220	0.938	-2.1	-5.649
230	1.131	-2.1	-5.842
250	1.485	-2.1	-6.196
240	1.308	-2.1	-6.019

The **Data panel** shows the global coordinates X, Y, Z for each node.

Property	Description	Unit Metric	Unit USA
Node name	Text or number	-	-
X	X global coordinate	m	ft
Y	Y global coordinate	m	ft
Z	Z global coordinate	m	ft

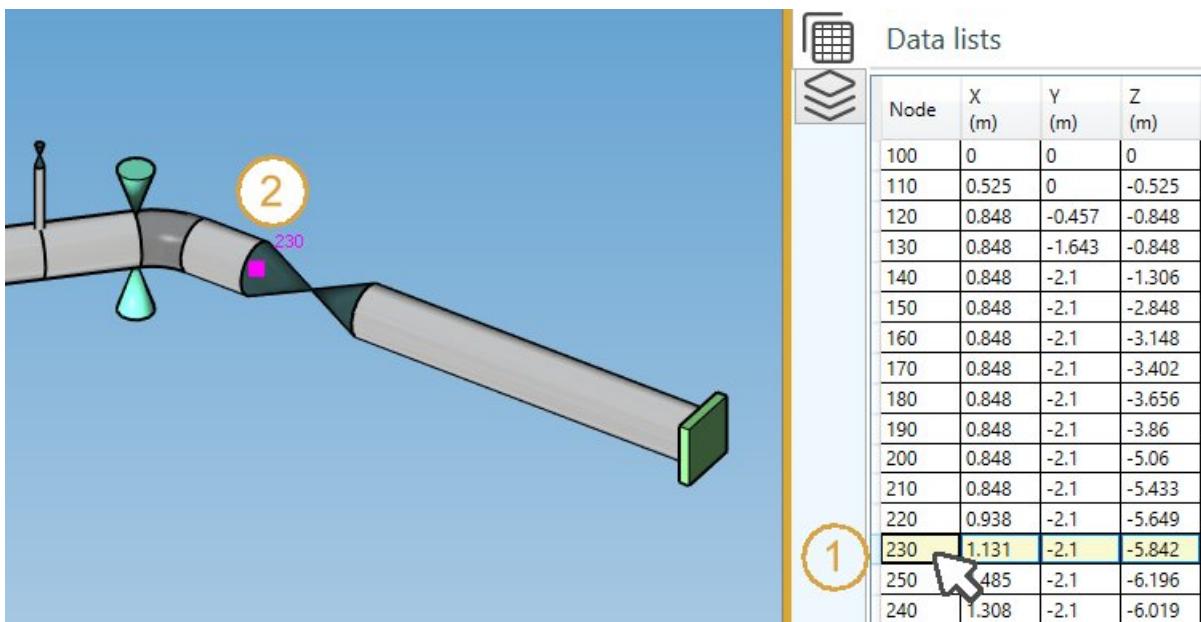
The **Selection mode** is automatically set to POINT

INTERACTIVITY :

- 1 Select a node in the model (1) will highlight the corresponding row in the data list (2) :

Node	X (m)	Y (m)	Z (m)
100	0	0	0
110	0.525	0	-0.525
120	0.848	-0.457	-0.848
130	0.848	-1.643	-0.848
140	0.848	-2.1	-1.306
150	0.848	-2.1	-2.848
160	0.848	-2.1	-3.148
170	0.848	-2.1	-3.402
180	0.848	-2.1	-3.656
190	0.848	-2.1	-3.86
200	0.848	-2.1	-5.06
210	0.848	-2.1	-5.433
220	0.938	-2.1	-5.649
230	1.131	-2.1	-5.842
250	1.485	-2.1	-6.196
240	1.308	-2.1	-6.019

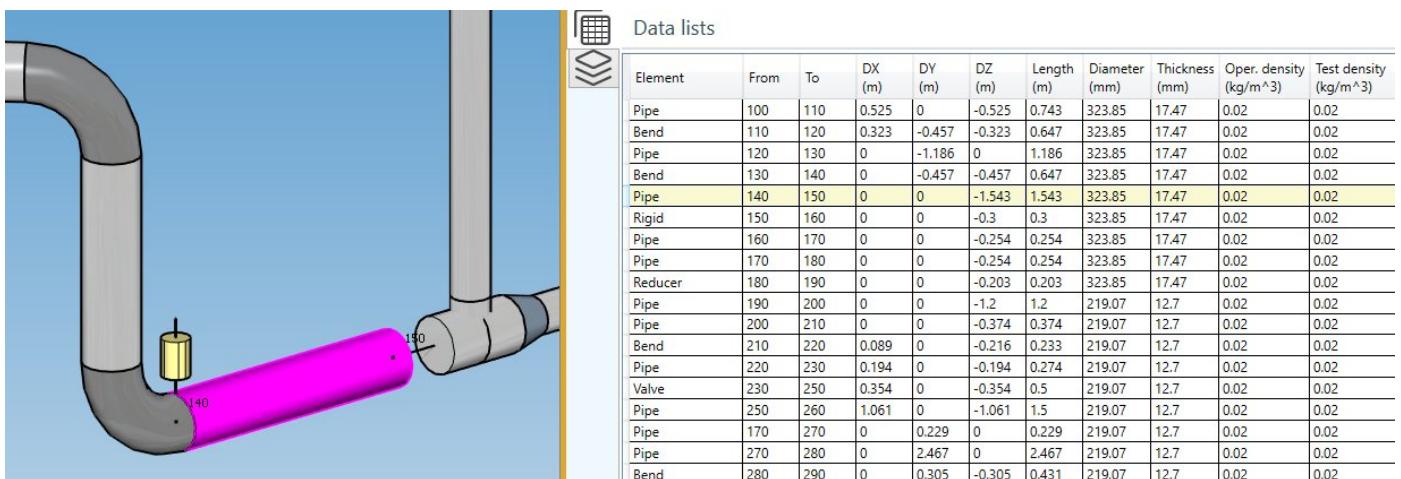
- 1 Select a row in the data list (1) will highlight the corresponding node in the model (2) :



The data are not editable

6. Elements

Click on the **Elements** button :



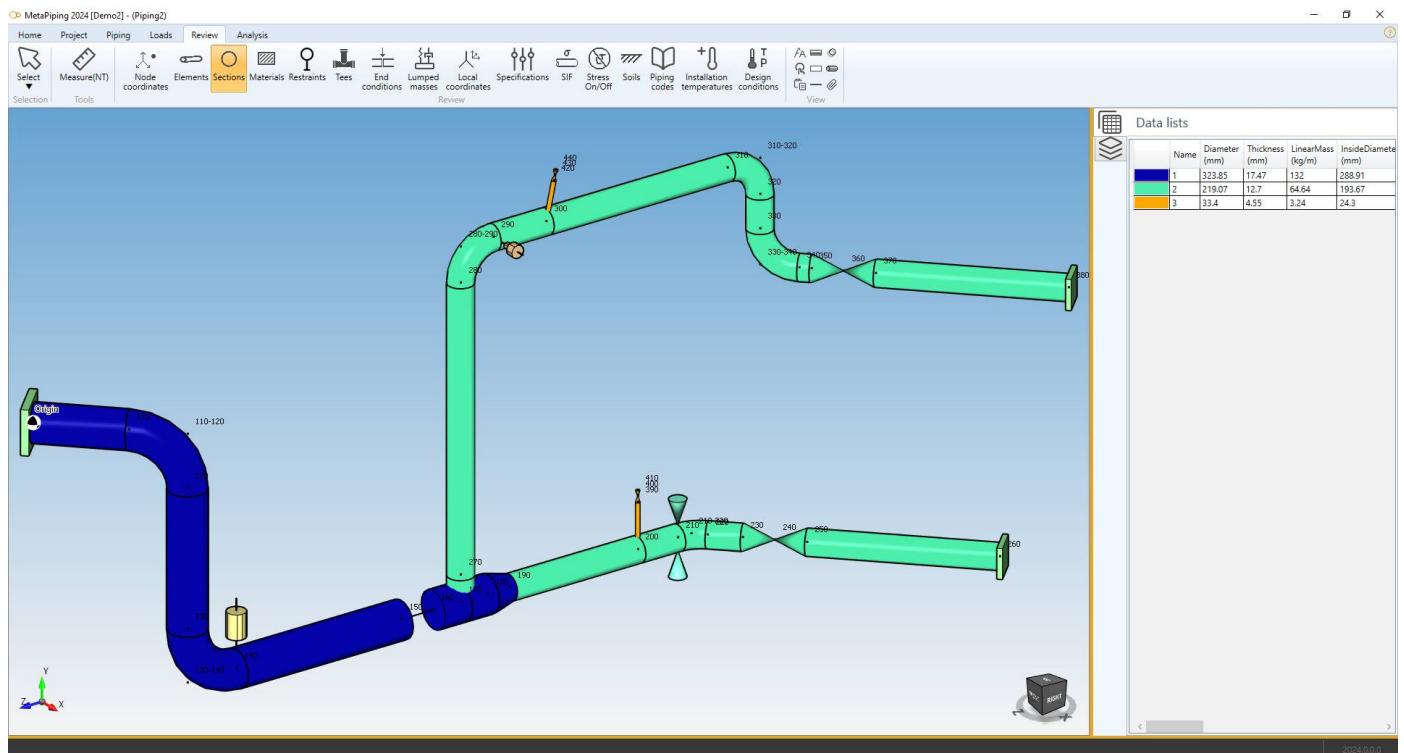
Property	Unit Metric	Unit USA
Element type	-	-
Node1 name	-	-
Node2 name	-	-
DX	m	ft
DY	m	ft
DZ	m	ft
Length	m	ft
Diameter	mm	in
Thickness	mm	in
Operating density	-	-
Test density	-	-

The **INTERACTIVITY** is the same as explained on §5.

The data are not editable

7. Sections

Click on the **Sections** button :



The piping elements are colorized in the color of their corresponding section.

Data lists

Name	Diameter (mm)	Thickness (mm)	LinearMass (kg/m)	InsideDiameter (mm)	OperatingDensity	TestDensity	InsulationThickness (mm)	Corrosion (mm)	Erosion (mm)	MillTolerance (mm)	QualityFactor	OvalizationRatio	LinerThickness (mm)	TopCoatThickness (mm)	SpecialThickness (mm)	BendThickness (mm)	Description
1	323.85	17.47	132	288.91	0.02	0.02	0	0	0	0	0	0	0	0	0	0	12" Sch 80
2	219.07	12.7	64.64	193.67	0.02	0.02	0	0	0	0	0	0	0	0	0	0	8" Sch 80 XS 80S
3	33.4	4.55	3.24	24.3	0.02	0.02	0	0	0	0	0	0	0	0	0	0	1" Sch 80 XS 80S

Property	Unit Metric	Unit USA
Section color	-	-
Name	-	-
Diameter	mm	in
Thickness	mm	in
Linear mass	kg/m	lb/ft
Inside diameter	mm	in
Operating density	-	-
Test density	-	-
Insulation thickness	mm	in
Corrosion	mm	in
Erosion	mm	in
Mill tolerance	mm	in
Quality factor	-	-
Ovalization ratio	-	-
Liner thickness	mm	in
Topcoat thikness	mm	in
Special thickness	mm	in
Bend thickness	mm	in
Description	-	-

The data are not editable

8. Materials

Click on the **Materials** button :

Data lists

	Name	RefTemperature (°C)	MaxTemperature (°C)	ThermalExpansionOption	Density (kg/m^3)	Poisson	Description
	100	21.1	427	2	0	0.3	SA-376 TP304 (18Cr - 8Ni)

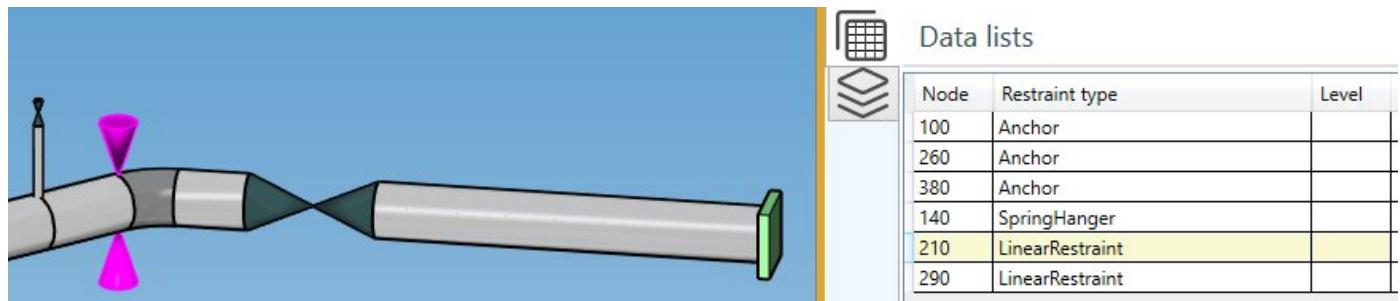
The piping elements are colorized in the color of their corresponding material.

Property	Unit Metric	Unit USA
Material color	-	-
Name	-	-
Ref temperature	°C	°F
Max temperature	°C	°F
Thermal expansion option	-	-
Density	kg/m ³	lb/ft ³
Poisson	-	-
Description	-	-

The data are not editable

9. Restraints

Click on the **Restraints** button :



The **Data panel** shows the **Restraint type** and the **Level** for each restraint.

The **INTERACTIVITY** is the same as explained on §5.

The **label** and modified **Spring Constants** are shown :

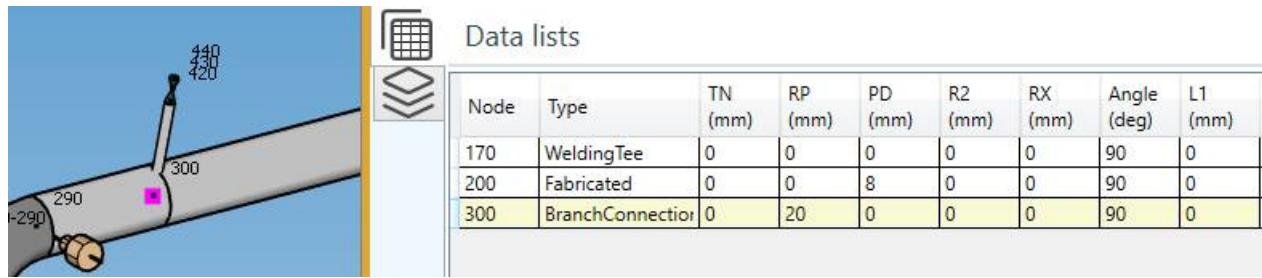
Data lists

Node	Restraint type	Level	X (m)	Y (m)	Z (m)	Label	Kx (kN/mm)	Ky (kN/mm)	Kz (kN/mm)	Krx (kN.m/rad)	Kry (kN.m/rad)	Krz (kN.m/rad)
10	Anchor		0	0	1							
64	Anchor		1.96	0.146	3.145							
27	LinearRestraint		0.399	-0.4	3.852							
41	LinearRestraint		-0.399	-5.926	3.852							
40	LinearRestraint		0.399	-5.926	3.852							
25	LinearRestraint		-0.399	-0.4	3.852	Support1						
67	Anchor		-1.96	0.146	3.145	Anchor1						

The data are not editable

10. Tees

Click on the **Tees** button :



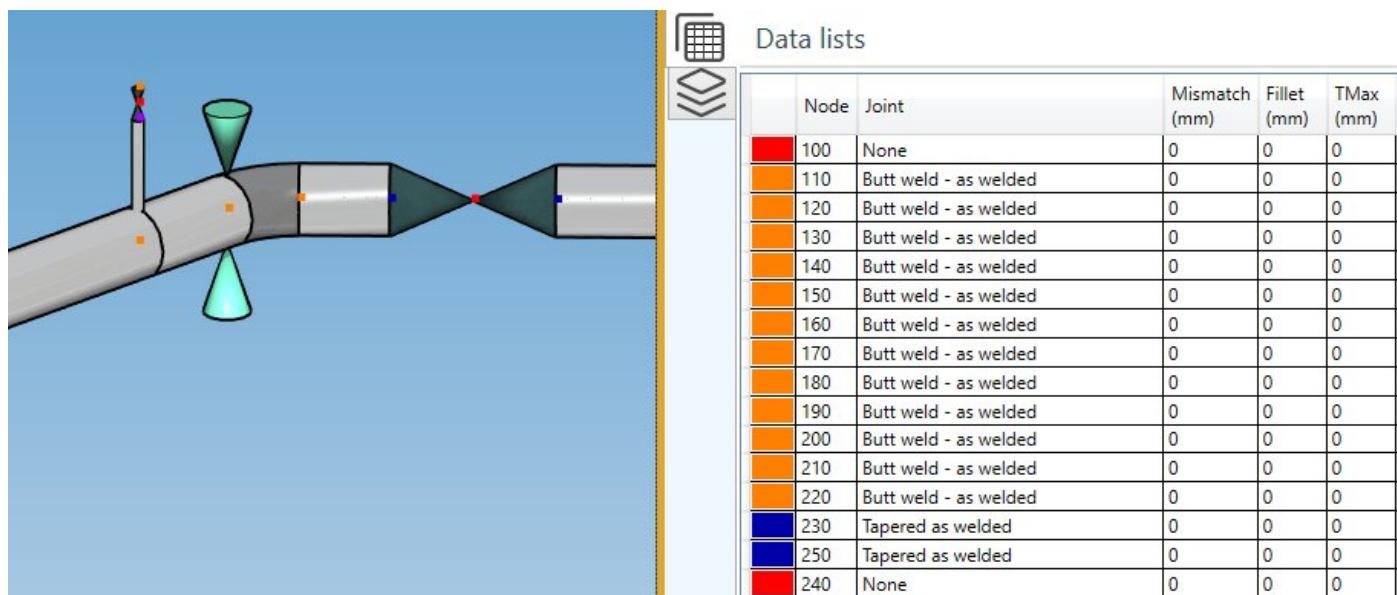
Property	Description	Unit Metric	Unit USA
TN	Branch connections and lateral connections reinforcement thickness	mm	in
RP	Branch Outer Radius	mm	in
PD	Pad thickness for reinforced fabricated tees	mm	in
R2	Branch-to-run fillet radius	mm	in
RX	Transition radius	mm	in
Angle	Angle header/branch	°	°
L1	Branch reinforcement length	mm	in

The **INTERACTIVITY** is the same as explained on §5.

| The data are not editable

11. End conditions

Click on the **End conditions** button :



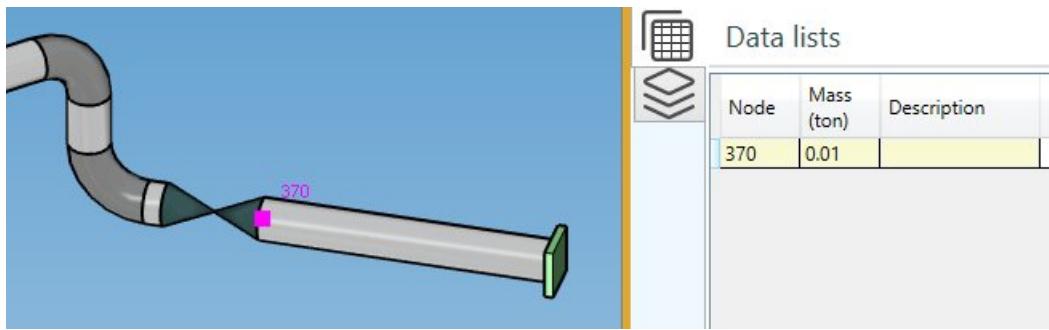
Property	Unit Metric	Unit USA
Color	-	-
Node name	-	-
Joint type	-	-
Mismatch	mm	in
Fillet length	mm	in
TMax	mm	in

The **INTERACTIVITY** is the same as explained on §5.

| The data are not editable

12. Lumped masses

Click on the **Lumped masses** button :



Property	Unit Metric	Unit USA
Node name	-	-
Mass	ton	kips
Description	-	-

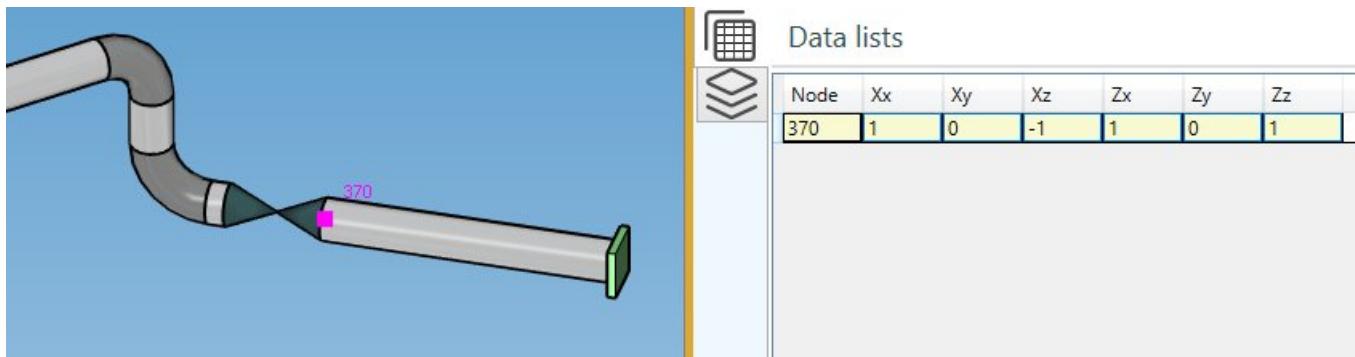
Click [here](#) for more information about creation of lumped mass on node.

The **INTERACTIVITY** is the same as explained on §5.

The data are not editable

13. Local coordinates

Click on the **Local coordinates** button :



Property	Unit Metric	Unit USA
Node name	-	-
Xx	-	-
Xy	-	-
Xz	-	-
Zx	-	-
Zy	-	-
Zz	-	-

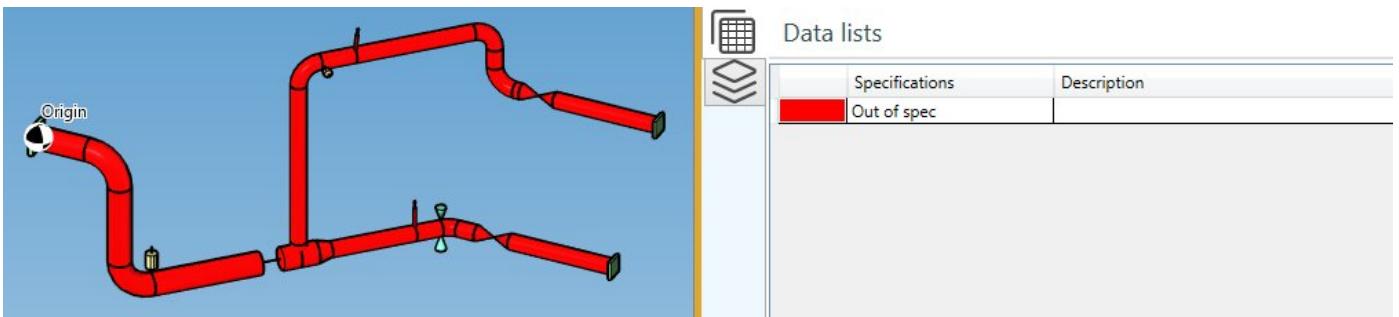
Click [here](#) for more information about creation of local coordinates on node.

The **INTERACTIVITY** is the same as explained on §5.

The data are not editable

14. Specifications

Click on the **Specifications** button :



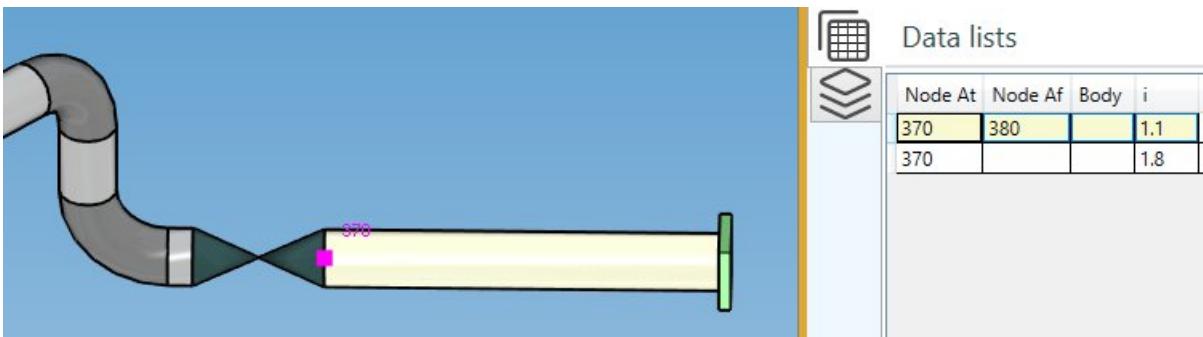
The data are not editable

Click [here](#) for more information about creation of specification.

15. SIF

MetaPiping lets you define **Stress Intensification Factors** on nodes and elements.

Click on the **SIF** button :



Property	Definition
At Node name	The node where the SIF are defined
Af Node name	Empty if SIF on all elements (extremities) - otherwise only on the element (extremity) defined by the Af node
Body	Yes if SIF on element - Empty if SIF on node

If Body = Yes, Node At and Node Af correspond to the nodes of the element

Click [here](#) for more information about creation of Stress Intensification Factors on nodes and elements.

Each **piping code** has its own factors.

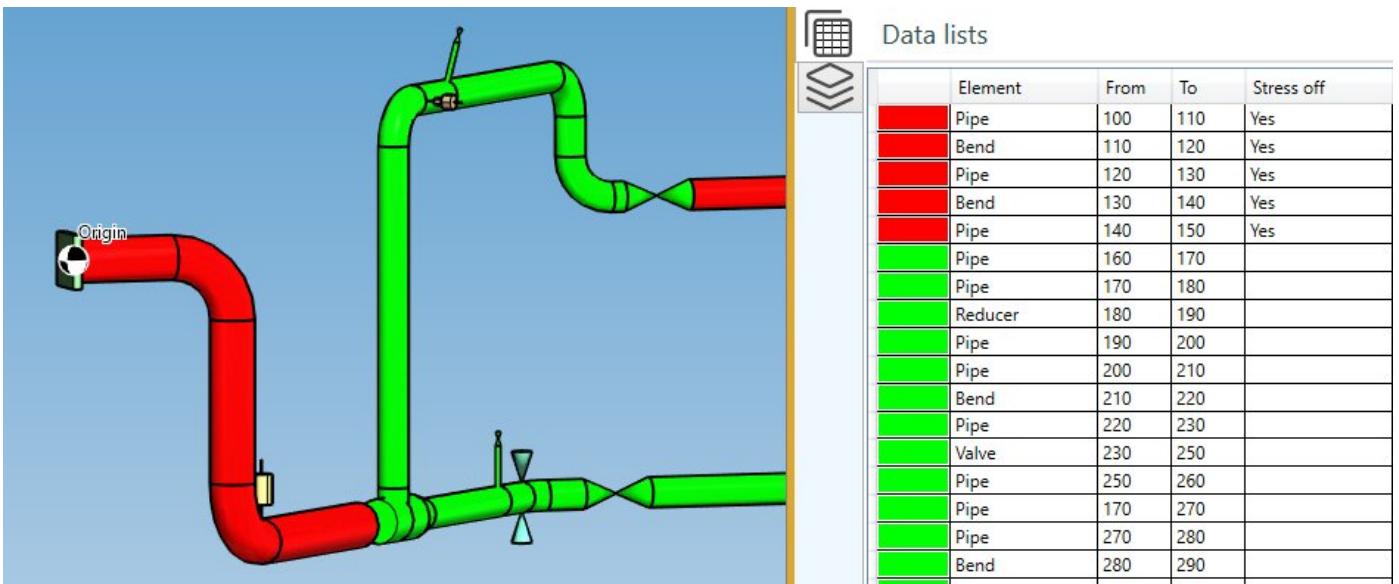
Code	SIFS
ASME Class 1	B1, C1, K1, B2, C2, K2, C3, CP, K3
ASME Class 2	i, B1, B2, B2', C2
B31.1	i
B31.1 B31.J	ii, io, it, ia, li, lo, It, la
EN 13480	io, ii
RCCM Class 2	i

The **INTERACTIVITY** is the same as explained on §5.

The data are not editable

16. Stress On/Off

Click on the **Stress On/Off** button :



Property	Definition
Color	Red = stress reporting off - Green = stress reporting on
Element type	-
From	Element Node1 name
To	Element Node2 name
Stress off	Yes or empty

Click [here](#) for more information about Stress On/Off definition.

The data are not editable

17. Soils

Click on the **Soils** button :

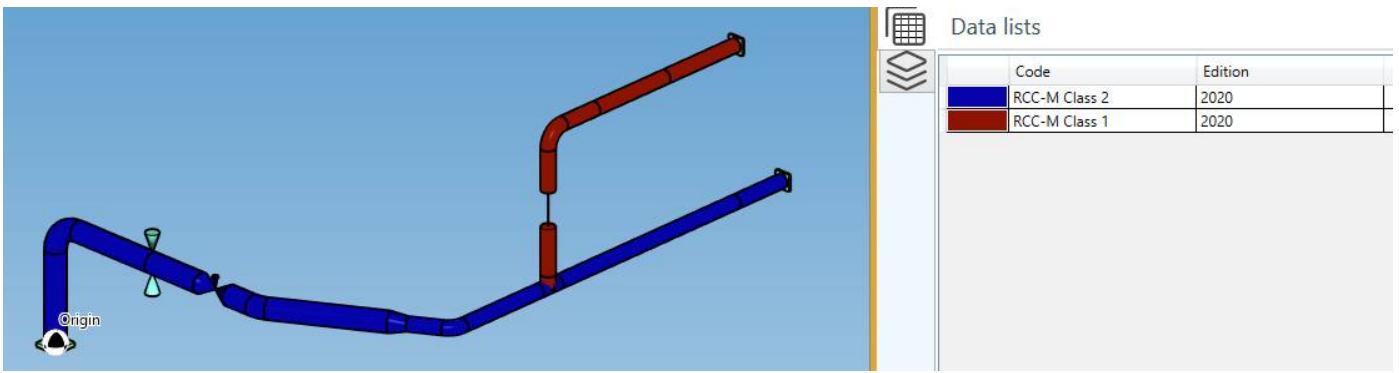


Click [here](#) for more information about soil definition.

The data are not editable

18. Piping codes

Click on the **Piping codes** button :

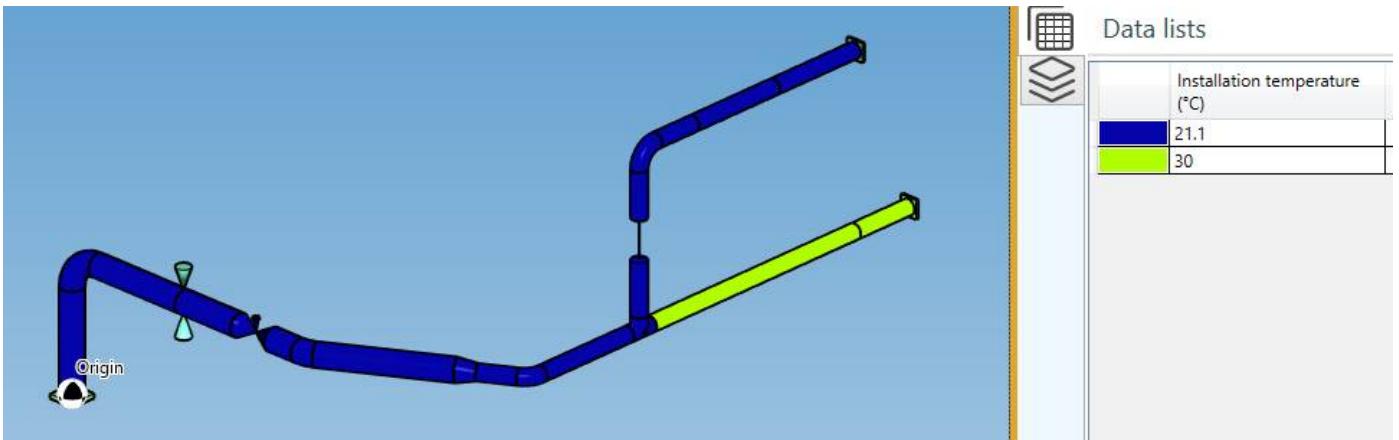


Click [here](#) for more information about piping code definition.

The data are not editable

19. Installation temperatures

Click on the **Installation temperatures** button :

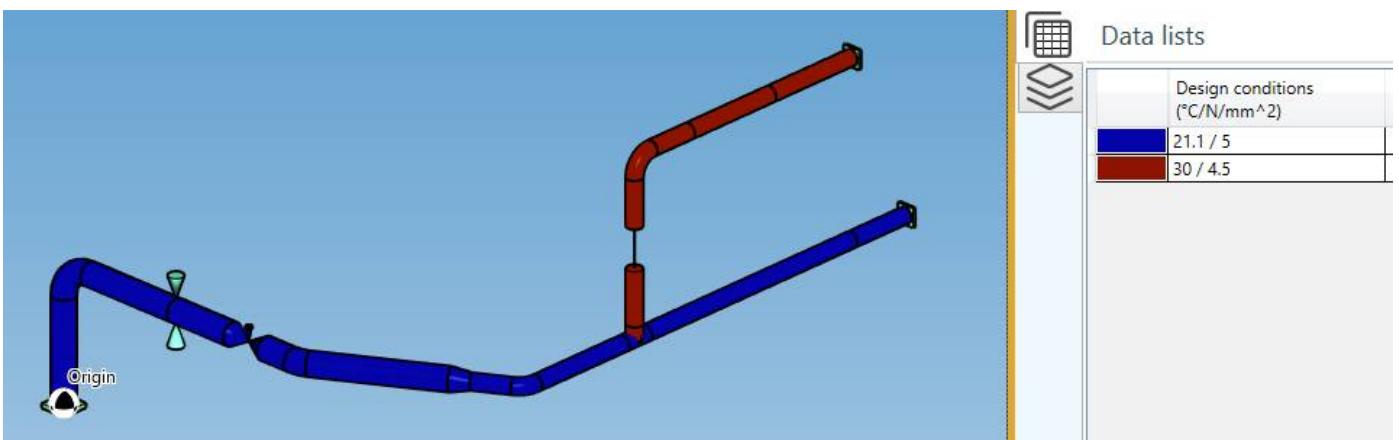


Click [here](#) for more information about installation temperature definition.

The data are not editable

20. Design conditions

Click on the **Design conditions** button :



Property	Unit Metric	Unit USA
Color	-	-
Design conditions (Temperature // Pressure)	°C // N/mm ²	°F // lb/in ²

Click [here](#) for more information about design condition definition.

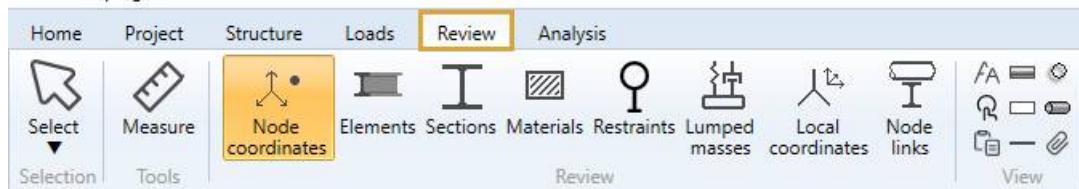
The data are not editable



Structure review

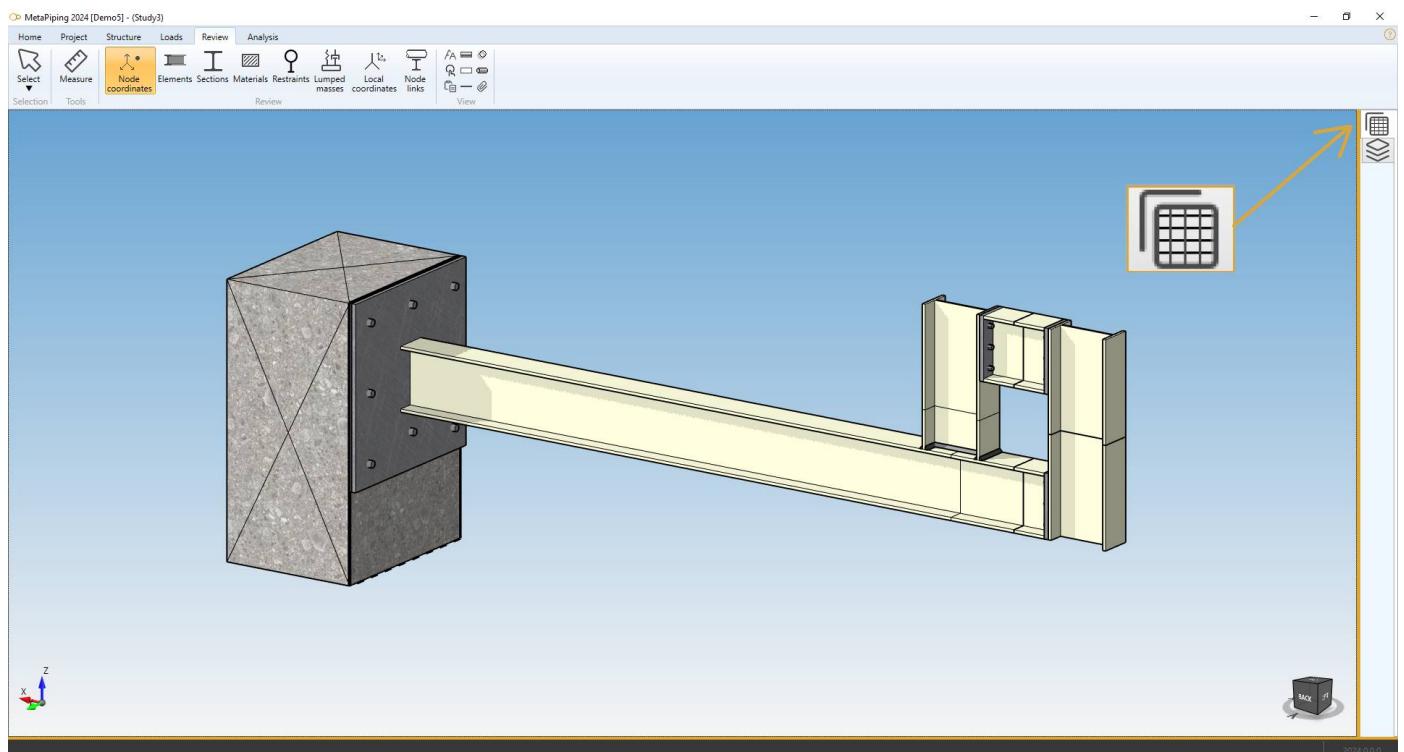
1. Ribbon menu

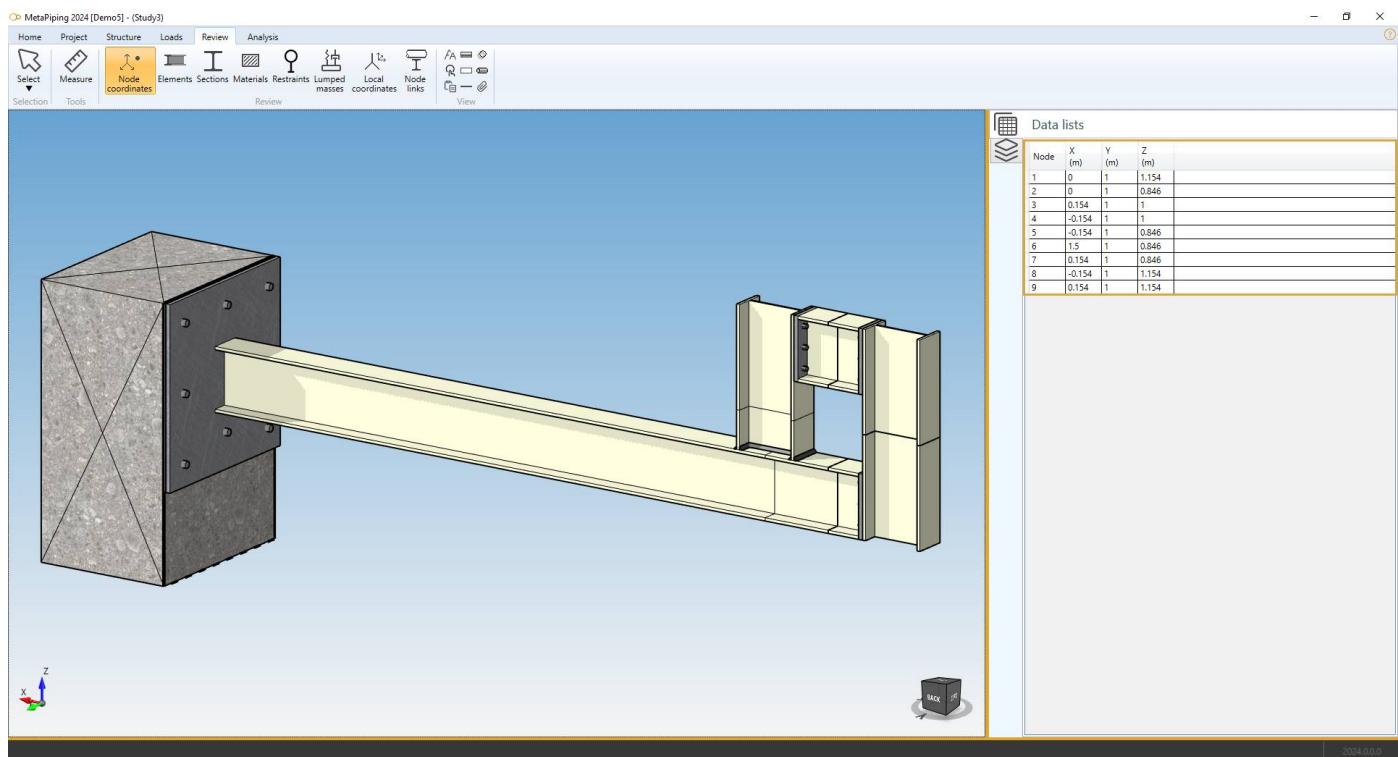
MetaPiping 2024



2. Data panel

A right panel can be opened by clicking on the top button. It presents the selected data :



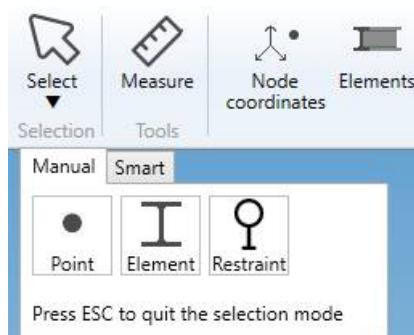


Click on the same button to hide the panel. Shortcut = F2.

TIP : you can copy the data by CTRL+C and paste on other software.

3. Select tool

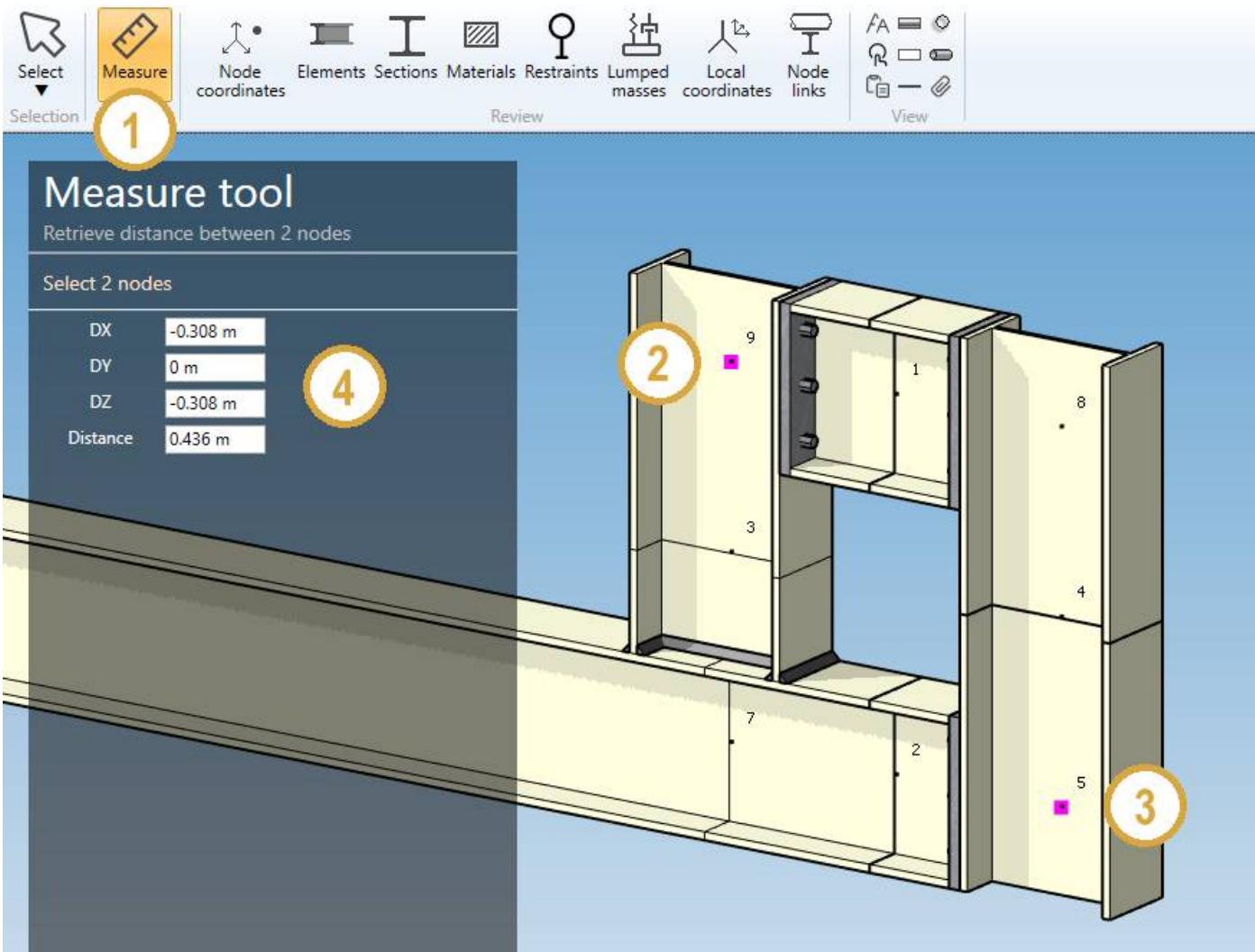
Click on the **Select** button to open the **Selection tool** :



Click [here](#) for more information about the selection tool.

4. Measure tool

MetaStructure lets you measure the **distance** between 2 nodes :



- 1 Select the **Measure** tool
- 2 Select the first node
- 3 Select the second node with CTRL pressed
- 4 The distances appear on the left panel

You can continue to select other nodes (with CTRL pressed), only the 2 last ones will be taken in account.

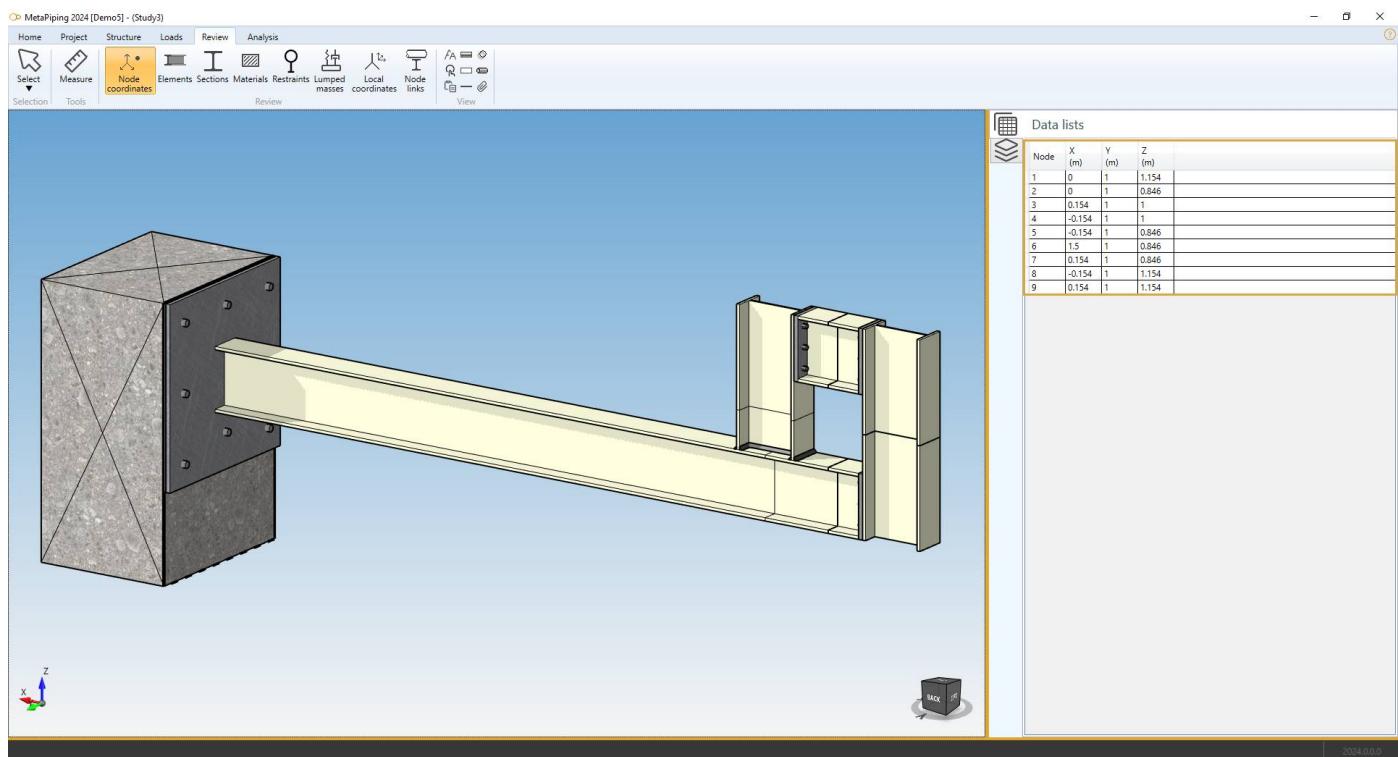
Property	Description	Unit Metric	Unit USA
DX	Distance on global X	m	ft
DY	Distance on global Y	m	ft
DZ	Distance on global Z	m	ft
Distance	Distance between the 2 nodes	m	ft

The **Selection mode** is automatically set to POINT when clicking the **Measure** button.

5. Node coordinates



Click on the **Node coordinates** button :



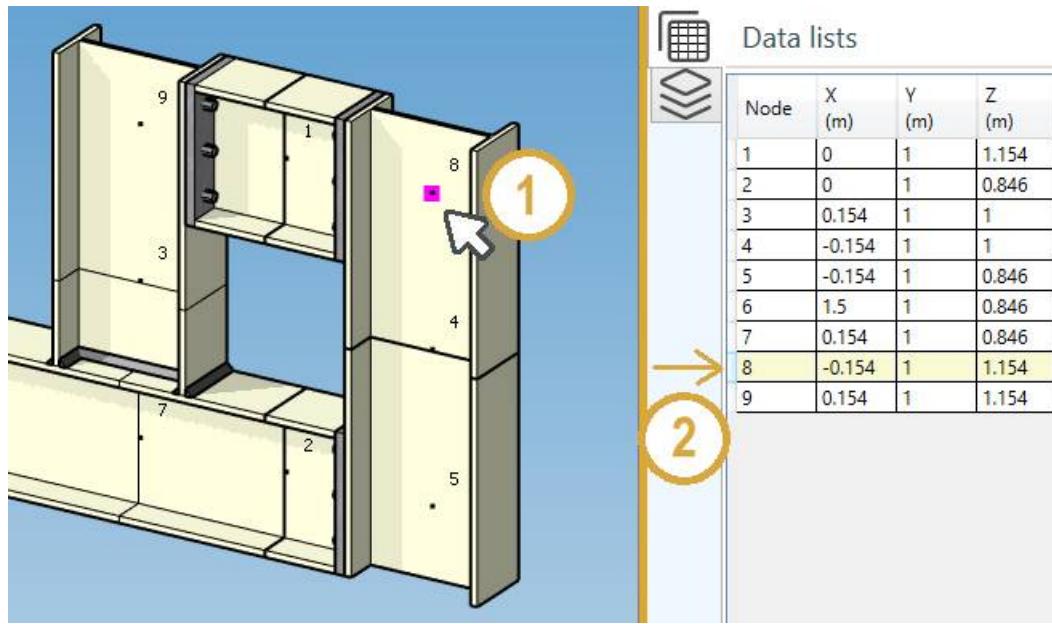
The **Data panel** shows the global coordinates X, Y, Z for each node.

Property	Description	Unit Metric	Unit USA
Node name	Text or number	-	-
X	X global coordinate	m	ft
Y	Y global coordinate	m	ft
Z	Z global coordinate	m	ft

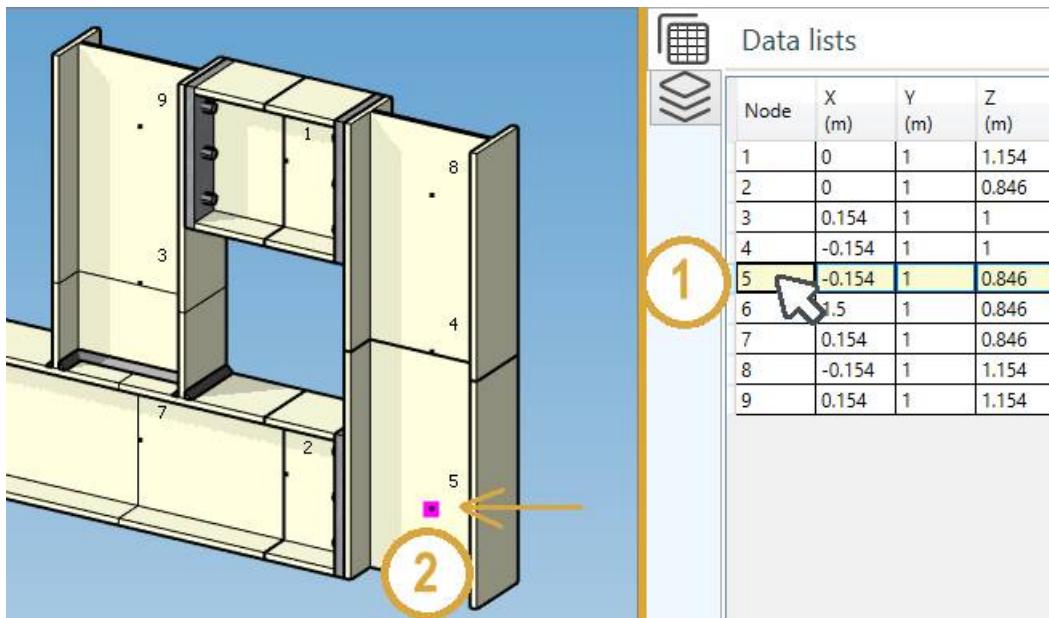
The **Selection mode** is automatically set to POINT

INTERACTIVITY :

- Select a node on the model (1) will highlight the corresponding row on the data list (2) :

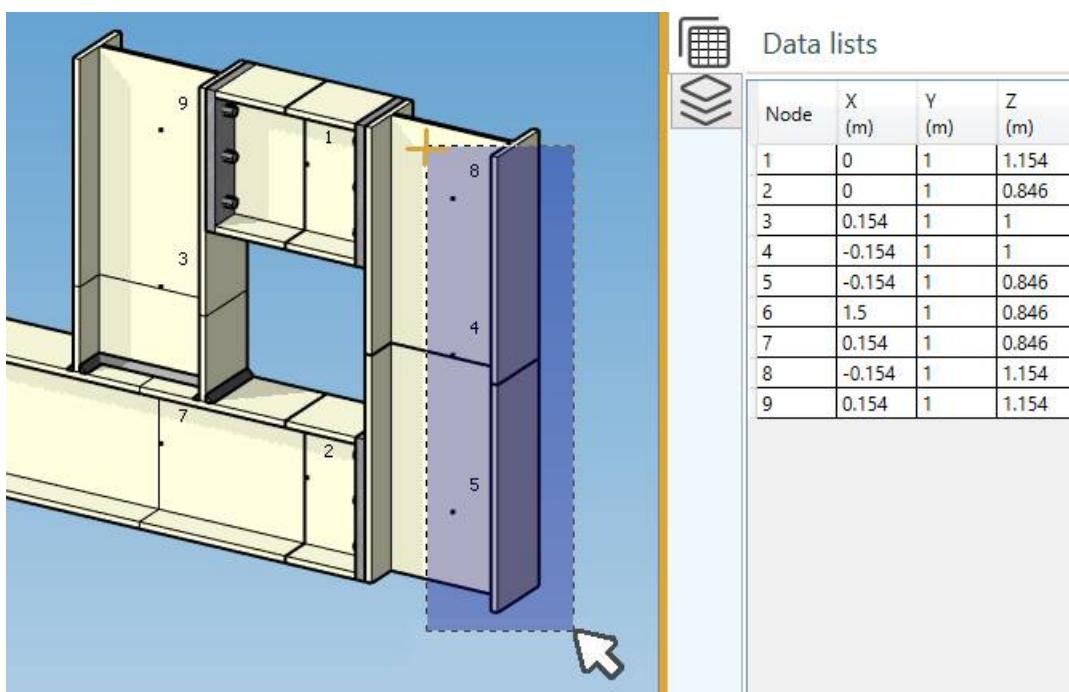


- Select a row on the data list (1) will highlight the corresponding node on the model (2) :

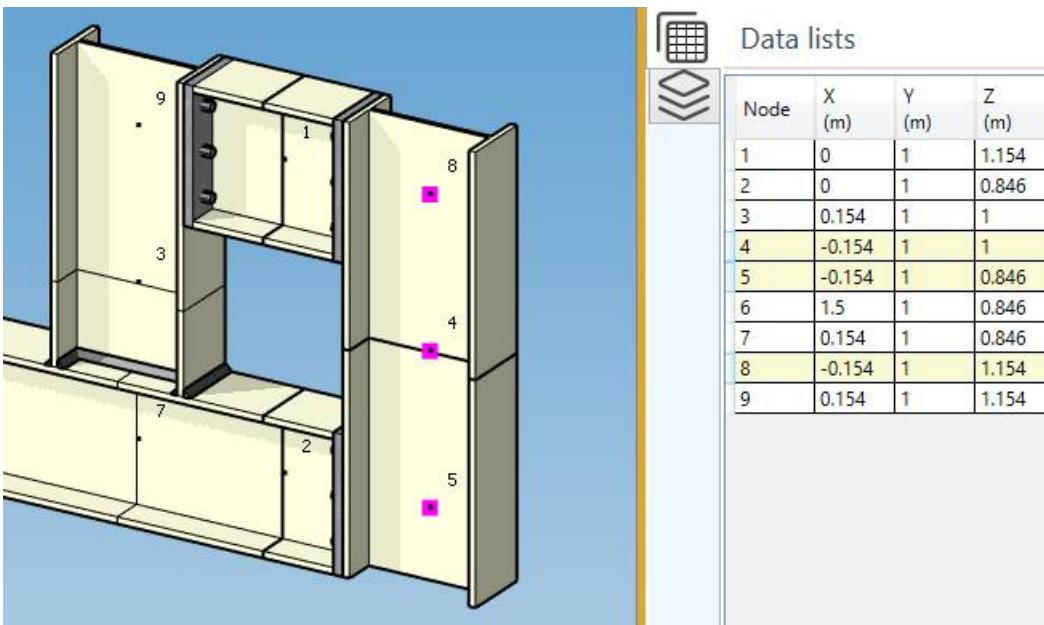


The data are not editable

- 1 Hold down the left mouse button to make a rectangular selection :



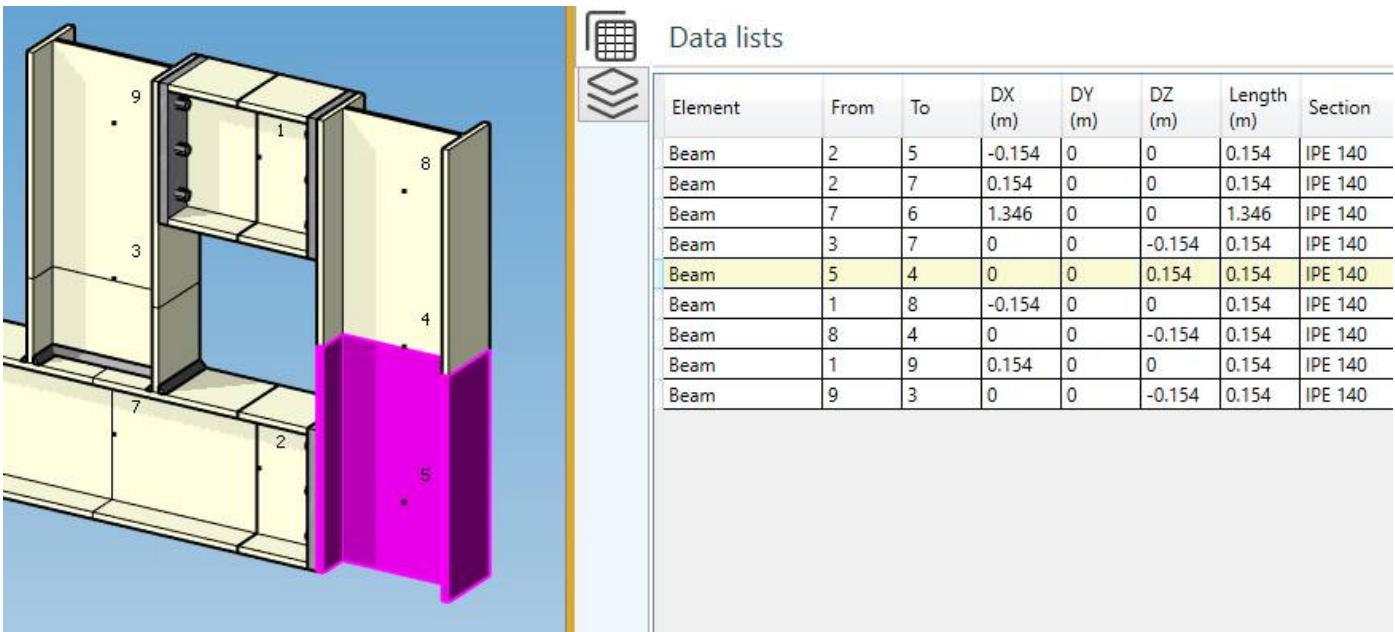
Release the mouse button, the data are highlighted :



6. Elements



Click on the **Elements** button :



Property	Unit Metric	Unit USA
Element type	-	-
Node1 name	-	-
Node2 name	-	-
DX	m	ft
DY	m	ft
DZ	m	ft
Length	m	ft
Section	-	-

The **INTERACTIVITY** is the same as explained on §5.

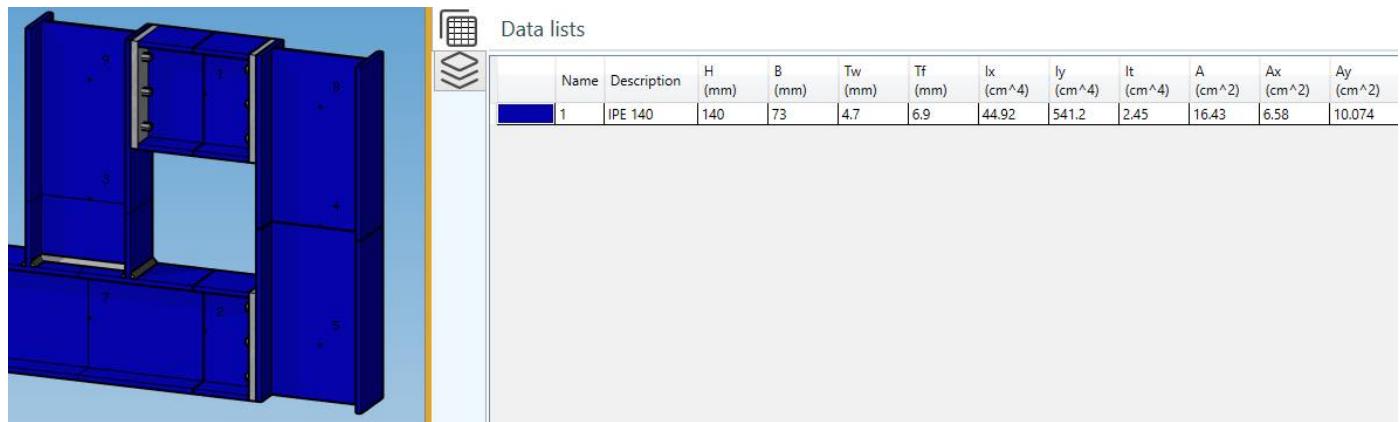
The data are not editable

7. Sections

Click on the **Sections** button :



The beam elements are colorized in the color of their corresponding section.



Property	Description	Unit Metric	Unit USA
Section color	-	-	-
Name	Section's name	-	-
Description	Section's description	-	-
h	Height	mm	in
b	Basis	mm	in
tw	Web thickness	mm	in
tf	Flange thickness	mm	in
Ix	Bending inertia X	cm ⁴	in ⁴
ly	Bending inertia Y	cm ⁴	in ⁴
It	Torsional inertia	cm ⁴	in ⁴
A	Area	cm ²	in ²
Ax	Shear area X	cm ²	in ²
Ay	Shear area Y	cm ²	in ²

The data are not editable

Other properties can be found on the **Data** screen. Click [here](#) for more information.

8. Materials



Click on the **Materials** button :

	Name	RefTemperature (°C)	MaxTemperature (°C)	ThermalExpansionOption	Density (kg/m³)	Poisson	Description
100	20	100	2		7850	0.3	S235

The elements are colorized in the color of their corresponding material.

Property	Unit Metric	Unit USA
Material color	-	-
Name	-	-
Ref temperature	°C	°F
Max temperature	°C	°F
Thermal expansion option	-	-
Density	kg/m³	lb/ft³
Poisson	-	-
Description	-	-

The data are not editable

9. Restraints



Click on the **Restraints** button :

Node	Restraint type	Level
6	AnchorPlate	

The **Data panel** shows the **Restraint type** and the **Level** for each restraint.

The **INTERACTIVITY** is the same as explained on §5.

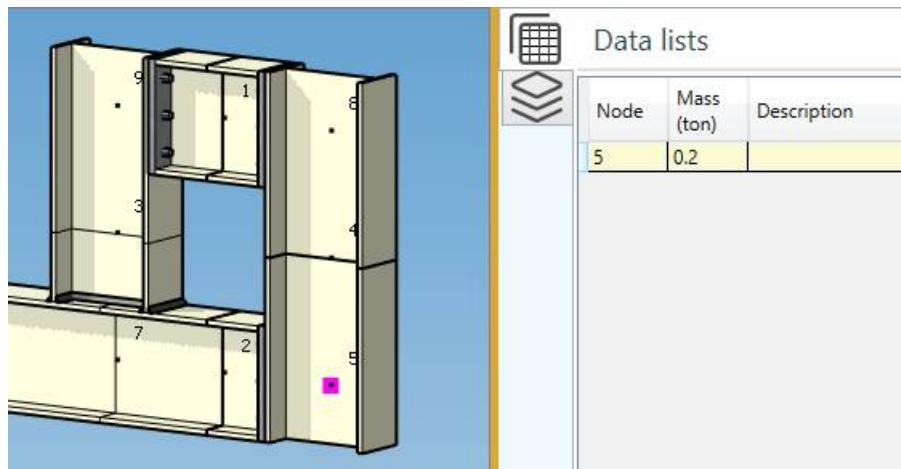
The data are not editable

Click [here](#) for more informations about **Level** during **Spectra** definition.

10. Lumped masses



Click on the **Lumped masses** button :



Property	Unit Metric	Unit USA
Node name	-	-
Mass	ton	kips
Description	-	-

Click [here](#) for more information about creation of lumped mass on node.

The **INTERACTIVITY** is the same as explained on §5.

The data are not editable

11. Local coordinates



It is useful to define a new coordinate system especially if we need to orient an **Anchor plate** on a node.

Node
(0, 0, 1)

Set node properties

Name	1	Modify	
Local X-axis and Z-axis directions			
Xx	0	Zx	-0.5
Xy	0	Zy	1
Xz	1	Zz	0
Modify			
Remove			

Lumped mass

Mass	0
Create	

Click on the **Local coordinates** button :

Data lists							
Node	Xx	Xy	Xz	Zx	Zy	Zz	
1	0	0	1	-0.5	1	0	

Property	Unit Metric	Unit USA
Node name	-	-
Xx	-	-
Xy	-	-
Xz	-	-
Zx	-	-
Zy	-	-
Zz	-	-

Click [here](#) for more information about creation a new coordinate system on node.

The **INTERACTIVITY** is the same as explained on §5.

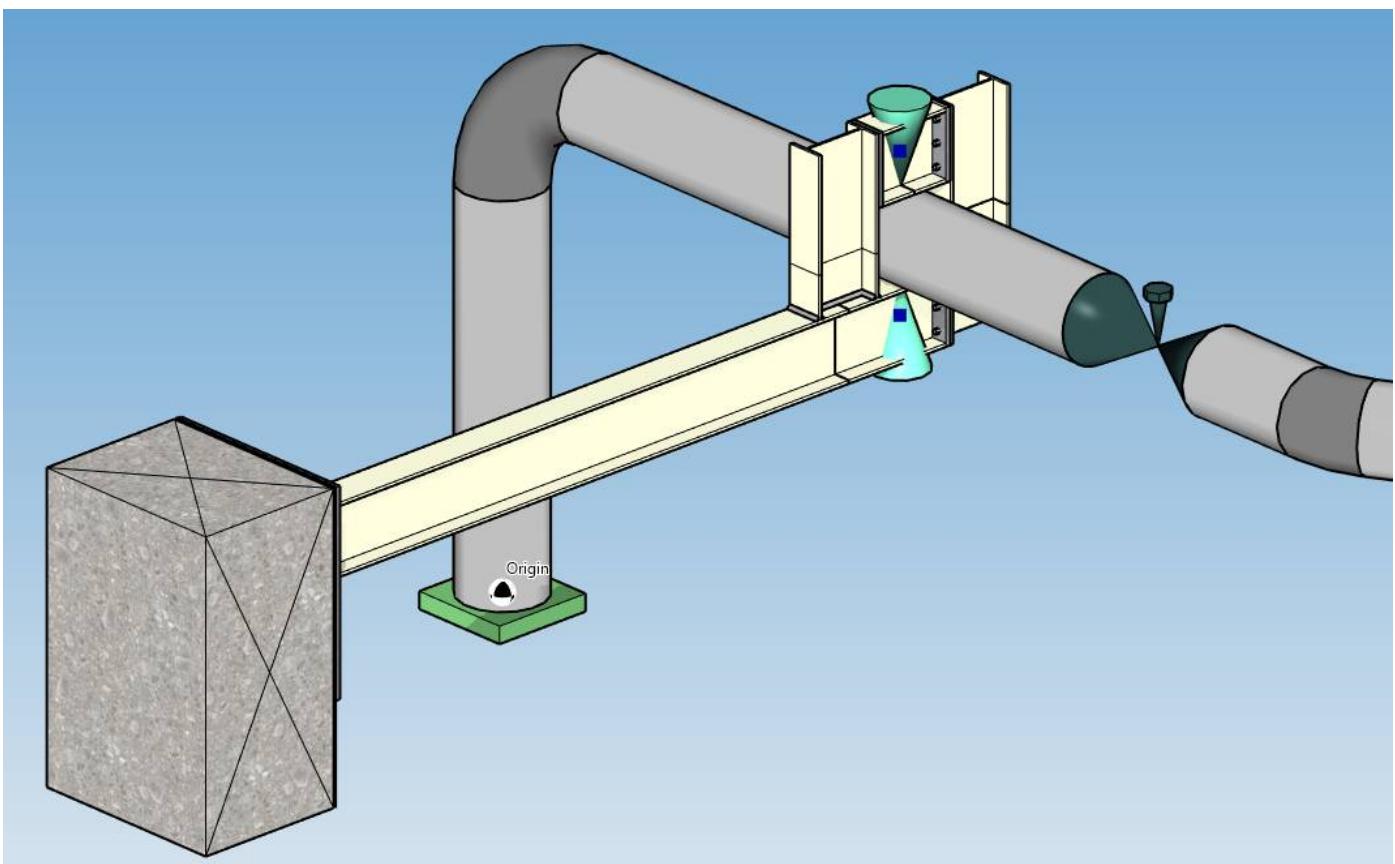
The data are not editable

12. Node links



This tool creates a Link between one piping node and multiple structure nodes.

This link is necessary for load definition (External load) and during analysis.



Click on the **Node links** button :

A screenshot of a software interface. On the left, there is a piping diagram with nodes numbered 1 through 9. Nodes 1 and 2 are highlighted with blue squares. On the right, there is a "Data lists" window with a table. The table has columns for Node 1, Node 2, Node 3, Node 4, Static Friction, Dynamic Friction, Study, and Piping node. The first row contains the values: Node 1 is 1, Node 2 is 2, Node 3 is empty, Node 4 is empty, Static Friction is 0.2, Dynamic Friction is 0.3, Study is Study2, and Piping node is 4. The "Node 1" column header is also highlighted with a blue square.

Property	Description
Color	-
Node 1	First structure node of the link
Node 2	Second structure node of the link
Node 3	Third structure node of the link
Node 4	Fourth structure node of the link
Static friction	Static friction factor
Dynamic friction	Dynamic friction factor
Study	Name of the linked piping study
Piping node	Name of the linked piping node

The data are not editable

Click [here](#) for more information about **links**.

13. View options



show/hide the node name

show/hide the node point

copy the 3D view to the clipboard

show the structure in shaded mode

show the structure in hidden lines mode

show the structure in wireframe mode

show/hide the shadows

shows/hides the silhouette drawing

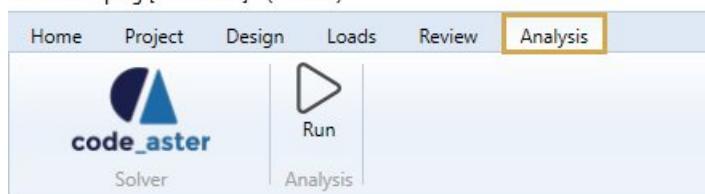
shows/hides the linked study models

Analysis

After designing the whole model, setting the loading and reviewing the data, click on the **ANALYSIS** button to launch a complete verification of the piping system.

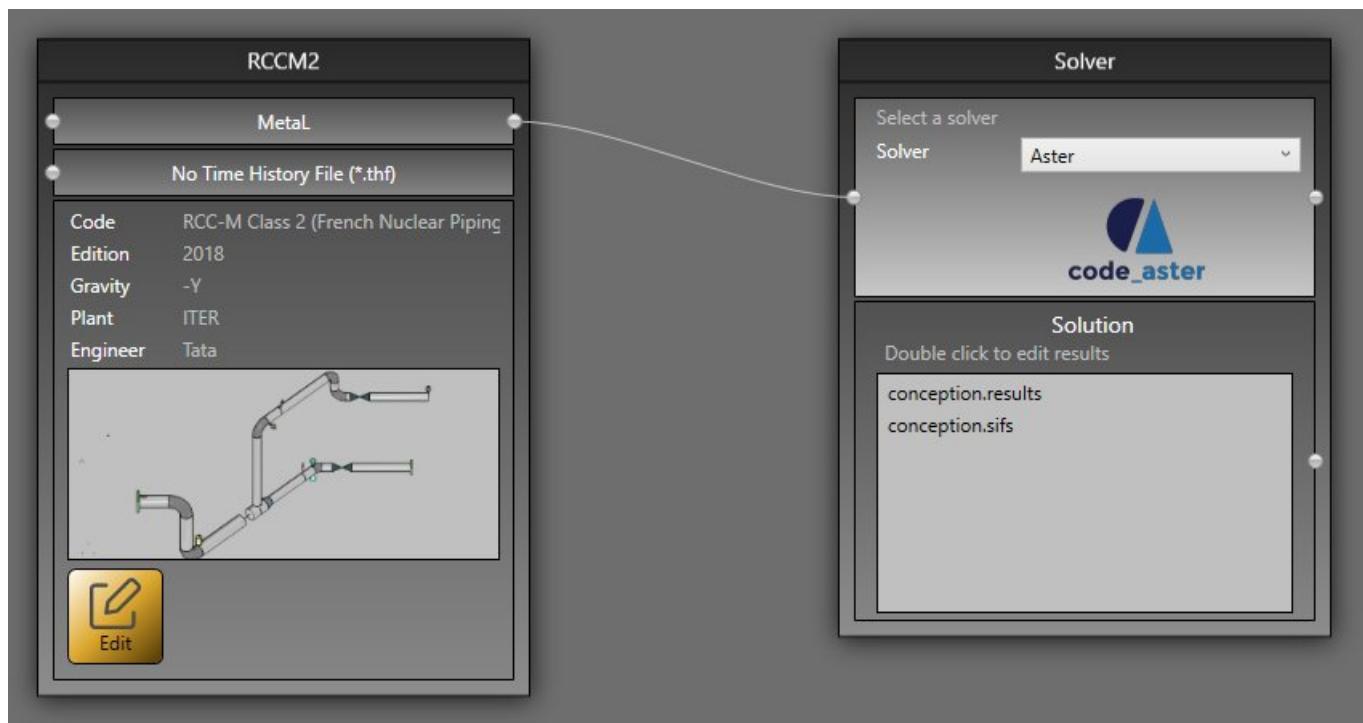
1. Ribbon menu

MetaPiping [COMPLEX] - (RCCM2)



1.1 Solver

Code Aster is the main SOLVER of MetaPiping and MetaStructure. The selection of the solver is done in the **Study** screen :



Click [here](#) to have more information about studies.

For **Editpipe** users, it is also possible to connect to PIPESTRESS solver.

Click [here](#) to have more information about PIPESTRESS settings.

TABLE OF CONTENTS

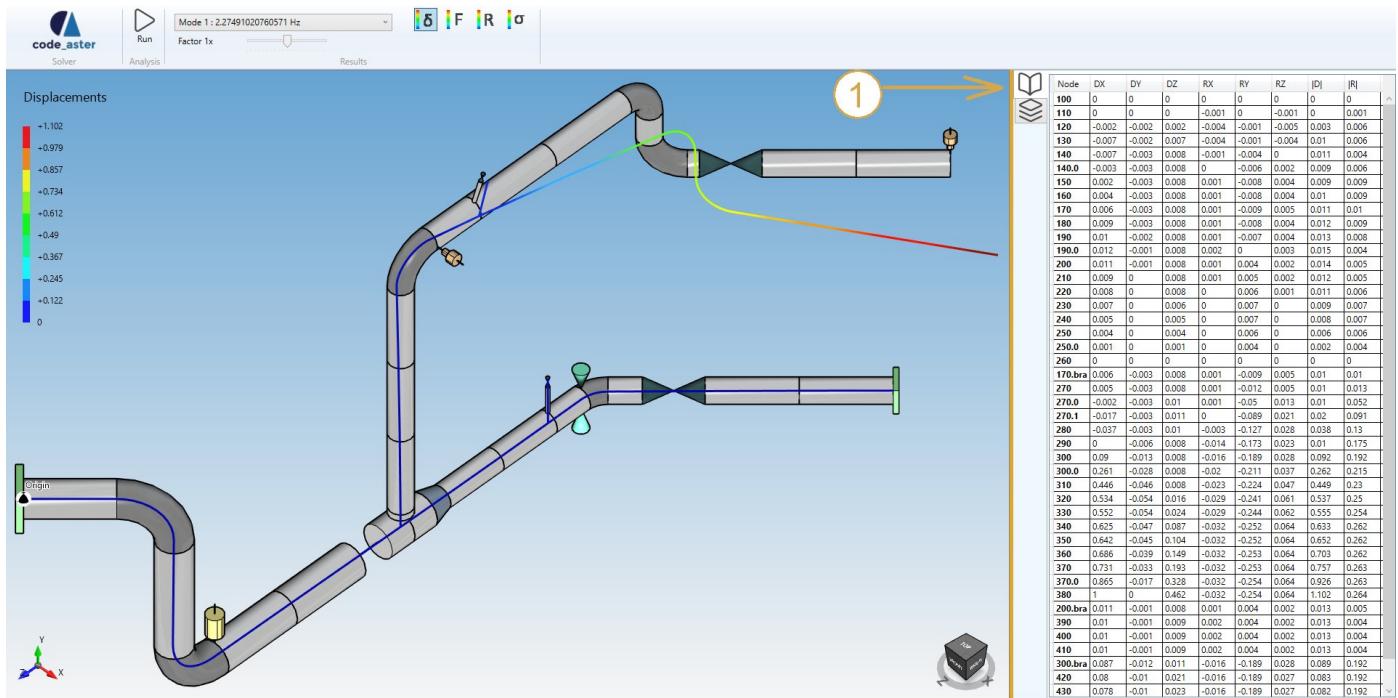
- [Piping](#)
- [Structure](#)
- [Finite Element Analysis](#)
- [Solvers](#)
- [Codes](#)



Piping analysis

1. Data panel

After analysis, a right panel can be opened by clicking on the top button (1). It presents the selected results :



Shortcut to open the data panel = F2.

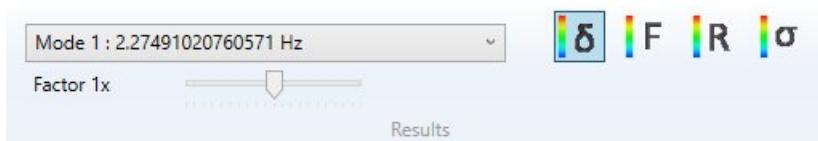
2. Results

Click on the **RUN** button :



Analysis

After a few seconds, the results of the analysis are shown. This panel will appear :

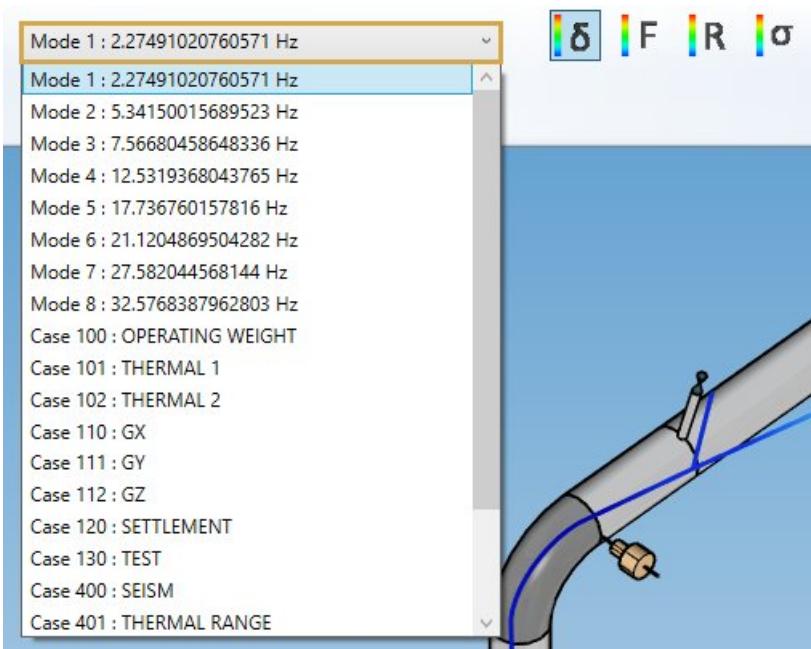


Select a case and the type of results :

- Displacements
- Forces and moments
- Reactions
- Stresses

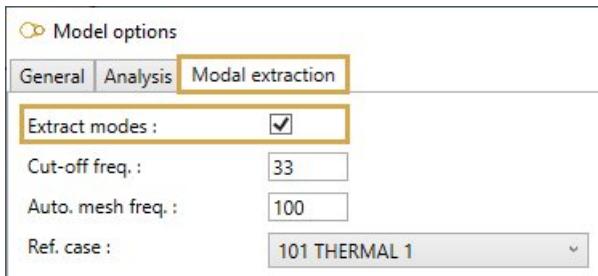
2.1 Load cases

The combobox shows all calculated load cases :



The loadings can be defined [here](#).

The **Modes** are generated if the **Extract modes** is checked in the **Model options** screen :

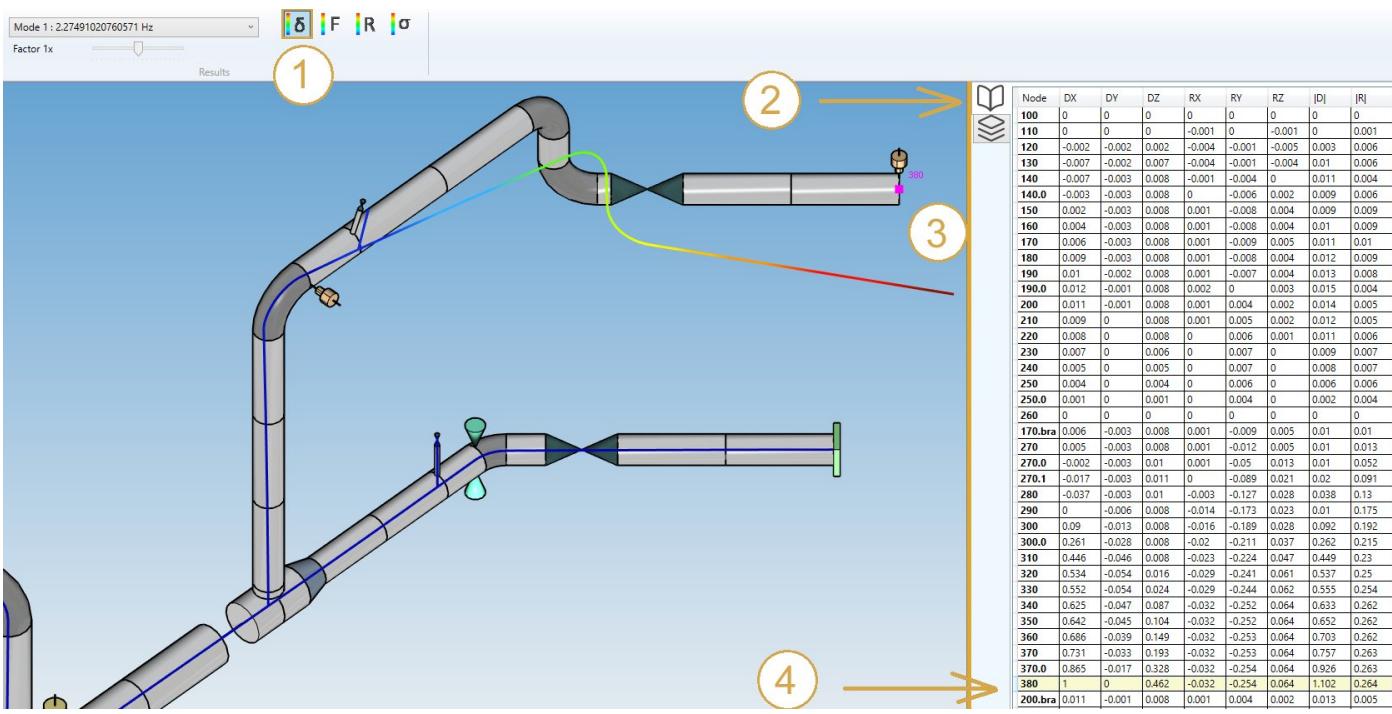


Click [here](#) to have more information about the model options.

A special load case "Fatigue" is added at the bottom of the combobox for Class 1 piping codes (Code Aster solver only). It allows to visualize the results of the detailed fatigue analysis : elastic shakedown (Eq. 10), usage factor, thermal ratchet, etc.

2.2 Displacements

- 1 Select the **Displacement** button
- 2 Open the right **Data panel**
- 3 Select a node
- 4 The results are highlighted



Shortcut to open the data panel = F2.

The **Data panel** shows the global displacements for each node.

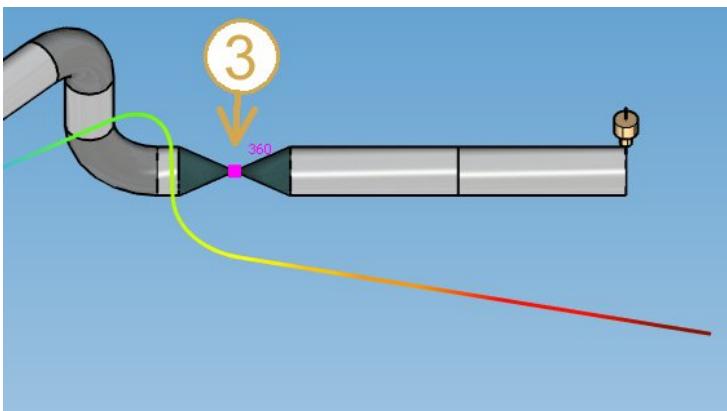
Property	Description	Unit Metric	Unit USA
Node name	Text or number	-	-
DX	X global displacement	mm	in
DY	Y global displacement	mm	in
DZ	Z global displacement	mm	in
RX	X global rotation	Rad	Rad
RY	Y global rotation	Rad	Rad
RZ	Z global rotation	Rad	Rad
D	global absolute displacement	mm	in
R	global absolute rotation	Rad	Rad

The **Selection mode** is automatically set to POINT.

You can amplify the displacement by changing the factor cursor.

INTERACTIVITY :

- 1 Click on a **Column header** to sort the results in ascending or descending order
- 2 Select a row
- 3 The node is highlighted

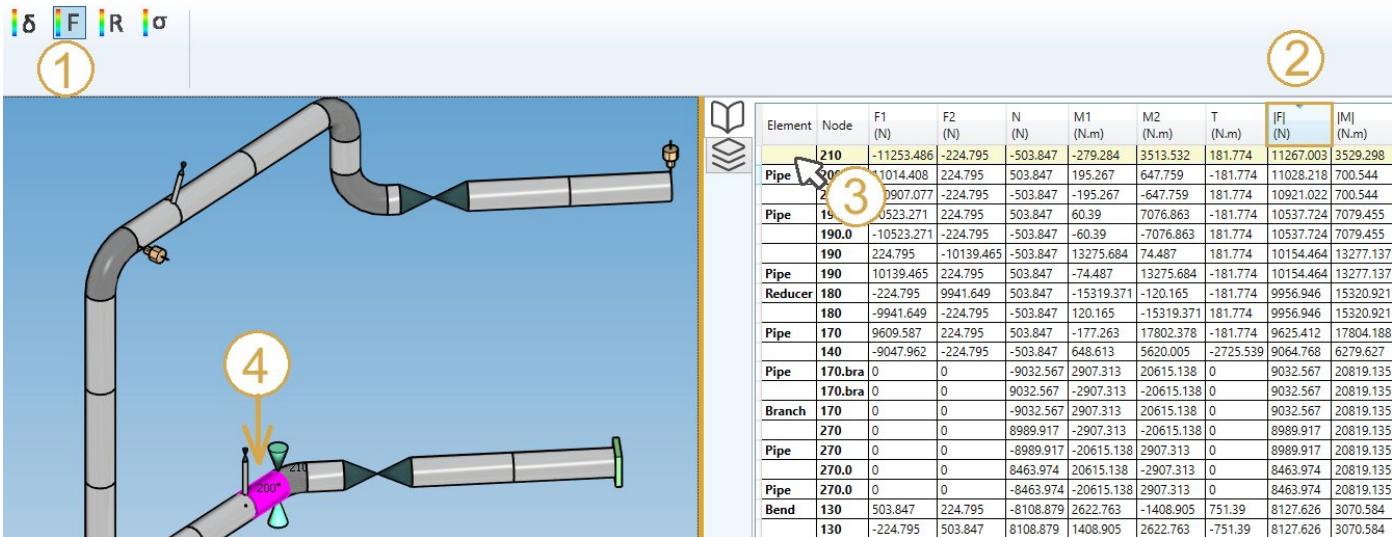


Node	DX	DY	DZ	RX	RY	JD1	JR1
380	1	0	0.462	-0.032	-0.254	1.102	0.264
370.0	0.865	-0.017	0.328	-0.032	-0.254	0.064	0.926
370	0.731	-0.033	0.193	-0.032	-0.253	0.064	0.757
360	0.686	-0.039	0.149	-0.032	-0.253	0.064	0.703
350	0.442	-0.045	0.109	-0.032	-0.252	0.064	0.652
340	0.625	-0.047	0.061	-0.032	-0.252	0.064	0.633
330	0.552	-0.054	0.016	-0.029	-0.244	0.062	0.555
320	0.534	-0.054	0.016	-0.029	-0.241	0.061	0.537
310	0.446	-0.046	0.008	-0.023	-0.224	0.047	0.449
300.0	0.261	-0.028	0.008	-0.02	-0.211	0.037	0.262
300	0.09	-0.013	0.008	-0.016	-0.189	0.028	0.092
300.bra	0.087	-0.012	0.011	-0.016	-0.189	0.028	0.089
420	0.08	-0.01	0.021	-0.016	-0.189	0.027	0.083
430	0.078	-0.01	0.023	-0.016	-0.189	0.027	0.082
440	0.077	-0.01	0.025	-0.016	-0.189	0.027	0.082
440	0.077	-0.01	0.025	-0.016	-0.189	0.027	0.082

TIP : you can copy the data by CTRL+C and paste in other software.

2.3 Forces and moments

- 1 Select the **Force** button
- 2 Click on a column of the **Data panel** to sort the data
- 3 Select a row
- 4 The corresponding element is highlighted



Element	Node	F1 (N)	F2 (N)	N (N)	M1 (N.m)	M2 (N.m)	T (N.m)	F (N)	M (N.m)
210	200	-11253.486	-224.795	-503.847	-279.284	3515.532	181.774	11267.003	3529.298
Pipe	200	1014.408	224.795	503.847	195.267	647.759	-181.774	11028.218	700.544
Pipe	2	0907.077	-224.795	-503.847	-195.267	-647.759	181.774	10921.022	700.544
Pipe	190	0523.271	224.795	503.847	60.39	7076.863	-181.774	10537.724	7079.455
Pipe	190.0	-10523.271	-224.795	-503.847	-60.39	-7076.863	181.774	10537.724	7079.455
Pipe	190	224.795	-10139.465	-503.847	13275.684	74.487	181.774	10154.464	13277.137
Pipe	190	10139.465	224.795	503.847	-74.487	13275.684	-181.774	10154.464	13277.137
Reducer	180	-224.795	9941.649	503.847	-15319.371	-120.165	-181.774	9956.946	15320.921
Reducer	180	-9941.649	-224.795	-503.847	120.165	-15319.371	181.774	9956.946	15320.921
Pipe	170	9609.587	224.795	503.847	-177.263	17802.378	-181.774	9625.412	17804.188
Pipe	140	-9047.962	-224.795	-503.847	648.613	5620.008	-2725.539	9064.768	6279.627
Pipe	170.bra	0	0	-9032.567	2907.313	20615.138	0	9032.567	20819.135
Pipe	170.bra	0	0	9032.567	-2907.313	-20615.138	0	9032.567	20819.135
Branch	170	0	0	-9032.567	2907.313	20615.138	0	9032.567	20819.135
Branch	270	0	0	8989.917	-2907.313	-20615.138	0	8989.917	20819.135
Pipe	270	0	0	-8989.917	-20615.138	2907.313	0	8989.917	20819.135
Pipe	270.0	0	0	8463.974	20615.138	-2907.313	0	8463.974	20819.135
Pipe	270.0	0	0	-8463.974	-20615.138	2907.313	0	8463.974	20819.135
Bend	130	503.847	224.795	-8108.879	2622.763	-1408.905	751.39	8127.626	3070.584
Bend	130	-224.795	503.847	8108.879	1408.905	2622.763	-751.39	8127.626	3070.584

Shortcut to open the data panel = F2.

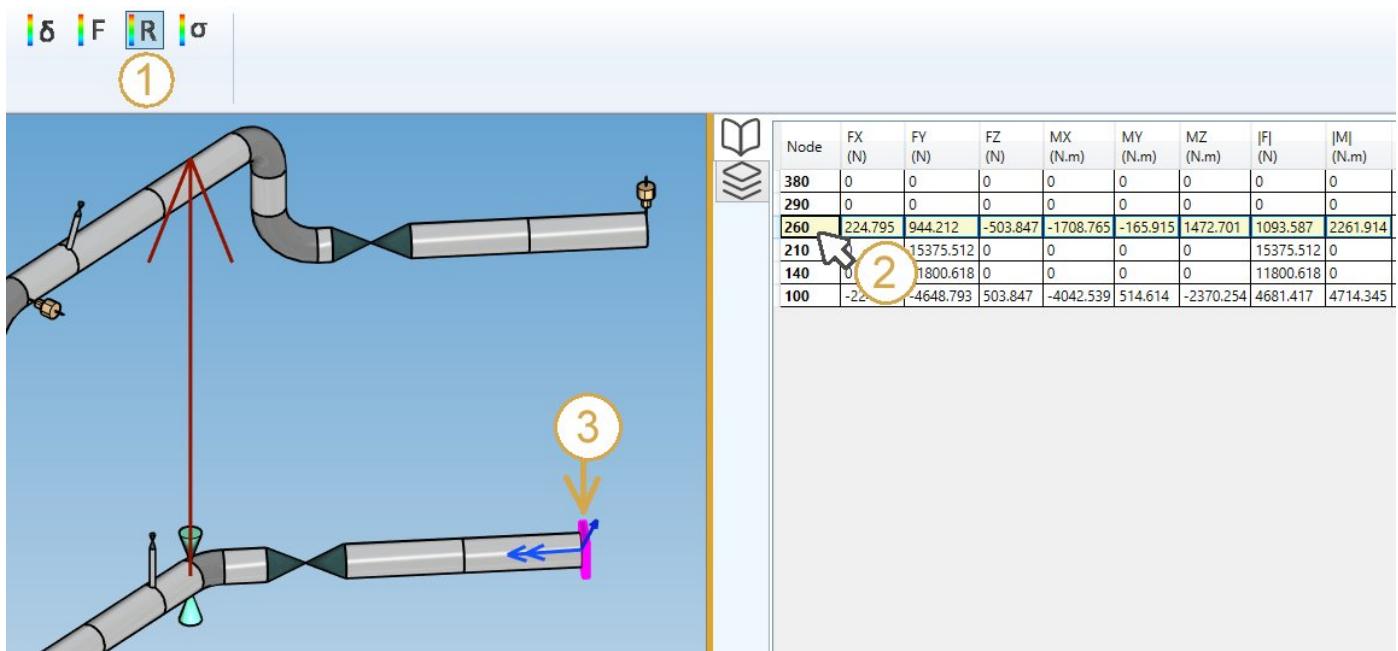
The **Data panel** shows the forces and moments for the 2 extremities of each element.

Property	Description	Unit Metric	Unit USA
Element	Type	-	-
Node	Node name of the extremity	-	-
F1	Force 1	N	lb
F2	Force 2	N	lb
N	Normal force	N	lb
M1	Moment 1	N.m	lb.ft
M2	Moment 2	N.m	lb.ft
T	Torsion	N.m	lb.ft
F	Absolute force	N	lb
M	Absolute moment	N.m	lb.ft

The **Selection mode** is automatically set to ELEMENT.

2.4 Reactions

- 1 Select the **Reaction** button
- 2 Select a row on the data panel
- 3 The corresponding restraint is highlighted



Or select directly a restraint to highlight the corresponding row.

Shortcut to open the data panel = F2.

The **Data panel** shows the forces and moments (reactions) for each node.

Property	Description	Unit Metric	Unit USA
Node	Node name	-	-
FX	X component of the reaction force	N	lb
FY	Y component of the reaction force	N	lb
FZ	Z component of the reaction force	N	lb
MX	X component of the reaction moment	N.m	lb.ft
MY	Y component of the reaction moment	N.m	lb.ft
MZ	Z component of the reaction moment	N.m	lb.ft
F	Absolute reaction force	N	lb
M	Absolute reaction moment	N.m	lb.ft

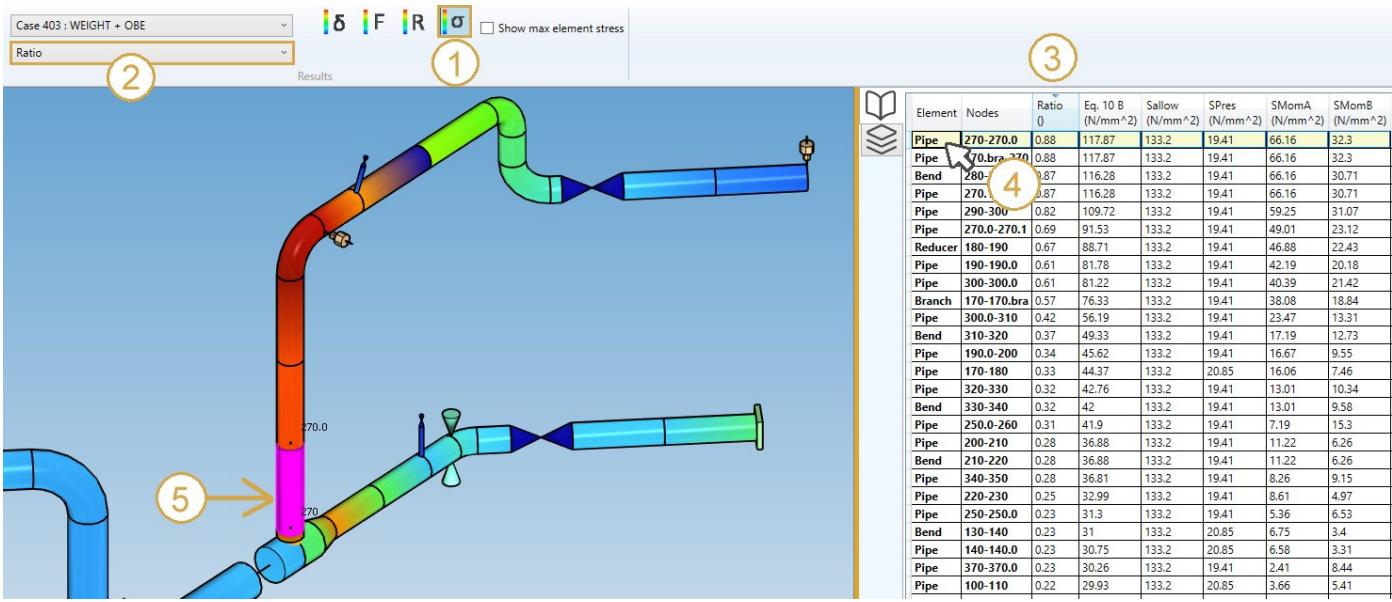
The **forces** are represented by a simple **arrow**.

The **moments** are represented by a double **arrow**.

The **Selection mode** is automatically set to RESTRAINT.

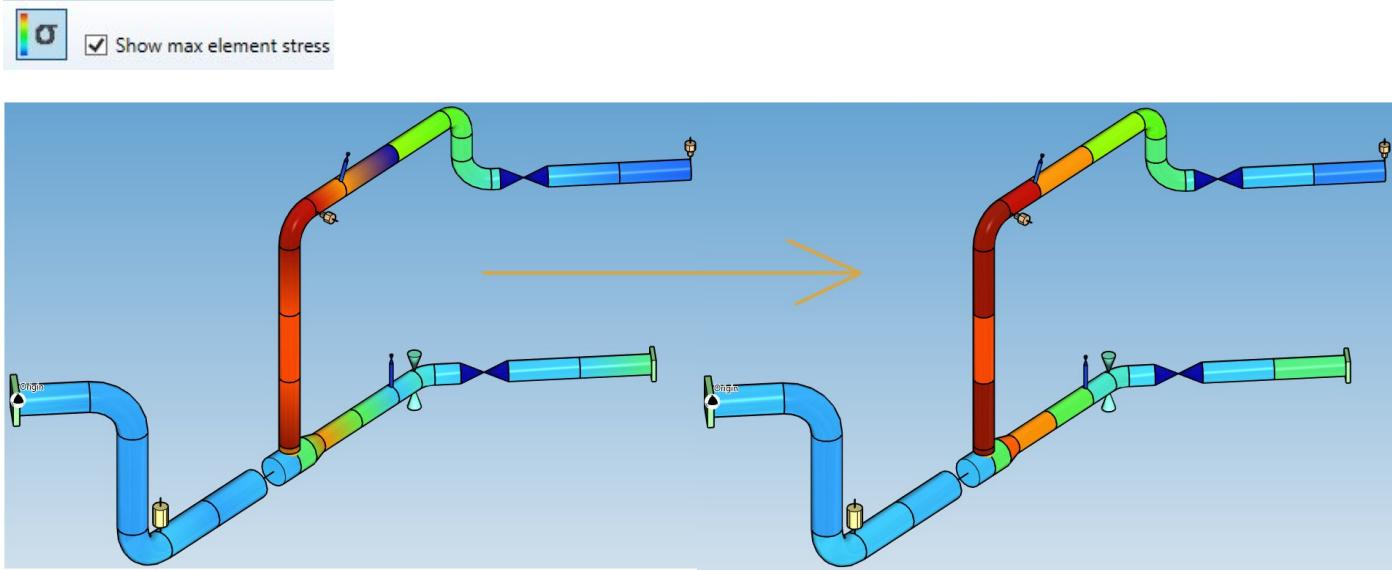
2.5 Stresses

- 1 Select the **Stress** button
- 2 Select the result type
- 3 Click on a column of the **Data panel** to sort the desired data
- 4 Select a row
- 5 The corresponding element is highlighted



The results are **code dependent**. The third column shows the stress ratio, defined as the calculated stress divided by the allowable stress.

The stresses are shown at each extremity of the elements. The maximum stresses of both extremities may be displayed instead by checking "Show max element stress" :



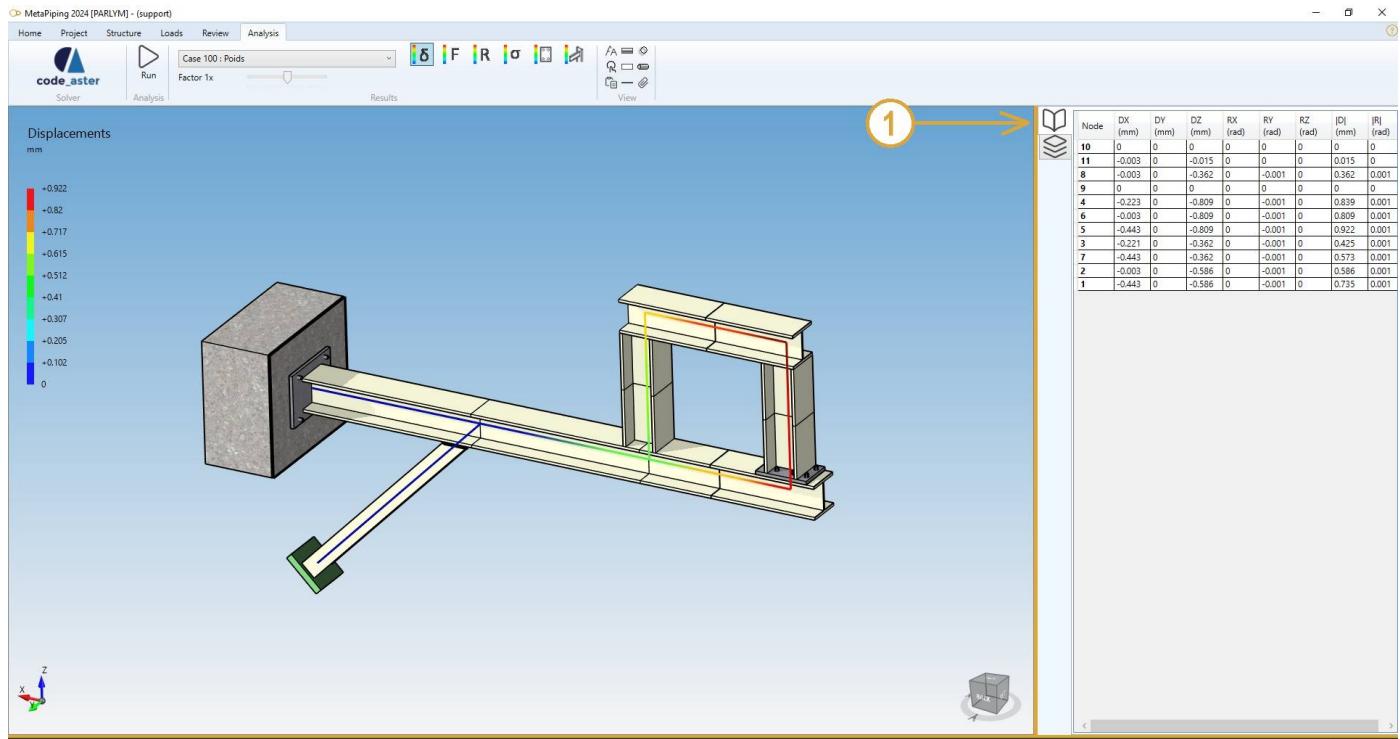
The **Selection mode** is automatically set to ELEMENT.



Structure analysis

1. Data panel

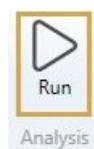
After analysis, a right panel can be opened by clicking on the top button (1). It presents the selected results :



Shortcut to open the data panel = F2.

2. Results

Click on the **RUN** button :



After a few seconds, the results of the analysis are shown. This panel will appear :

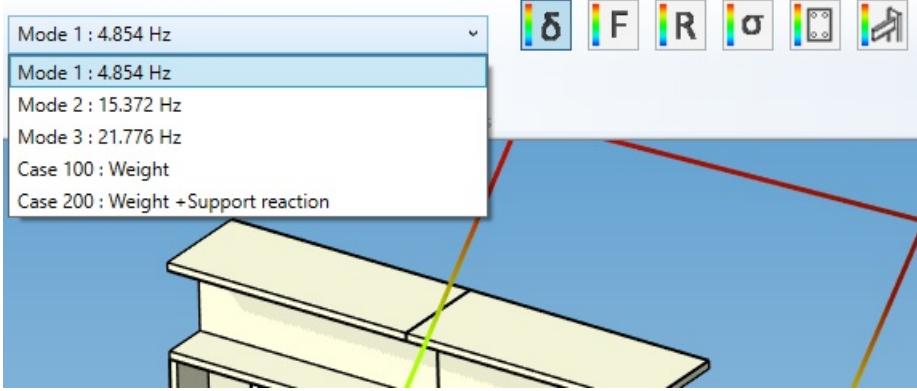


Select a case and the type of results :

- Displacements
- Forces and moments
- Reactions
- Stresses
- Anchor plates
- Joints

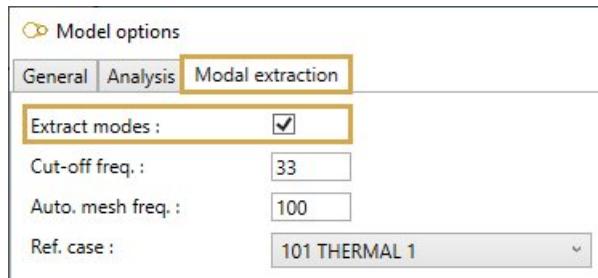
2.1 Load cases

The combobox shows all calculated load cases :



The loadings can be defined [here](#).

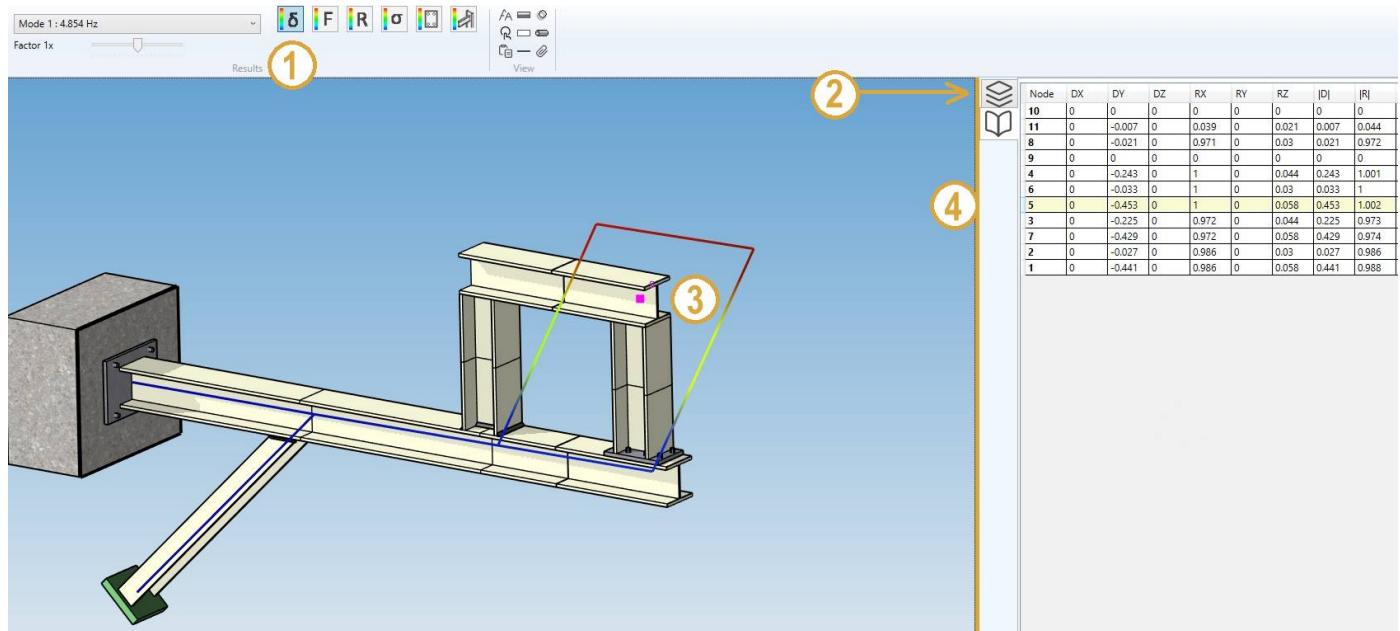
The **Modes** are generated if the **Extract modes** is checked in the **Model options** screen :



Click [here](#) to have more information about the model options.

2.2 Displacements

- 1 Select the **Displacement** button
- 2 Open the right **Data panel**
- 3 Select a node
- 4 The results are highlighted



Shortcut to open the data panel = F2.

The **Data panel** shows the global displacements for each node.

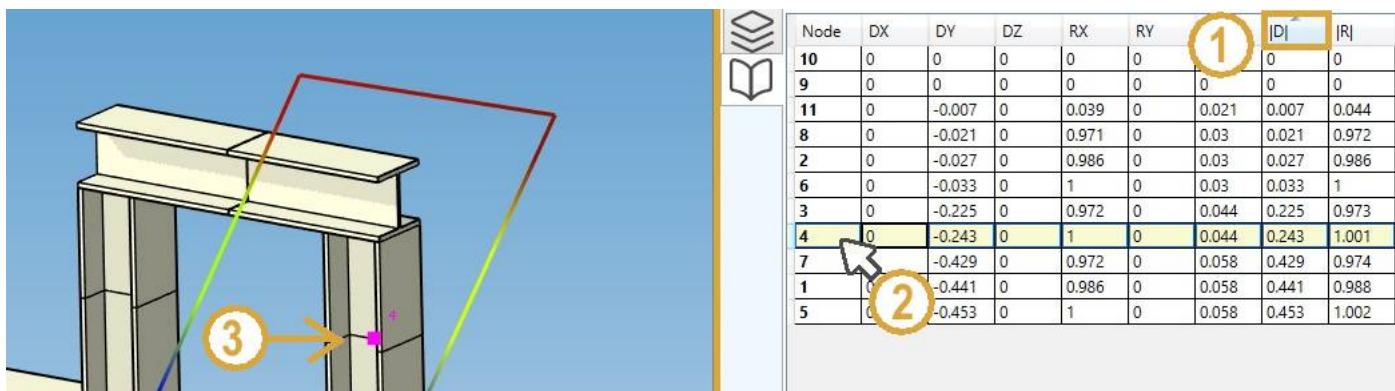
Property	Description	Unit Metric	Unit USA
Node name	Text or number	-	-
DX	X global displacement	mm	in
DY	Y global displacement	mm	in
DZ	Z global displacement	mm	in
RX	X global rotation	Rad	Rad
RY	Y global rotation	Rad	Rad
RZ	Z global rotation	Rad	Rad
D	global absolute displacement	mm	in
R	global absolute rotation	Rad	Rad

The **Selection mode** is automatically set to POINT.

You can amplify the displacement by changing the factor cursor.

INTERACTIVITY :

- 1 Click on a **Column header** to sort the results in ascending or descending order
- 2 Select a row
- 3 The node is highlighted



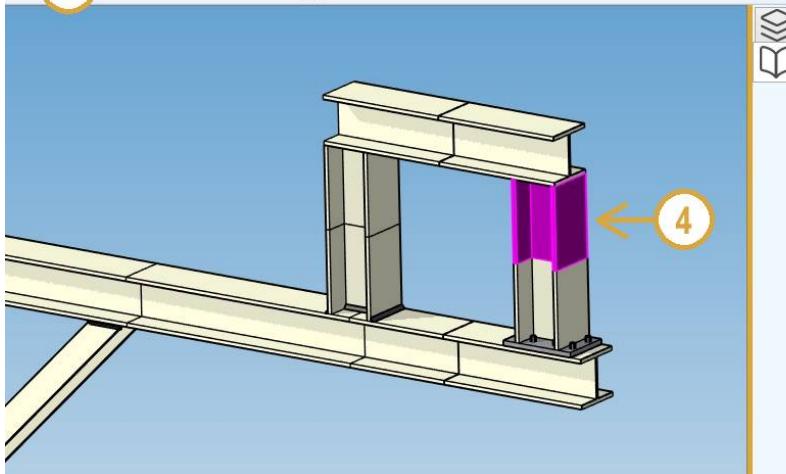
TIP : you can copy the data by CTRL+C and paste in other software.

2.3 Forces and moments

- 1 Select the **Force** button
- 2 Click on a column of the **Data panel** to sort the data
- 3 Select a row
- 4 The corresponding element is highlighted



View



2

Element	Node	F1 (N)	F2 (N)	N (N)	M1 (N.m)	M2 (N.m)	T (N.m)	F (N)	M (N.m)
6		-129.726	0	11.521	0	29.005	0	130.237	29.005
6		11.521	0	129.726	0	-29.005	0	130.237	29.005
Beam 4		129.726	0	-21.268	0	-1.773	0	131.458	1.773
Beam 2		309	0	-129.726	0	23.145	0	137.085	23.145
Beam 5		726	0	54.056	0	-25.46	0	140.538	25.46
5		4.056	0	-129.726	0	-25.46	0	140.538	25.46
Beam 1		86.845	0	129.726	0	10.671	0	156.112	10.671
Beam 1		-86.845	0	129.726	0	10.671	0	156.112	10.671
7		-119.633	0	-129.726	0	11.001	0	176.468	11.001
Beam 7		-129.726	0	-119.633	0	11.001	0	176.468	11.001
3		129.726	0	152.422	0	-38.234	0	200.153	38.234
3		-129.726	0.156	-152.422	0.046	38.234	0	200.153	38.234
Beam 2		1044.309	0	-129.726	0	23.145	0	1052.336	23.145
8		-1077.098	0	129.726	0	199.523	0	1084.882	199.523
8		129.726	-0.156	3431.266	-0.079	-65.467	0	3433.718	65.467
Beam 8		4508.364	0	0	0	-264.99	0	4508.364	264.99
11		-4586.46	0	0	0	2538.696	0	4586.46	2538.696
9		3013.419	0	-11313.888	0	-343.696	0	11708.32	343.696
Beam 11		-3091.515	0	11313.888	0	-1182.537	0	11728.662	1182.537
11		0	2570.979	13429.274	1356.158	0	0	13673.161	1356.158
Beam 10		0	-2521.333	-13478.92	444.246	0	0	13712.71	444.246

Shortcut to open the data panel = F2.

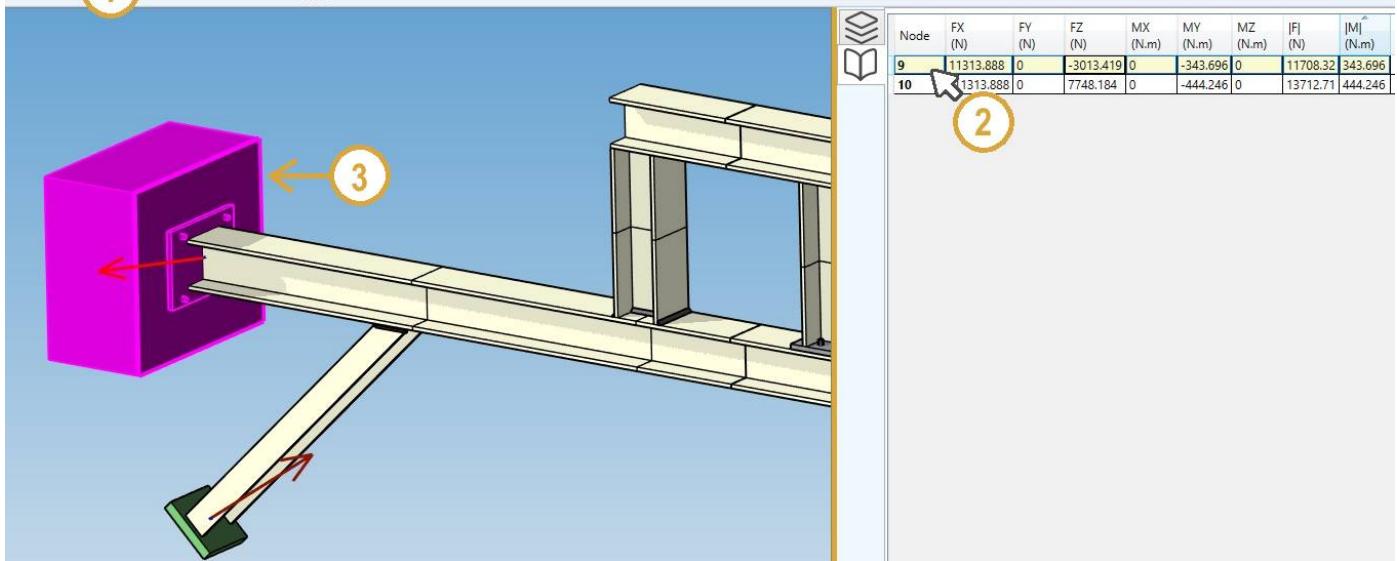
The **Data panel** shows the forces and moments for the 2 extremities of each element.

Property	Description	Unit Metric	Unit USA
Element	Type	-	-
Node	Node name of the extremity	-	-
F1	Force 1	N	lb
F2	Force 2	N	lb
N	Normal force	N	lb
M1	Moment 1	N.m	lb.ft
M2	Moment 2	N.m	lb.ft
T	Torsion	N.m	lb.ft
F	Absolute force	N	lb
M	Absolute moment	N.m	lb.ft

The **Selection mode** is automatically set to ELEMENT.

2.4 Reactions

- 1 Select the **Reaction** button
- 2 Select a row on the data panel
- 3 The corresponding restraint is highlighted



Or select directly a restraint to highlight the corresponding row.

Shortcut to open the data panel = F2.

The **Data panel** shows the forces and moments (reactions) for each node.

Property	Description	Unit Metric	Unit USA
Node	Node name	-	-
FX	X component of the reaction force	N	lb
FY	Y component of the reaction force	N	lb
FZ	Z component of the reaction force	N	lb
MX	X component of the reaction moment	N.m	lb.ft
MY	Y component of the reaction moment	N.m	lb.ft
MZ	Z component of the reaction moment	N.m	lb.ft
F	Absolute reaction force	N	lb
M	Absolute reaction moment	N.m	lb.ft

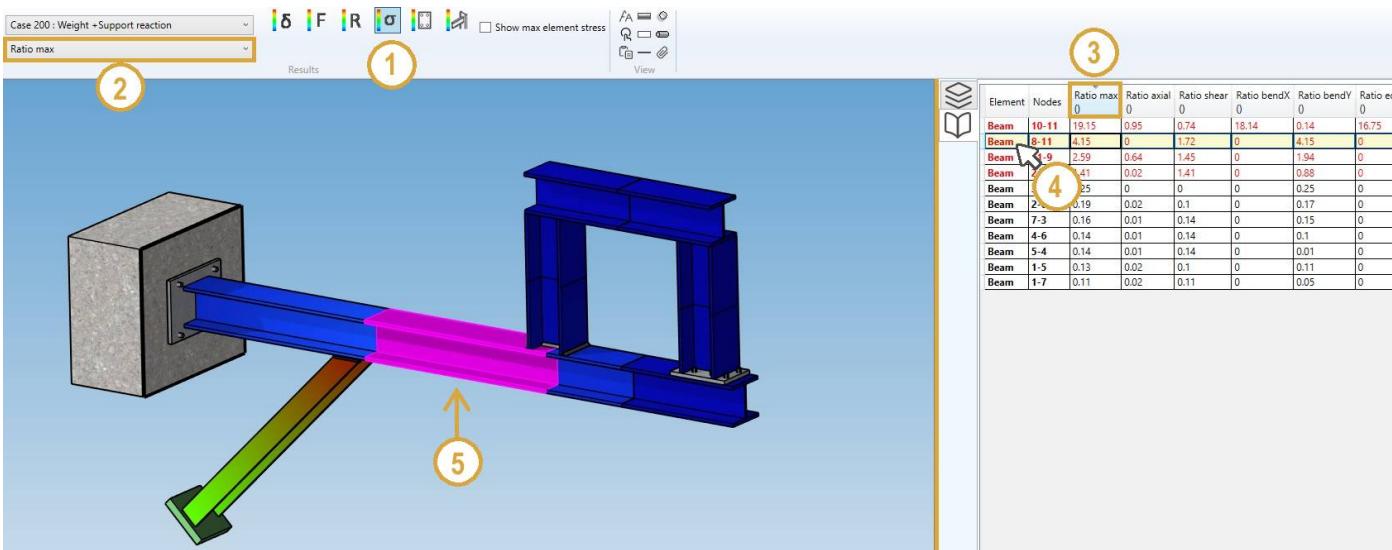
The **forces** are represented by a simple **arrow**.

The **moments** are represented by a double **arrow**.

The **Selection mode** is automatically set to RESTRAINT.

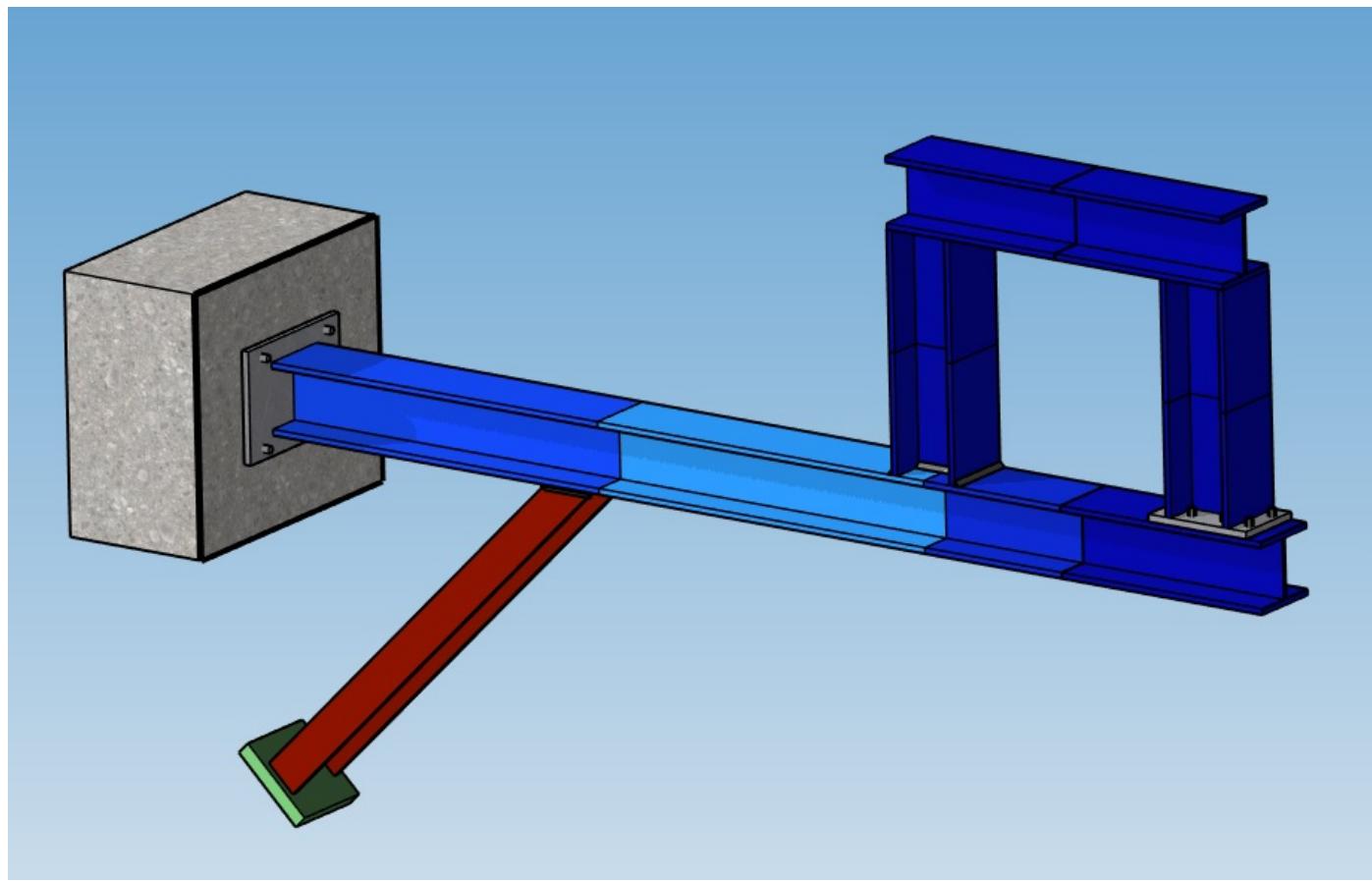
2.5 Stresses

- 1 Select the **Stress** button
- 2 Select the result type
- 3 Click on a column of the **Data panel** to sort the desired data
- 4 Select a row
- 5 The corresponding element is highlighted



The results are **code dependent**. The third column shows the stress ratio, defined as the calculated stress divided by the allowable stress.

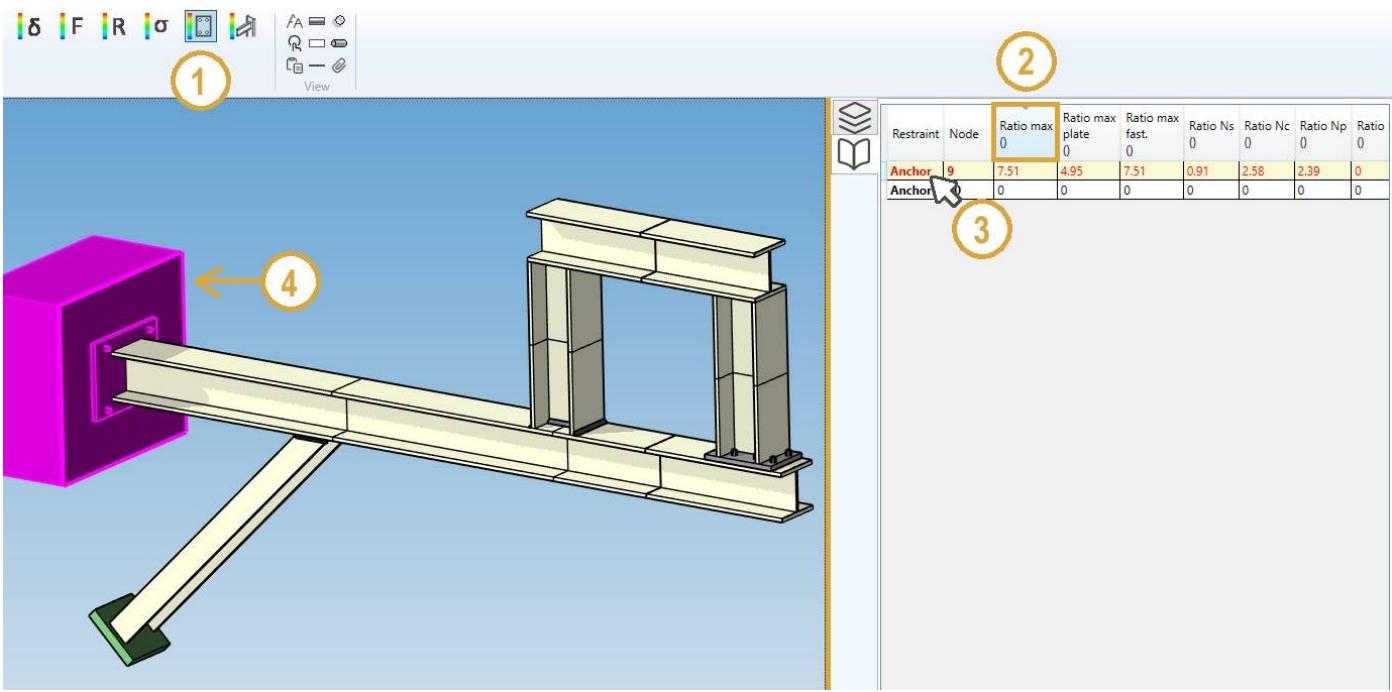
The stresses are shown at each extremity of the elements. The maximum stresses of both extremities may be displayed instead by checking "Show max element stress" :



The **Selection mode** is automatically set to ELEMENT.

2.6 Anchor plates

- 1 Select the **Anchor plate** button
- 2 Click on a column of the **Data panel** to sort the desired data
- 3 Select a row
- 4 The corresponding anchor is highlighted



Or select directly an anchor to highlight the corresponding row.

The results are **code dependent**. The plates are verified according to the structure code (RCC-M, ASME, etc.) and the fasteners according to the anchor code (only EN-1992-4 in the current software version).

Plates are only checked for regular configurations with 2 or 4 fasteners in the current version of MetaStructure.

The ratios output in **Data panel** are the calculated values (stress or force) divided by the allowable ones :

Property	Description
Restraint	Type of restraint
Node	Node name
Ratio max	Envelope of plate stress ratio and fastener ratios
Ratio max plate	Plate stress ratio
Ratio max fast.	Maximum ratio for the fasteners
Ratio Ns	Ratio of steel failure in tension
Ratio Nc	Ratio of concrete cone failure
Ratio Np	Ratio of pull-out failure
Ratio Npc	Ratio of combined pull-out and concrete failure (bonded fasteners)
Ratio Nsp	Ratio of failure by splitting
Ratio Ncb	Ratio of failure in tension by blow-out at the edge of the slab (cast-in fasteners only)
Ratio Vs	Ratio of steel failure in shear
Ratio NVs	Ratio of steel failure in combined tension and shear
Ratio Vcp	Ratio of concrete failure by pry-out effect
Ratio Vc	Ratio of failure in shear at the edge of the slab
Ratio NcMax	Max concrete ratio in tension
Ratio VcMax	Max concrete ratio in shear
Ratio NVc	Ratio of concrete failure in combined tension and shear
Error	Error number

The error numbers are listed below :

Error number	Description
0	No error
-1	Non regular arrangement, no check is carried out
2	Insufficient concrete thickness
3	Insufficient edge distance
4	Insufficient anchor spacing
5	Insufficient edge distance for selected spacing
6	Insufficient edge distance c1 in the case of mounting with lever arm
7	Insufficient edge distance c1 in case $t \geq 0.25 \text{ hef}$
9	Spacing must be $> 4 \text{ dnom}$ for slab edge failure

The **Selection mode** is automatically set to RESTRAINT.

2.7 Bolted and welded joints

- 1 Select the **Joint** button
- 2 Click on a column of the **Data panel** to sort the desired data
- 3 Select a row
- 4 The corresponding beam is highlighted

Element	Node	Ratio max	Ratio weld	Ratio lam. tear.	Ratio bolt tens.	Ratio bolt shear	Ratio bolt comb.	Ratio bearing
11	0	16.35	4	16.35	0	0	0	0
8	0.57	0.14	0.57	0	0	0	0	0
6	0.34	0	0	0.34	0.18	0.15	0.03	0
Beam 6	0	0	0	0	0	0	0	0
Beam 2	0	0	0	0	0	0	0	0
7	0	0	0	0	0	0	0	0
Beam 1	0	0	0	0	0	0	0	0
8	0	0	0	0	0	0	0	0
Beam 2	0	0	0	0	0	0	0	0
3	0	0	0	0	0	0	0	0
Beam 7	0	0	0	0	0	0	0	0
Beam 3	0	0	0	0	0	0	0	0
4	0	0	0	0	0	0	0	0
Beam 5	0	0	0	0	0	0	0	0
Beam 4	0	0	0	0	0	0	0	0
9	0	0	0	0	0	0	0	0
Beam 11	0	0	0	0	0	0	0	0
11	0	0	0	0	0	0	0	0
Beam 8	0	0	0	0	0	0	0	0
Beam 1	0	0	0	0	0	0	0	0
5	0	0	0	0	0	0	0	0

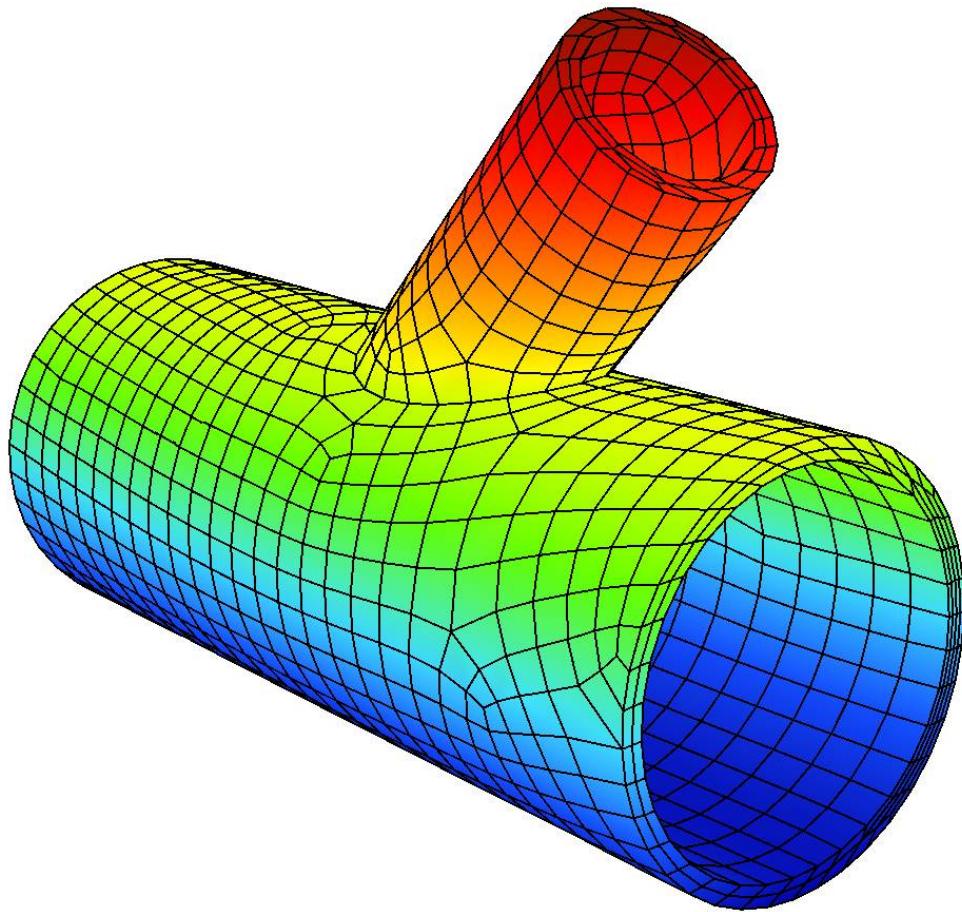
Or select directly a beam to highlight the corresponding row.

The results are **code dependent**. The ratios output in **Data panel** are the calculated values (stress or force) divided by the allowable ones :

Property	Description
Element	Type of element
Node	Node of the element where the joint is located
Ratio max	Max ratio of the welds or the bolts
Ratio weld	Ratio of the welds (0 if bolted joint)
Ratio lam. tear.	Ratio for lamellar tearing on the base metal (0 if bolted joint)
Ratio bolt tens.	Ratio of the bolts in tension
Ratio bolt shear.	Ratio of the bolts in shear
Ratio bolt comb.	Ratio of the bolts in combined tension and shear
Ratio bearing	Ratio of the bearing stress

The **Selection mode** is automatically set to ELEMENT.

Finite Element Analysis



MetaPiping/MetaStructure proposes detailed **finite element analysis** of part of piping system or structure.

The selected assembly is meshed with shell or brick element depending on the assembly type.

Loads are then applied on the resulting mesh and the boundary conditions are defined.

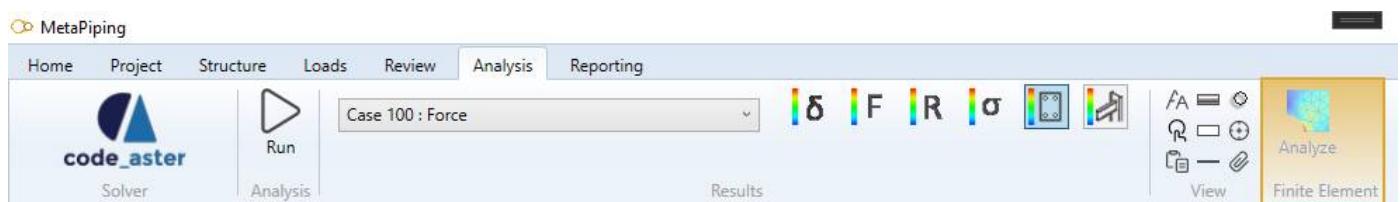
The solution is calculated with our native **Code_Aster** solver.

A diagnostic is shown depending on the assembly type.

A report can finally be generated.

1. Analysis

The **Finite Element Analysis** button is available at the end of the Analysis ribbon :



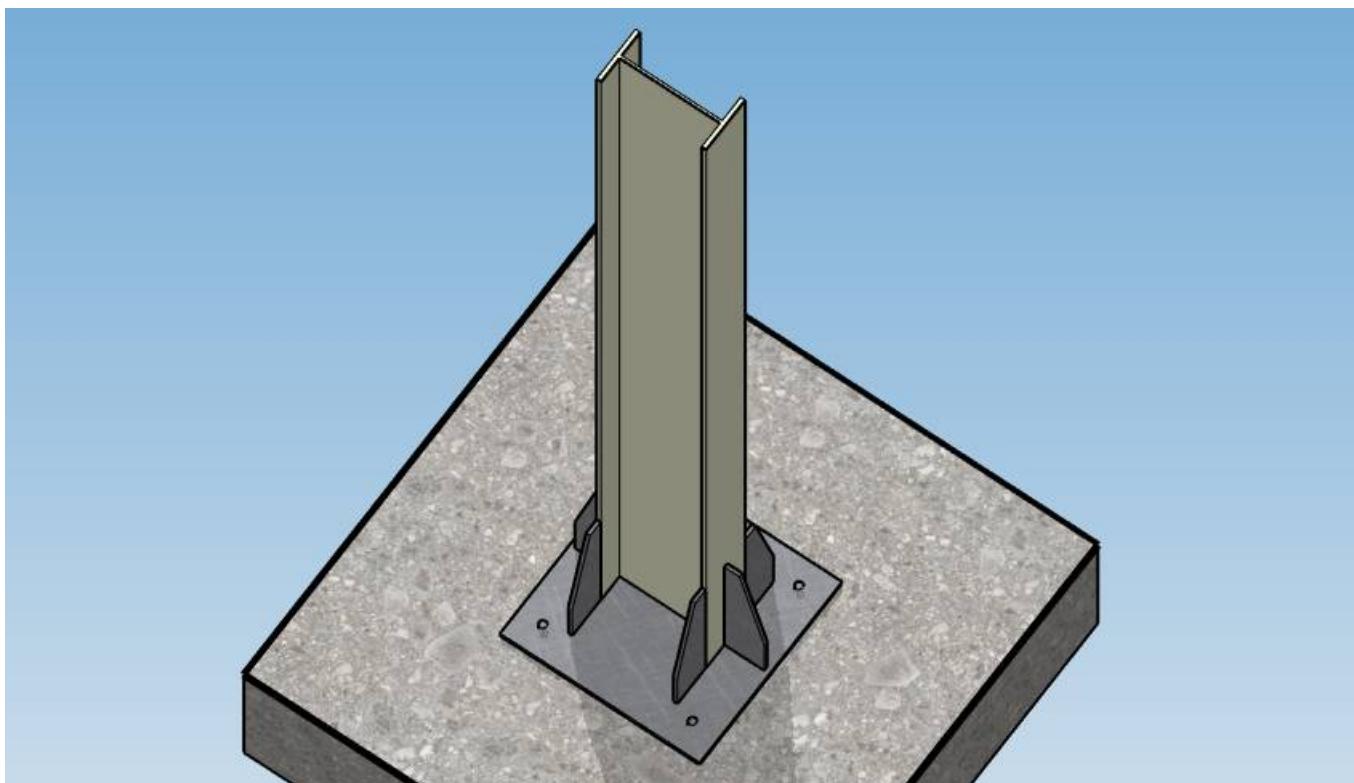
The button is only accessible for certain selected element.

The possible analysis are listed in the menu.

- [Anchor plate](#)

Anchor Plate Analysis

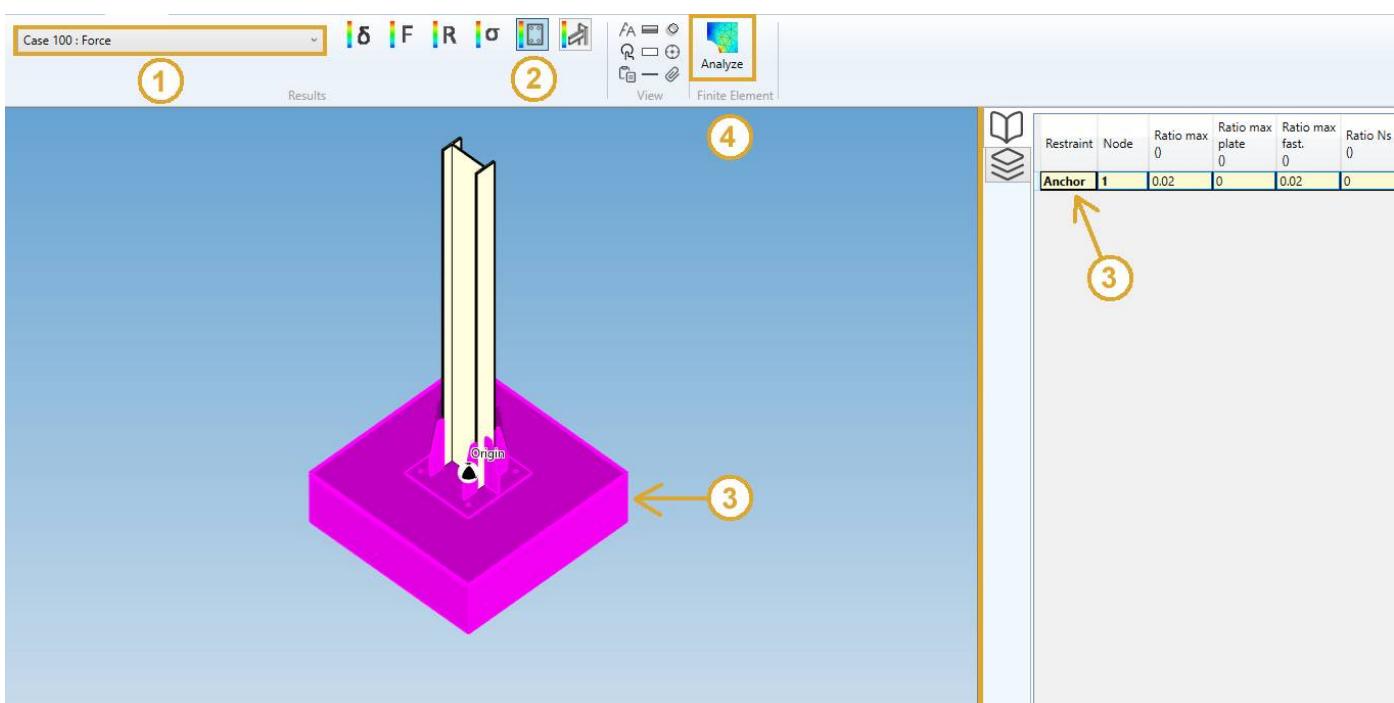
MetaStructure proposes a detailed analysis of anchor plate and its fasteners.



Click [here](#) for more information about the anchor plate definition.

1. Structure analysis

After structure analysis, the anchor plates of the structure can be examined.

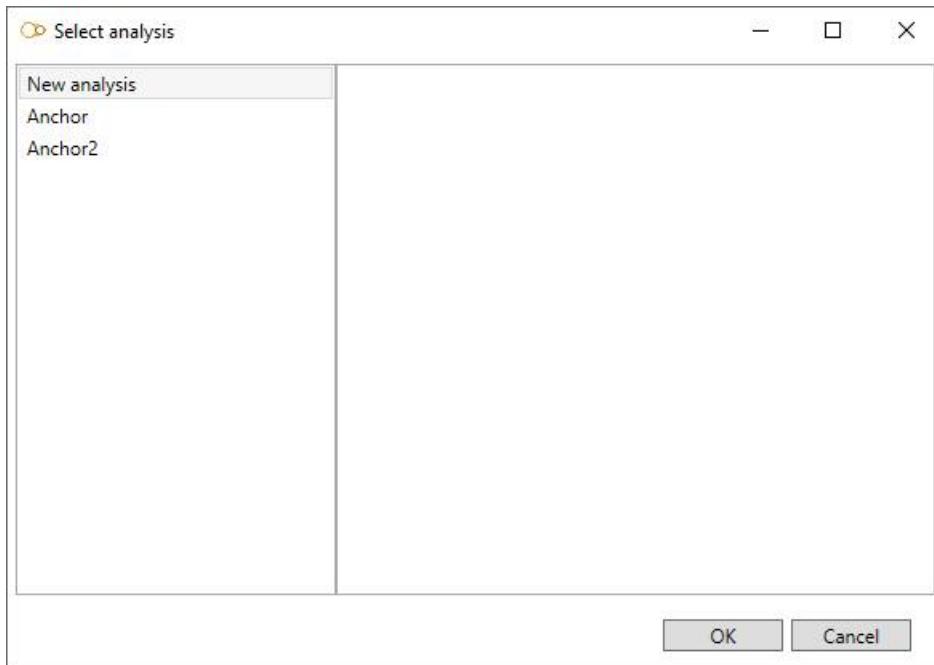


- Select a load case or mode (1).
- Select the **anchor plate** button (2).

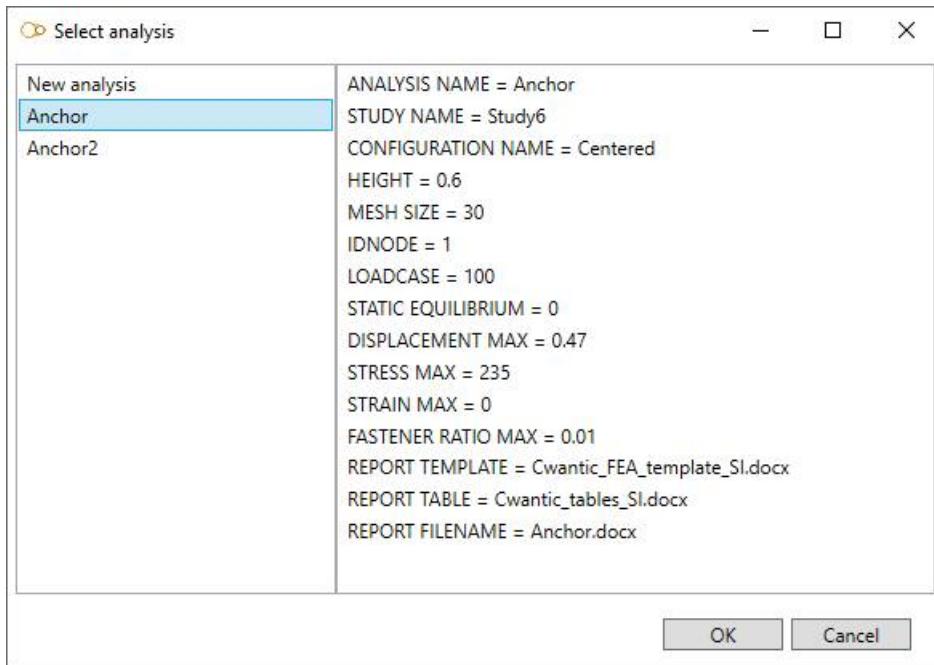
- Select either an anchor plate in the 3D space or in the results table (3).
- Click on the **Analysis** button (4).

The **selection mode** is automatically set to **Restraint** when clicking on **anchor plate** button.

If other analysis exists for the same **Restraint** and the same **Load**, a window will appear :



- Select **New analysis** to start a new analysis from scratch.
- Or select an existing analysis to reopen it :

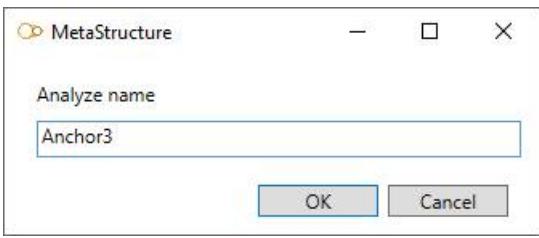


Some properties and results are shown.

Click OK.

2. New analysis

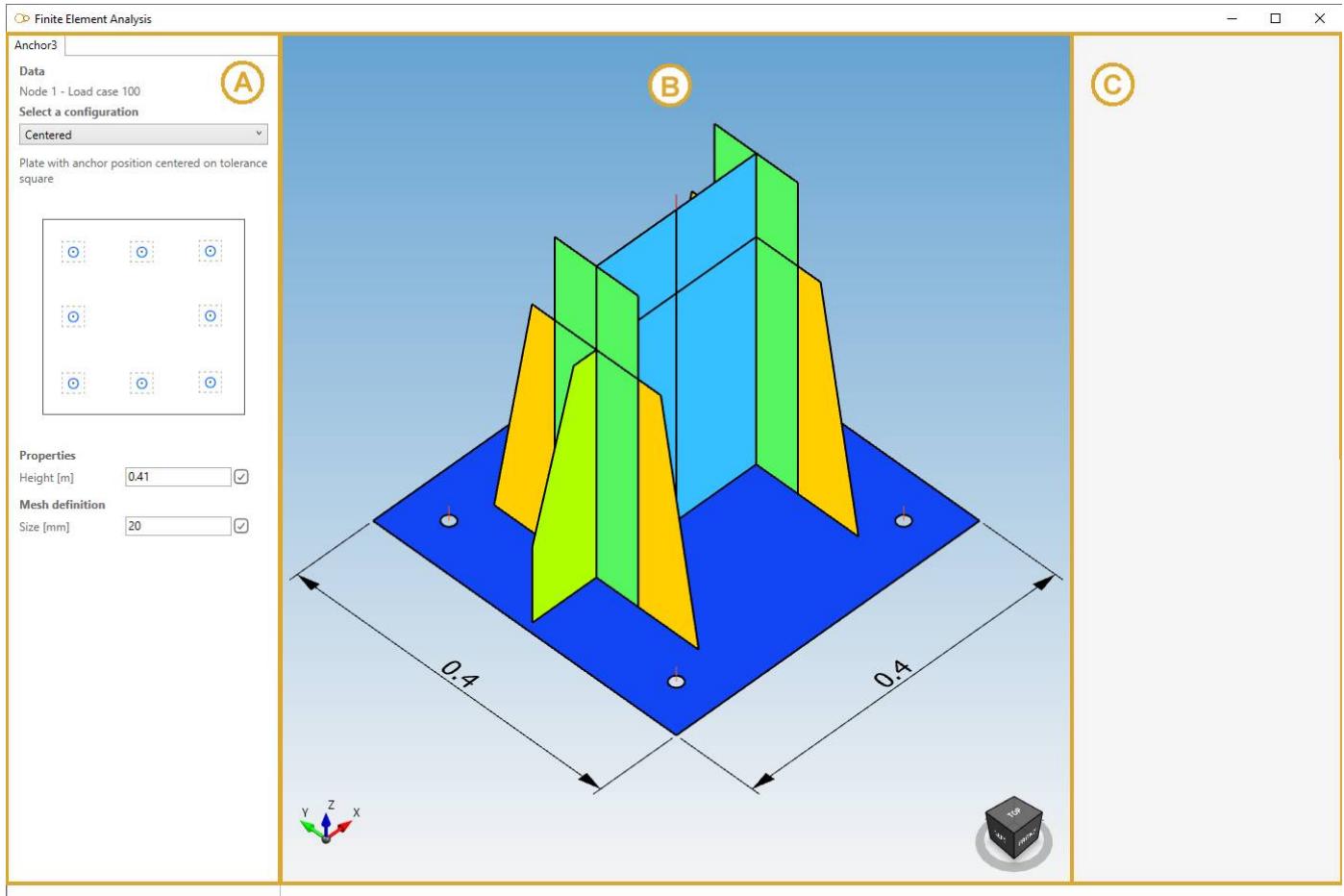
If you choose to create a **New analysis**, you have to define a name to the analysis (that doesn't already exist) :



The **Finite Element Analysis Window** appears :

MetaStructure automatically transforms the actual plate, anchor positions, stiffeners and a part of the beam to surfacing elements (on neutral fiber).

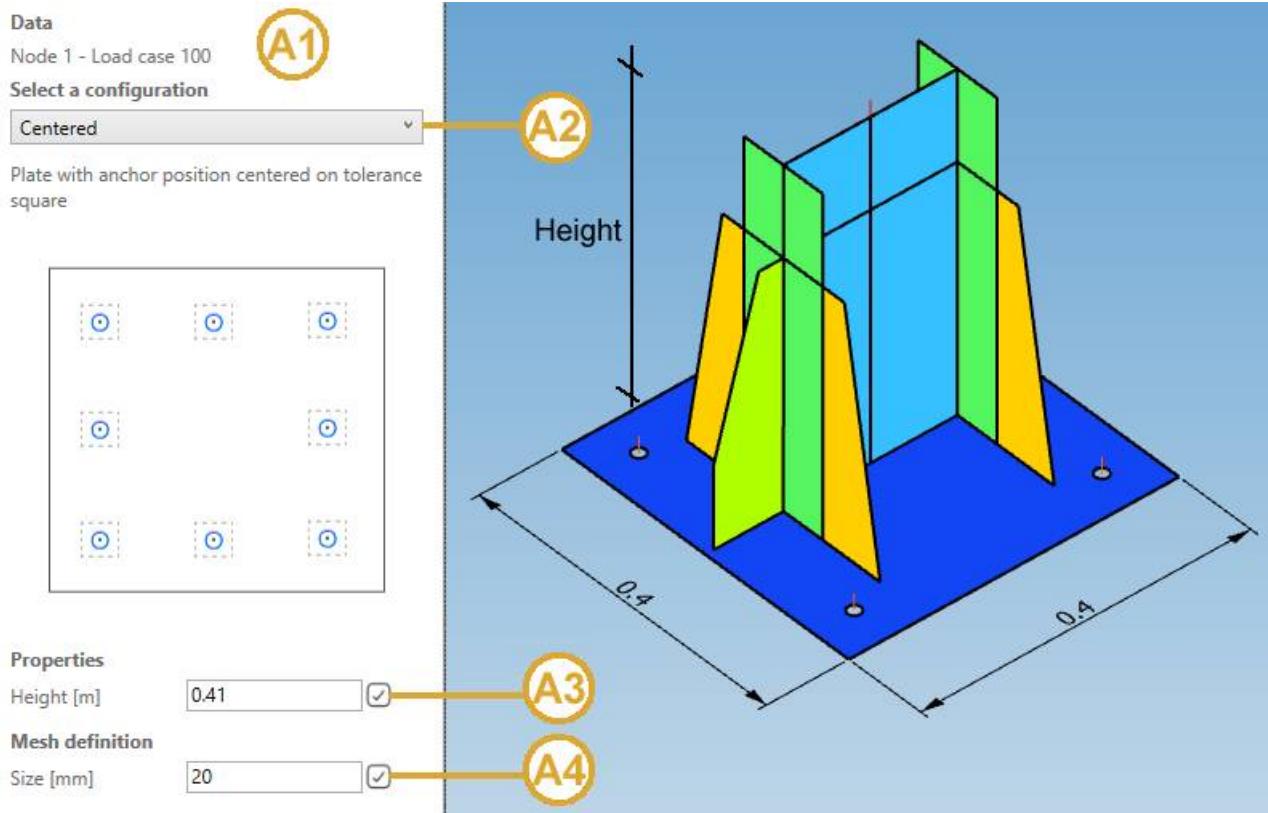
These surfaces are grouping by types, thicknesses and materials :



The window is divided into 3 areas :

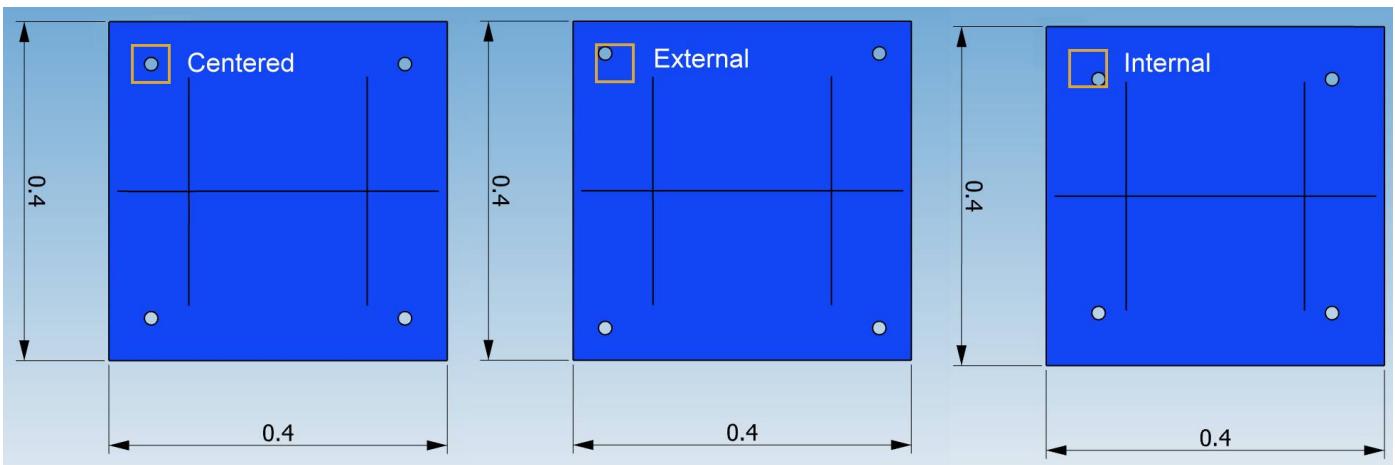
- A : Definition of the assembly, mesh, results and report
- B : Model 3D
- C : Groups of elements with same properties (type, thickness, material)

The first area contains 4 properties :



- A1 : The node of the anchor plate (restraint) and the current load
- A2 : The configuration
- A3 : The total height of the assembly. Click on **v** button to modify.
- A4 : The desired mesh size. Click on **v** button to **generate the meshing**.

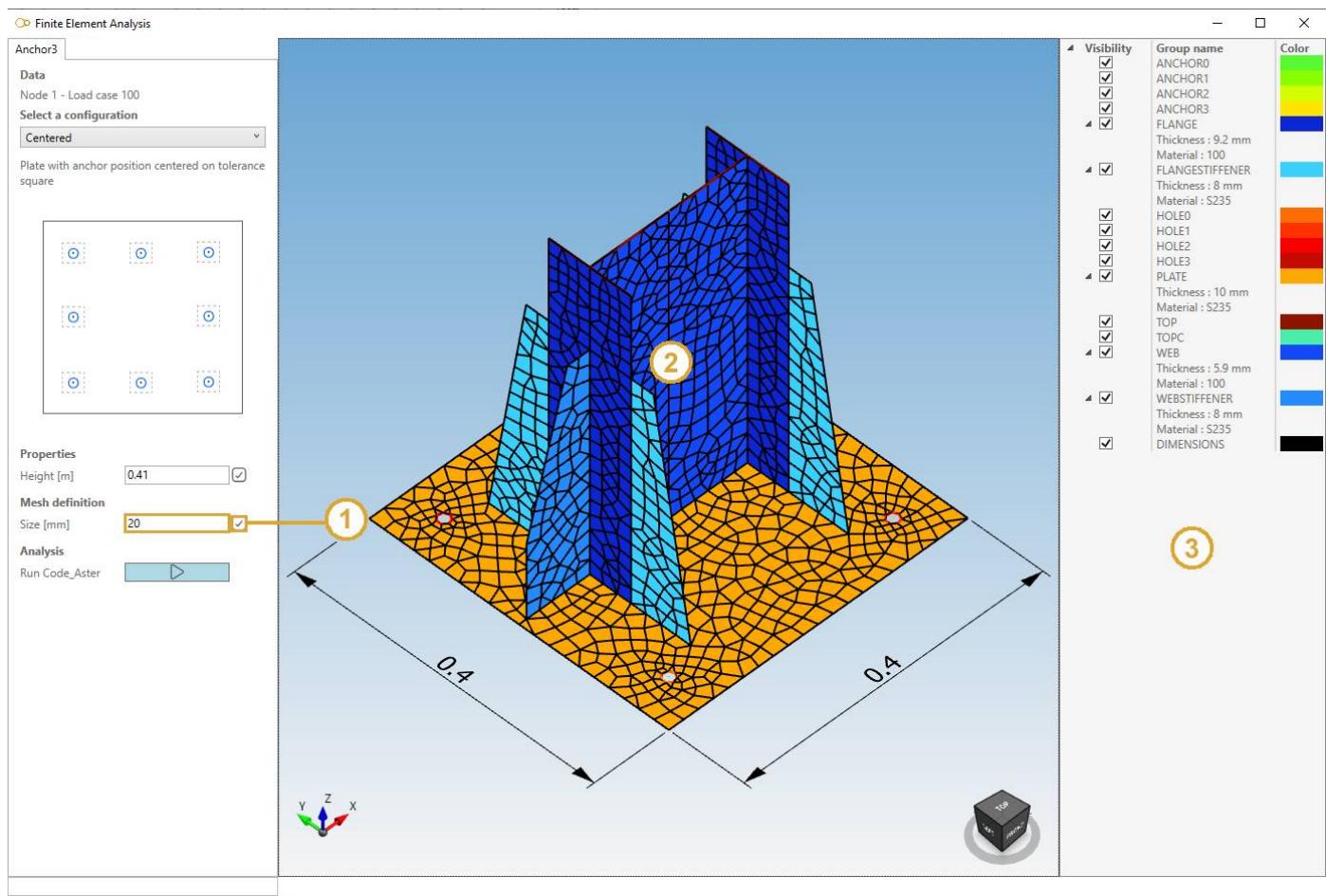
If the **anchor plate** has a non-zero value for the **Tolerance** property of its fasteners, 2 other configurations will be added in the list representing extreme positions of fasteners.



Click [here](#) for more information about the fastener definition.

3. Meshing

Choose a **mesh size** and click on **v** button (1) :



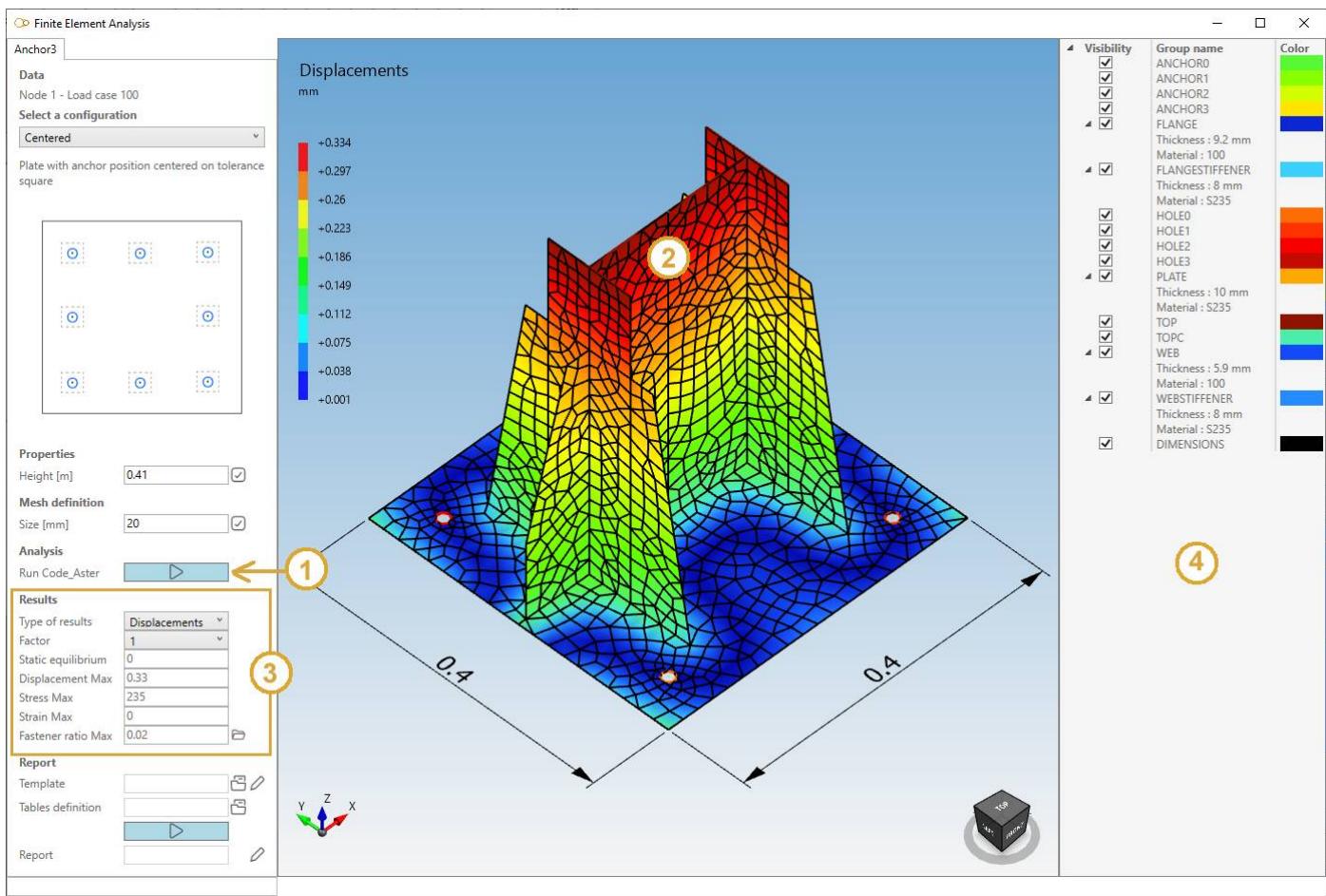
After several seconds, the assembly is totally meshed (2).

All groups appears on the right (3). You can show/hide each group for a better visualization.

The **Code_Aster** button is now available for a complete calculation.

4. Finite element analysis

Click on the **Code_Aster** button to launch a detailed calculation (1) :



Colored results appears (2) with a corresponding legend.

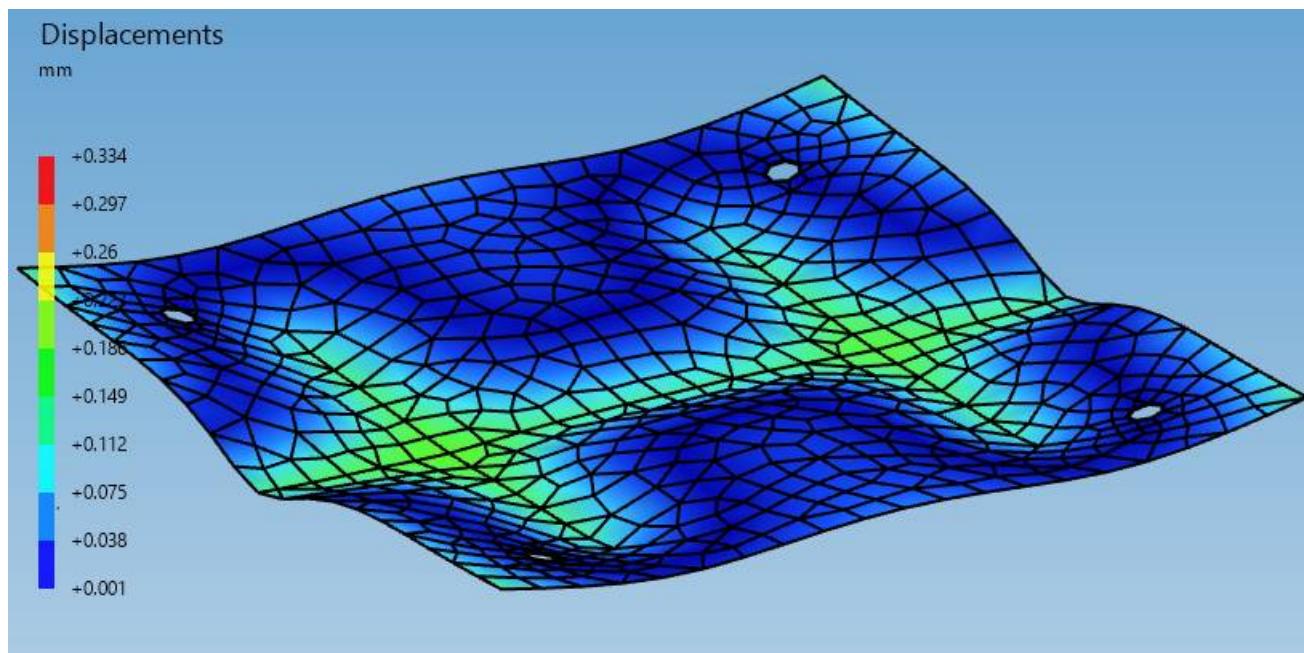
A result panel appears where the type of results can be choose and some informations are shown (3).

You can show/hide each group for a better visualization (4).

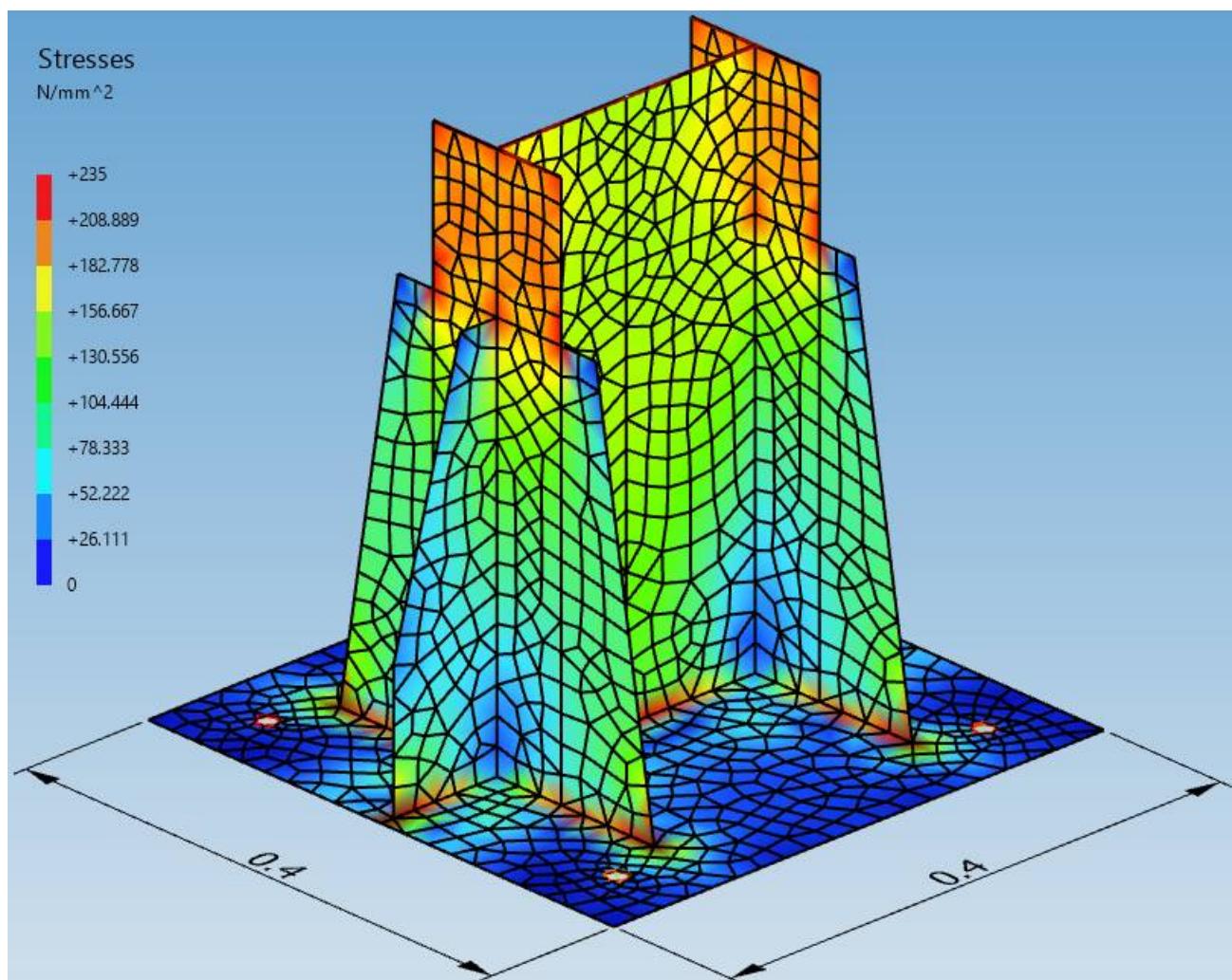
Type of results :

Property	Unit Metric	Unit USA	Remark
Groups	-	-	
Displacements	mm	in	Use Factor to amplify the deformation
Stresses	N/mm ²	ksi	
Strains	%	%	
Compressions	N/mm ²	ksi	
Iso-displacements	mm	in	Use Factor to amplify the deformation
Iso-stresses	N/mm ²	ksi	
Iso-Strains	%	%	
Iso-Compressions	N/mm ²	ksi	

Example of the deformation of the plate (with factor = 128) :



Change for example the type to **Stresses**:



Results

Type of results	Stresses
Factor	1
Static equilibrium	0
Displacement Max	0.33
Stress Max	235
Strain Max	0
Fastener ratio Max	0.02

The **Static equilibrium** is also evaluated (value near 0 reaches the perfect equilibrium).

Static Equilibrium refers to the physical state in which a system is at rest and the net force acting on it is null. It is a state in which all the forces acting on an object are balanced out and the object is not found to be in motion to the relative plane.

The **Fastener ratio max** indicates the max value of all ratios of all fasteners of the plate (must be < 1).

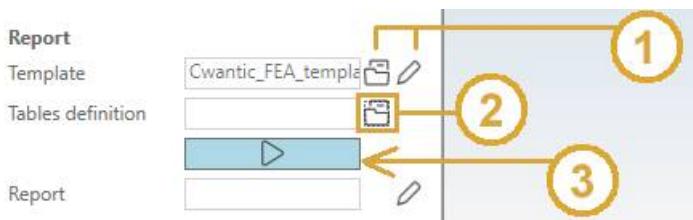
Click on the **Open** button to show a summary report of the **fasteners' results**:

C W A N T I C - M E T A P I P I N G Version 2024.0										Page 1			
CALCULATION CODE RCC-M June 2007 + EN-1992-4 2018													
MAX CONCRETE PRESSURE = 10.73 N/mm ²													
RESULTING TENSION = 1347 N (-16.6 mm, 2.8 mm)													
RESULTING COMPRESSION = 561347 N (-2.0 mm, 0.0 mm)													
FORCES ON FASTENERS													
No. Tension Shear-X Shear-Y													
(N) (N)													
1	372	-258	0										
2	289	-258	0										
3	376	-258	0										
4	310	-258	0										
STEEL FAILURE IN TENSION													
Fastener	alphaEq (N)	NRk,s gammaM,s	NRd,s	NEd	Ratio								
	3	125600	1.50	83733	376	0.004							
CONCRETE CONE FAILURE													
Fasteners	h'ef (mm)	ccr,N (mm)	Ac,NB (mm ²)	Ac,N	psis,N	psire,N	ec1,N (mm)	ec2,N	psiec,N	z	psim,N		
	1,2,3,4	100.0	150.0	90000	360000	1.00	1.00	16.6	2.8	0.88	14.8		
											1.00		
											10.10		
											45169		
											159727		
											1.50		
											106485		
											1347		
											0.013		
STEEL FAILURE IN SHEAR													
Fastener	VRk,s0 (N)	k7	alphaGap	alphaEq	VRk,s gammaM,s	VRd,s	VED	Ratio					
	1	120000	1.00		120000	1.25	96480	250	0.003				
COMBINED STEEL FAILURE													
Fastener	NRd,s (N)	NEd	VRd,s	VED	Ratio								
	3	83733	376	96480	250	0.000							
PRYOUT FAILURE													
Fasteners	h'ef (mm)	ccr,N (mm)	Ac,NB (mm ²)	Ac,N	psig,N	psis,N	psire,N	e,V (mm)	psiec,N	NRk,c (N)	NRk,c		
	1,2,3,4	100.0	150.0	90000	360000	1.00	1.00	0.0	1.00	45169	180674		
											2.00		
											361349		
											1.50		
											240899		
											1000		
											0.004		
CONCRETE EDGE FAILURE -x													
Fasteners	c1 (mm)	l1	dnom	k9	alpha	beta	Ac,V0 (mm ²)	Ac,V	psis,V	psih,V	e,V (mm)		
	1,3	233.3	100.0	24.0	2.40	0.07	0.06	245000	300000	1.00	1.08		
											0.0		
											1.00		
											1.00		
											63089		
											83441		
											1.50		
											55627		
											1000		
											0.018		
COMBINED CONCRETE FAILURE													
RatioNc	RatioVc	RatioNvc											
	0.013	0.018	0.004										
SHORT TERM LOAD DISPLACEMENTS : deltaN = 0.00 mm, deltaV = 0.00 mm													
LONG TERM LOAD DISPLACEMENTS : deltaN = 0.00 mm, deltaV = 0.00 mm													

Each fastener is deeply examined according to the current **standard**.

5. Report

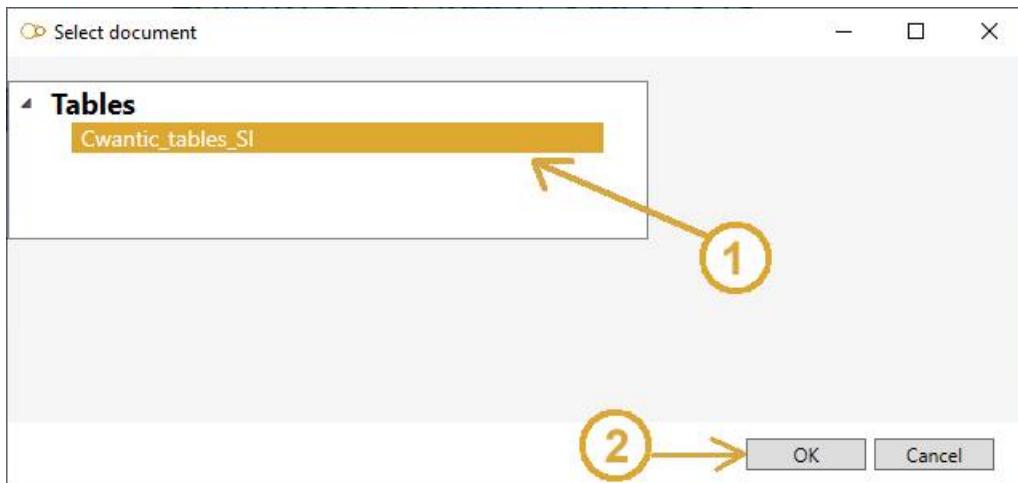
A report can be generated based on a template and "tables" file.



- Select a template (**open** button) or edit template (**pencil** button) (1).
- Select a **tables** file (**open** button) (2).
- Click on the **Report** button to generate the report (3).

The template is copy from the settings to the analysis' directory and can be locally modified before report generation (requires Microsoft Word).

The **select document window** (example for table) :



- Select the document (1)
- Click OK (2)

After generation, the report receive the name of the analysis.

You can **edit** the final report by clicking on the **pencil** button (requires Microsoft Word) (1) :

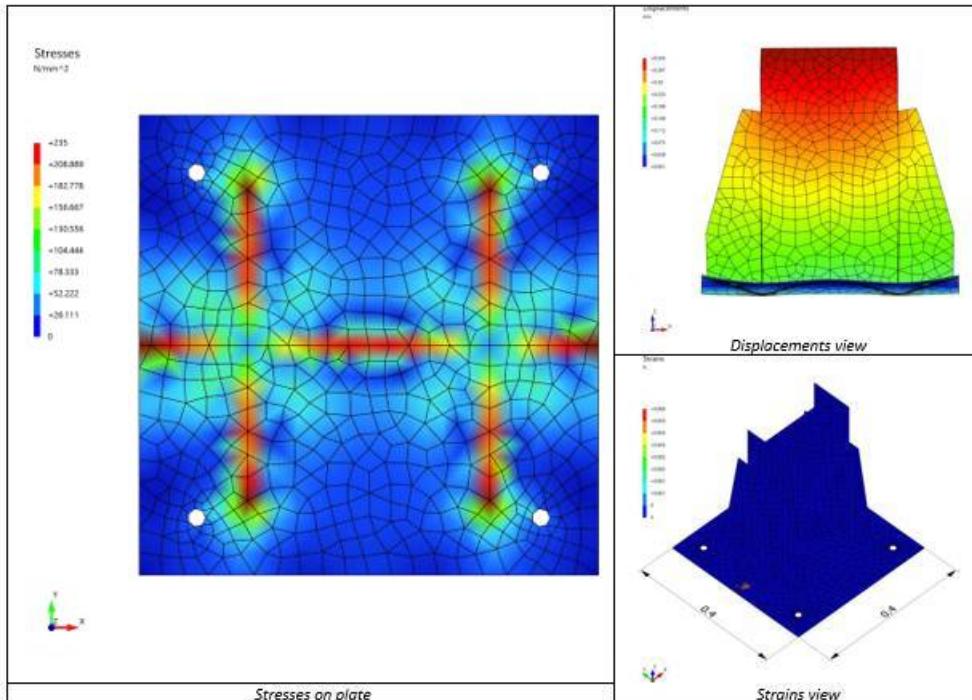


Finite Element Analysis

Anchor plate with fasteners

Analyse : Anchor3

Configuration: Centered
Node: 1
Load: 100
Height: 0.41 m
Mesh size: 20 mm



Static equilibrium: 0 %
Displacement max: 0.33 mm
Stress max: 235 MPa
Strain max: 0 %
Fastener ratio max: 0.02

Click [here](#) to have more information about the reporting mechanism.

5.1 Keywords

The keyword is useful to make a correspondence between the **template** and the **table** document but with specific decorators :

```
$$keyword$$ for the template  
[keyword] for the table
```

Keyword	Description	Remark
STUDY NAME	The name of the current study	No table
ANALYSIS NAME	The name of the current analysis	No table
PICTURE	Take a picture (§5.2)	No table
MESH RESULTS	The list of all mesh results	table [MESH RESULTS]
CONFIGURATION NAME	The name of the current configuration	No table
HEIGHT	The height of the current assembly	No table
MESH SIZE	The meshing size	No table
IDNODE	The name of the current node	No table
LOADCASE	The current load case	No table
STATIC EQUILIBRIUM	The static equilibrium value	No table
DISPLACEMENT MAX	The displacement max	No table
STRESS MAX	The stress max	No table
STRAIN MAX	The strain max	No table
FASTENER RATIO MAX	The max ratio on all fasteners	No table

5.2 Picture

It is possible to include pictures in the report with use of the keyword **PICTURE**.

When the software encounters this keyword, it simply makes a screenshot of the 3D engine.

It is possible to change the kind of visualization via a **JSON structure** just after the keyword separated by a semicolon character :

```
$$PICTURE;{...}$$
```

JSON parameters :

Parameter	Description	Default value
Groups	An array of visible group name	Empty list = all groups will be visible
View	Orientation of the camera	35 (= FrontFaceTopLeft - see below)
ResultType	Result category	0 (= Group - see below)
Factor	Amplification factor of the displacements	1
Dim	1 = show dimensions	1

View values :

```

Front          = 0
Right         = 1
Rear          = 2
Left          = 3
Top           = 4
Bottom        = 5
Isometric     = 6
Dimetric      = 7
Trimetric     = 8
FrontFaceBottom = 9
FrontFaceRight = 10
FrontFaceTop   = 11
FrontFaceLeft  = 12
RightFaceBottom = 13
RightFaceRight = 14
RightFaceTop   = 15
RightFaceLeft  = 16
BackFaceBottom = 17
BackFaceRight  = 18
BackFaceTop    = 19
BackFaceLeft   = 20
LeftFaceBottom = 21
LeftFaceRight  = 22
LeftFaceTop    = 23
LeftFaceLeft   = 24
BottomFaceBottom = 25
BottomFaceRight = 26
BottomFaceTop   = 27
BottomFaceLeft  = 28
TopFaceBottom  = 29
TopFaceRight   = 30
TopFaceTop     = 31
TopFaceLeft    = 32
FrontFaceBottomLeft = 33
FrontFaceBottomRight = 34
FrontFaceTopLeft = 35
FrontFaceTopRight = 36
BackFaceBottomLeft = 37
BackFaceBottomRight = 38
BackFaceTopLeft = 39
BackFaceTopRight = 40

```

Result type values :

```

Group = 0
Displacement = 1
Stress = 2
Strain = 3
Compression = 4
IsoDisplacement = 5
IsoStress = 6
IsoStrain = 7
IsoCompression = 8

```

Example :

```
$$PICTURE;{"ResultType":8,"Groups":["PLATE"],"View":4,"Dim":0}$$
```

ResultType = 8 for an "Iso-compression" view

Groups = ["PLATE"] will show only the group plate

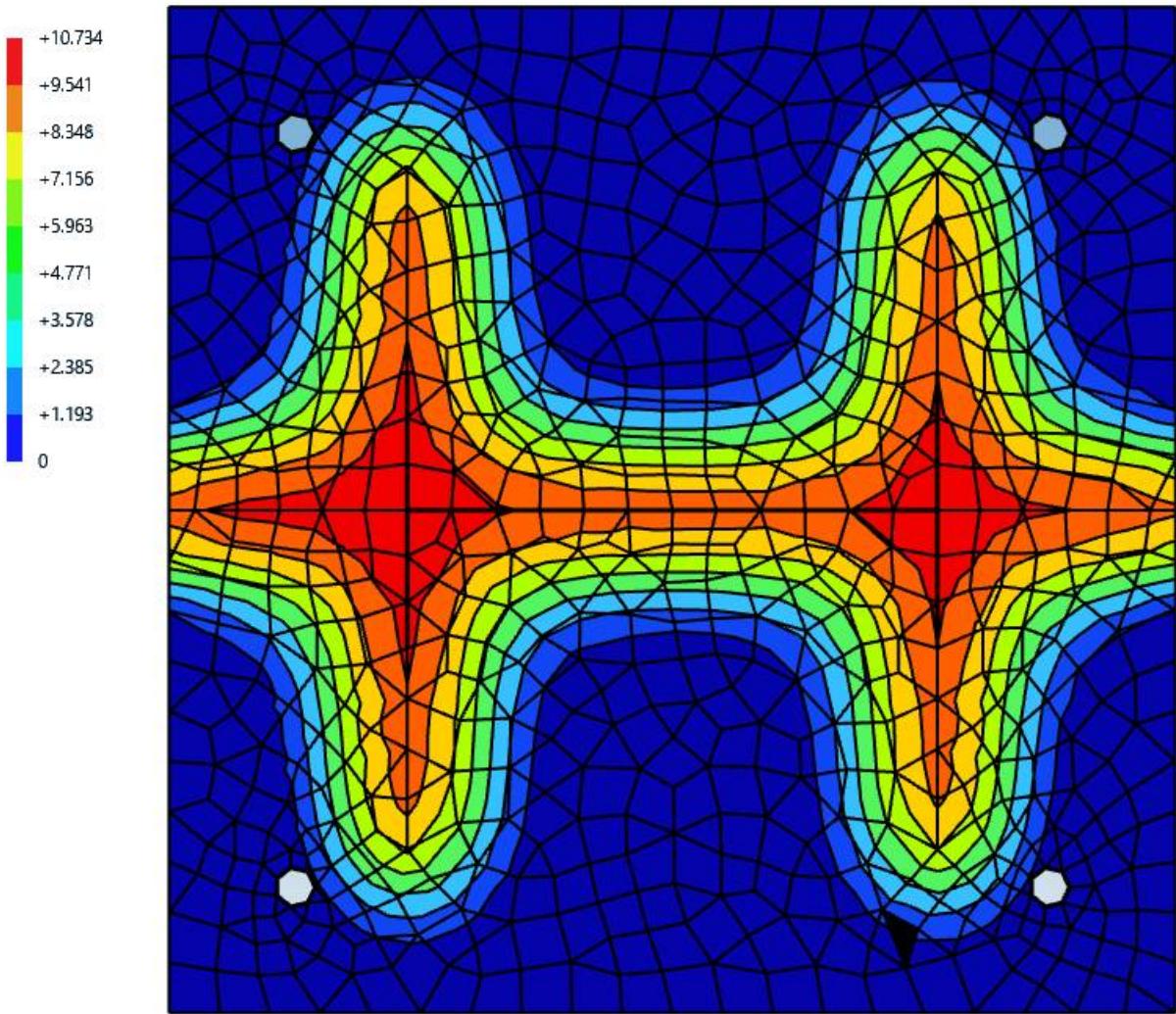
View = 4 for a top view

Dim = 0 will hide the dimensions

Result :

Compressions

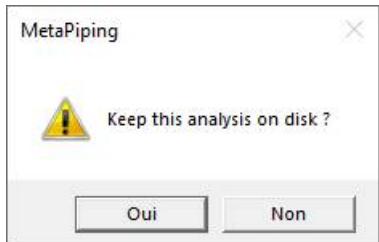
N/mm²



6. Conclusion

The analysis is terminated.

You can keep this analysis on disk by closing the window and answer **Yes** to the question :



This analysis will be proposed on the window of §1 for the same anchor plate (node) and load.



Solvers

MetaPiping comes with its built-in solver based on the finite element program Code_Aster from EDF. It can also connect to previously installed PIPESTRESS program.

1. Code_Aster (piping & structure)

Code_Aster offers a full range of multiphysical analysis and modeling methods that go well beyond the standard functions of a thermomechanical calculation code: from seismic analysis to porous media via acoustics, fatigue, stochastic dynamics, etc.

The following features are currently implemented in Code_Aster based solver :

- complex geometries made of straight pipes, elbows, miter bends, tees, reducers, valves, rigids, expansion joints, flanges, structural elements, socket welded fittings, beams, matrices and springs, lumped masses
- supports : anchors, translational and rotational restraints, multiple restraints, snubbers, variable and constant spring hangers
- static loads : dead weight, thermal expansion, anchor movements, nodal forces and moments, distributed forces, wind and snow, static acceleration, cold springing, variable fluid density and stratification
- load combinations
- automatic mass point generation
- modal extraction
- time history analysis with applied forces and moments
- floor response spectrum analysis

Supported codes :

- EN 13480-3 Editions 2017 and 2020 with Table H.3 (bidirectionnal SIFs)
- RCC-M Class 1 & 2 Editions 2002, 2007, 2012, 2016, 2017, 2018, 2020
- ASME Section III Class 1, 2 & 3 Editions 2004 + A06, 2007, 2010, 2013, 2015, 2017, 2019, 2021
- ANSI/ASME B31.1 Editions 1998, 2004, 2007, 2010, 2012, 2014, 2016, 2018, 2020 (with B31J SIFs)

2. PIPESTRESS (piping)

ONLY FOR USERS WITH A VALID PIPESTRESS LICENCE

PIPESTRESS is a pipe stress analysis software developed by DST Computer Services.

Supported codes :

- ASME Section III Classes 1, 2 and 3 (1967 to 2021)
- ANSI/ASME B31.1 (1972 to 2020)
- ANSI/ASME B31.3 (1999 to 2018)
- CODETI (1982 to 2016)
- RCC-M Classes 1 and 2 (1983 to 2020)
- RCC-MX (2008)
- KTA Classes 1 and 2 (1980 to 1992)
- EDF Piping Code for Composite Materials, Indice D
- EN 13480-3 Piping Code (2002 to 2017 - A4:2021)
- ASME HDPE Piping Code (2011 to 2017) includes Code Case N-7555 and Section III Appendix XXVI



Codes

Documentation will come soon...



Reporting

After designing the whole model, setting the loading, reviewing the data and analysing the study, click on the **REPORTING** button to generate a complete report.

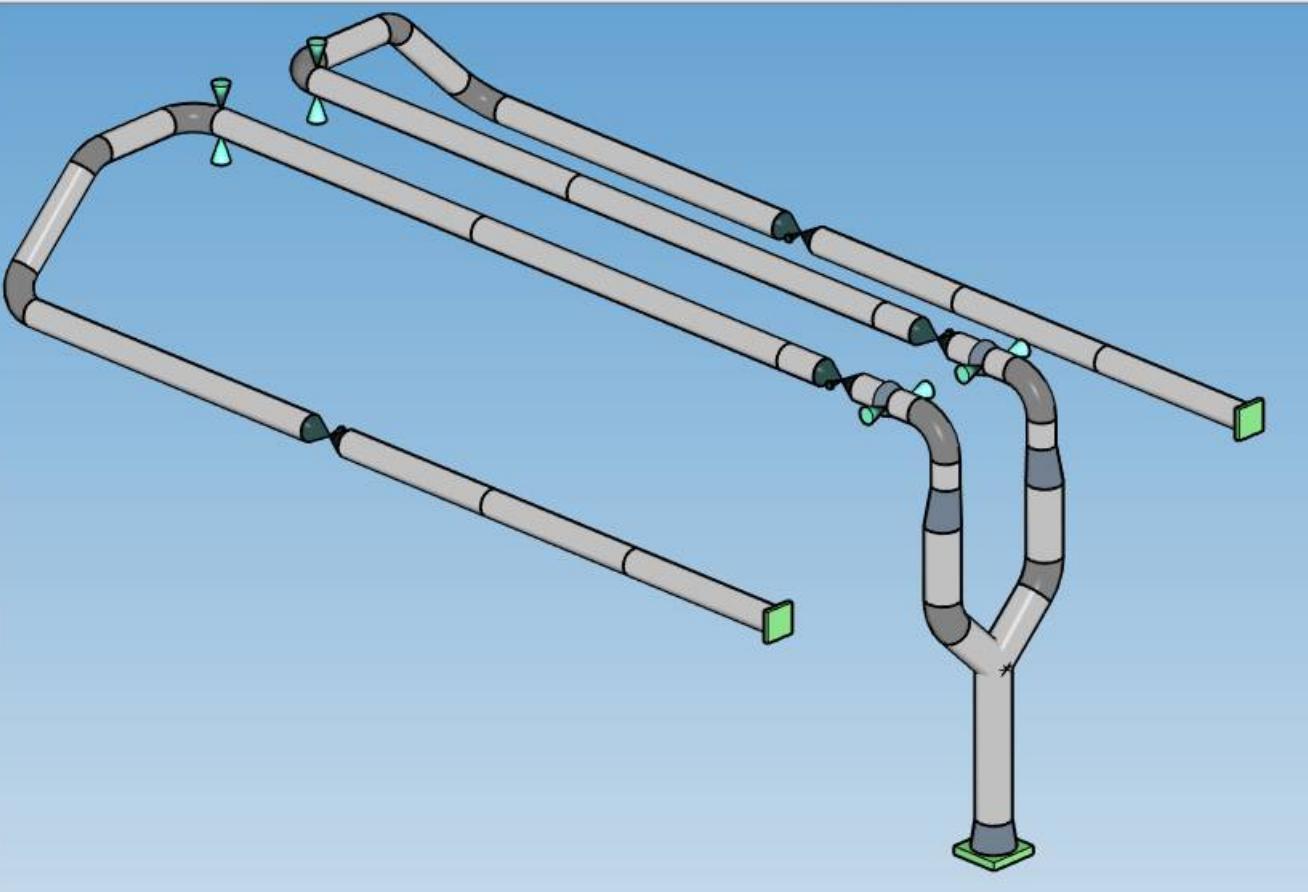
This explanation is available for MetaPiping and MetaStructure.

MetaPiping

Home Project Piping Loads Review Analysis **Reporting**

Template file : Report name :
Tables file : Edit template

Import templates Templates Report



1. Templates

The first step is to define the **template** of the report and the **table** document that exposes the content and the properties of each **keywords**.

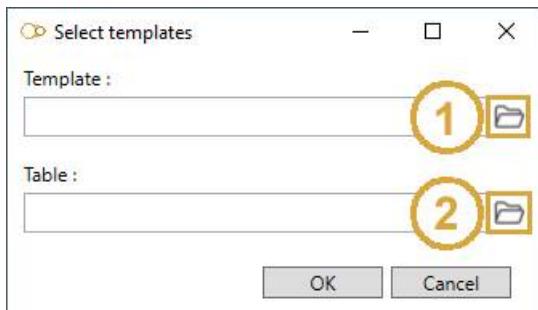
Import templates Template file :
Tables file : Edit template

Templates

1

- Click on the **Import templates** button (1)

Select from the settings, the **template** document and the **table** document :



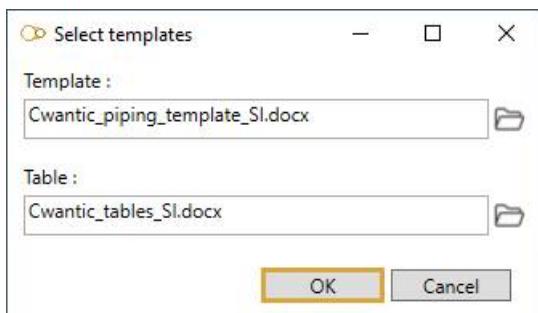
- Select a template document (1)
- Select a table document (2)

Click [here](#) to have more information about the reporting mechanism, the definition of a template and a table document.

The **select document window** (example for table) will appear :

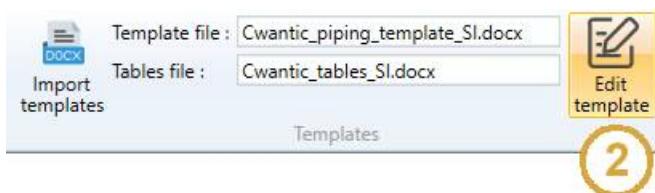


- Select the document (1)
- Click OK (2)



- Click OK to validate

The **template** document is copied to the study directory (the original will stay unchanged) and can now be modified to adapt some content to current model.



- Click the **Edit template** (2) to modify the local document (in Microsoft Word)

Enregistrement automatique Cwantic_piping_template_SI.docx - Word

Fichier Accueil Insertion Dessin Conception Mise en page Références Publipostage Révision Affichage Aide

Coller Presse-papiers Police Paragraphes Styles Édition Dicter Rédacteur Compléments

Report
Piping system analysis

Table of contents

1. Model definition	4
1.1 Materials.....	4
1.2 Sections.....	4
1.3 Nodes.....	4
1.4 Elements.....	4
1.5 Node local coordinate systems	4
1.6 Element local coordinate systems	4
1.7 Lumped masses	4
1.8 Supports.....	4
2. Loads	6
2.1 Forces and moments	6
2.2 Movements	6
2.3 Operating conditions	6
2.4 Distributed forces	6
2.5 Wind	6
2.6 Snow	6
2.7 Densities	6
2.8 Stratifications	6
2.9 Cold springs	6
2.10 Accelerations	6
2.11 Transients	7
2.12 Load sets	7
2.13 Static cases	7
2.14 Dynamic events	7
2.15 Dynamic cases	7
2.16 Spectra	7
2.17 Primary floor response	7
2.18 Secondary floor response	7
2.19 Combination cases	7
2.20 Stress combination cases	7
3. Results	8

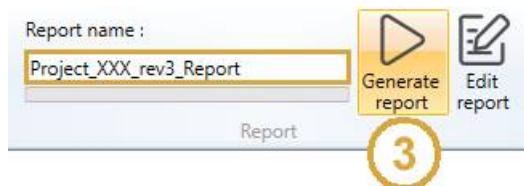
Page 1 sur 11 493 mots Anglais (États-Unis) Saisie semi-automatique : activée CWANTIC 03-09-24 Focus 2 03-09-24 CWANTIC 03-09-24 62%

WARNING

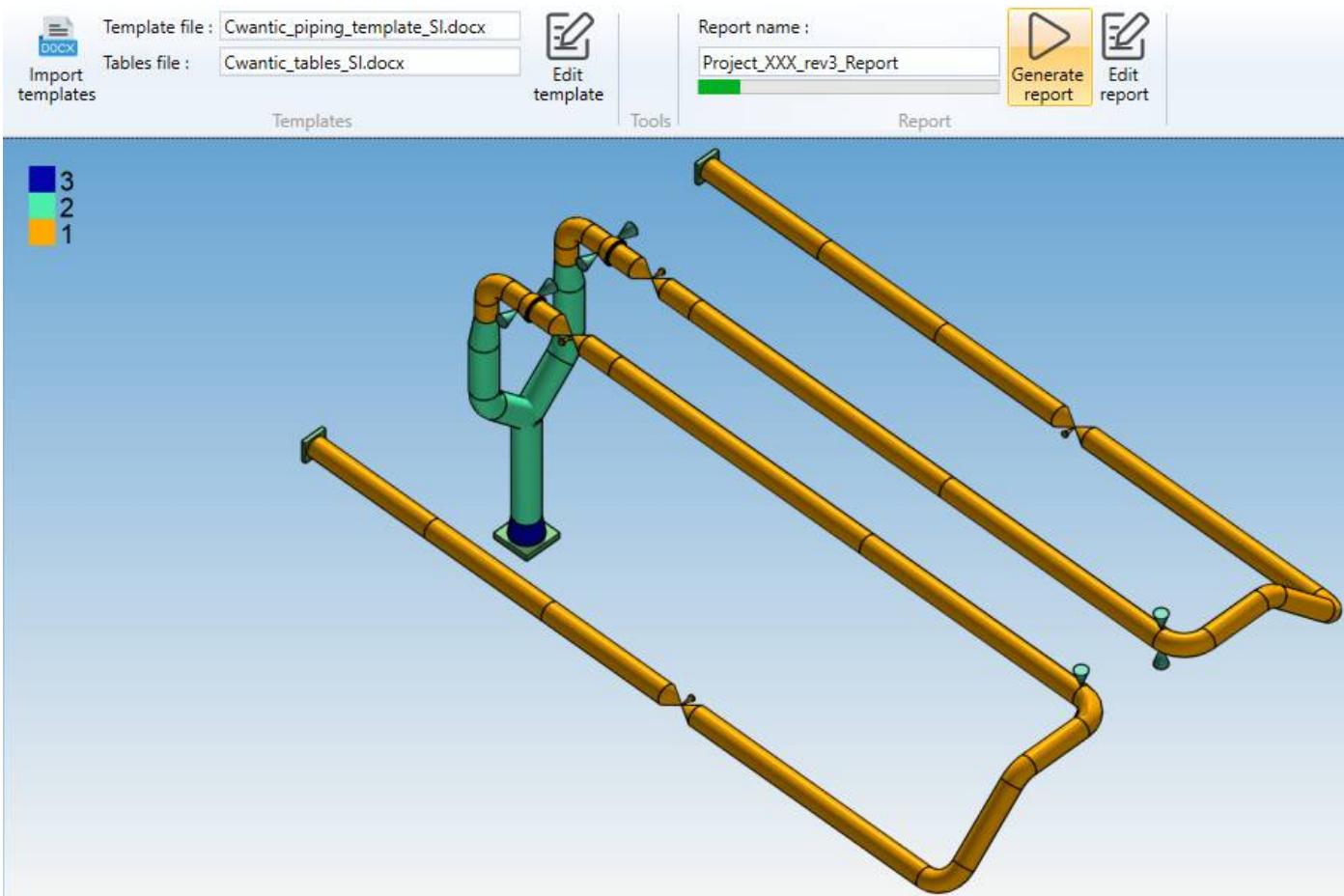
ATTENTION, don't forget to save (not save as) your modifications.

2. Report

Once the **template** defined and adapted for the current model, it is time to generate the report.

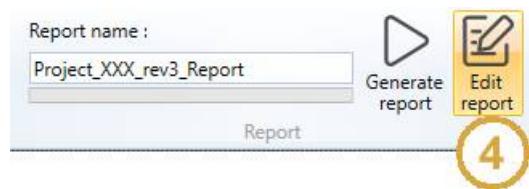


- Fill the report name cell
- Click the **Generate report** button (3)



A progress bar will display the progress of generation. The screen will change depending on the **picture** instructions.

You can finally edit the resulting **report**:



- Click the **Edit report** button (4) to modify the final report (in Microsoft Word)

Enregistrement automatique Project_XXX_rev3_Report.docx - Word

Fichier Accueil Insertion Dessin Conception Mise en page Références Publipostage Révision Affichage Aide

Presse-papiers Coller Police Paragraphes Styles Édition Dicter Rédacteur Compléments

Report
Piping system analysis

Table of contents

1.	Model definition	4
1.1	Materials.....	4
1.2	Sections.....	4
1.3	Nodes.....	4
1.4	Elements.....	4
1.5	Node local coordinate systems.....	4
1.6	Element local coordinate systems.....	4
1.7	Lumped masses	4
1.8	Supports.....	4
2.	Loads	6
2.1	Forces and moments	6
2.2	Movements.....	6
2.3	Operating conditions	6
2.4	Distributed forces	6
2.5	Wind	6
2.6	Snow	6
2.7	Densities	6
2.8	Stratifications	6
2.9	Cold springs	6
2.10	Accelerations	6
2.11	Transients	7
2.12	Load sets	7
2.13	Static cases	7
2.14	Dynamic events	7
2.15	Dynamic cases	7
2.16	Spectra	7
2.17	Primary floor response	7
2.18	Secondary floor response	7
2.19	Combination cases	7
2.20	Stress combination cases.....	7
3.	Results	8

CWANTIC 1 03-09-24 CWANTIC 2 03-09-24

Page 1 sur 45 9187 mots Français Saisie semi-automatique : activée Focus

WARNING

ATTENTION, don't forget to update the **table of contents** and to save (not save as) your modifications.

You can quit the study, the template and report are still in the directory.

WARNING

ATTENTION, if you make any modification in the model, the loads or the analysis, you have to re-generate the report. It will restart from the template document but the latest manual modifications on the final report will be lost.



TABLE OF CONTENTS

- [2024.1](#)
- [2024](#)
- [2023.2](#)
- [2023.1](#)
- [2023.0](#)



MetaPiping/MetaStructure 2024.1

Launch date

- October 2024

Main features

New codes/features in ASTER solver :

- ASME III Appendix XXVI (Polyethylene HDPE Piping)
- EDF Composite Piping (E.N.S.IN/91-025 indice D)
- Mixed-class and mixed-material piping

Databases :

- Rating database for post-processing purpose

Piping study :

- New post-processing brick (MetaPiping) for flange check according to ASME III Appendix XI, RCC-M Appendix Z and EN 13480-3 Appendix D
- New post-processing brick (MetaPiping) for support load combinations

Load :

- Time-history file generator

Analysis :

- Evaluation of cut-off frequency for time-history cases

Reporting

- Template and table (*.docx) definition in the settings
- Report generator with programmable pictures

Finite element analysis

- Detailed analysis of flexible anchor plates (MetaStructure) by means of plate and shell finite elements for various stiffener arrangements, taking into account non-linear plate/concrete contact and material plasticity.
- Report generation

Secondary features

Explorer

- Rename folder and project

Study

- Import *.Metal file (via file brick)

Design :

- Absolute coordinate of point
- Show/hide origin symbol and concrete
- Merge of bends
- Bend creation between 2 nodes
- New property "Section" that enable user to change the section of multiple piping elements
- Label on restraints and elements
- Selection based on node name or element label

Miscellaneous :

- Insert capabilities in some database's tables
- Graduated 3D wheel with 1° of precision
- New anchor representation if at least one spring constant released (disk instead of box)



MetaPiping 2024 (Current release)

Launch date

- March 2024

Main features

New application MetaStructure :

- Advanced structural/pipe supports/assemblies/anchor plates calculation - fully integrated to MetaPiping
- Beam section database
- Fastener and bolt database
- New study type (structure)
- Linked studies between piping and supports
- Beam analysis according to ASME III NF, RCC-M, EN 13480-3 and EN 1993-1-1
- Bolted joint analysis according to ASME III NF, RCC-M, EN 13480-3 and EN 1993-1-1
- Weld analysis according to ASME III NF, RCC-M, EN 13480-3 or EN 1993-1-1.
 - + The geometry of the weld beads is automatically determined according to the cross-section and angle between the assembled sections.
 - + Multiple weld bead configurations.
 - + Verification of weld stresses.
 - + The forces transmitted in the welds are recalculated if the connection is not right-angled.
 - + Verification of the LAMELLAR TEARING in the base metal.
- Anchor plate analysis according to EN 1992-4 Edition 2018
 - + Lescouarc'h method for calculating forces in fasteners (rigid plate)
 - + All regular configurations supported + possibility of modifying fastener positions to avoid concrete reinforcing bars
 - + No limit to the number of fasteners
 - + Concept of drilling tolerance
 - + Notion of virtual concrete edge to avoid interference between fastener influence zones
 - + Complete definition of concrete
 - + Free orientation of the anchor plate
 - + Consideration of profile eccentricity on the plate
 - + Consideration of assembly with spacing and grouting
 - + Consideration of configurations with spade to take up shear forces
 - + Realistic visualization of anchoring
- Parametric structure by python script
- Import database from BeamStress (materials, sections, fasteners)
- Import *.bst files

Non-linear static analysis :

- Soil definition (buried piping properties)
- Non-linear restraint

Secondary features

Explorer :

- Import/export of project

Design :

- Properties on elements : mounting temperature, design conditions, calculation code
- Move nodes tool
- Change length of element tool
- Cut bend in N elements

Review :

- Improvement of measuring tool

Database :

- Drag&Drop capabilities in hierarchy trees



MetaPiping 2023.2

Launch date

- October 2023

Main features

New codes in ASTER solver :

- RCC-M Class 1 Editions 2002, 2007, 2012, 2016, 2017, 2018, 2020
- ASME Section III Class 1 Editions 2015, 2017, 2019, 2021
- EN-13480 Editions 2017

Databases :

- Component databases for valves, flanges, bellows and sockets
- Support databases for rotational and translational restraints, snubbers, variable spring hangers and constant spring hangers
- Spectra databases

User manual :

- First version of the MetaPiping user manual

Python :

- New type of script : Simple script without virtual environment (faster execution)

Secondary features

General settings :

- New options in METAL settings for automatic node naming
- New options in PIPESTRESS PLUGIN settings for fre generation

Project management :

- The selected project shows a preview of all studies in 2 columns
- The selected study shows a preview + main informations + USER informations by a SIMPLE python script

Study diagram :

- Show an explanation of how to navigate in the lower left corner
- New comment brick with a text and a background color

Model options :

- New Class 1 options

Design :

- New save button
- New eccentric reducer capabilities : distance + angle values to define the tangent side
- New miter bend capabilities
- New stress intensification factor (SIF) capabilities on node and elements
- New stress report option capabilities (ON/OFF)
- New cut capabilities : pipe in N parts
- New view options : hide shadows and silhouettes

Pipestress plugin :

- New button to show advices/errors on fre file
- Delete temporary files (*.FOR) when calculation with PIPESTRESS solver
- Check if *.ppo is up to date with the result file

Loads :

- New thermal cases with transient definition (Class 1)
- New load set case (Class 1)

Review :

- New stress intensification factor (SIF) page
- New stress report option page

Analysis :

- Wireframe view of the results
- Show ASTER solver errors

Miscellaneous :

- Optimization of project loading : 8x faster
- For all treeview controls : memorization of the position when adding or deleting element
- For all treeview controls : navigation with the keyboard arrows
- F2 open/close the right tab in Design/Loads/Review/Analysis ribbon page
- ChatGPT uses now gpt-3.5-turbo model (released in March 2023)



MetaPiping 2023.1

Launch date

- May 2023

Main features

- Spec driven mode
- ASME Section III Class 2 & 3 Editions 2004 + A06, 2007, 2010, 2013, 2015, 2017, 2019, 2021
- ANSI/ASME B31.1 Editions 1998, 2004, 2007, 2010, 2012, 2014, 2016, 2018, 2020 (with B31J SIFs)
- POSTR brick in PIPESTRESS plugin
- PIPSYN import plugin
- PCF import plugin (beta)
- Copy data from review, load and analysis to clipboard
- Screenshot to clipboard
- Review improvements + distance tool
- Python script user manual



MetaPiping 2023.0

Launch date

- February 2023

Main features

- High quality graphical modeler dedicated to nuclear piping systems
- Powered by EDF's Code_Aster solver with RCC-M Class 2 and EN-13480 v2020 codes
- Integration of Python user scripts
- Data management
- Certified quality assurance
- Plugin architecture (NUPPIPE and PIPESTRESS)



Roadmap

CWANTIC has a very ambitious roadmap in order to deliver qualified professional solutions, written in the state of the art, using the latest computer technologies and open to Python developers.

MetaPiping 2024.1 (Current release)

[See the main features](#)

MetaPiping 2025

Expected date

- April 2025

Main features

MetaPiping :

- Code updates
- Finite element analysis of piping elements
- Pipe-support coupling (injection of *MetaStructure supports* into MetaPiping to take account of stiffness and mass)
- (RCC-MRx)
- Requests from the user community

MetaStructure :

- New assembly types
- Cable element for bracing modelling
- Requests from the user community

Disclaimer

The following is intended for informational purposes only, and may not be incorporated into any contract. No purchasing decisions should be made based on the following materials. CWANTIC is not committing to deliver any functionality, features or code. The development, timing and release of all products, functionality and features are at the sole discretion of CWANTIC, and are subject to change.

System Requirements



In order to exploit the full power of the software, we recommend the following configuration :

- **Windows** : 8.1 or upper
- **Processor** : Intel® Core I5 or upper
- **Memory** : 8 Gb or more
- **Hard disk** : 1 Gb of free space or more
- **Graphic card** : NVIDIA GeForce® GTX series or upper, AMD Radeon® RX series or upper
- **Screen resolution** : Full HD (1920x1080) or more
- **Mouse** : 3 buttons (scroll wheel button)