Version
February 2016

Programs

RF-PIPING

Add-on Modules for PIPELINES

Program Description

Dlubal

Am Zellweg 2
D-93464 Tiefenbach
Germany

Tel.: +49 9673 9203-0
Fax: +49 9673 9203-51
E-mail: info@dlubal.com
Web: www.dlubal.com

Contents

1 Introduction

1.1 RF-PIPING and RF-PIPING Design Add-On Modules

Pipelines are used to transport fluids. They are used in all areas of construction. For the design and calculation of pipelines, the two US standards ASME B31.1 [1] and ASME B31.3 [2] are ground-breaking. DIN EN 13480-3 [3] contains the appropriate guidelines for metallic industrial piping.

With the **RF-PIPING** add-on module, DLUBAL offers a powerful tool for modeling piping systems. There is no need to adjust to a new interface because the add-on module is displayed as an extension in the RFEM user interface when modeling pipelines.

You can model the pipelines like members in the RFEM user interface. It is possible to graphically enter connections such as tees and piping components using corresponding dialogs. Helpful libraries for pipelines, flanges and other components are included in the program.

You can enter loads and combine load cases using the usual procedures. You can decide if the load cases should be automatically or manually superimposed.

The calculation of internal forces, deformations and support forces corresponds to the general principle of RFEM. You can use the graphical RFEM user interface with all functions to evaluate the results.

With the second module, **RF-PIPING Design**, you can carry out piping analysis according to the previously defined standard ([1], [2] or [3] - in preparation). In the current state of development of the module, stress designs are carried out due to permanent, occasional or accidental loads.

You can clearly document all data, from modeling to design, in the global printout report of RFEM.

We hope you will enjoy working with RF-PIPING and RF-PIPING Design.

Your DLUBAL Team

1.2 Using the Manual

Since the topics installation, graphical user interface, results evaluation and printout are described in detail in the RFEM manual, no further description is given here. The present manual focuses on typical features of RF-PIPING and RF-PIPING Design.

The descriptions in this manual follow the sequence and structure of the module's input and result tables. In the text, the described **buttons** are given in square brackets, for example [Preset all members]. They are also displayed on the left. The **expressions** that appear in dialog boxes, tables, and menus are set in *italics* to clarify the explanations.

At the end of the manual, you find the index. In addition, feel free to use the search function on our blog website www.dlubal.com/blog/en where you can find tips and tricks in our articles about the add-on modules RF-PIPING.

2 RF-PIPING

In this chapter, you will find information about the different dialog boxes and functions of the add-on module RF-PIPING.

2.1 Opening RF-PIPING

RF-PIPING is an extension of RFEM. The functions of the piping module are available when **Piping analysis** in the *General Data* dialog box, *Options* tab is activated.

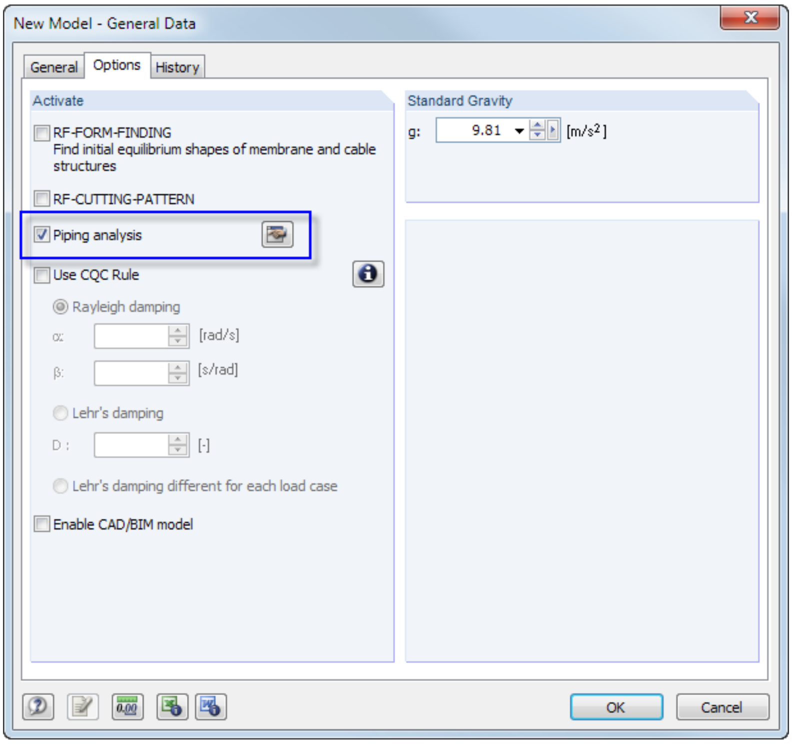
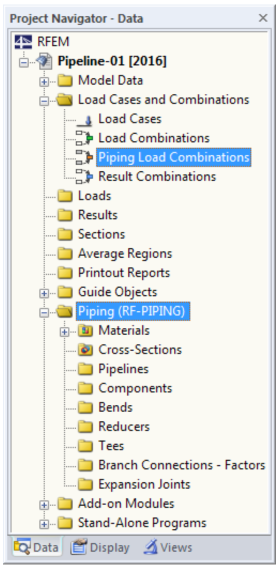


Figure 2.1: Dialog box *New Model - General Data*, *Options* tab



Data Navigator

After closing the dialog box, additional navigator items and tables as well as a new toolbar are available. They are described in Chapter 2.1.2.

Figure 2.2: Toolbar *Piping*

2.1.1 Piping Analysis - Settings

To get to the detail settings for RF-PIPING, use the [Edit] button (see Figure 2.1). A dialog box appears with important settings for the piping analysis.

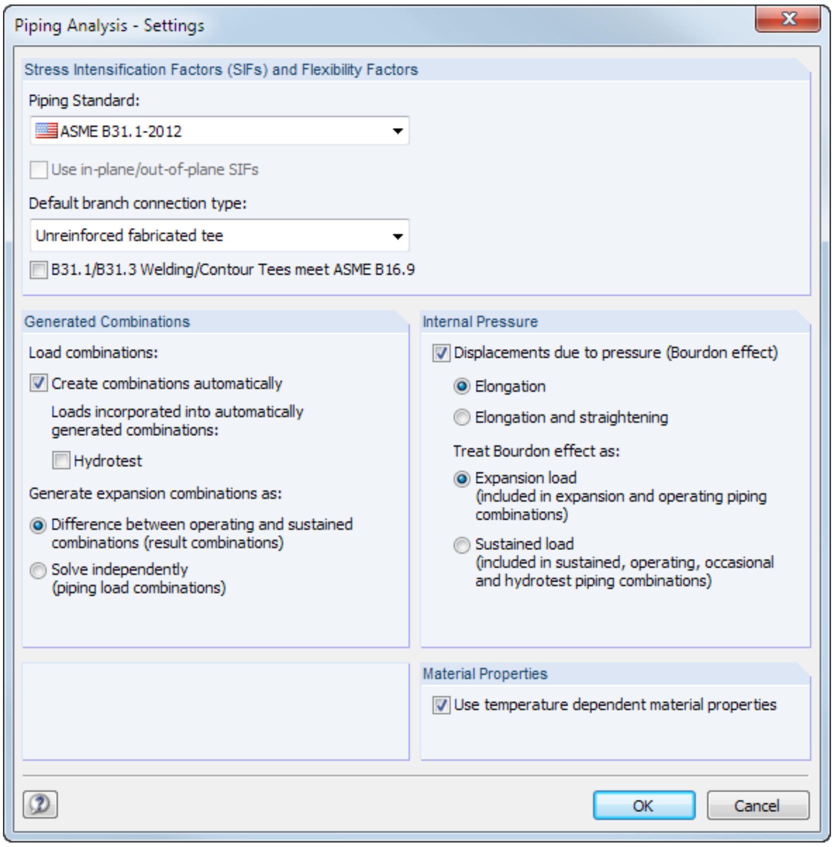
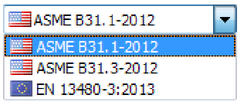


Figure 2.3: Dialog box *Piping Analysis - Settings*

Stress Intensification Factors (SIFs) and Flexibility Factors



The following standards are available in the list **Piping Standard**:

- ASME B31.1-2012 [1]
- ASME B31.3-2012 [2]
- EN 13480-3:2013 [3] (in preparation)

The option **Use in-plane/out-of-plane SIFs** is intended to be used for an analysis according to [3].

For a correct analysis in the module RF-PIPING Design, detailed data on the design of the tee is necessary. The list **Default branch connection type** contains the following options:

- Welding forged tee
- Reinforced fabricated tee with pad or saddle
- Unreinforced fabricated tee
- Extruded welding tee
- Welded-in contour insert
- Branch welded-on fitting (integrally reinforced)
- User-defined

The check box **B31.1/B31.3 Welding/Contour Tees meet ASME B16.9** determines if the parameters r_x and T_c are available for the branch connection types *Welding forged tee* and *Welded-in contour insert* (see Chapter 2.2.10, page 27). This option thus refers to the note (7) in [1] Table D-1 and Note (8) in [2] Table D300.

Generated Combinations

This dialog section manages how the load cases are combined.

Create Combinations Automatically

The superposition of load cases is carried out in *piping combinations*. If the check box is selected, these combinations are generated automatically. You can define other combinations individually here.

If the check box *Hydrotest* is selected, additional piping combinations for the hydrostatic test pressure are generated.

Expansion Combinations

The temperature combinations can be generated according to the following two methods:

- Difference between operating and sustained combination (result combinations)
- Solve independently (piping load combinations)

In the first case, result combinations are generated. The second case generates load combinations. The difference between the two possible combinations is described in the RFEM manual, Chapter 5.5 and 5.6.

It is generally recommended to use result combinations: Here, the thermal load cases are entered separately which interact with the other loads.

Internal Pressure

An internal pressure results in elongation and straightening of pipes. The check box *Displacements due to pressure (Bourdon effect)* and the two radio buttons below specify if and how this effect is considered.

Furthermore, if the Bourdon effect is considered, you have to specify if the internal pressure acts as *expansion load* or as *sustained load*.

Material Properties

This check box specifies if the temperature dependent material properties are used which are stored in the library (see Chapter 2.2.1.3, page 13). If the standard setting is deactivated, those values will be used which correspond to the reference temperature T_{ref} indicated in Table *7.1 - Materials*.

2.1.2 Extension for Toolbars, Navigator and Tables

RF-PIPING is displayed in the RFEM user interface as an extension in the toolbars, navigator and tables.

Toolbar

By activating the module RF-PIPING, the corresponding toolbar is displayed (see Figure 2.2, page 4). The functions contained therein enable the modeling of the pipeline.

Button	Description
	New Pipeline
	New Reducer
	New Valve
	New 3-way Valve
	New 4-way Valve
	New Bend
	New Flange
	New Blind Flange
	New Tee
	New Branch Connection - Factors

Table 2.1: Buttons of RF-PIPING

The buttons and the corresponding dialog boxes are described in Chapter 2.2.

Navigator

In the navigator, the category *Load Cases and Combinations* is extended by the entry **Piping Load Combinations**.

Moreover, the new category **Piping (RF-PIPING)** is added. All data concerning materials, cross-sections and pipelines are saved here.

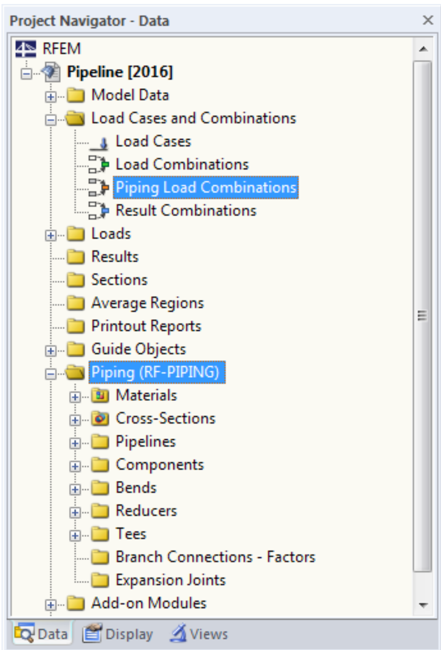
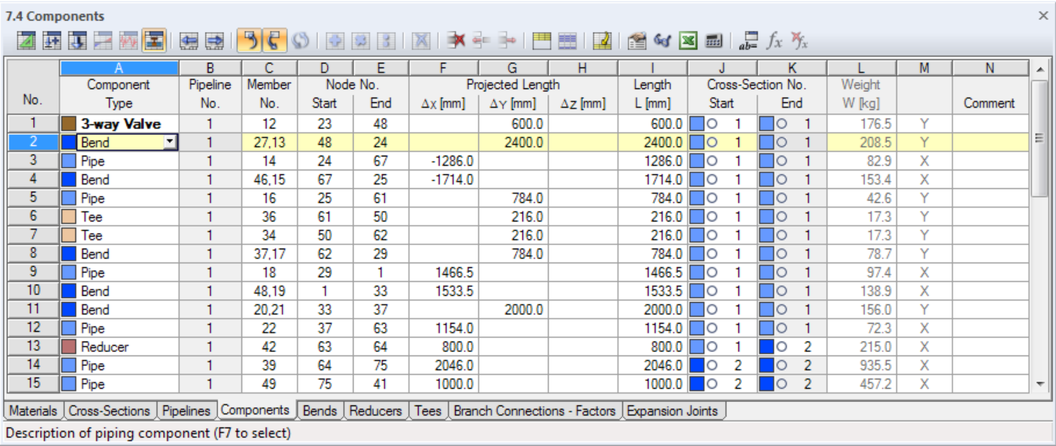


Figure 2.4: *Data Navigator* for RF-PIPING

Tables

All information about the pipelines are saved in the tables 7.1 to 7.10. To open these tables, use the button shown on the left.


7.4 Components

No.	A Component Type	B Pipeline No.	C Member No.	D Node No. Start	E Node No. End	F Projected Length Δx [mm]	G Projected Length ΔY [mm]	H Projected Length ΔZ [mm]	I Length L [mm]	J Cross-Section No. Start	K Cross-Section No. End	L Weight W [kg]	M	N Comment
1	3-way Valve	1	12	23	48		600.0		600.0	1	1	176.5	Y	
2	Bend	1	27,13	48	24		2400.0		2400.0	1	1	208.5	Y	
3	Pipe	1	14	24	67	-1286.0			1286.0	1	1	82.9	X	
4	Bend	1	46,15	67	25	-1714.0			1714.0	1	1	153.4	X	
5	Pipe	1	16	25	61		784.0		784.0	1	1	42.6	Y	
6	Tee	1	36	61	50		216.0		216.0	1	1	17.3	Y	
7	Tee	1	34	50	62		216.0		216.0	1	1	17.3	Y	
8	Bend	1	37,17	62	29		784.0		784.0	1	1	78.7	Y	
9	Pipe	1	18	29	1	1466.5			1466.5	1	1	97.4	X	
10	Bend	1	48,19	1	33	1533.5			1533.5	1	1	138.9	X	
11	Bend	1	20,21	33	37		2000.0		2000.0	1	1	156.0	Y	
12	Pipe	1	22	37	63	1154.0			1154.0	1	1	72.3	X	
13	Reducer	1	42	63	64	800.0			800.0	1	2	215.0	X	
14	Pipe	1	39	64	75	2046.0			2046.0	2	2	935.5	X	
15	Pipe	1	49	75	41	1000.0			1000.0	2	2	457.2	X	

Materials | Cross-Sections | Pipelines | Components | Bends | Reducers | Tees | Branch Connections - Factors | Expansion Joints

Description of piping component (F7 to select)

Figure 2.5: Tables 7.xx for RF-PIPING

The functioning of these tables is described in the RFEM manual, Chapter 11.5.

2.2 Dialog Boxes of RF-PIPING

2.2.1 Pipeline

A pipeline is a continuous string of members of the type *piping*. By clicking the button [New Pipeline], you can model new member strings. A dialog box appears where you can provide detailed information about the pipeline.

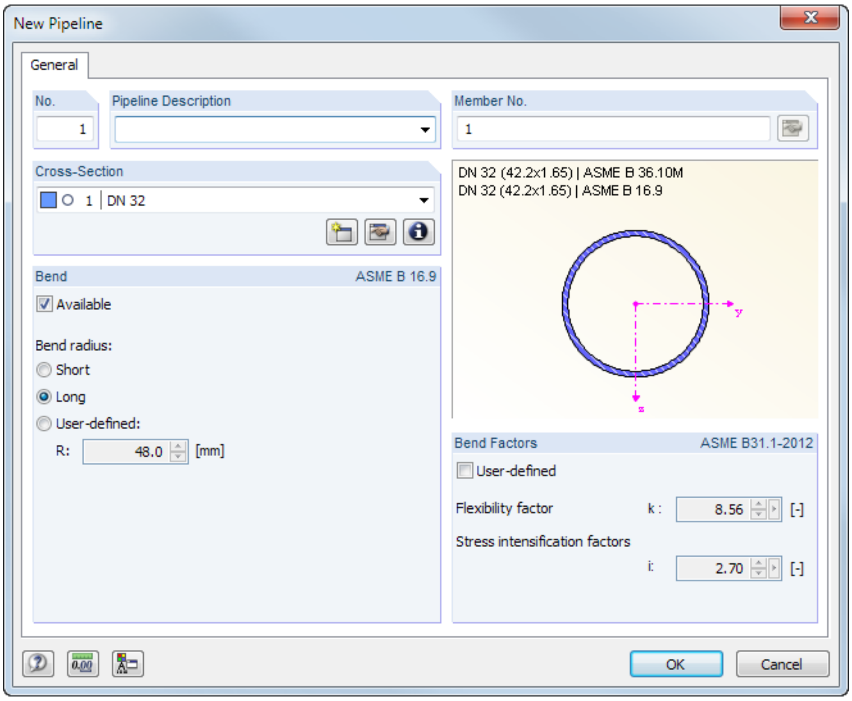


Figure 2.6: Dialog box *New Pipeline*

No.

You can assign any number. It is not possible to modify it afterwards.

Pipeline Description

You can enter any description for the pipeline.

Member No.

This number defines the first member of the new pipeline. You can freely define it. Other members of the same pipeline will have sequential numbers.

Cross-Section

You have to define the cross-section of the pipeline in the text box. By using the button [button], you can select a cross-section from the pipeline library (see Chapter 2.2.1.2, page 11).

For general information about the cross-sections, have a look at the RFEM manual, Chapter 4.13.

Bend

In this dialog section, you can define if the bends are generated automatically during the following modeling and which bend radius is considered.

Bend Factors

The values of the *Flexibility factor* as well as of the *Stress intensification factors* indicated in this dialog section are automatically calculated. However, it is also possible to define them manually.

The flexiblity factor k has an influence on the bending stiffness of the pipe bend. The stress intensification factors i only take effect at the piping design with RF-PIPING Design.

2.2.1.1 Defining Pipelines

If you have entered all data, you can close the dialog box by clicking [OK]. You can now define the pipeline as polyline in the work window. The following dialog box appears:

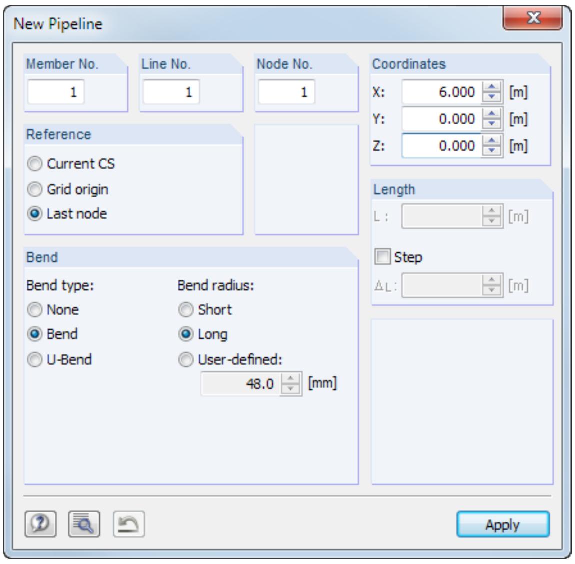
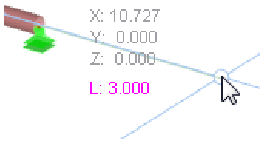


Figure 2.7: Dialog box *New Pipeline*

In the *New Pipeline* dialog box, the numbers of the new members, lines and nodes are preset, but it is possible to modify them.

For entering the coordinates, you have the following three options:

- *Current CS*: The coordinates always refer to the origin of the coordinate system. The grid is fixed.
- *Grid origin*: The coordinates refer to the origin of the grid.
- *Last node*: The coordinates always refer to the last defined node. The position of the grid origin is also in the last node.



The *Bend* dialog section controls if bends are generated when gradually defining the polyline and which form they have. The parameters are explained on the previous page.

If you select the check box *Step*, the length of the pipeline L is displayed at the cursor which may help to model the pipeline easier.

Buttons

Two buttons in this dialog box are useful for modeling.

Details

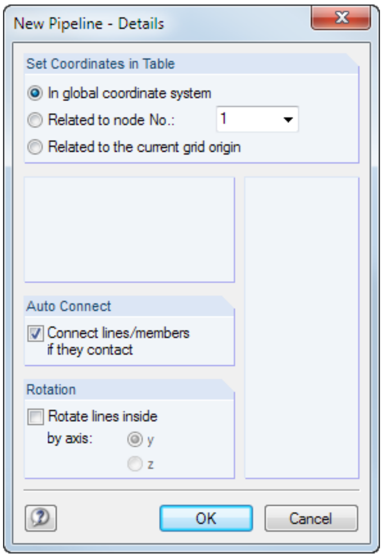


Figure 2.8: Dialog box *New Pipeline - Details*

Set Coordinates in Table: The input data is saved in the RFEM Table *1.1 Nodes*. It is possible to save the nodes in the global coordinate system or with reference to another node.

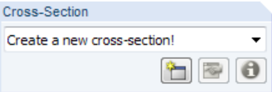
Auto Connect: This check box controls if a connection is generated between two pipelines, when, for example, the end point is placed on an existing pipeline. If this check box is selected, the existing pipeline in this node is separated; both pipelines are connected. If this option is deactivated, the existing pipeline will not be separated. Depending on whether the option *Use division for members with nodes lying on them* is activated in the *FE Mesh Settings* dialog box, the two pipelines are either connected or not.

Rotation: This option may be helpful for bends which look drilled in the rendering. The drilling is caused by a change of the local z-axis direction.

Undo

Modeling a pipeline is equal to modeling a continuous line. From the very start to the very end, all parts are continuously placed. If you have made any mistakes, you can undo your entries step-by-step by using this button without deleting the pipe.

2.2.1.2 New Piping Cross-Section



In the dialog box *New Piping*, you have to define a cross-section (see Figure 2.6, page 8). You can use the button .

The cross-section library is displayed where you can select the piping cross-section. This dialog box is described in the RFEM manual, Chapter 4.13.

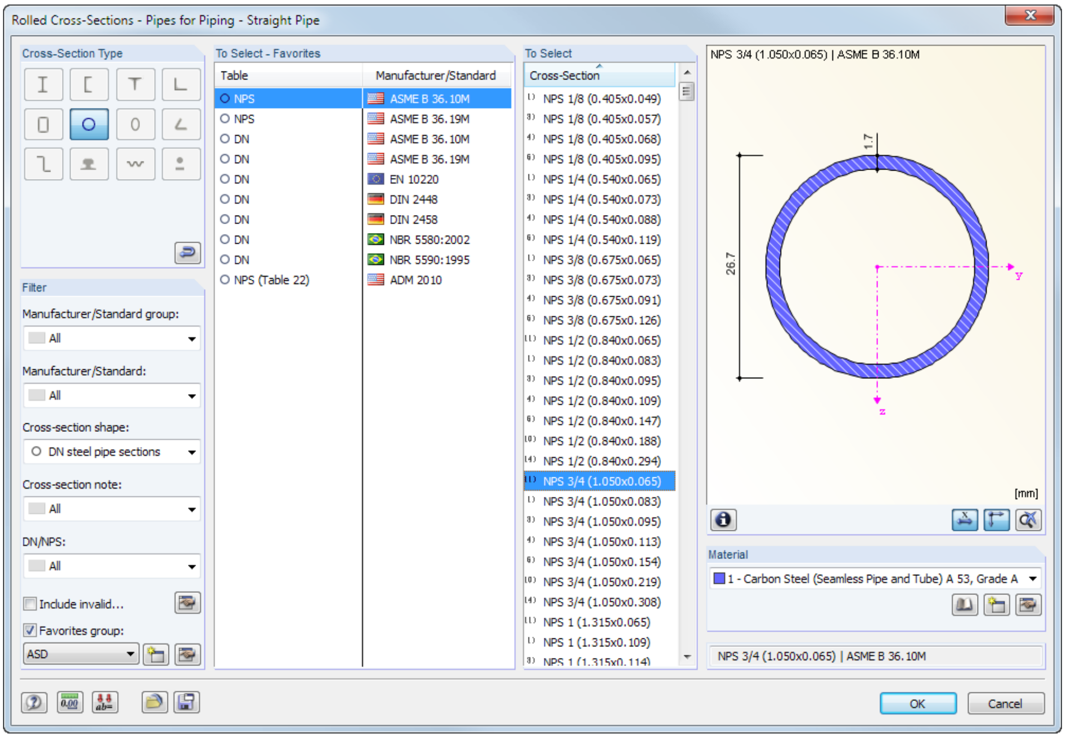


Figure 2.9: Dialog box *Rolled Cross-Sections - Pipes for Piping - Straight Pipe*

After clicking [OK], the *New Piping Cross-Section* dialog box is displayed (see Figure 2.10). In this dialog box, you can provide specific information about the cross-section, the bend, the structure of the pipeline and the stress analysis.

In this dialog box, you can enter specifications separately for straight sections and bends of a pipeline.

Straight Pipe

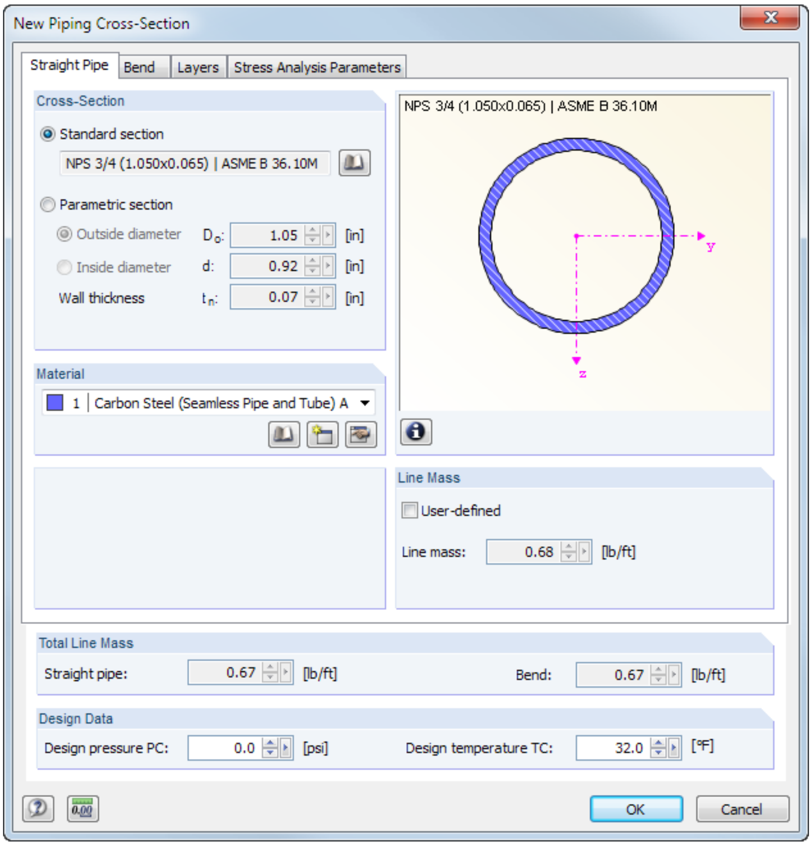
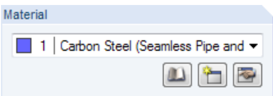


Figure 2.10: Dialog box *New Piping Cross-Section*

In the first tab, you can provide information about the cross-section of straight sections. You can select between a *Standard section* from the library or a *Parametric section* defined by outside and inside diameter with corresponding wall thickness.



You can select the *Material* of the cross-section from the list of already defined materials. With the buttons below the list, you can open the material library as well as create or edit a material. For more information about the material library, have a look at Chapter 2.2.1.3, page 13 and at the RFEM manual, Chapter 4.3.

The *Line Mass* is automatically calculated by default from the cross-section and the material. However, you can define it manually.

In the dialog section below, the *Total Line Mass* of the straight pipe sections and bends is displayed. It contains additional masses from the *Layers* tab.

In the last dialog section, you can define the *Design pressure PC* and the *Design temperature TC*.

If you use piping components such as flanges, the list of possibilities will be reduced automatically according to the design pressure to appropriate elements.

Bend

The setting options of this tab apply solely to bends of pipelines. The concept corresponds to the one for pipes (see above).

Layers

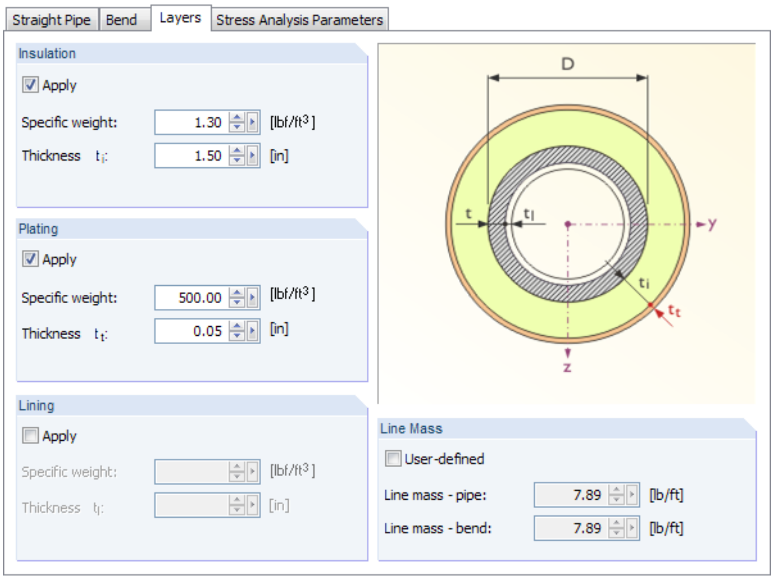


Figure 2.11: Tab *Layers*

In this tab, you can provide further information on the structure of the pipe. You can define the thickness of the layer as well as the specific weight each for *Insulation*, *Plating* and *Lining*.

If you enter the *Line Mass* manually, the values of the last calculated line masses are available so that individual values for pipe and bend can be entered.

2.2.1.3 Material Library

RFEM manages the materials for RF-PIPING separately in the category *Piping (RF-PIPING)*. You can access the material library as usual by using the navigator shortcut menu or the button [...] in Table *7.1 Materials*, Column A.

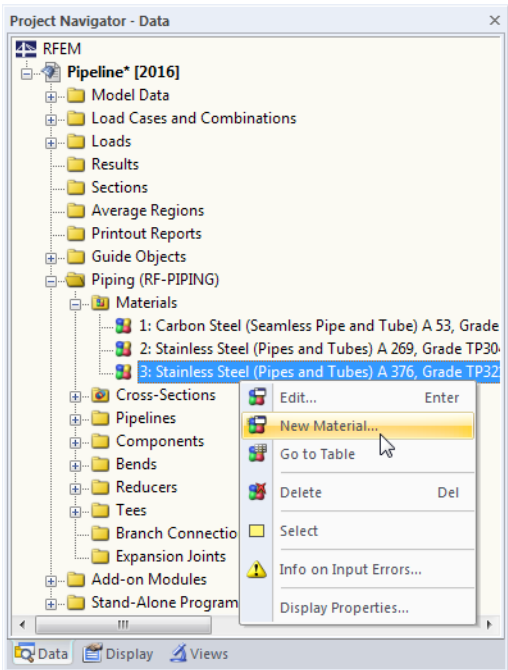


Figure 2.12: Shortcut menu *Materials*

In the library, the materials for the pipelines are preset by the *Filter* function.

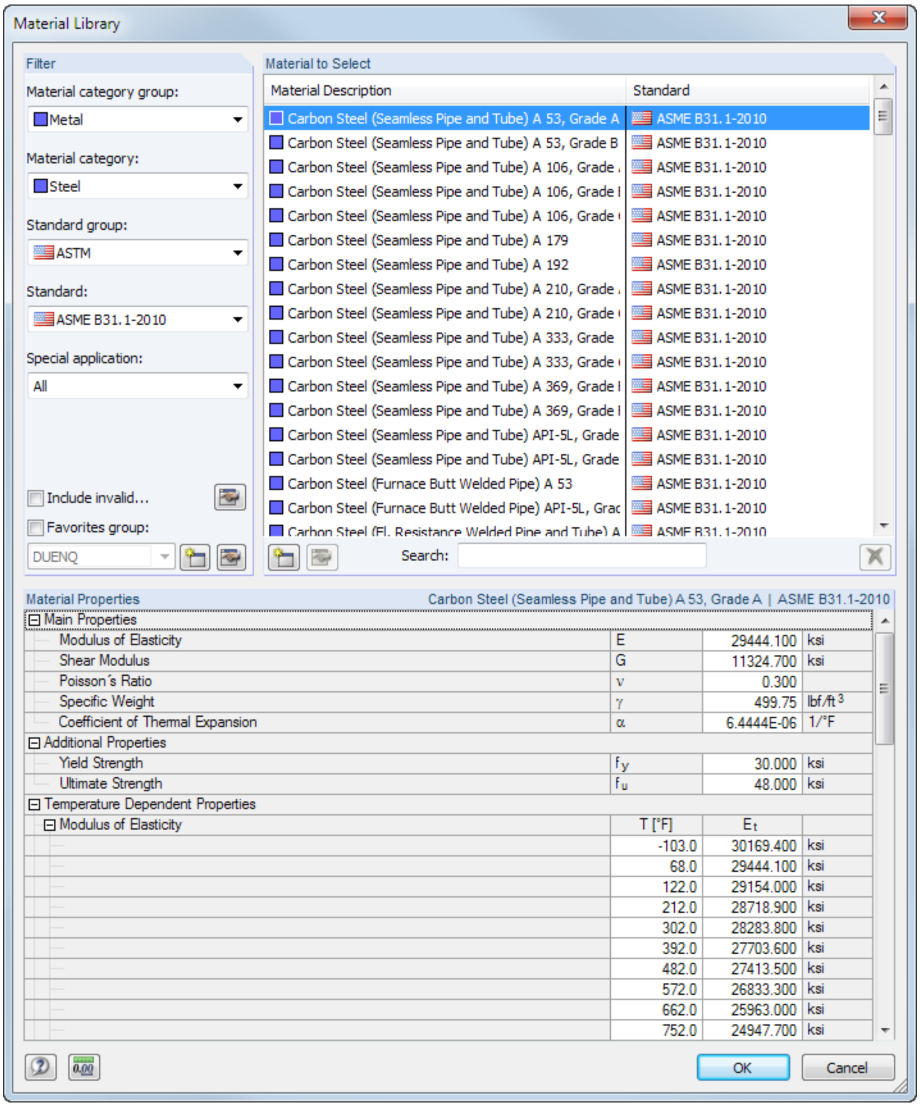


Figure 2.13: Dialog box *Material Library*

The characteristic features of pipeline materials are the temperature dependent strengths which are specified in the *Material Properties* dialog section.

Chapter 4.3 of the RFEM manual describes how to select, modify or add materials in the library.

2.2.2 Reducer

Reducers are used to modify the piping cross-section. When clicking the [New Reducer] button, you can select the member in the work window where you want to apply the reducer.

Reducers refer to nodes. If a cross-section is modified within a member, you have to divide this member in advance at the appropriate position.

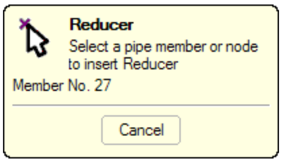


Figure 2.14: Select member in the work window

When clicking on the member, the *New Reducer* dialog box appears.

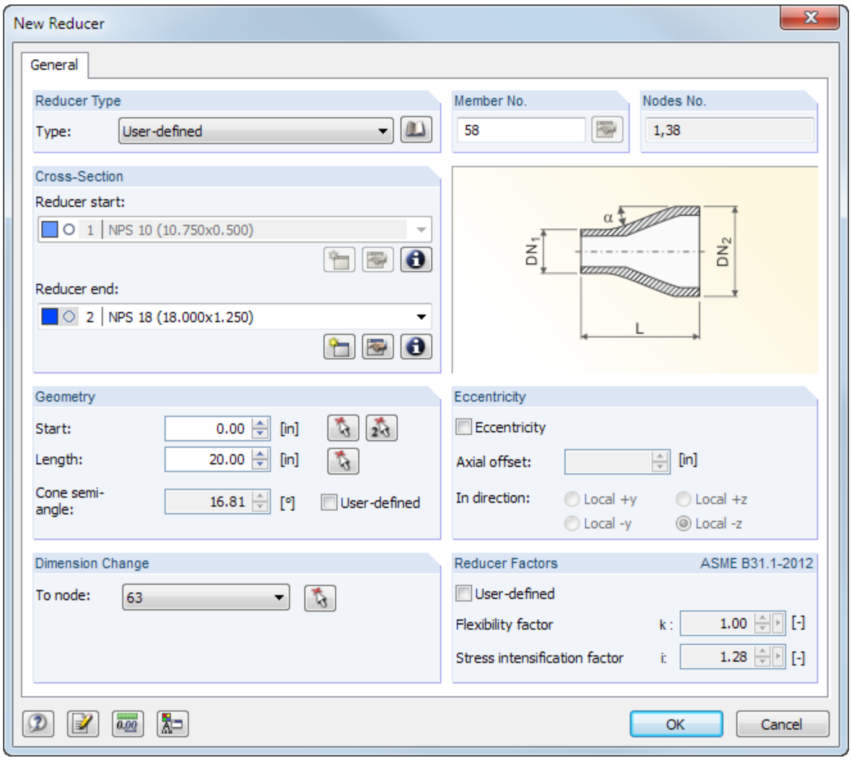


Figure 2.15: Dialog box *New Reducer*

Reducer Type

If the reducer is specified as *User-defined*, the geometry of the reducer can be defined manually in the other dialog sections.

With the button [book icon], you can access a library with standardized reducers (see Figure 2.16).

If you have already defined a cross-section for the *Reducer end* (dialog section below), solutions combatible with DN are preset in the library.

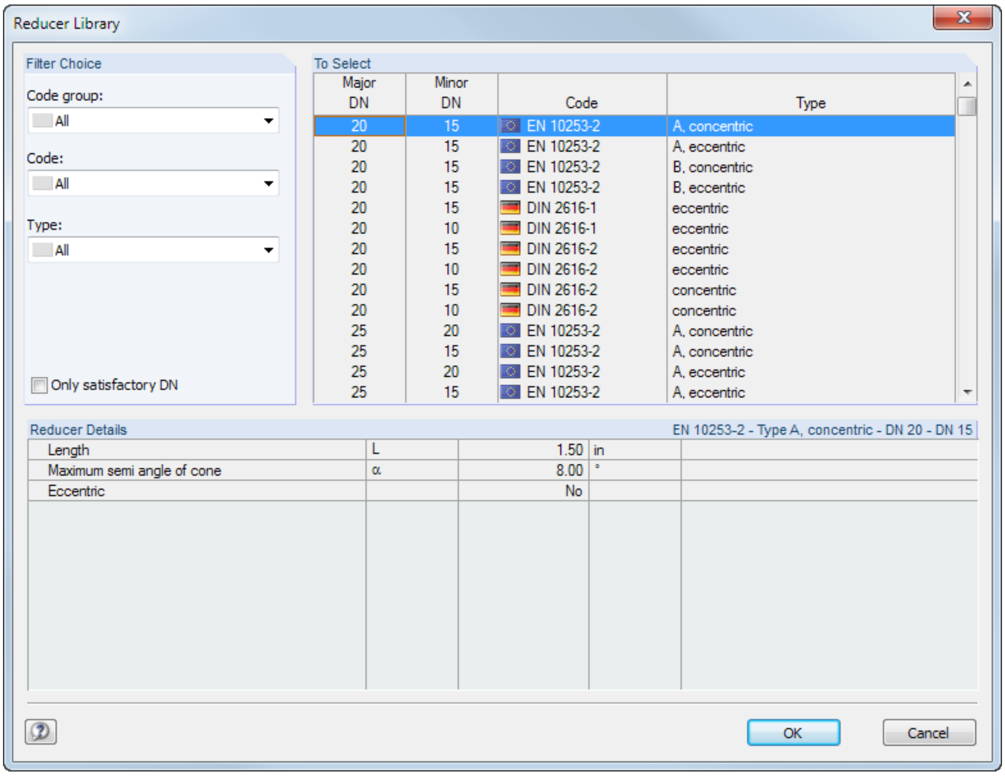


Figure 2.16: Reducer library

Cross-Section

A reducer is – similar to a taper – defined by a start and end cross-section. Depending on which node of the member was selected in the *Dimension Change* dialog section, either the *Reducer start* or the *Reducer end* can be changed.

Geometry

If you have manually defined the reducer, you have to specify the start point (x-location along the member axis) as well as the length of the reducer in addition to the cross-sections. You can also define it graphically with and .

The *Cone semi-angle* can be specified as user-defined if necessary.

Dimension Change

The *node* specified here defines which end of the pipe is concerned by the modification of the cross-section.

Eccentricity

Pipes always refer to the cross-section's centroid (wireframe model). For reducers, this leads to the fact that the pipe is continued centrally with a smaller cross-section perimeter. If, for example, the bottom side of the pipe has to be continuous, you can define an *Axial offset* of the local member axes after having selected the check box *Eccentricity*.

Reducer Factors

You can manually define the *Flexibility factor* and the *Stress intensification factor* where necessary.

2.2.3 Valve

The definition of a valve is used within the program to correctly enter the weight and the stiffness. When clicking on the [New Valve] button, you can select the member where you want to apply the valve.

Valves refer to members. Therefore, they can be arranged within a member.

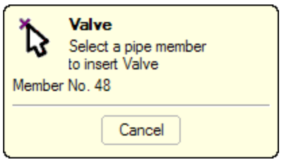


Figure 2.17: Select member in the work window

When clicking on the member, the *New Valve* dialog box appears.

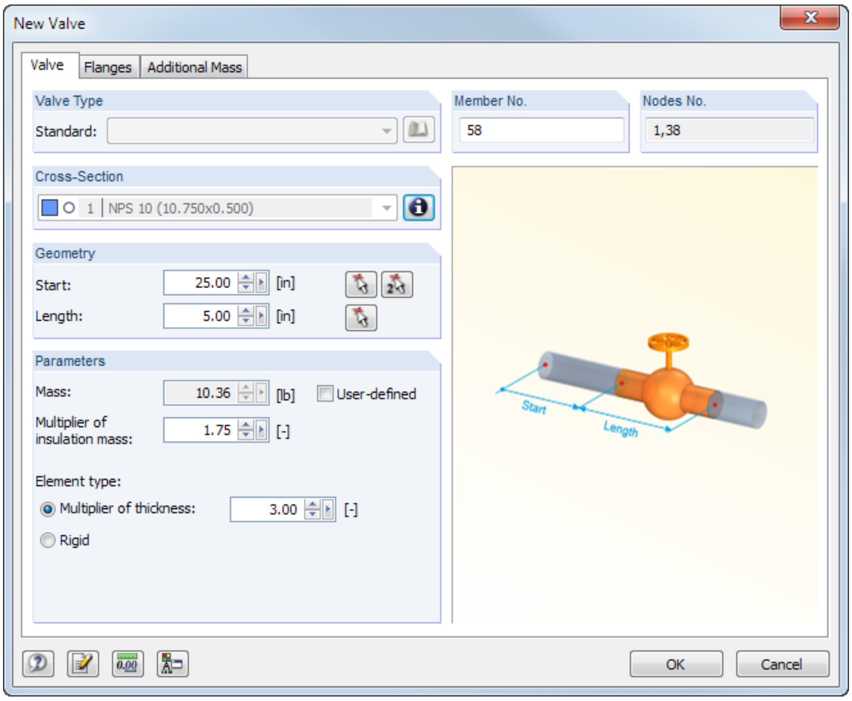


Figure 2.18: Dialog box *New Valve*

Valve Type

Currently, there are no preset valves available.

Cross-Section

The cross-section of the member is displayed for information purposes.

Geometry

To place the valve at the member, you have to define the *Start* (x-location along the member axis) and *Length* of the valve (end point). By using [button] and [button], you can graphically define both positions.

Parameters

In this dialog section, you can find options to adjust the *Mass* of valve and insulation as well as setting options concerning the stiffness. The mass is uniformly "distributed" over the length of the valve.

Flanges

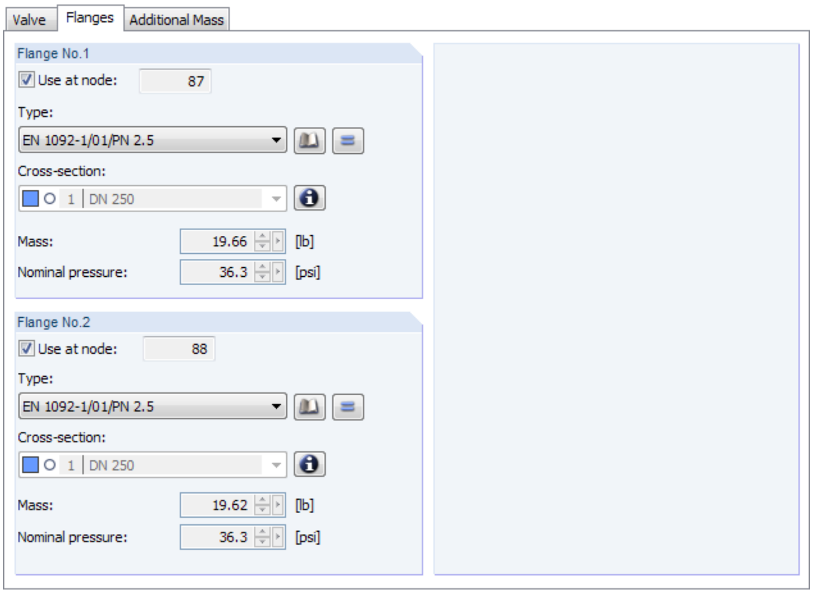


Figure 2.19: Dialog box *New Valve*, tab *Flanges*

Use the second tab of the dialog box to assign flanges on both sides of the valve. By using this function, you can correctly determine the load situation. The [Library] helps to find the correct flange type and thus the correct load (see Figure 2.20).

As an alternative, you can manually define the flanges with *Mass* and the *Nominal Pressure*.

If you want to use the same flange type at both ends, you can transfer the flange with the button to the other part.

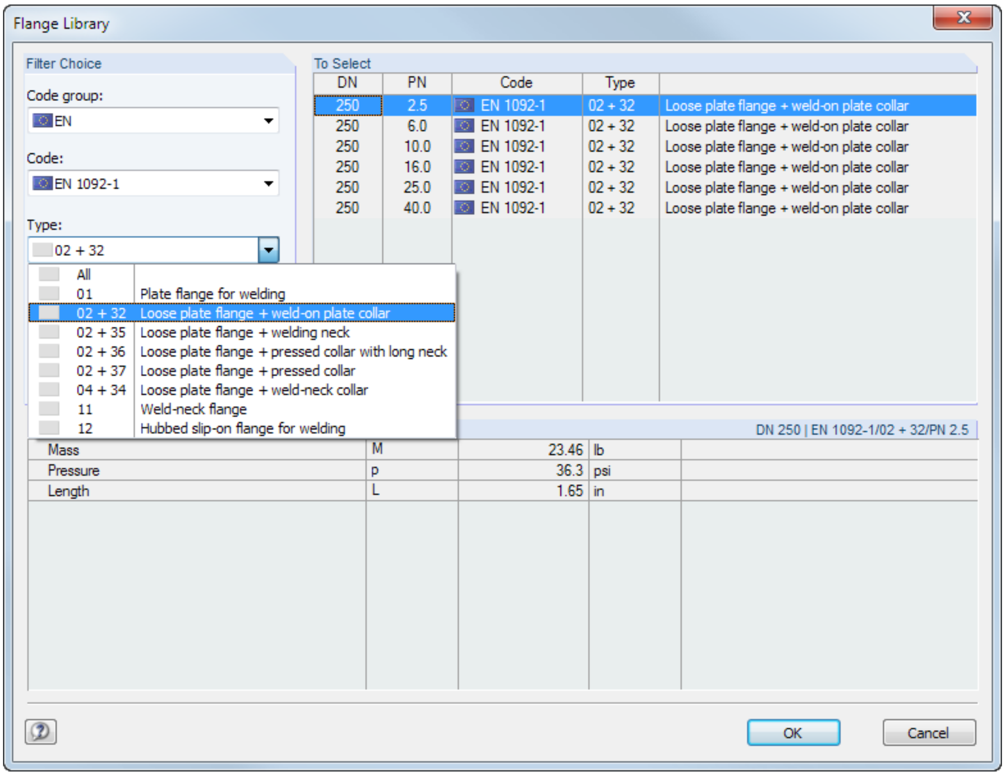


Figure 2.20: Flange library

Additional Mass

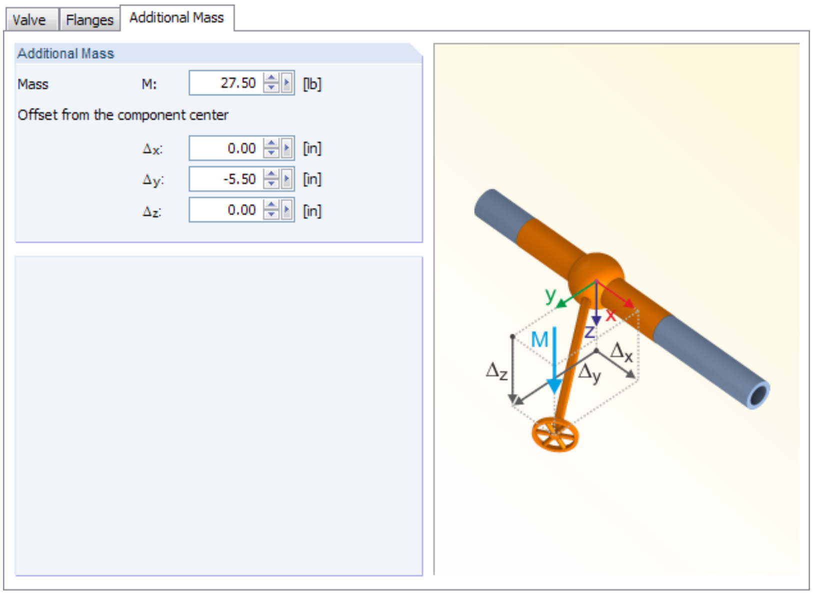


Figure 2.21: Dialog box *New Valve*, tab *Additional Mass*

To consider the eccentric mass of the handwheel, you can define the *Mass* and the *Offset* in this tab from the center of the pipe.

2.2.4 3-Way Valve

You can arrange 3-way valves on nodes where three pipes are connected. When clicking on the button [New 3-way Valve], you can select the corresponding node.

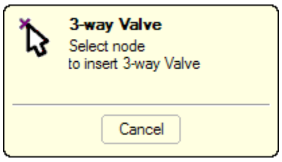


Figure 2.22: Select node in the work window

When clicking on the node, the *New 3-way valve* dialog box appears.

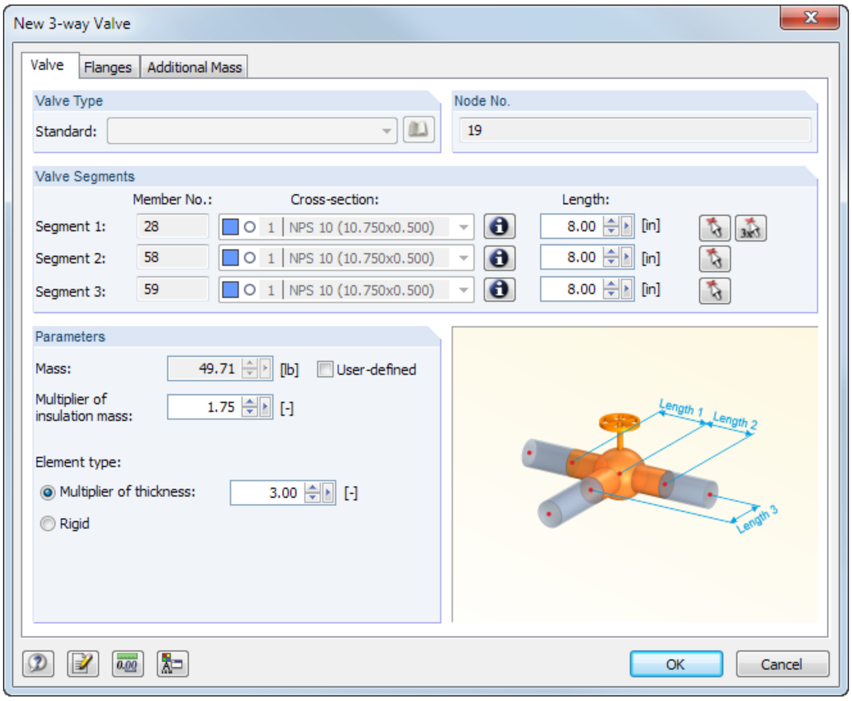


Figure 2.23: Dialog box *New 3-way Valve*

Valve Type

Currently, there are no preset 3-way valves available.

Valve Segments

In this dialog section, you have to define the *Length* of the individual segments. By using [button] and [button], you can also graphically define them.

The *Parameters* dialog section and the other tabs *Flanges* and *Additional Mass* are described in Chapter 2.2.3.

2.2.5 4-Way Valve

You can arrange 4-way valves on nodes where four pipes are connected. When clicking on the button [New 4-way Valve], you can select the corresponding node.

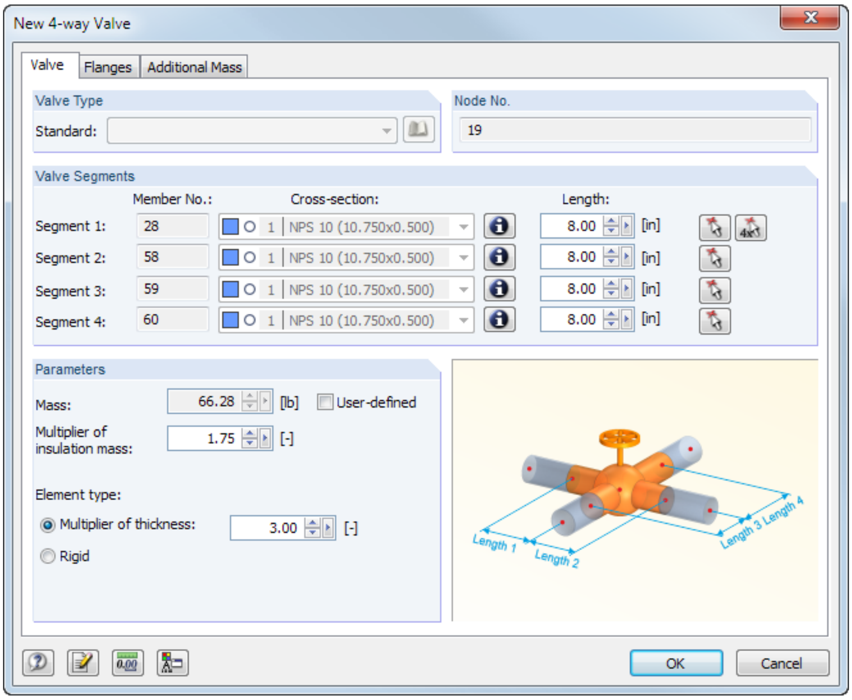


Figure 2.24: Dialog box *New 4-way Valve*

Currently, there are no preset 4-way valves available.

The sections of this dialog box are described in Chapter 2.2.3 and Chapter 2.2.4.

2.2.6 Bend

If the check box for generating bends is selected in the *New Pipeline* dialog box (see Figure 2.6, page 8), bends are generated automatically between straight pipeline sections when defining a pipeline. By using the [New Bend] button, you can subsequently define a bend to connect two straight pipeline sections.

When clicking on the corresponding node, the *New Bend* dialog box appears.

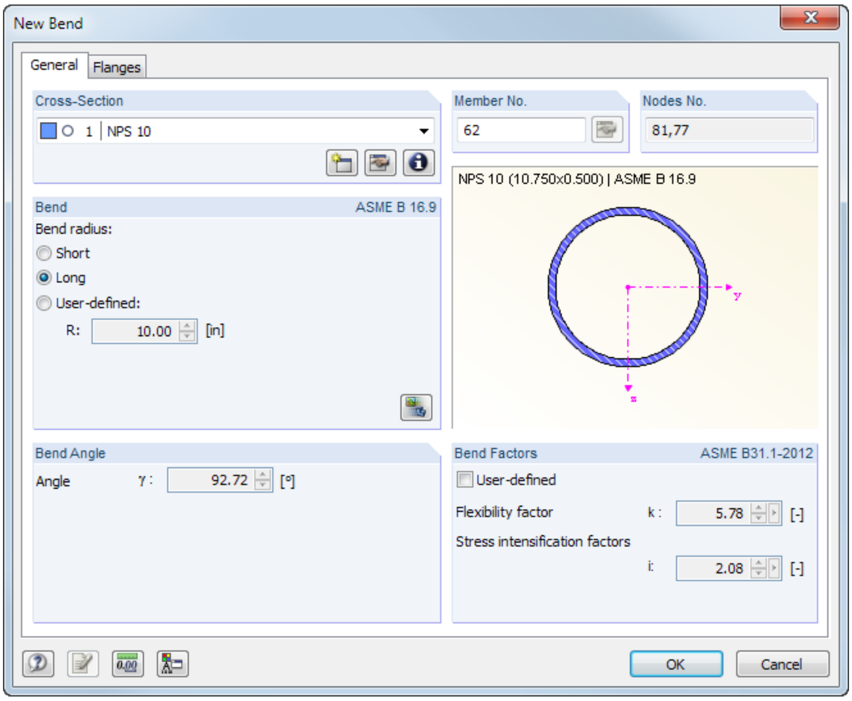


Figure 2.25: Dialog box *New bend*

In the first tab, you define the cross-section as well as the member number of the bend.

The *Bend* and *Bending Factors* dialog sections are explained in Chapter 2.2.1, the *Flanges* tab in Chapter 2.2.3.

The *Bend Angle* is displayed for information purposes.

2.2.7 Flange

For detailed views of the loading, you can define flanges. When clicking on the [New Flange] button, you can select the node or member in the work window where you want to apply the flange.

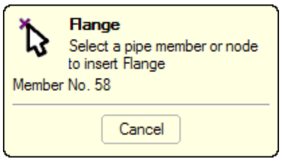


Figure 2.26: Select member in the work window

When clicking on the node or member, the *New Flange* dialog box appears.

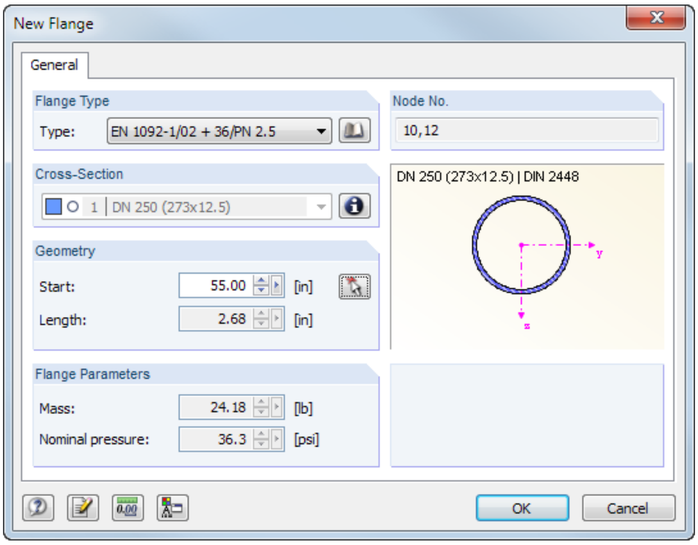


Figure 2.27: Dialog box *New Flange*

Flange Type

If the flange is specified as *User-defined*, the parameters of the reducer can be defined manually in the other sections.

With the button [button], you can access a library with standardized flanges (see Figure 2.19, page 18). Here are solutions preset which are suitable for the cross-section. In the library, you can filter according to DN and PN compatible entries.

Cross-Section

The current piping cross-section is displayed here for information purposes.

Geometry

If you have manually defined the flange, you have to specify the *Start* (x-location along the member axis) as well as the *Length* of the flange. You can also define it graphically with [button].

When you use flanges from the library, you just have to indicate the position of the flange.

2.2.8 Blind Flange

Blind flanges are used to lock nozzles and piping ends. When clicking on the button [New Blind Flange], you can select the corresponding node in the work window.

When clicking on the node, the *New Blind Flange* dialog box appears.

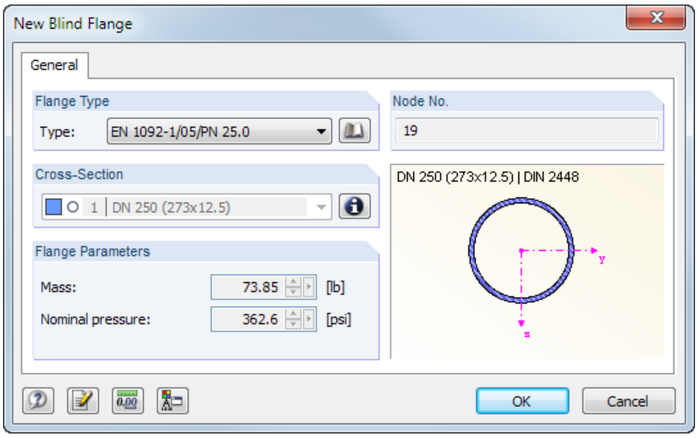


Figure 2.28: Dialog box *New Blind Flange*

Flange Type

If the flange is specified as *User-defined*, the parameters can be defined manually in the dialog section below.

With the button [library], you can access a library with standardized blind flanges (see Figure 2.29). Here are preset solutions which are suitable for the cross-section. In the library, you can filter according to DN and PN compatible entries.

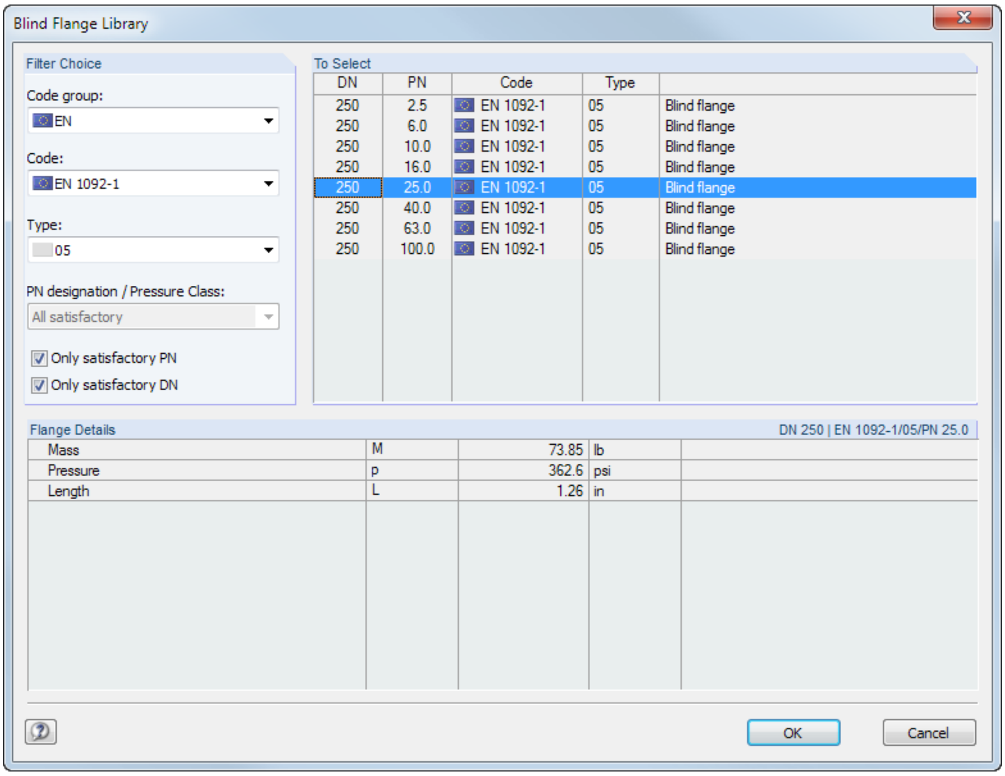


Figure 2.29: Blind flange library

Cross-Section

The current piping cross-section is displayed here for information purposes.

Flange Parameters

If you have selected the blind flange from the library, *Mass* and *Nominal Pressure* are preset. The pressure classes refer to the design pressure which you can define in the piping cross-section.

If the blind flange is specified as *User-defined*, both parameters can be defined manually. Only the mass is considered as an additional load for the calculation.

2.2.9 Tee

If a pipe is connected to an existing pipeline, prefabricated tees can be used for connection. When clicking on the [New Tee] button, you can select the corresponding node in the work window.

Only nodes are allowed where two pipes go straight through. The third pipe can be connected in any way.

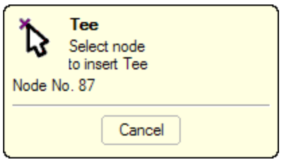


Figure 2.30: Select node in the work window

When clicking on the node, the *New Tee* dialog box appears.

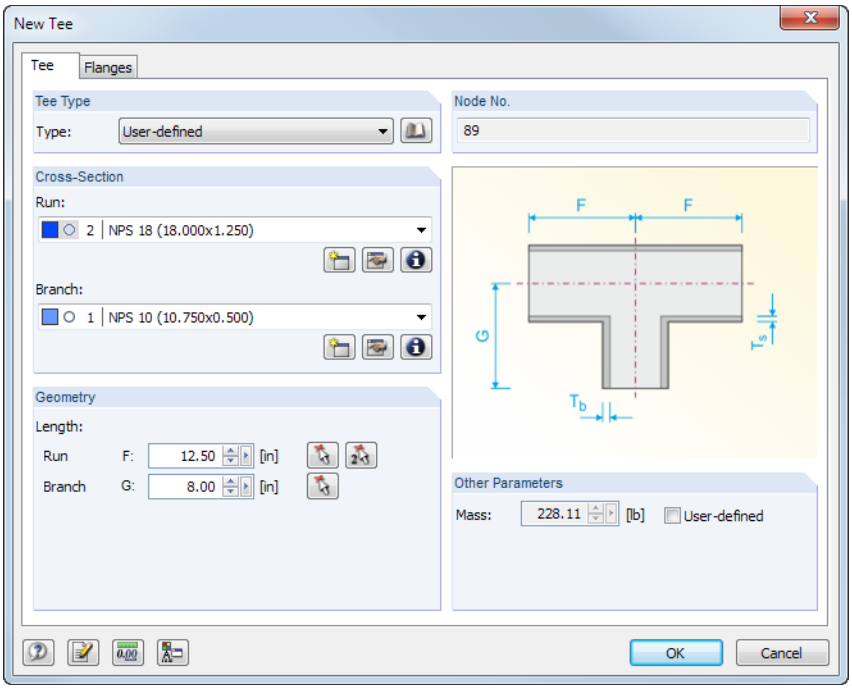


Figure 2.31: Dialog box *New Tee*

Tee Type

If the tee is specified as *User-defined*, the parameters can be defined manually in the dialog sections below.

With the button, you can access a library with standardized tees (see Figure 2.32). Here are preset solutions which are suitable for the connection. In the library, you can filter according to DN compatible entries.

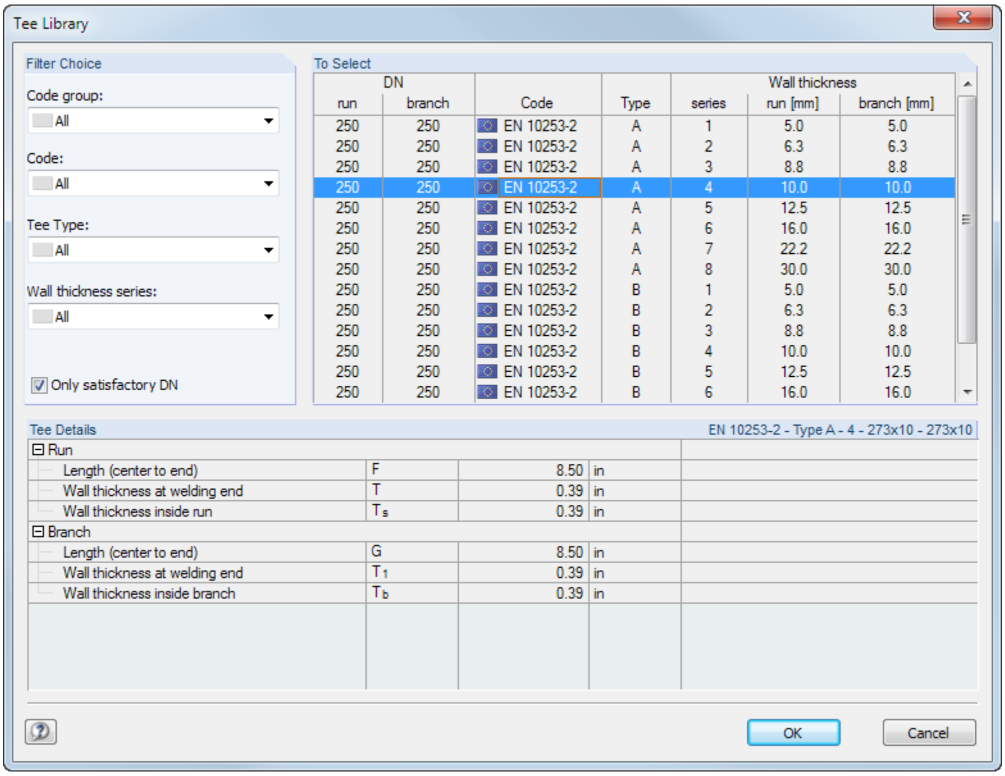


Figure 2.32: Tee Library according to EN 10253-2 [4]

Cross-Section

If the tee is specified as user-defined, you can define the cross-sections for the *Run* and the *Branch*.

Geometry

If you select the tee from the library, the geometry parameters of the connection will be entered automatically. If you manually enter the data, you have to define the *Length* and inner *Wall thickness* for the run and the branch. You can define the lengths also graphically by using the buttons and .

Other Parameters

The *Mass* of the tee is uniformly distributed as a distributed load over the parts of the tee. Only the corresponding pipe weight is considered for the value 0.

Flanges

In the second tab of the dialog box, you have the possibility to arrange flanges on the connection. The functions of this tab are described in Chapter 2.2.3 on page 18.

2.2.10 Branch Connection

The stress intensification factors (SIF) relevant for the design differ according to the type of tee. The type preset in the basic settings is used by default (see Chapter 2.2.9). You can manually define the factors for each connection by using the [New Branch Connection] button.

When clicking on the corresponding node, the *New Branch Connection - Factors* dialog box appears.

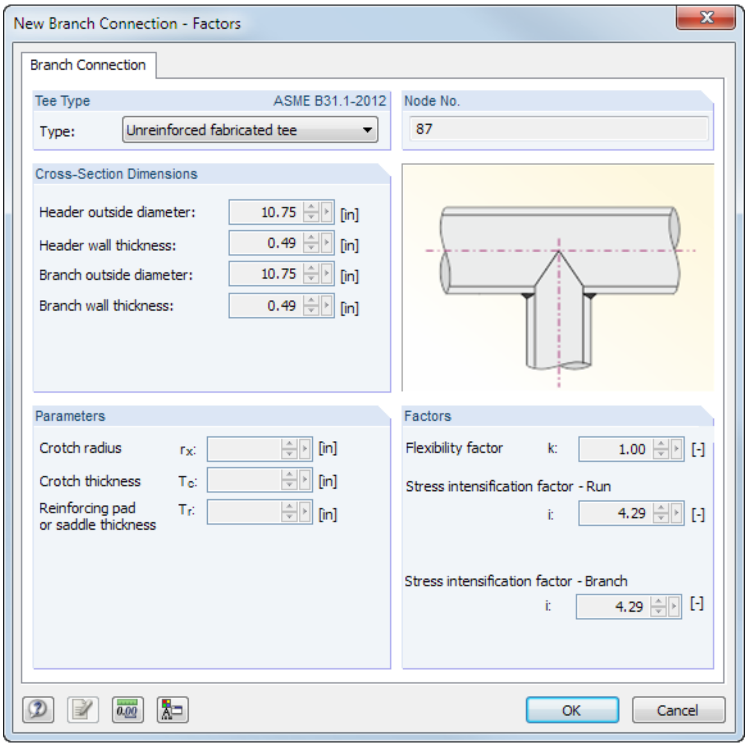


Figure 2.33: Dialog box *New Branch Connection - Factors*

Tee Type

This list contains different types of tees according to [1] (see also Chapter 2.1.1, page 5). Depending on the selection, more information about, for example, diameters or wall thicknesses is necessary.

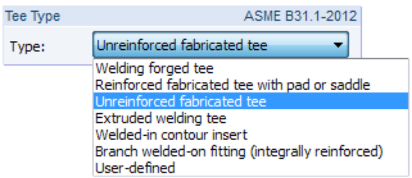


Figure 2.34: Tee Types

With the option *User-defined*, you can manually define the flexibility and stress intensification factors.

2.3 Load Cases and Combinations

The loads acting in the piping structure are organized in different load cases. It is possible to superimpose these load cases – manually or automatically – in **Piping Load Combinations** (*PC*) and in **Result Combinations** (*RC*).

The functionality of these two combination possibilities is similar to the load and result combinations in RFEM. They are described in the RFEM manual, Chapter 5. Only the specific characteristics of piping construction will be mentioned in the following.

2.3.1 Load Cases

Load cases enable sorting of loads according to their action category. By activating RF-PIPING in the *General Data* dialog box, the list of possible action categories is extendend.

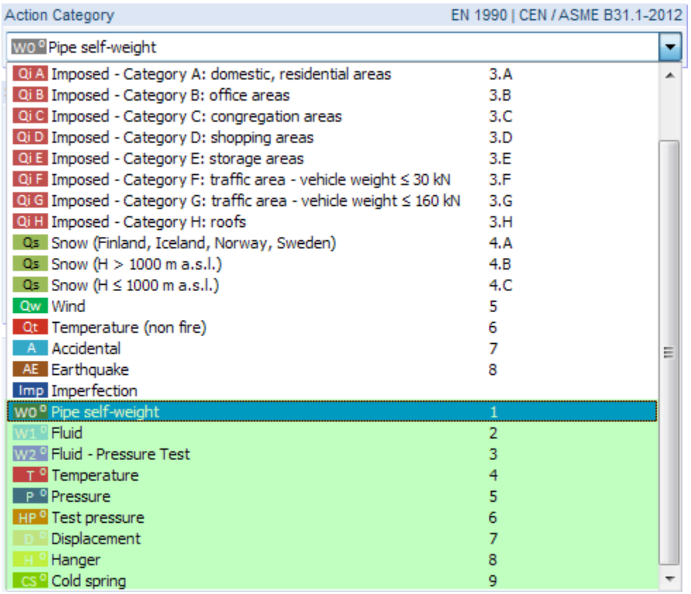


Figure 2.35: Action categories for piping load cases (in green)

The action categories for the classification of piping loads are:

- Pipe self-weight
- Fluid
- Fluid - Pressure Test
- Temperature
- Pressure
- Test pressure
- Displacement
- Hanger
- Cold spring

A load case which is adequate for piping design has to be classified in one of the piping-specific action categories. These categories are described, for example, in [3] Section 4.2.5 as design conditions.

If the load combinations are generated automatically according to the default settings in the *Settings* dialog box (see Figure 2.3, page 5), only those load cases which have been classified as piping load cases are considered for creating piping combinations and result combinations. Load cases of other action categories are ignored.

2.3.2 Piping Load Combinations (PC)

The superposition of load cases is normally carried out in *Piping load combinations*. They are based on the principle of the load combinations: The loading of the single load cases is combined to a "big load case" according to the combination criterion.

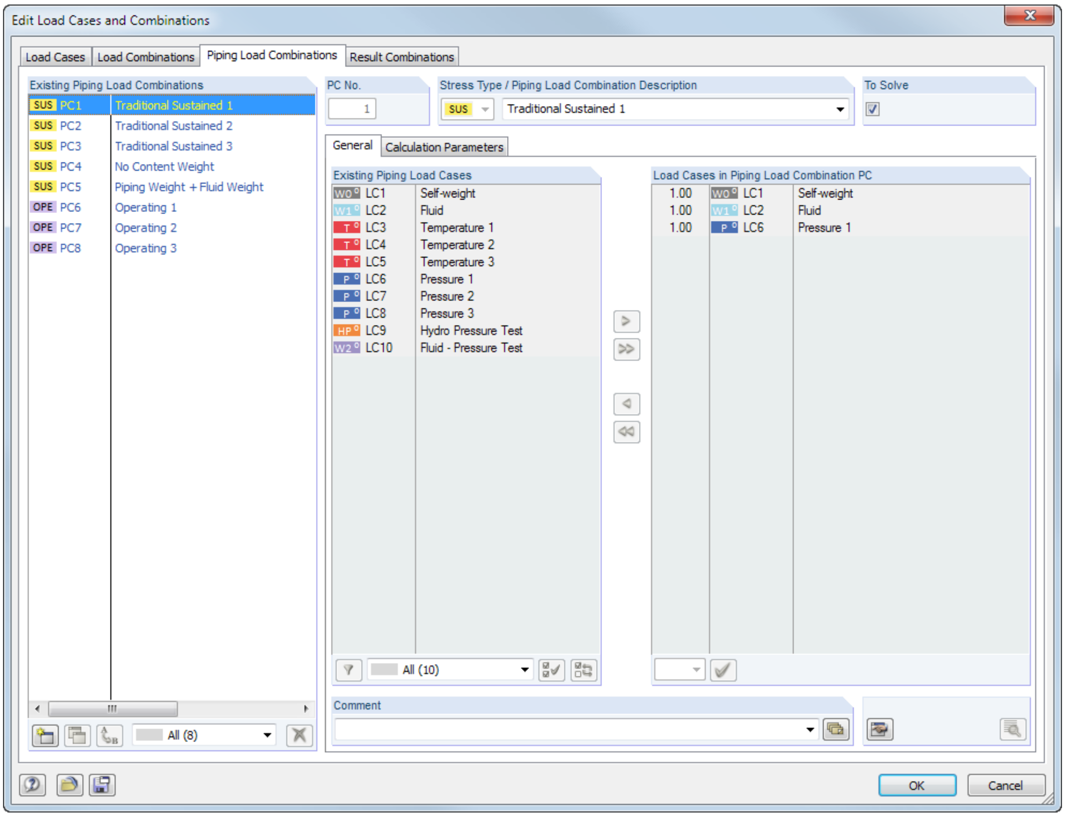


Figure 2.36: Dialog box *Edit Load Cases and Combinations*, tab *Piping Load Combinations*

Depending on the load situation, you may have to generate different combinations of the load cases. They are classified in *Stress Types*. If you manually define a combination, you have the following options:

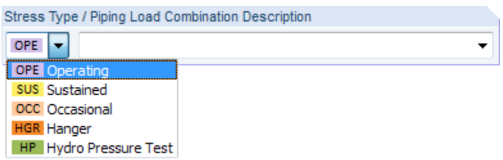


Figure 2.37: Stress types

Later on during the design in the module RF-PIPING Design, the stress type of a piping combination determines which analysis is carried out with the resulting internal forces. If you manually define combinations, you have to take care that they are correctly allocated.

If combinations are created automatically (see *Settings* dialog box, Figure 2.3, page 5), the piping load combinations are automatically allocated to the corresponding stress types.

If there are more relevant temperature/pressure load cases , the pairs have to be defined appropriately before the combination. An appropriate message will be displayed before the combination. However, it is also possible to assign it later by using the button  at the botton in the *Edit Load Cases and Combinations* dialog box. The *Grouping of Thermal and Internal Pressure Load Cases for Operating Combinations* dialog box will be displayed (see Figure 2.38).

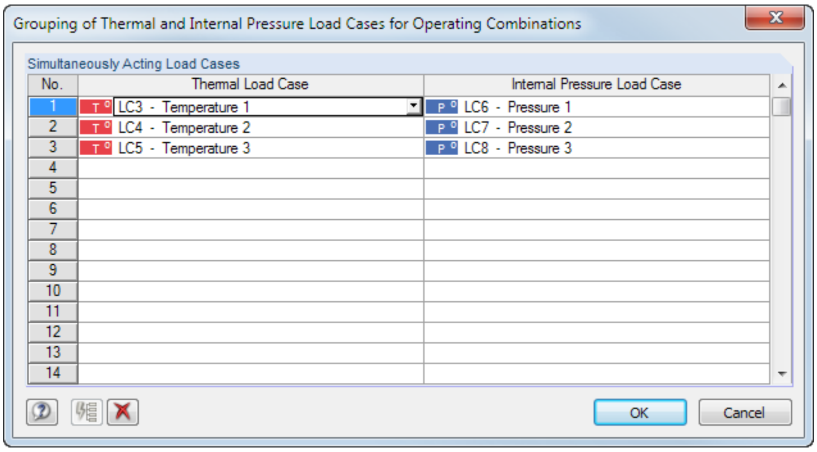


Figure 2.38: Dialog box *Grouping of Thermal and Internal Pressure Load Cases for Operating Combinations*

2.3.3 Result Combinations (RC)

The function of result combinations is described in the RFEM manual, Chapter 5.6.

Result combinations are used in piping construction to generate temperature combinations. If you have selected the automatic generation of combinations in the *Settings* dialog box (see Figure 2.3, page 5), the combinations of the stress type *Expansion* will be created as a result combination. The internal forces result from the difference between the operating stress piping combinations (OPE) and the sustained piping combinations (SUS).

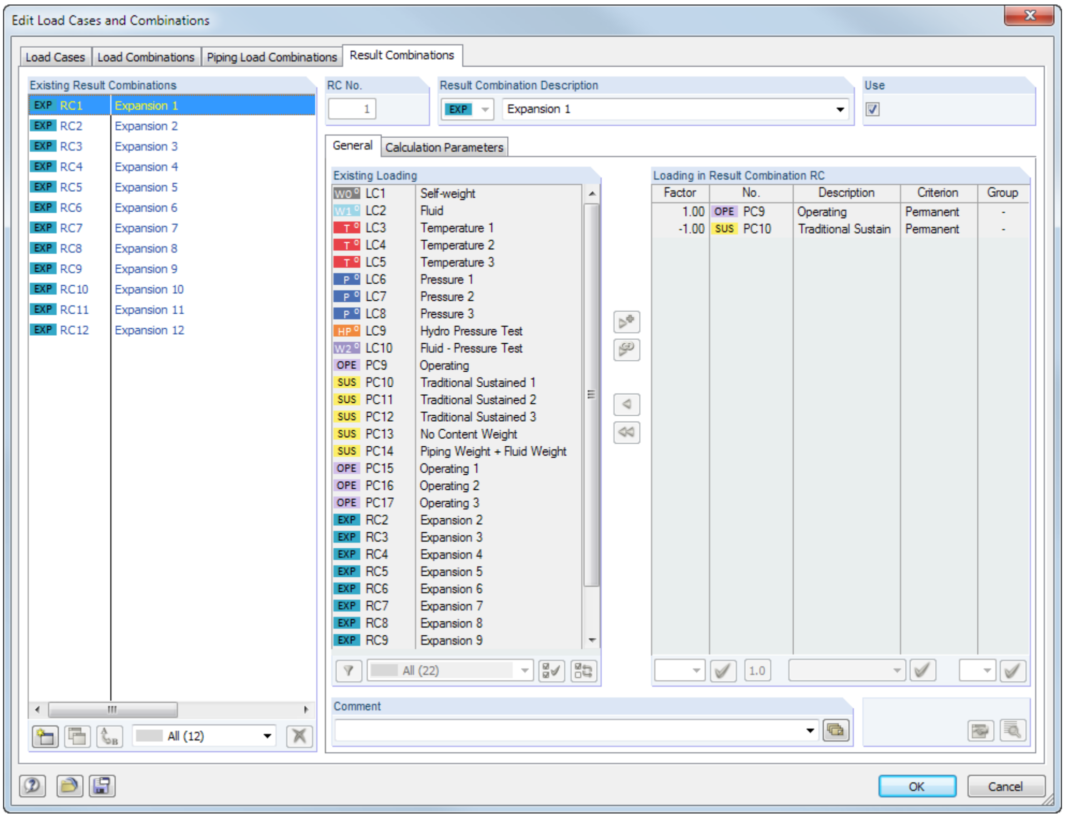
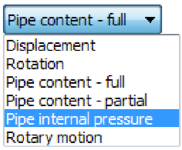


Figure 2.39: Dialog box *Edit Load Cases and Combinations*, tab *Result Combinations*

If the option *Solve independently (piping load combinations)* in the *Settings* dialog box (see Figure 2.3, page 5) is selected, no result combinations will be generated. The situations *Expansion* are represented in this case by piping combinations.

2.4 Piping Loads

The loading of the pipeline can be carried out with nodal and member loads.



For member loads, you can also select piping-specific load types such as *Pipe content - full/partial* and *Pipe internal pressure* in addition to temperature loads.

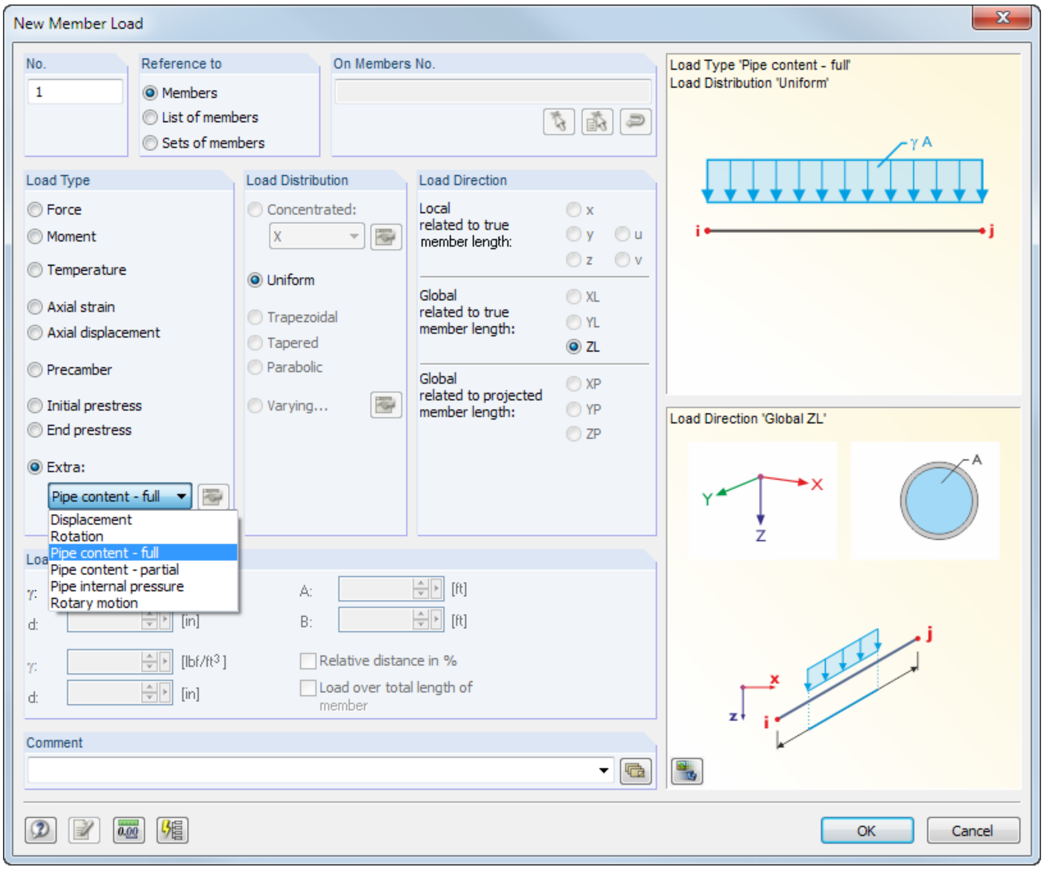


Figure 2.40: Dialog box *New Member Load* with options for piping loads

The piping-specific load types are briefly described in the following table:

Load Type	Description
Temperature	Temperature load evenly distributed over the member cross-section. You have to define the heating T_c with reference to the member center line.
Pipe content - full	Distributed load due to complete filling of a pipe. You have to define the specific weight γ of the pipe content.
Pipe content - partial	Distributed load due to partial filling of a pipe. In addition to the specific weight γ of the pipe content, you have to define the filling level *d*.
Pipe internal pressure	Constant internal pressure of a pipe. You have to define the value of the internal pressure.

Table 2.2: Specific load types for pipelines

The definition of node and member loads is described in the RFEM manual, Chapter 6.1 and 6.2.

2.5 Selection and Modification of Pipelines

Before you can edit pipes or pipelines, you have to select the relevant objects. You have different options:

- Selection in the work window by clicking the object
- Selection in the *Data* Navigator by clicking the entry
- Selection in the table by clicking the line

The selection of pipelines in the work window is similar to the selection of members: Left-click to select the object, double-click to open the editing dialog box of the pipe or a component (pipe, bend, reducer, etc.).

When you right-click the object, a shortcut menu is displayed where you have different editing options.

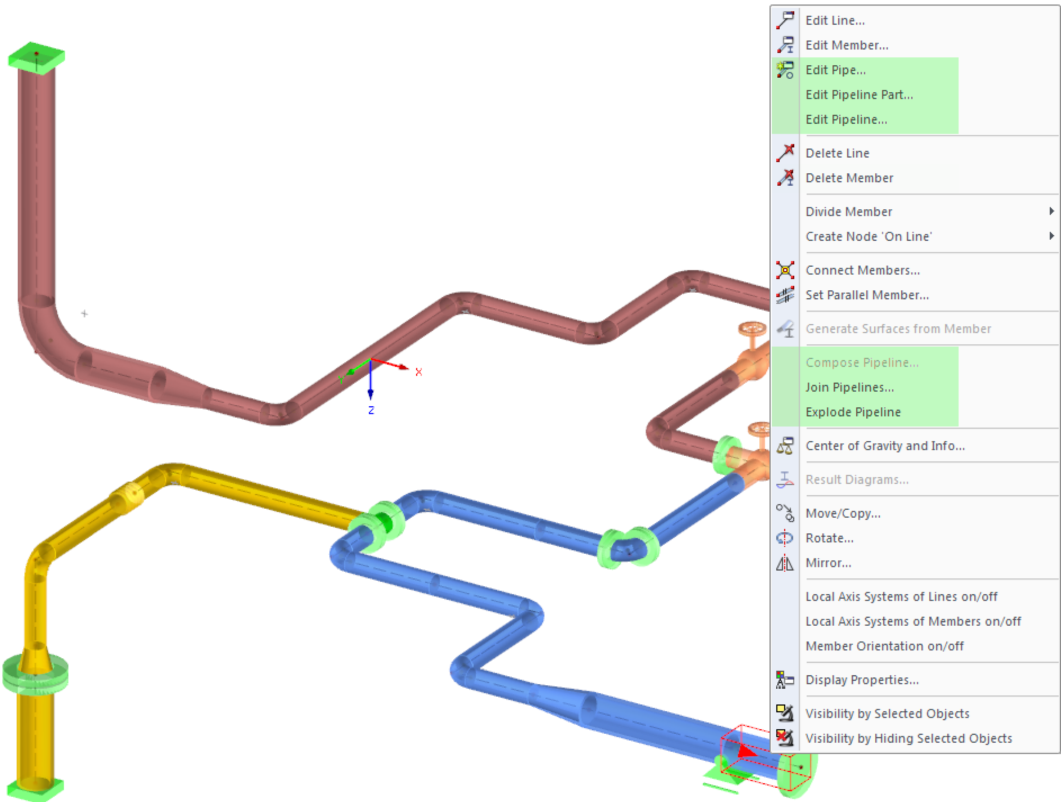


Figure 2.41: Piping shortcut menu with editing options (highlighted)

Edit Pipe

This option is the same as double-clicking the object. The dialog box is displayed and you can edit the pipe (or the valves, flanges, etc.).

Edit Pipeline Part

With this function, the whole pipeline can be edited. In the *Edit Pipeline Part ...* dialog box (see Figure 2.42), you can define, for example, a new bend radius for all bends or a new cross-section for the pipeline.

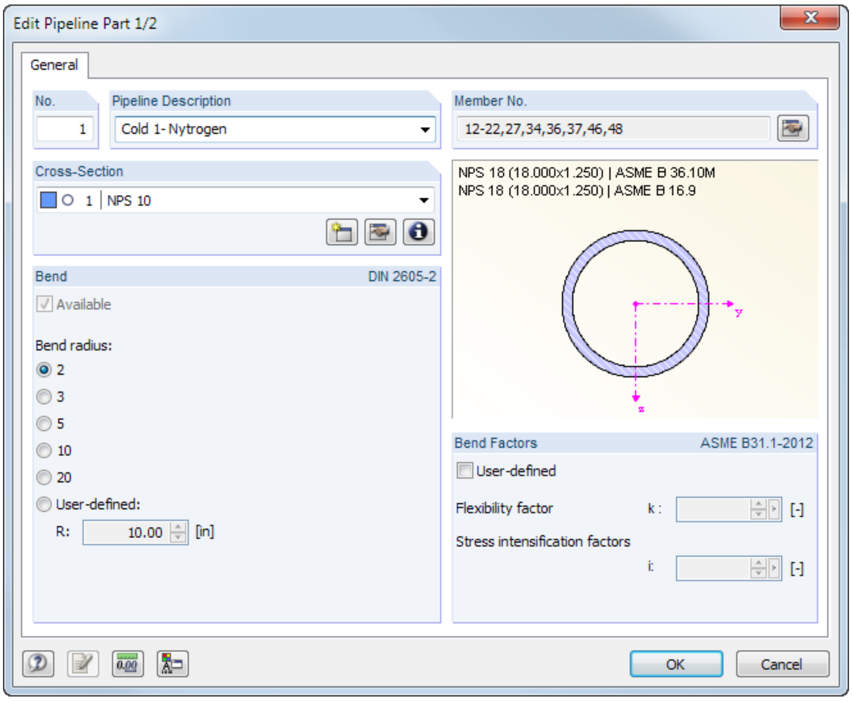


Figure 2.42: Dialog box *Edit Pipeline Part 1/2*

Edit Pipeline

With this function, you can edit the *description* of the pipeline and change the *members* which display the pipeline. Thus, you can add members at a later time.

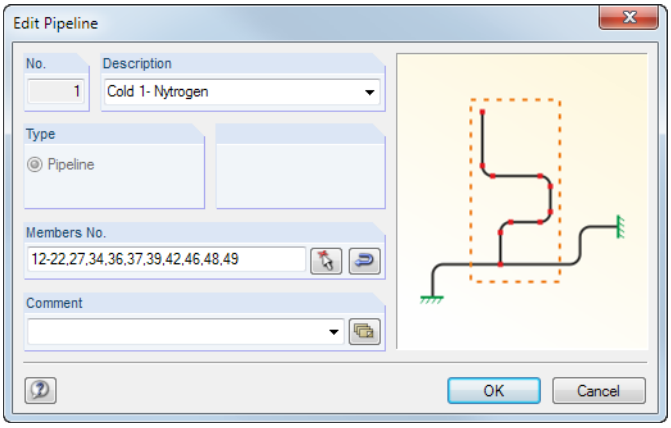


Figure 2.43: Dialog box *Edit Pipeline*

Delete Piping

The selected pipeline or the selected object will be deleted.

Compose Pipeline

If connected piping elements (members of the member type *Piping*) are present, which have not been assigned to any pipeline, you can define a new pipeline with this function.

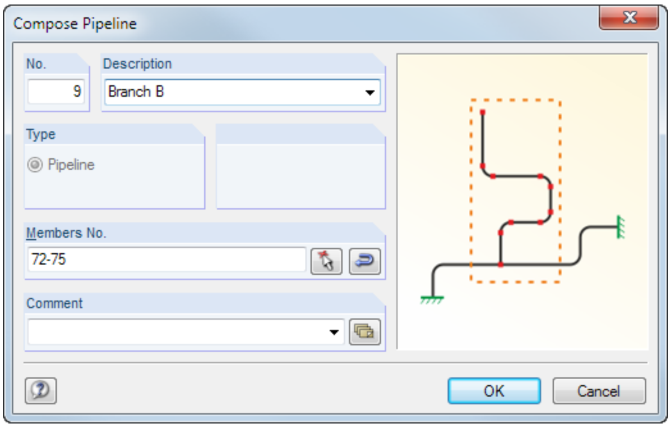


Figure 2.44: Dialog box *Compose Pipeline*

Join Pipelines

With this function, you can connect multiple pipelines. However, they have to match the criteria of a continuous member, i.e. the members of each pipeline must not branch.

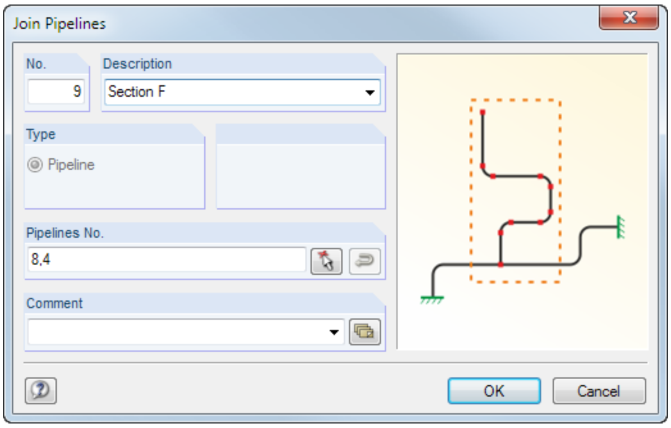


Figure 2.45: Dialog box *Join Pipelines*

Explode Pipeline

The selected pipeline is divided in its single elements. Afterwards, you can rearrange these elements.

Pipes which do not belong to a pipeline cannot be designed at a later stage with the module RF-PIPING Design.

2.6 Graphical Display

For visual control and a clearly arranged display, it is possible to display each pipeline in a different color. You can set this function in the *Display* navigator: **Colors in Graphics According to → Pipelines**.

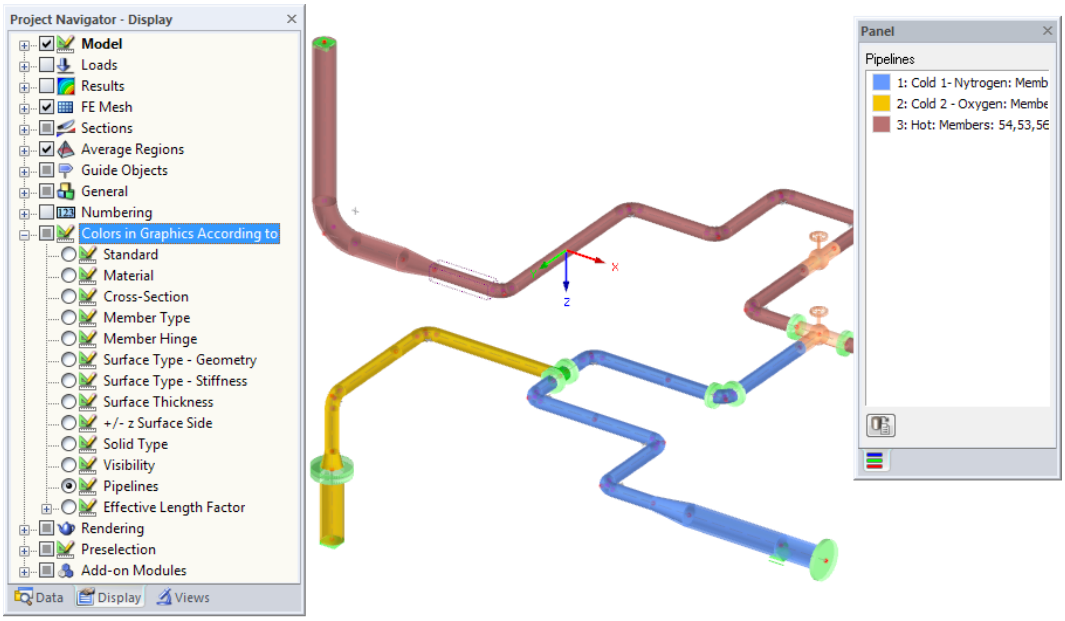


Figure 2.46: Pipelines in rendering

The functions of the graphical user interface are described in the RFEM manual, Chapter 11.

2.7 Calculation Settings

You can modify piping-specific settings for the analysis in the *Piping Analysis - Settings* dialog box (see Chapter 2.1.1, page 5). In this dialog box, the global settings for stress intensification factors, load combinations, internal pressure and material parameters can be adjusted.

The settings for the internal pressure are checked with the *Calculation Parameters* RFEM dialog box, *Global Calculation Parameters* tab: If you select, for example, the Bourdon effect for the add-on module RF-PIPING, the check box for the RFEM analysis will be also activated.

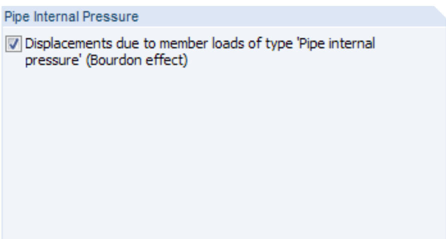


Figure 2.47: Dialog section *Pipe Internal Pressure* in the dialog box *Calculation Parameters*

The Bourdon effect is only fully considered for members of the type *Piping*: Regular members are only subject to elongation, the curvatures are considered additionally for pipelines (option *Elongation and straightening*).

2.8 Results

You have the same possibilities for the results display and evaluation in RF-PIPING as for normal load cases and combinations. They are described in the RFEM manual, Chapter 8 and 9.

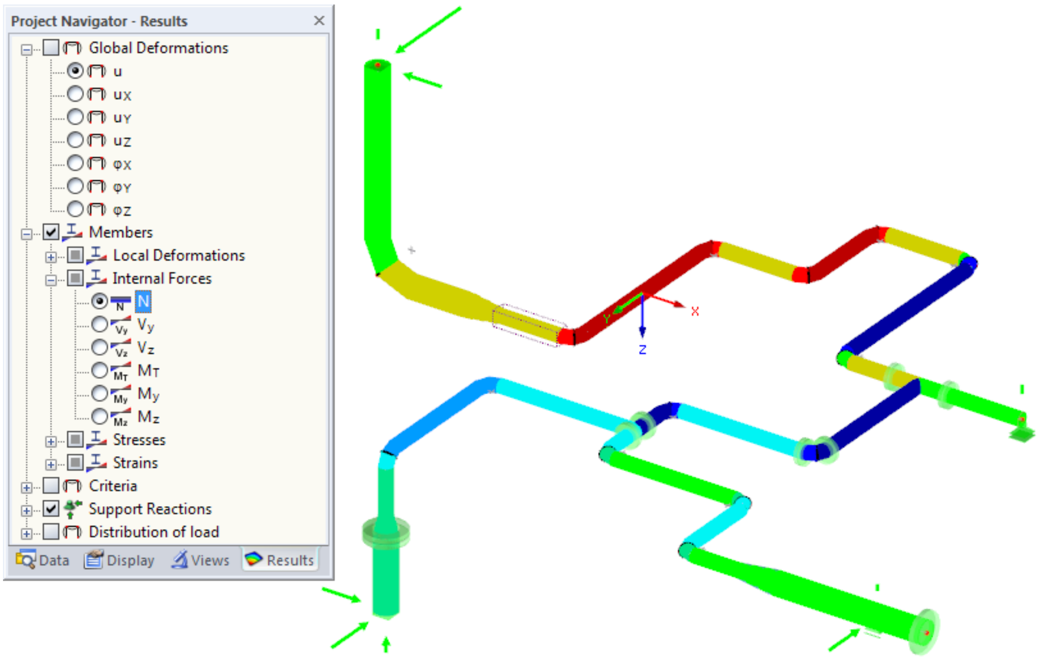


Figure 2.48: Axial forces of RF-PIPING load case with display mode *Cross-Sections*

You can include the results in the printout report (see Chapter 4, page 62).

3 RF-PIPING Design

The add-on module RF-PIPING Design is a tool for piping analysis according to the standards ASME B31.1 [1], ASME B31.3 [2] and DIN EN 13480-3 [3] (in preparation). The stress designs due to permanent and occasional loads and from thermal expansion are carried out.

RF-PIPING Design is integrated in RFEM. Thus, the design-relevant input data is already preset when you start the module. After the design, you can use the graphical RFEM user interface to evaluate the results. You can include the designs in the printout report.

3.1 Start RF-PIPING Design

In RFEM, you have the following possibilities to start the add-on module RF-PIPING Design.

Menu

You can open the add-on module with the RFEM menu

Add-on Modules → **Piping** → **RF-PIPING Design**.

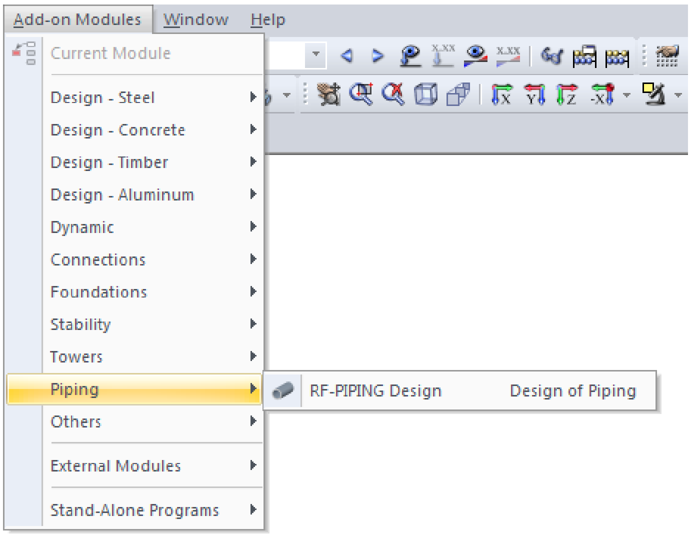


Figure 3.1: Menu *Add-on Modules* → *Piping* → *RF-PIPING Design*

Navigator

Alternatively, you can open the add-on module in the *Data Navigator* by clicking

Add-on Modules → RF-PIPING Design.

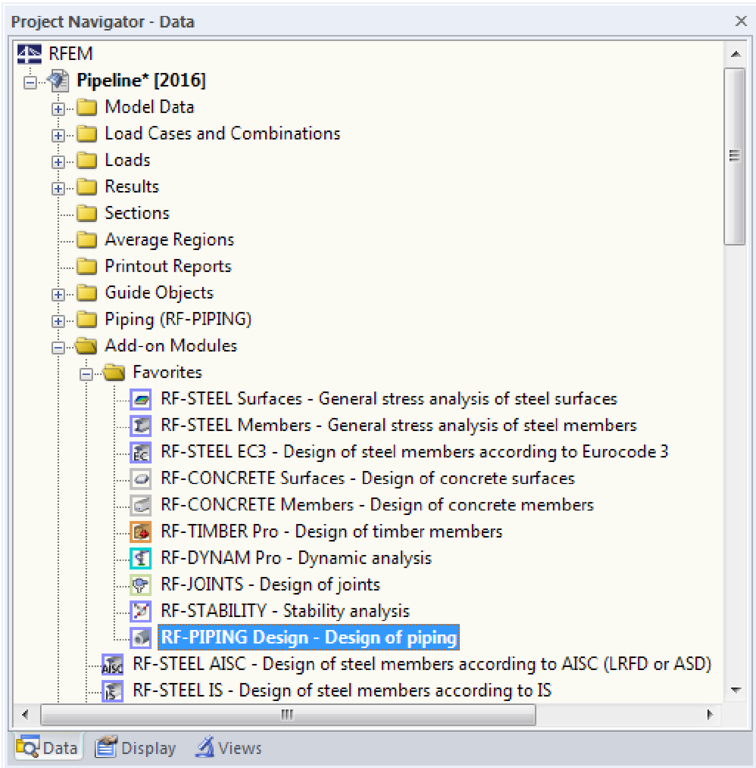


Figure 3.2: Data Navigator *Add-on Modules → RF-PIPING Design*

3.2 Input Data

When you start the add-on module, a new window is displayed. A navigator is displayed on the left, managing the currently available tables. The drop-down list above the navigator contains the design cases (see Chapter 3.7.1, page 59).

The design-relevant data is stored in three tables. When you open RF-PIPING for the first time, the following parameters are imported automatically:

- Pipelines
- Piping Combinations (*PC*) and Piping Result Combinations (*RC*)
- Materials
- Cross-Sections

To select a table, click the corresponding entry in the navigator. To set the previous or next table, use the buttons shown on the left. Alternatively, you can use the function keys to go to the next [F2] or previous [F3] table.

To save the results, click [OK]. You exit RF-PIPING Design and return to RFEM. To exit the add-on module without saving the data, click [Cancel].

3.2.1 General Data

In Table 1.1 *General Data*, you select the pipelines and combinations that you want to design. The standard is already preset due to the default setting for RF-PIPING (see Chapter 2.1.1, page 5).

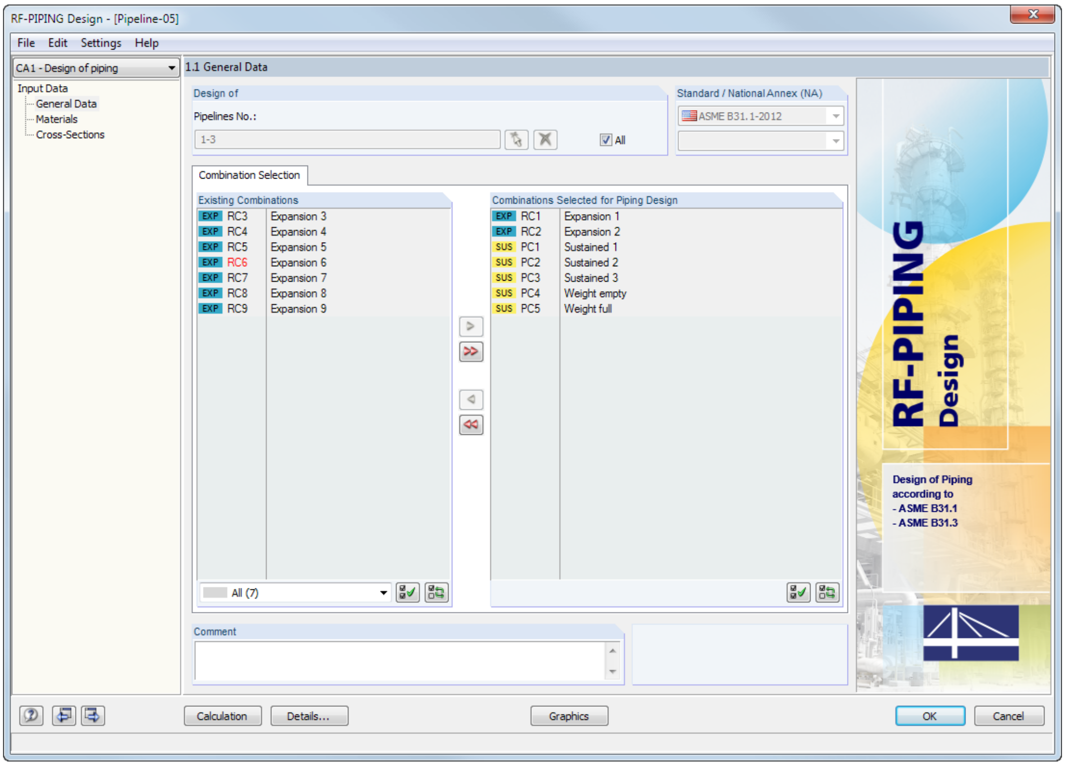


Figure 3.3: Table 1.1 *General Data*

Design

It is only possible to design *Pipelines*. Single piping components which are not assigned to a pipeline are not available.

If you want to analyze only selected pipelines, clear the check box *All*: Then you can access the text box to enter the numbers of the relevant pipelines. Use the [Delete] button to clear the list of preset numbers in the box. By using the [Select] button, you can also select the pipelines graphically in the RFEM work window.

Standard/ National Annex (NA)

In this dialog section, the design is displayed which is preset in the *General Data* RFEM dialog box for the calculation with RF-PIPING (see Figure 2.3, page 5). You have the following opportunities:

- ASME B31.1-2012 [1]
- ASME B31.3-2012 [2]
- EN 13480-3:2013 [3] (in preparation)

Select Load Case

The column *Existing Load Combinations* lists all piping combinations *PC* and result combinations *RC* that have been created in RFEM.

Use the button to transfer selected entries to the list *Selected for Piping Design* on the right. Alternatively, you can double-click the entries. To transfer the complete list to the right, use the button .

To transfer multiple entries of combinations, click the entries while pressing the [Ctrl] key as common for Windows applications. Thus, you can transfer several combinations at the same time.

If the number of a combination is displayed in red, such as RC 6 in Figure 3.3, you cannot design it: This happens when a combination is defined without any load data. When you transfer such a load case, a corresponding warning appears.

At the end of the list, several filter options are available. They will help you assign the entries sorted according to certain criteria.

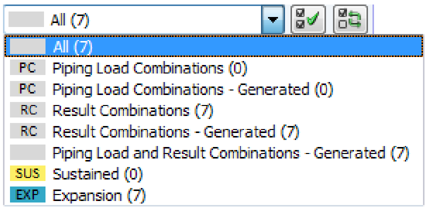


Figure 3.4: Filter options for combinations

The buttons have the following functions:

	All combinations in the list are selected.
	The selection of the combinations is inverted.

Table 3.1: Button in the tab *Load Combination Selection*

The column *Selected for Piping Design* lists all piping combinations and result combinations that have been selected for design. Click or double-click to remove selected entries from the list. The button empties the whole list.

Comment

In this text box, you can type user-defined notes describing, for example, the current design case.

3.2.2 Materials

This table is split into two parts. The upper part lists all materials created in RFEM. In the *Material Properties* section, the properties of the current material, e.g. the table row currently selected in the upper section, are displayed.

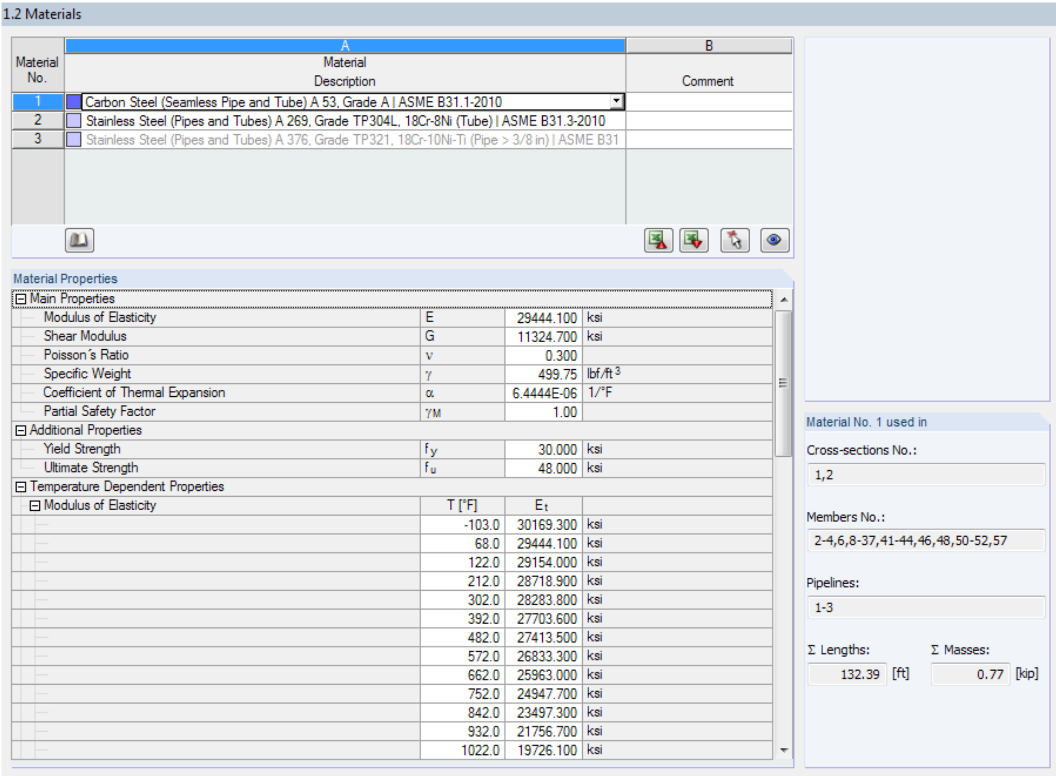


Figure 3.5: Table 1.2 *Materials*

Materials that won't be used in the design appear in gray lettering. Materials that are not allowed are highlighted red, modified materials are displayed in blue.

The material properties required for the determination of internal forces (*Main Properties*) are described in Chapter 4.3 of the RFEM manual. The material properties required for design are stored in the global material library. These values are preset (*Additional Properties*).

To adjust the units and decimal places of material properties and stresses, select **Settings** → **Units and Decimal Places** (see Chapter 3.7.2, page 60).

Material Description

The materials defined in RFEM are already preset, but it is always possible to modify them: To activate the field, click the material in column A. Then click the button or press function key [F7] to open the material list.

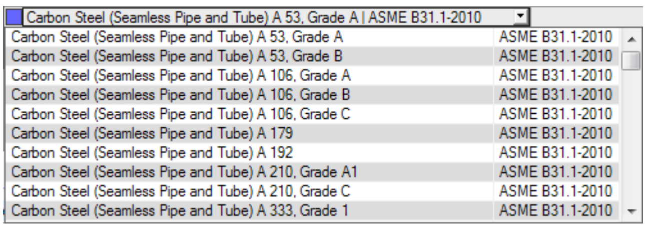


Figure 3.6: List of materials

In compliance with the design concept, you can select only piping materials.

Material Library

Numerous materials are already available in the library. To open the library, select

Edit → Material Library

or use the button shown on the left.

The material library is described in Chapter 2.2.1.3 on page 13.

3.2.3 Cross-Sections

This table lists the cross-sections that are used for the design.

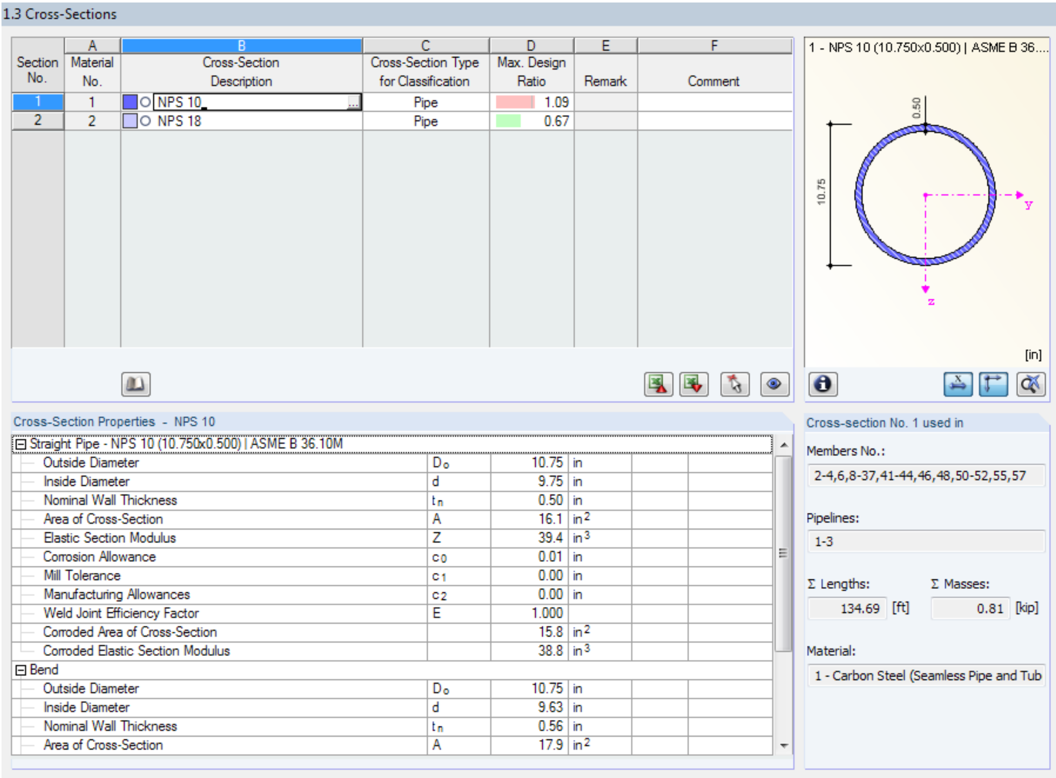


Figure 3.7: Table 1.3 *Cross-Sections*

Cross-Section Description

The cross-sections defined in RFEM are preset together with the assigned material numbers.

If you want to modify a cross-section, click the entry in column B to activate this field. Click [Cross-Section Library] or [...] in the field or press function key [F7] to open the dialog box *Edit Piping Cross-Section* (see Figure 2.10, page 12).

Modifying the cross-section also affects the model in RFEM!

Max. Design Ratio

This column will be shown after the calculation. It serves as a decision support for the optimization: By means of the design ratios and colored relation scales, you can see which piping cross-sections are little utilized and thus oversized, or overloaded and thus undersized.

Remark

This column shows remarks in the form of footers that are described in detail below the cross-section list.

Info About Cross-Section

In the *Info About Cross-Section* dialog box, you can view the cross-section properties, stress points, and c/t-parts.

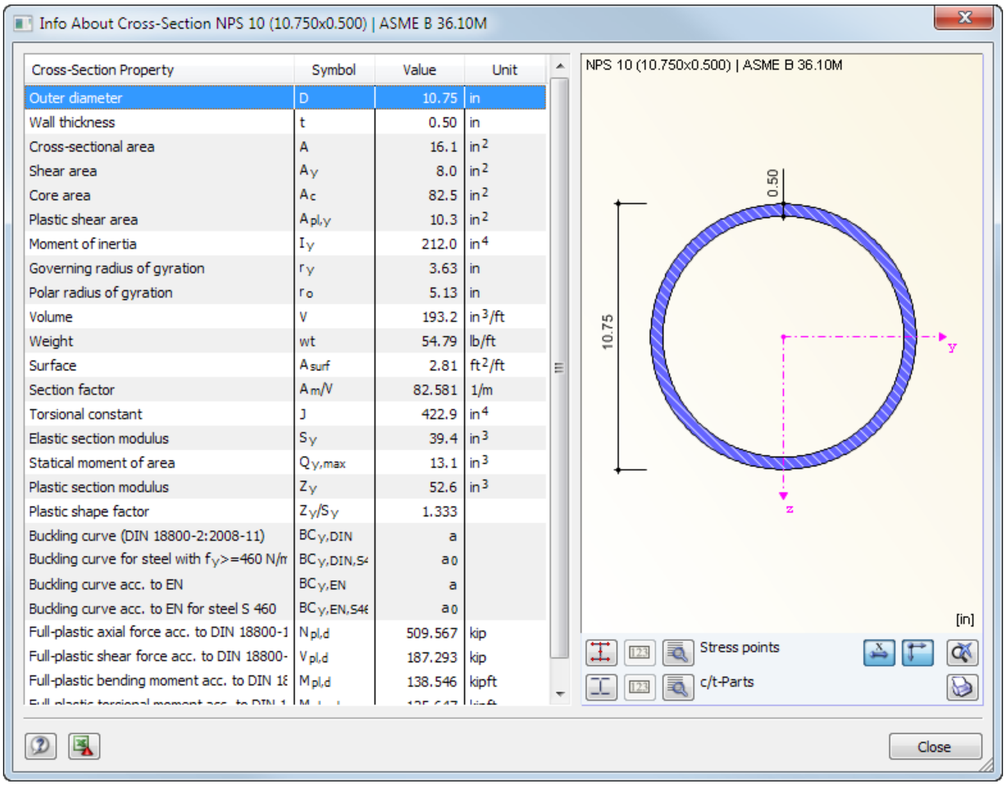


Figure 3.8: Dialog box *Info About Cross-Section*

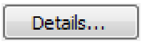
The buttons below the cross-section graphic have the following functions:

Button	Function
	Displays or hides the stress points
	Displays or hides the c/t-parts
	Displays or hides the numbers of stress points or c/t-parts
	Displays the details of the stress points or c/t-parts
	Displays or hides the dimensions of the cross-section
	Displays or hides the main axes of the cross-section
	Resets the full view of the cross-section graphic
	Prints the cross-sections properties and graphic

Table 3.2: Buttons of the cross-section graphic

3.3 Calculation

3.3.1 Detail Settings



Before starting the calculation, you should check the design details. To open the corresponding dialog box, use the [Details] button available in every window of the add-on module. The content of this dialog box varies depending on the design standard. The following description refers to the detail settings for ASME B31.1 [1].

The *Details* dialog box contains the following tabs:

- Details
- Used Literature

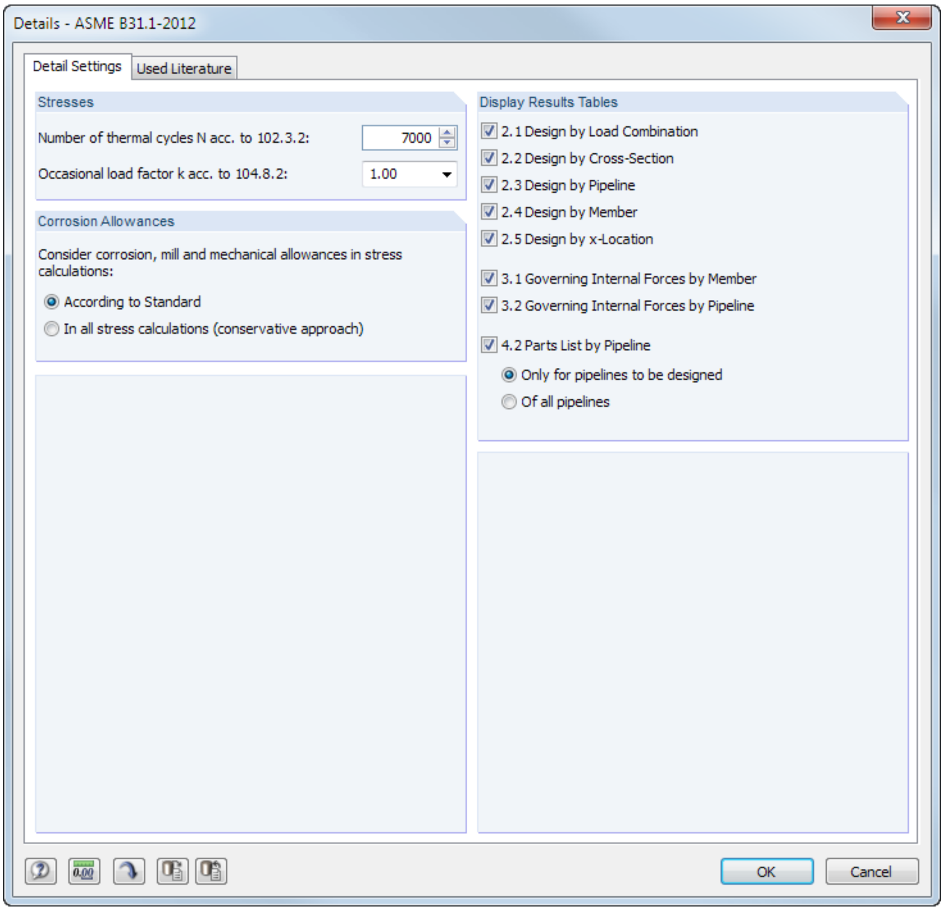


Figure 3.9: Dialog box *Details*, tab *Details*

Stresses

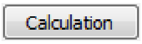
The limit stresses based on the designs may vary. They are, amongst other things, dependent on two parameters which you can define manually in this dialog section:

- *Number of thermal cycles N* according to [1] sec. 102.3.2
- *Occasional load factor k* according to [1] sec. 104.8.2

Display Result Tables

In this dialog section, you can select the result tables including parts list that you want to display. These tables are described in Chapter 3.4 from page 46.

3.3.2 Starting Calculation



To start the calculation, click the [Calculation] button that is available in all input tables of the RF-PIPING Design add-on module.

RF-PIPING Design searches for the results of the piping combinations and result combinations selected for design. If no results can be found, the program starts the RFEM calculation to determine the design-relevant internal forces.

You can also start the calculation in the RFEM user interface: The *To calculate* dialog box (menu **Calculation** → **To Calculate**) includes, amongst other things, design cases of the add-on modules.

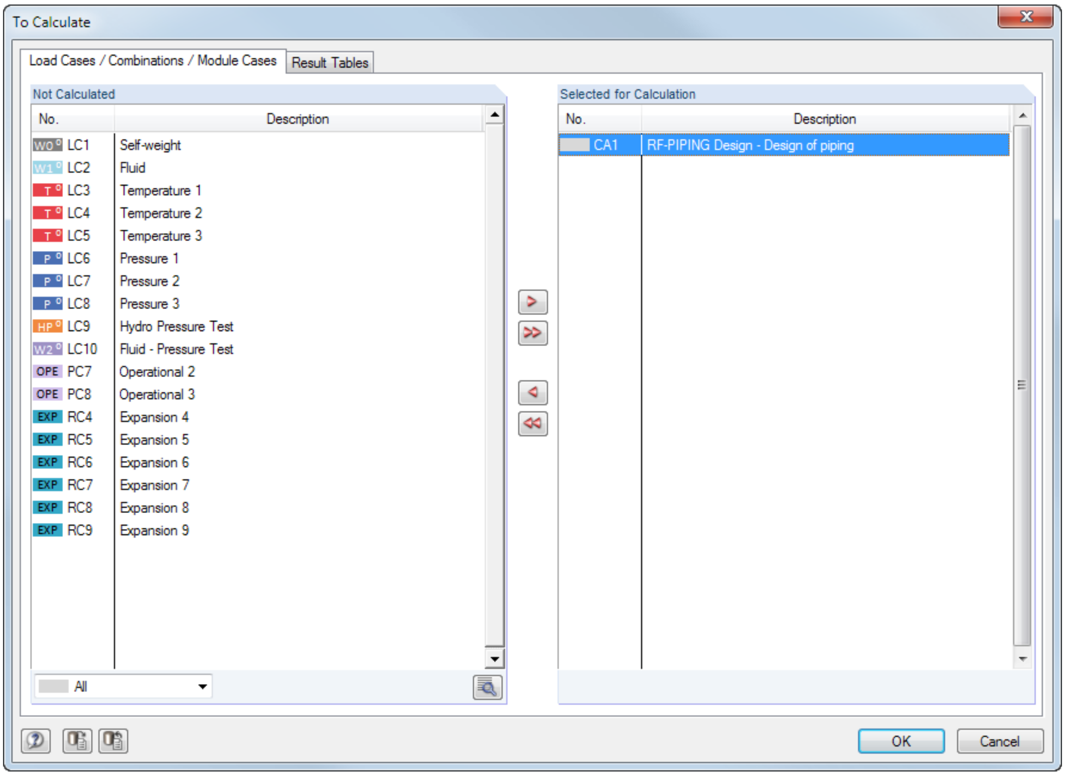
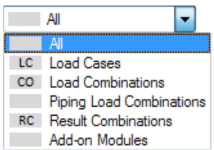


Figure 3.10: Dialog box *To Calculate*



If the RF-PIPING Design cases are missing in the *Not Calculated* list, select *All* or *Add-on Modules* in the drop-down list at the end of the dialog section.

To transfer the selected RF-PIPING Design cases to the section on the right, click [▷]. Click [OK] to start the calculation.

Subsequently, you can observe the design process in a separate dialog box.

3.4 Results

Immediately after the calculation, the Table *2.1 Design by Load Combination* is displayed.

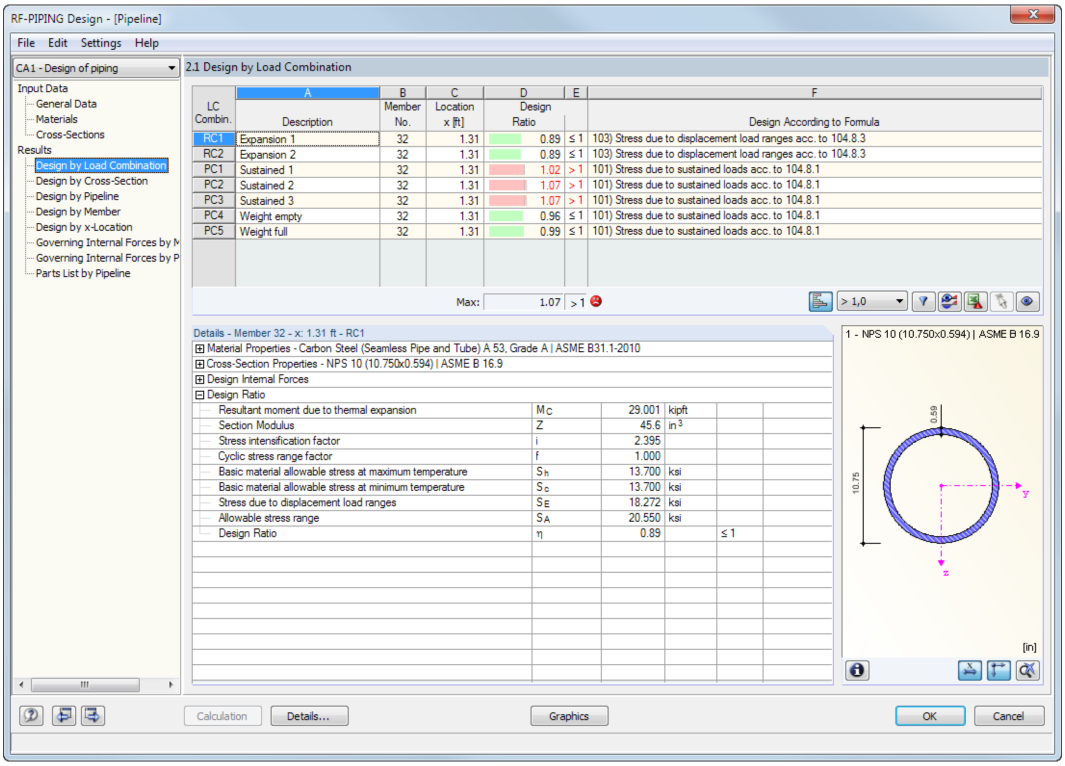
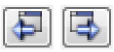


Figure 3.11: Result table with designs and details

The designs are sorted in the result tables 2.1 to 2.5 by different criteria. The Tables 3.1 and 3.2 display the governing internal forces, the Table 4.2 displays the parts list.



Each table can be selected by clicking the corresponding entry in the navigator. To go to the previous or next table, use the buttons shown on the left. Alternatively, you can use the function keys to select the next [F2] or previous [F3] table.

OK

Click [OK] to save the results. You exit RF-PIPING Design and return to RFEM.

The buttons below the table have the following functions:

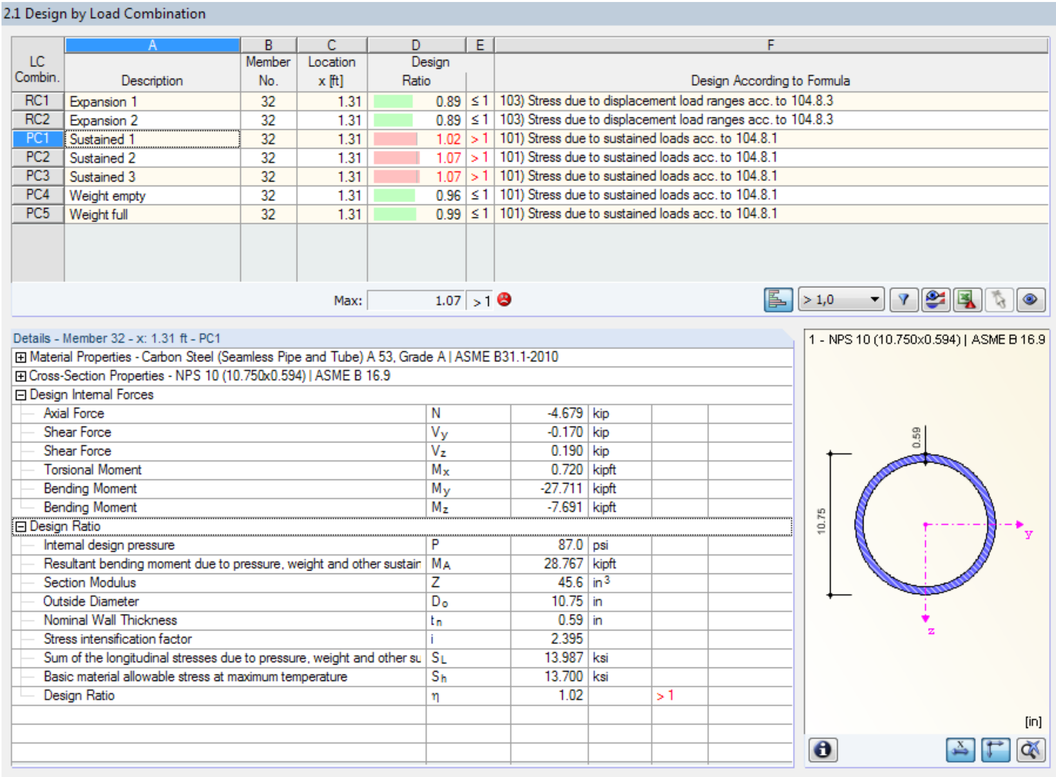
Button	Description	Function
	Color bar	Displays or hides the colored reference scales in the result tables
> 1,0 > 1,0 Max Define	Filter parameters	Represents the criterion for filtering the results in tables: Design ratios greater than 1, maximum value, or user-defined limit
	Apply filter	Shows only rows where the filter parameters are valid (ratio > 1, maximum, user-defined value)
	Result diagrams	Opens the dialog box *Result Diagram on Member* → Chapter 3.5.2, page 57
	Member selection	Allows for the graphical selection of a member to display its results in the table
	View mode	Goes to the RFEM work window for changing the view

Table 3.3: Buttons in the result tables 2.1 to 2.5

3.4.1 Design by Load Combination

The upper part of the table shows a summary of the governing checks sorted by piping combinations and result combinations.

The lower part provides detailed information about the cross-section properties, design internal forces, and check parameters for the combination selected above.


2.1 Design by Load Combination

LC Combin.	A Description	B Member No.	C Location x [ft]	D Design Ratio	E	F Design According to Formula
RC1	Expansion 1	32	1.31	0.89	≤ 1	103) Stress due to displacement load ranges acc. to 104.8.3
RC2	Expansion 2	32	1.31	0.89	≤ 1	103) Stress due to displacement load ranges acc. to 104.8.3
PC1	Sustained 1	32	1.31	1.02	> 1	101) Stress due to sustained loads acc. to 104.8.1
PC2	Sustained 2	32	1.31	1.07	> 1	101) Stress due to sustained loads acc. to 104.8.1
PC3	Sustained 3	32	1.31	1.07	> 1	101) Stress due to sustained loads acc. to 104.8.1
PC4	Weight empty	32	1.31	0.96	≤ 1	101) Stress due to sustained loads acc. to 104.8.1
PC5	Weight full	32	1.31	0.99	≤ 1	101) Stress due to sustained loads acc. to 104.8.1
			Max:	1.07	> 1	

Details - Member 32 - x: 1.31 ft - PC1

⊞ Material Properties - Carbon Steel (Seamless Pipe and Tube) A 53, Grade A | ASME B31.1-2010				
⊞ Cross-Section Properties - NPS 10 (10.750x0.594) | ASME B 16.9				
⊟ Design Internal Forces				
Axial Force	N	-4.679	kip	
Shear Force	Vy	-0.170	kip	
Shear Force	Vz	0.190	kip	
Torsional Moment	Mx	0.720	kipft	
Bending Moment	My	-27.711	kipft	
Bending Moment	Mz	-7.691	kipft	
⊟ Design Ratio				
Internal design pressure	P	87.0	psi	
Resultant bending moment due to pressure, weight and other sustair	MA	28.767	kipft	
Section Modulus	Z	45.6	in³	
Outside Diameter	Do	10.75	in	
Nominal Wall Thickness	tn	0.59	in	
Stress intensification factor	i	2.395		
Sum of the longitudinal stresses due to pressure, weight and other su	SL	13.987	ksi	
Basic material allowable stress at maximum temperature	Sh	13.700	ksi	
Design Ratio	η	1.02		> 1

Figure 3.12: Table *2.1 Design by Load Combination*

Description

This column shows the descriptions of the piping and result combinations used for the designs.

Member No.

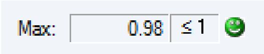
This column shows the number of the member with the maximum design ratio of the analyzed action.

Location

This column shows the respective x-location where the member's maximum stress ratio occurs. For the table output, the program uses the following member locations x:

- Start and end node
- Division points according to possibly defined member division (see RFEM Table 1.16)
- Member division according to specification for member results (*Calculation Parameters* RFEM dialog box, *Global Calculation Parameters* tab)
- Extreme values of internal forces

Design Ratio



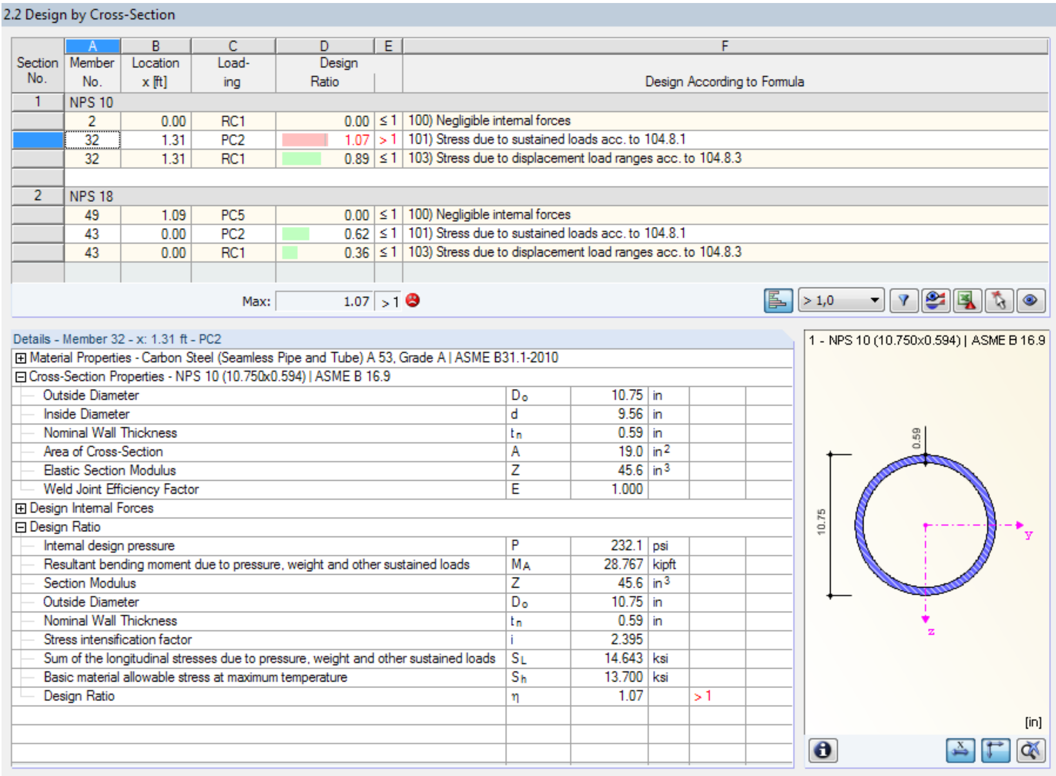
Columns D and E display the design conditions with the ratios which result from the design according to the set standard.

The length of the color scale displays graphically the respective ratio.

Design According to Formula

This column displays more information about the performed design.

3.4.2 Design by Cross-Section



2.2 Design by Cross-Section

Section No.	A Member No.	B Location x [ft]	C Loading	D Design Ratio	E	F Design According to Formula
1	NPS 10					
	2	0.00	RC1	0.00	≤ 1	100) Negligible internal forces
	32	1.31	PC2	1.07	> 1	101) Stress due to sustained loads acc. to 104.8.1
	32	1.31	RC1	0.89	≤ 1	103) Stress due to displacement load ranges acc. to 104.8.3
2	NPS 18					
	49	1.09	PC5	0.00	≤ 1	100) Negligible internal forces
	43	0.00	PC2	0.62	≤ 1	101) Stress due to sustained loads acc. to 104.8.1
	43	0.00	RC1	0.36	≤ 1	103) Stress due to displacement load ranges acc. to 104.8.3
			Max:	1.07	> 1	

Details - Member 32 - x: 1.31 ft - PC2

⊞ Material Properties - Carbon Steel (Seamless Pipe and Tube) A 53, Grade A | ASME B31.1-2010				
⊟ Cross-Section Properties - NPS 10 (10.750x0.594) | ASME B 16.9				
Outside Diameter	D_o	10.75	in	
Inside Diameter	d	9.56	in	
Nominal Wall Thickness	t_n	0.59	in	
Area of Cross-Section	A	19.0	in^2	
Elastic Section Modulus	Z	45.6	in^3	
Weld Joint Efficiency Factor	E	1.000		
⊞ Design Internal Forces				
⊟ Design Ratio				
Internal design pressure	P	232.1	psi	
Resultant bending moment due to pressure, weight and other sustained loads	M_A	28.767	kipft	
Section Modulus	Z	45.6	in^3	
Outside Diameter	D_o	10.75	in	
Nominal Wall Thickness	t_n	0.59	in	
Stress intensification factor	i	2.395		
Sum of the longitudinal stresses due to pressure, weight and other sustained loads	S_L	14.643	ksi	
Basic material allowable stress at maximum temperature	S_h	13.700	ksi	
Design Ratio	η	1.07		> 1

Figure 3.13: Table *2.2 Design by Cross-Section*

This table shows the maximum design ratios of all pipelines selected for design, sorted by cross-section.

The *Member No.* column shows the number of the member with the maximum design ratio for the individual design criteria.

3.4.3 Design by Pipeline

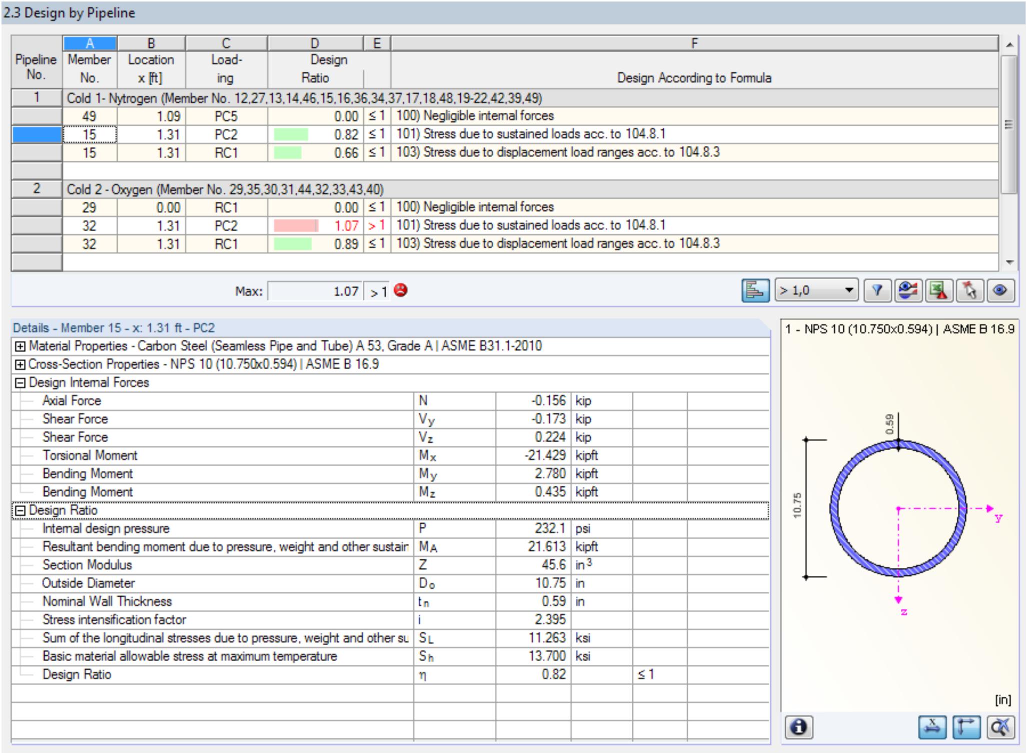


Figure 3.14: Table 2.3 *Design by Pipeline*

This result table lists the maximum design ratios sorted by pipelines.

3.4.4 Design by Member

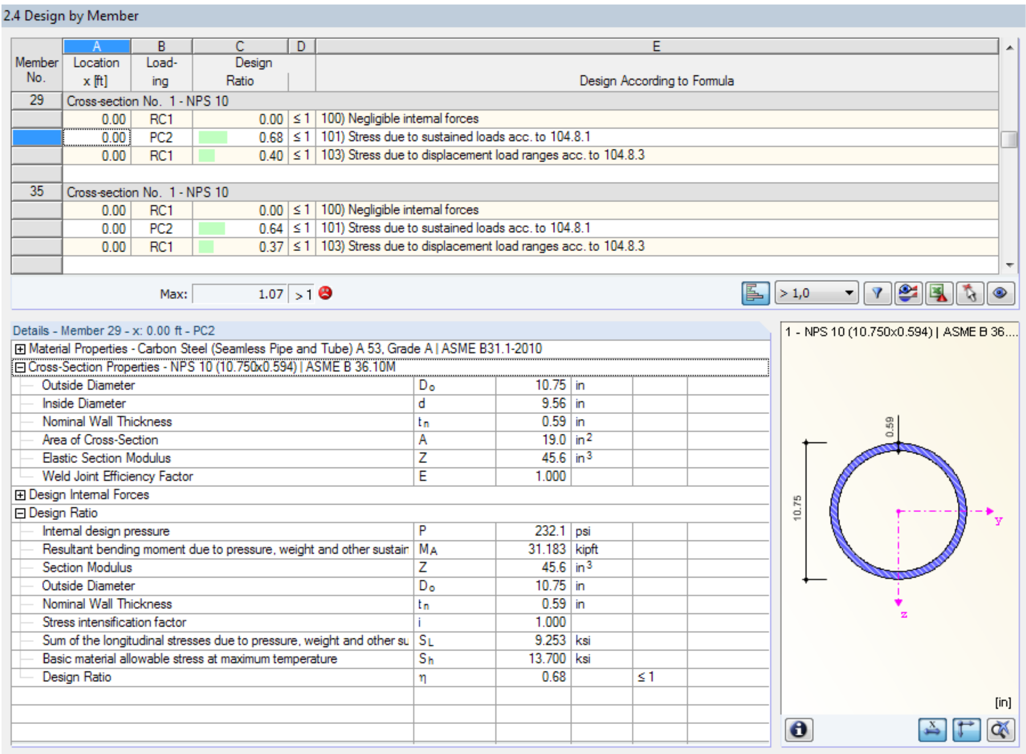


Figure 3.15: Table 2.4 *Design by Member*

The maximum design ratios are displayed sorted by member number.

3.4.5 Design by x-Location

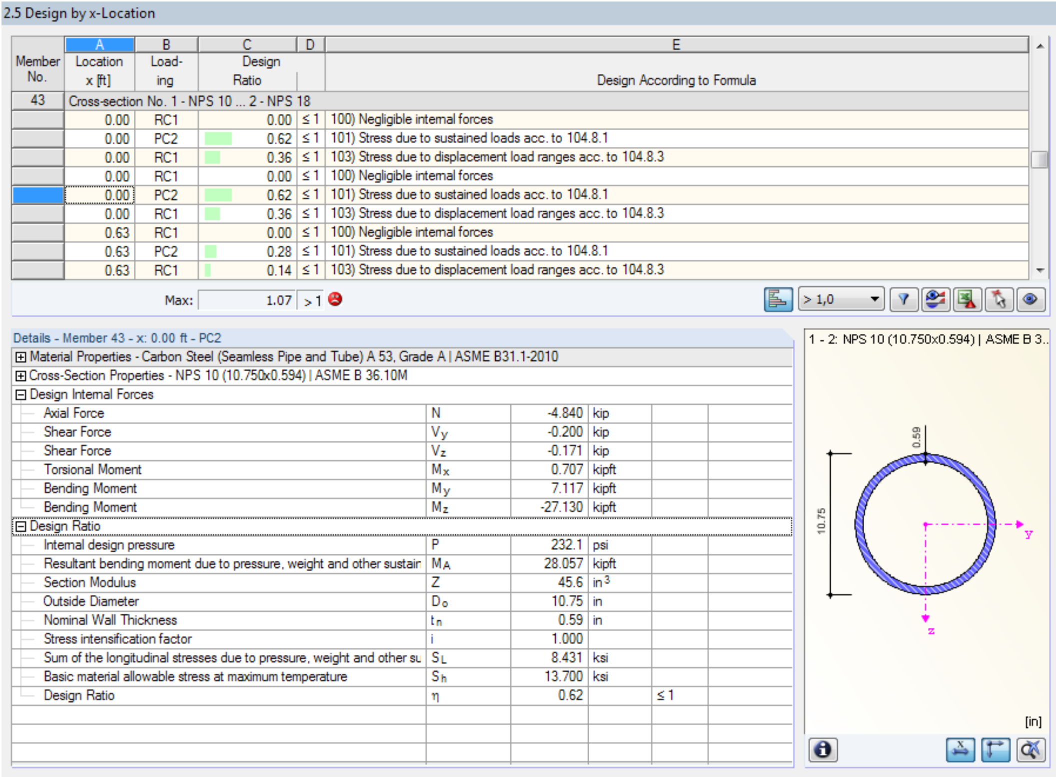


Figure 3.16: Table *2.5 Design by x-Location*

This result table lists the maxima for each member at any locations **x** resulting from the division points in RFEM:

- Start and end node
- Division points according to possibly defined member division (see RFEM Table 1.16)
- Member division according to specification for member results (RFEM dialog box *Calculation Parameters*, tab *Global Calculation Parameters*)
- Extreme values of internal forces

3.4.6 Governing Internal Forces by Member

3.1 Governing Internal Forces by Member

	A	B	C	D	E	F	G	H	I
Member No.	Location x [ft]	Load-ing	Forces [kip] N	Vy	Vz	Moments [kipft] MT	My	Mz	Design According to Formula
12	Cross-section No. 1 - NPS 10								
	0.00	RC1	0.000	0.000	0.000	0.000	0.000	0.000	100) Negligible internal forces
	0.00	PC2	-0.157	-0.152	1.858	-15.359	-10.052	0.275	101) Stress due to sustained loads acc. to 104.8.1
	0.00	RC1	-0.327	-0.222	1.839	-15.440	-9.905	0.871	103) Stress due to displacement load ranges acc. to 104.8.3
27	Cross-section No. 1 - NPS 10								
	0.00	RC1	0.000	0.000	0.000	0.000	0.000	0.000	100) Negligible internal forces
	0.00	PC2	-0.156	-0.157	1.455	-15.360	-6.791	0.575	101) Stress due to sustained loads acc. to 104.8.1
	0.00	RC1	-0.326	-0.227	1.436	-15.442	-6.681	1.308	103) Stress due to displacement load ranges acc. to 104.8.3
13	Cross-section No. 1 - NPS 10								
	0.00	RC1	0.000	0.000	0.000	0.000	0.000	0.000	100) Negligible internal forces
	1.31	PC2	-0.169	0.157	0.879	2.523	16.275	1.706	101) Stress due to sustained loads acc. to 104.8.1
	1.31	RC1	-0.239	0.328	0.860	2.484	16.344	2.845	103) Stress due to displacement load ranges acc. to 104.8.3
14	Cross-section No. 1 - NPS 10								
	0.00	RC1	0.000	0.000	0.000	0.000	0.000	0.000	100) Negligible internal forces
	3.39	PC2	-0.169	0.155	0.645	2.522	18.705	1.187	101) Stress due to sustained loads acc. to 104.8.1
	3.39	RC1	-0.238	0.325	0.626	2.481	18.708	1.751	103) Stress due to displacement load ranges acc. to 104.8.3
46	Cross-section No. 1 - NPS 10								
	0.00	RC1	0.000	0.000	0.000	0.000	0.000	0.000	100) Negligible internal forces
	4.79	PC2	-0.171	0.155	0.315	2.520	21.004	0.443	101) Stress due to sustained loads acc. to 104.8.1
	4.79	RC1	0.171	-0.155	-0.315	-2.520	-21.004	-0.443	103) Stress due to displacement load ranges acc. to 104.8.3
15	Cross-section No. 1 - NPS 10								
	0.00	RC1	0.000	0.000	0.000	0.000	0.000	0.000	100) Negligible internal forces
	1.31	PC2	-0.156	-0.173	0.224	-21.429	2.780	0.435	101) Stress due to sustained loads acc. to 104.8.1
	1.31	RC1	0.156	0.173	-0.224	21.429	-2.780	-0.435	103) Stress due to displacement load ranges acc. to 104.8.3
16	Cross-section No. 1 - NPS 10								
	0.00	RC1	0.000	0.000	0.000	0.000	0.000	0.000	100) Negligible internal forces
	1.74	PC2	-0.154	-0.172	0.104	-21.240	3.020	0.736	101) Stress due to sustained loads acc. to 104.8.1
	1.74	RC1	0.154	0.172	-0.104	21.240	-3.020	-0.736	103) Stress due to displacement load ranges acc. to 104.8.3

Figure 3.17: Table *3.1 Governing Internal Forces by Member*

This table shows for each member the governing internal forces - those internal forces which result in maximum design ratios at the individual designs.

Location x

This column shows the respective x-location where the member's maximum stress ratio occurs.

Loading

This column displays the number of the piping or result combination whose internal forces produce the maximum design ratio.

Forces / Moments

For each member, this column displays the axial and shear forces as well as the torsional and bending moments producing the maximum design ratio in the respective designs.

Design According to Formula

The final column provides information on the type of design and the formulas used in the designs according to the specified standard.

3.4.7 Governing Internal Forces by Pipeline

3.2 Governing Internal Forces by Pipeline

	A	B	C	D	E	F	G	H	I
Pipeline No.	Location x [ft]	Load-ing	Forces [kip] N	V_y	V_z	Moments [kipft] M_T	M_y	M_z	Design According to Formula
1	Cold 1- Nytrogen (Member No. 12,27,13,14,46,15,16,36,34,37,17,18,48,19-22,42,39,49)								
	1.09	PC5	-0.005	0.000	0.810	0.000	-1.205	0.000	100) Negligible internal forces
	1.31	PC2	-0.156	-0.173	0.224	-21.429	2.780	0.435	101) Stress due to sustained loads acc. to 104.8.1
	1.31	RC1	0.156	0.173	-0.224	21.429	-2.780	-0.435	103) Stress due to displacement load ranges acc. to 104.8.3
2	Cold 2 - Oxygen (Member No. 29,35,30,31,44,32,33,43,40)								
	0.00	RC1	0.000	0.000	0.000	0.000	0.000	0.000	100) Negligible internal forces
	1.31	PC2	-4.679	-0.170	0.190	0.720	-27.711	-7.691	101) Stress due to sustained loads acc. to 104.8.1
	1.31	RC1	-4.701	-0.239	0.286	0.910	-27.862	-7.995	103) Stress due to displacement load ranges acc. to 104.8.3
3	Hot (Member No. 54,53,56,55,52-50,2-4,41,6,57,8,9,24,25,10,11,26,23,28)								
	0.00	RC1	0.000	0.000	0.000	0.000	0.000	0.000	100) Negligible internal forces
	0.00	PC2	0.182	-0.165	2.512	-24.797	-0.104	0.761	101) Stress due to sustained loads acc. to 104.8.1
	0.00	RC1	0.252	-0.337	2.507	-24.899	-0.216	0.303	103) Stress due to displacement load ranges acc. to 104.8.3

Figure 3.18: Table *3.2 Governing Internal Forces by Pipeline*

This table shows the internal forces of each pipeline which result in the maximum design ratios for the individual designs.

3.4.8 Parts List by Pipeline

Finally, RF-PIPING Design provides a summary of all cross-sections that are included in the design case.

4.2 Parts List by Pipeline

	A	B	C	D	E	F	G	H	I
Part No.	Pipeline Description	Number of Pipelines	Length [ft]	Total Length [ft]	Surface Area [ft²]	Volume [ft³]	Unit Weight [lb/ft]	Weight [lb]	Total Weight [kip]
1	Cold 1- Nytrogen	1	57.27	57.27	182.63	11.21	95.96	5495.11	0.56
2	Cold 2 - Oxygen	1	28.81	28.81	91.40	5.56	94.54	2723.80	0.28
3	Hot	1	81.29	81.29	264.92	16.89	101.81	8276.31	0.84
Sum		3		167.37	538.94	33.66			1.68

Figure 3.19: Table *4.1 Parts List by Pipeline*

Details...

By default, this list contains only the designed pipelines. If you need a parts list for all pipelines of the model, select the corresponding option in the *Details* dialog box (see Figure 3.9, page 44).

Part No.

The program automatically assigns part numbers to similar pipelines.

Pipeline Description

This column shows the pipeline descriptions.

Number of Pipelines

This column shows for each part how many similar pipelines exist.

Length

This column displays the length of a single pipeline.

Total Length

The values in this column are the product from the previous two columns.

Surface Area

For each part, the program indicates the surface areas relative to the total length. They are determined from the *surface area* of the cross-sections that can be viewed in the Tables 1.3 and 2.1 to 2.5 in the cross-section properties (see Figure 3.8, page 43).

Volume

The volume of a part is determined from the cross-sectional area and the total length.

Unit Weight

The unit weight is the weight of the section relative to the length of one meter.

Weight

The values of this column are determined from the respective product of the entries in column C and G.

Total Weight

The final column indicates the total weight of each part.

Sum

At the bottom of the list, you find a summary of the values in columns B, D, E, F, and I. The last row of the *Total Weight* column informs you about the total amount of required steel.

3.5 Evaluation of Results

You can evaluate the design results in different ways. The buttons in the Tables 2.1 to 2.5 may help you to evaluate the results. You can find them below the upper table.

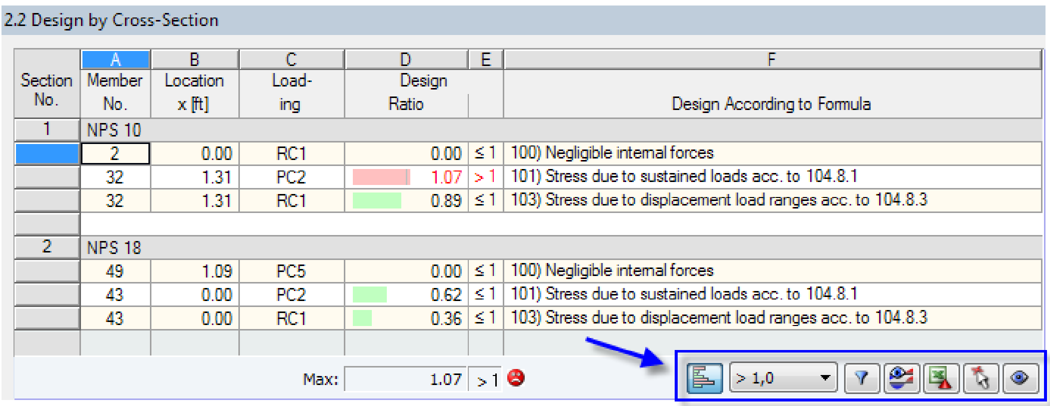


Figure 3.20: Buttons for evaluation of results

The functions of the buttons are described in Table 3.3 on page 46.

3.5.1 Results on RFEM Model

You can also evaluate the design results in the RFEM work window.

RFEM Background Graphic and View Mode

The RFEM graphic in the background may be useful when you want to check the position of a particular pipeline in the model: The member selected in the RF-PIPING Design result table is highlighted in color in the background graphic. Furthermore, an arrow indicates the member's x-location displayed in the currently selected table row.

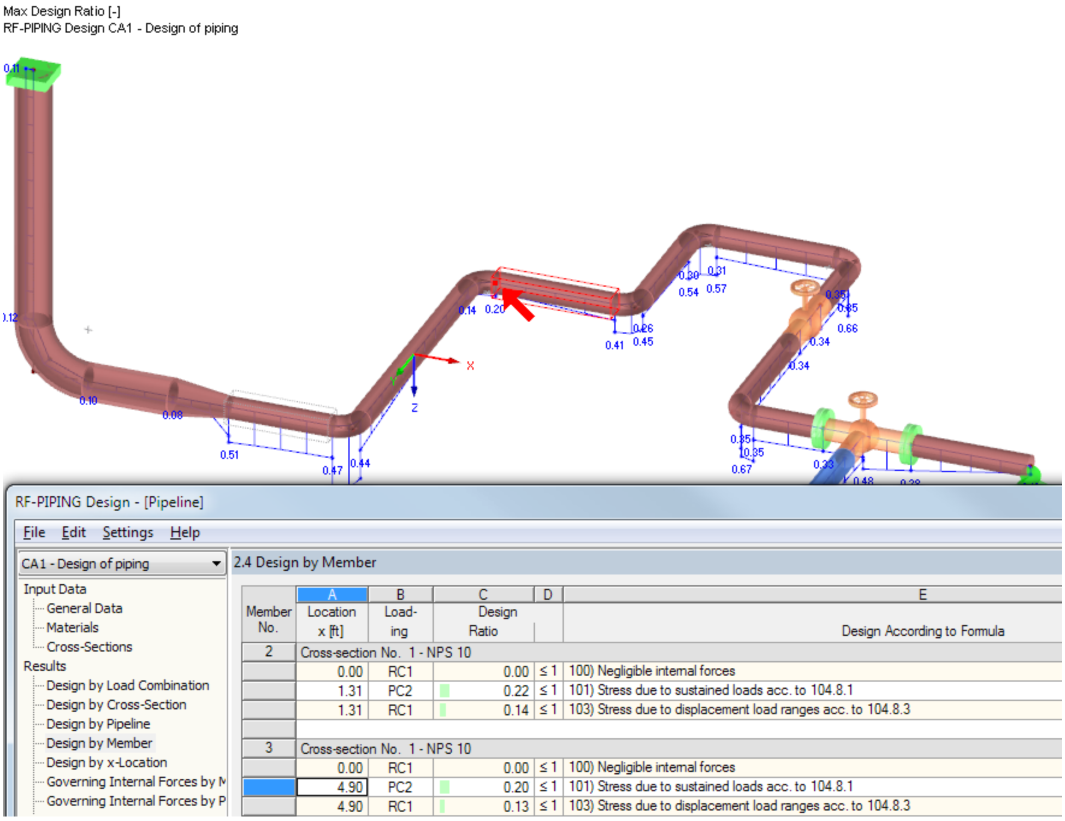
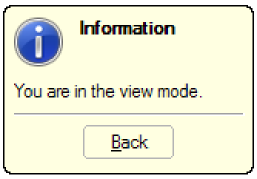


Figure 3.21: Indication of the member and the current *Location x* in the RFEM model



If you cannot improve the display by moving the module window, click [Jump to Graphic] to activate the *View Mode*: The program hides the module window so that you can modify the display in the RFEM user interface. In the view mode, you can use the functions of the *View* menu, for example zooming, moving, or rotating the display. The indication arrow remains visible.

Click [Back] to return to the add-on module RF-PIPING Design.

RFEM Work Window

The design ratios can also be checked graphically in the RFEM model: Click the [Graphics] button to exit the design module. In the RFEM work window, the design ratios are now displayed, such as the internal forces of a load case.

To display or hide design results, use the [Show Results] button known from the display of internal forces. To display the result values, click the [Show Result Values] button in the toolbar to the right.

To set the design cases (see Chapter 3.7.1, page 59), you can use the list in the RFEM menu bar.

To adjust the results display, use the *Display* navigator below the entry **Results** → **Members**. The display of the design ratios is *Two-Colored* by default.

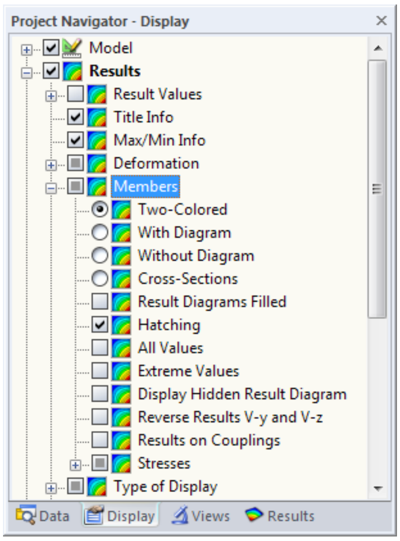


Figure 3.22: *Display* Navigator: *Results* → *Members*

When you select a multicolor representation (options *With/Without Diagram* or *Cross-Sections*), the color scale panel becomes available. It provides the common control functions described in the RFEM manual, Chapter 3.4.6.

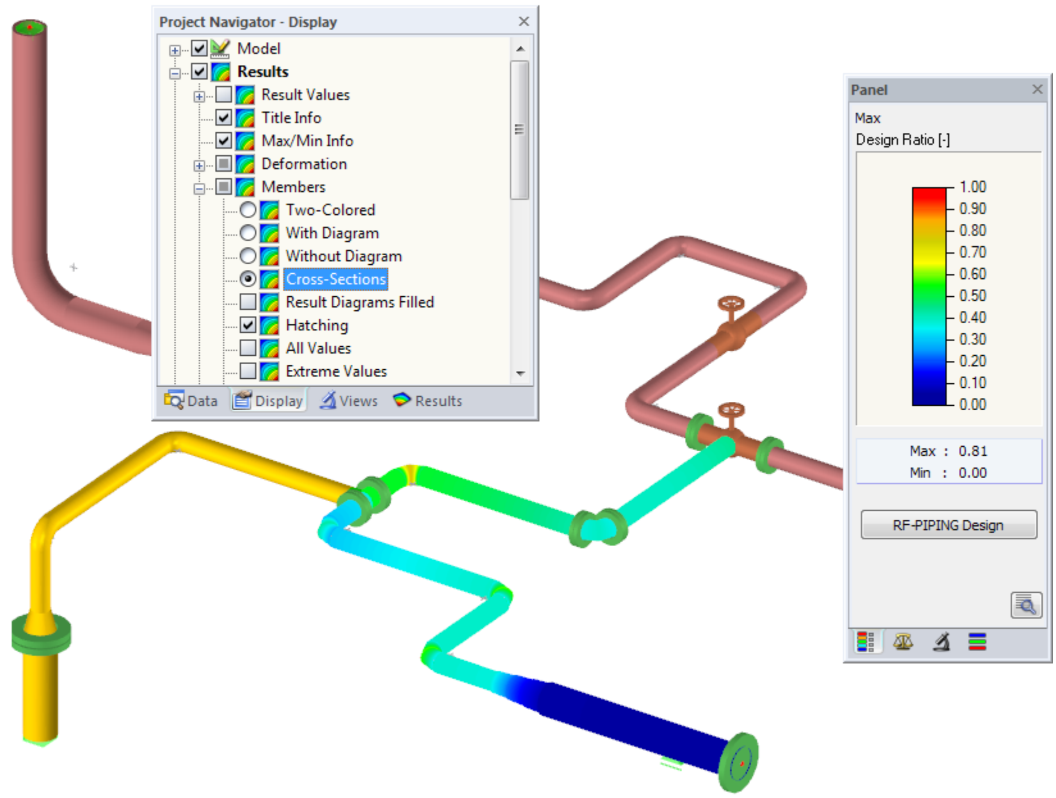


Figure 3.23: Design ratios with display option *Cross-Sections*

RF-PIPING Design

To return to the add-on module, click [RF-PIPING Design] in the panel.

3.5.2 Result Diagrams

You can also graphically evaluate a member's result distributions in a result diagram.

Select the member in the RF-PIPING Design result table by clicking in the table row of the member. Then, open the *Result Diagram on Member* dialog box by clicking the button shown on the left. You can find the button below the upper result table (see Figure 3.20, page 54).

To display the result diagrams in the RFEM graphic, select in the menu

Results → Result Diagrams for Selected Members

or use the corresponding button in the RFEM toolbar.

A window opens, graphically presenting the distribution of the maximum design ratios on the member.

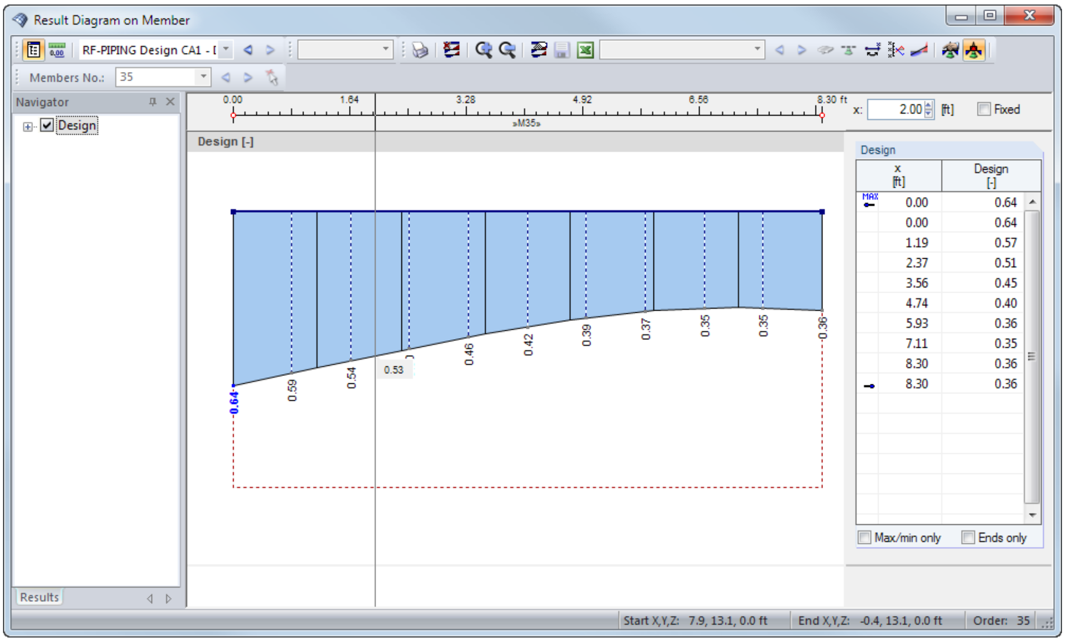
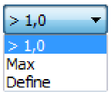


Figure 3.24: Dialog box *Result Diagram on Member*

Use the list in the toolbar to choose the relevant RF-PIPING Design design case (see Chapter 3.7.1, page 59).

The *Result Diagram on Member* dialog box is described in the RFEM manual, Chapter 9.5.

3.6 Filter for Results



The RF-PIPING Design result tables allow you to sort the results by various criteria. In addition, you can filter options for the tables (see Figure 3.20, page 54) to limit the numerical output by design ratios. This function is described in the following article of the DLUBAL Blog:
https://www.dlubal.com/blog/11217

Furthermore, you can use the filter options described in Chapter 9.9 of the RFEM manual in order to evaluate the results graphically.

In RF-PIPING Design, you can also use the *Visibility* option to filter the members in order to evaluate them (see RFEM manual, Chapter 9.9.1).

Filtering Designs

The design ratios can easily be used as filter criteria in the RFEM work window which you can access by clicking [Graphics]. To apply this filter function, the panel must be displayed. If the panel is not active, select in the RFEM menu

View → Control Panel (Colour scale → Factors → Filter)

or use the toolbar button shown on the left.

You define the filter settings for the results in the first panel tab (Color spectrum). Because this tab is not available for the two-colored results display, you have to use the *Display* navigator and set the display options *With/Without Diagram* or *Cross-Sections* first (see Figure 3.22, page 56).

The panel is described in the RFEM manual, Chapter 3.4.6. In Chapter 9.9.3, you can find detailed information about filter options for result values.

Filtering Members

In the *Filter* tab of the control panel, you can specify the numbers of particular members to display only those results. This function is described in the RFEM manual, Chapter 9.9.3.

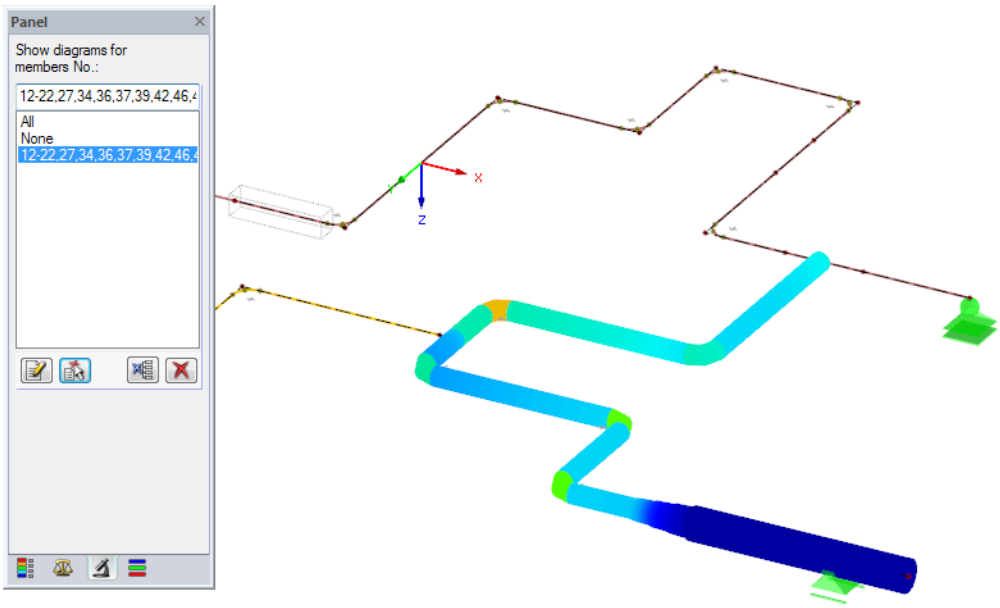


Figure 3.25: Member filter for the design ratios of a pipeline

Unlike the partial view function (Visibilities), the graphic displays the entire pipeline model. The figure above shows the design ratios of a pipeline. The remaining pipelines are displayed in the model, but are shown without design ratios.

3.7 General Functions

3.7.1 Design Cases

Design cases allow you to group pipelines for a design or analyze members with particular design specifications. It is no problem to analyze the same pipeline in different design cases.

To calculate a design case, you can also use the load case list in the RFEM toolbar.

Create New Design Case

To create a new design case, use the RF-PIPING Design menu and click

File → **New Case**.

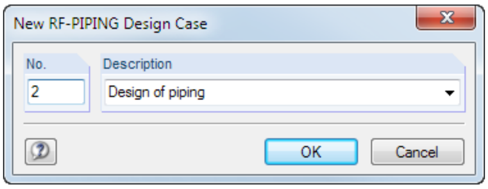


Figure 3.26: Dialog box *New RF-PIPING Design Case*

In this dialog box, enter a *No.* (one that is still available) for the new design case. An appropriate *Description* will make the selection in the load case list easier.

When you click [OK], Table *1.1 General Data* opens where you can enter the new design data.

Rename Design Case

To change the description of a design case, use the RF-PIPING Design menu and click

File → **Rename Case**.

Copy Design Case

To copy the input data of the current design case, use the RF-PIPING Design menu and click

File → **Copy Case**.

Delete Design Case

To delete design cases, use the RF-PIPING Design menu and click

File → **Delete Case**.

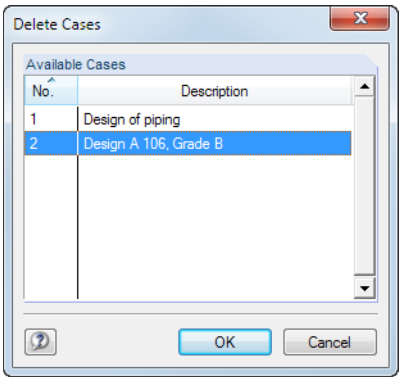


Figure 3.27: Dialog box *Delete Cases*

You can select the relevant design case in the list of *Available Cases*. To delete the selected case, click [OK].

3.7.2 Units and Decimal Places

Units and decimal places for RFEM and the add-on modules are managed in one dialog box. In RF-PIPING Design, you can use the menu to adjust the units. To open the corresponding dialog box, click

Settings → Units and Decimal Places.

The following dialog box appears which you already know from RFEM. RF-PIPING Design is preset in the *Program / Module* list.

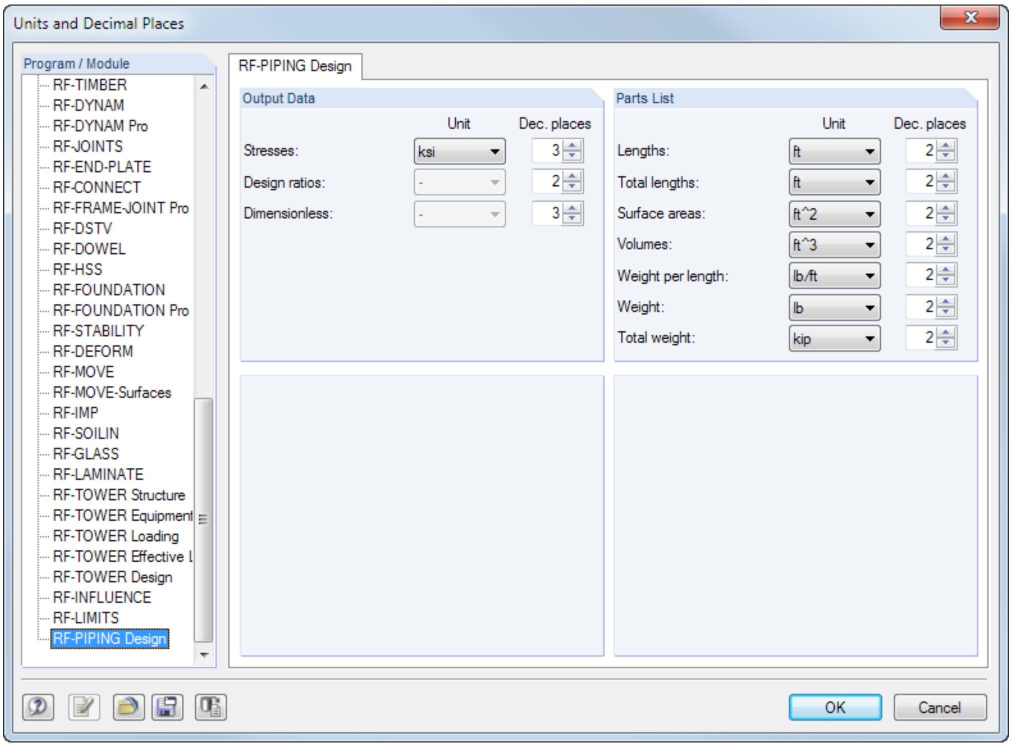


Figure 3.28: Dialog box *Units and Decimal Places*

You can save the settings as a user profile to reuse them in other models. These functions are described in Chapter 11.1.3 of the RFEM manual.

3.7.3 Data Transfer

3.7.3.1 Exporting Materials to RFEM

If you adjust the materials in RF-PIPING Design for design, you can export the modified materials to RFEM: To do this, go to the Table *1.2 Materials*, and then select the command in the menu

Edit → Export All Materials to RFEM.

Alternatively, you can export the modified materials to RFEM by using the shortcut menu in Table 1.2.

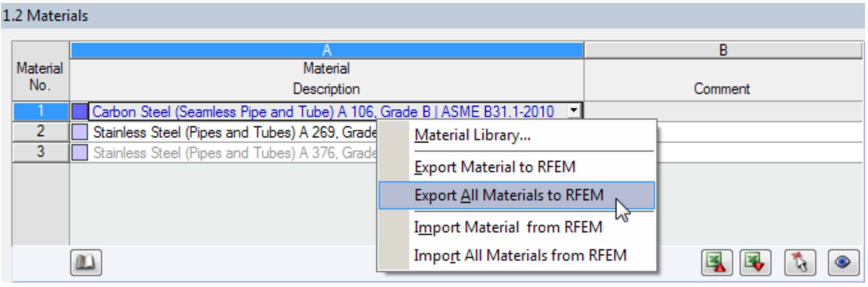
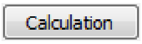


Figure 3.29: Shortcut menu of Table *1.2 Materials*



Before the modified materials are transferred to RFEM, a query appears as to whether you want to delete the results in RFEM. By confirming the query and starting the [Calculation] in the RF-PIPING Design module, the RFEM internal forces as well as the design ratios will be determined in a single calculation run.

If the modified materials have not been exported to RFEM yet, you can reimport the original materials in the design module by using the options shown in Figure 3.29. Please note that this option is only available in Table *1.2 Materials*.

3.7.3.2 Exporting Results

You can use the RF-PIPING Design results also in other programs.

Clipboard

To copy cells selected in the result tables to the Clipboard, use the keys [Ctrl]+[C]. Press [Ctrl]+[V] to insert the cells, for example in a word processing program. The headers of the table columns will not be transferred.

Printout Report

You can print the data of RF-PIPING Design into the printout report (see Chapter 4.1, page 62). To export them, click

File → **Export to RTF**.

The function is described in the RFEM manual, Chapter 10.1.11.

Excel / OpenOffice

RF-PIPING Design provides a function for the direct data export to MS Excel, OpenOffice Calc or the file format CSV. To open the corresponding dialog box, select in the module menu

File → **Export Tables**.

The following export dialog box appears:

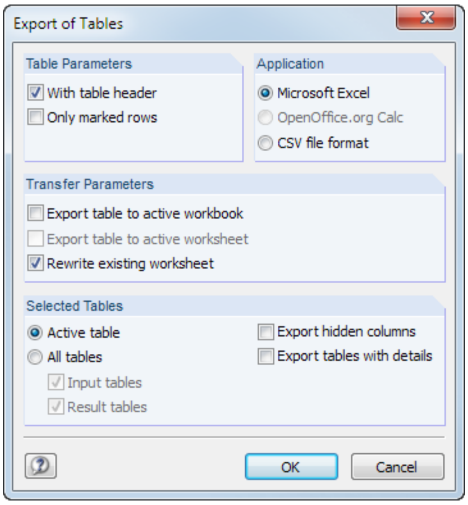


Figure 3.30: Dialog box *Export - Export of Tables*

When you have selected the relevant options, you can start the export by clicking [OK]. Excel or OpenOffice will be started automatically, you do not need to open the programs first.

4 Printout

4.1 Printout Report

Like in RFEM, the program generates a printout report for the RF-PIPING Design results, to which you can add graphics and descriptions. The selection in the printout report determines which data from the add-on modules will be included in the final printout.

The printout report is described in the RFEM manual. Chapter 10.1.3 *Define Contents of Printout Report* explains how to prepare the input and output data for the printout.

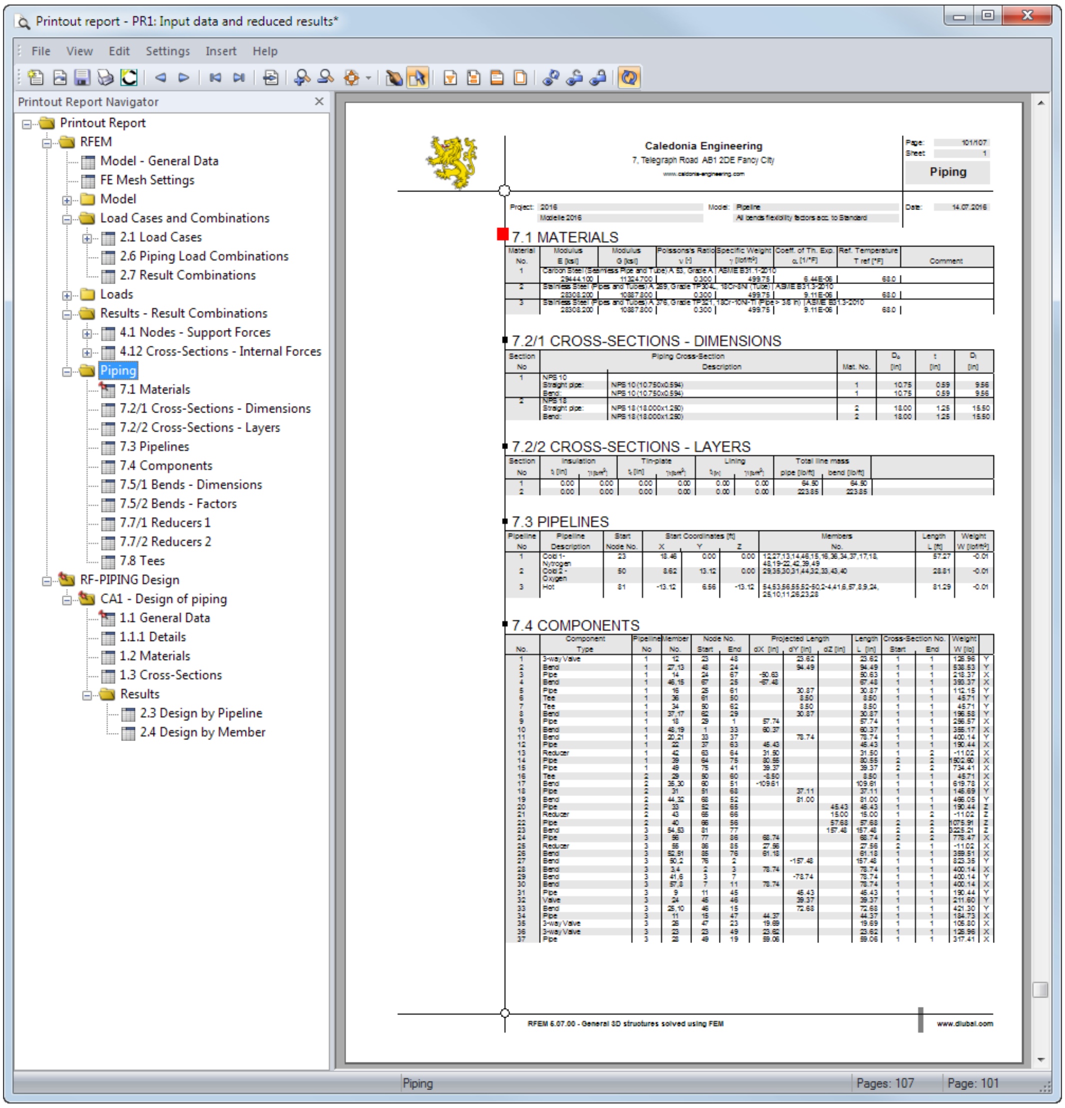


Figure 4.1: Printout Report for RF-PIPING and RF-PIPING Design

For large piping structures, it is recommended to split the data into several printout reports, thus allowing for a clearly-arranged printout.

4.2 Graphic Printout

In RFEM, you can add every picture displayed in the work window to the printout report or send it directly to a printer. In this way, you can also prepare the internal forces, deformations or design ratios displayed on the RFEM model for the printout.

To print the currently displayed graphic click

File → Print Graphic

or use the toolbar button shown on the left.

Also in the *Result Diagram* dialog box (see Figure 3.24, page 57), you can use the button to transfer the graphic of internal forces, deformations or design ratios to the printout report or print it directly.

The following dialog box appears:

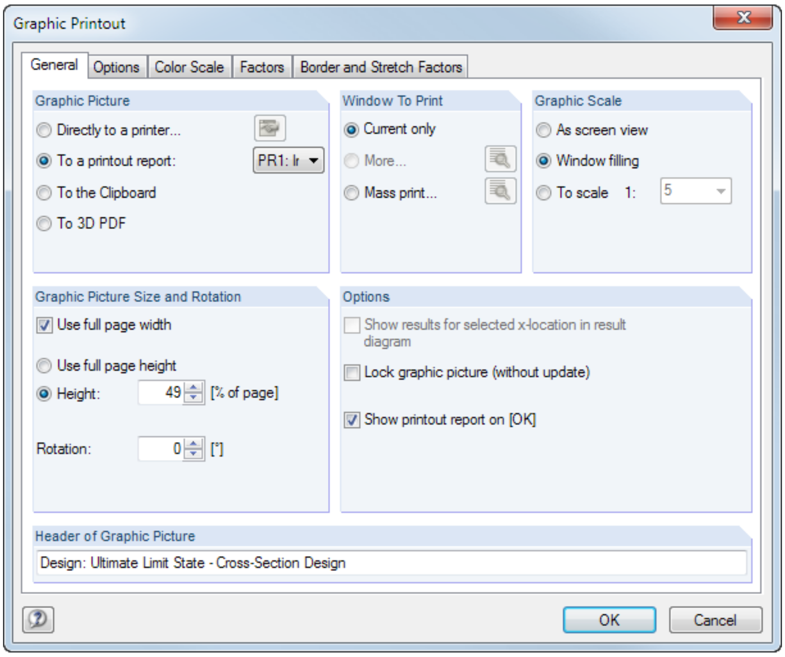


Figure 4.2: Dialog box *Graphic Printout*

Printing graphics is described in the RFEM manual, Chapter 10.2. In this chapter, the other tabs of the *Graphic Printout* dialog box are explained.

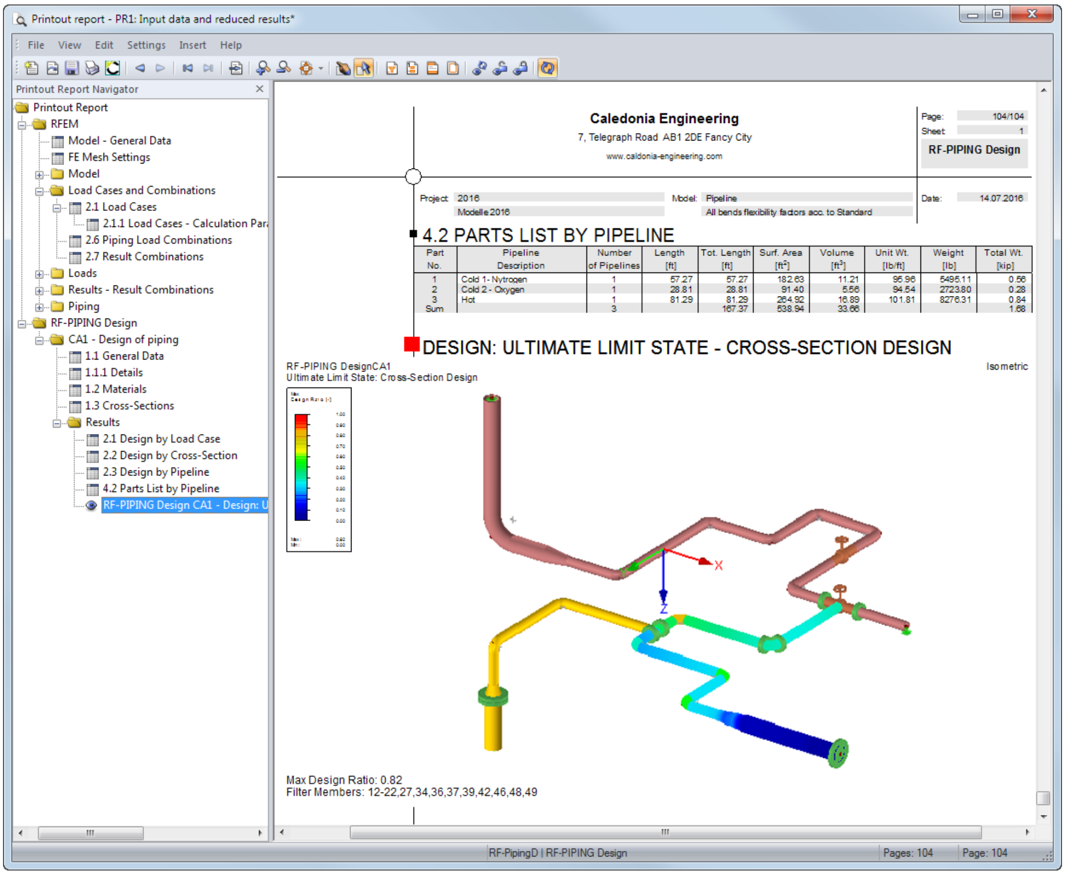
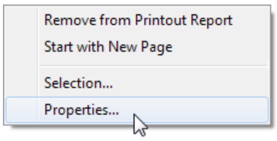


Figure 4.3: Printout report with pipeline graphic



To move a graphic within the printout report, use the drag-and-drop function. To adjust a graphic in the printout report at a later time, right-click the corresponding entry in the navigator of the printout report. The *Properties* option in the shortcut menu opens the *Graphic Printout* dialog box, offering various options for adjustment.

Literature

[1] *ASME B31.1-2012: Power Piping*. The American Society of Mechanical Engineers, 2012.

[2] *ASME B31.3-2012: Process Piping*. The American Society of Mechanical Engineers, 2013.

[3] *DIN EN 13480-3: Metallische industrielle Rohrleitungen – Teil 3: Konstruktion und Berechnung*. Beuth Verlag GmbH, 2014.

[4] *EN 10253-2: Formstücke zum Einschweißen – Teil 2: Unlegierte und legierte ferritische Stähle mit besonderen Prüfanforderungen*. Beuth Verlag GmbH, 2008.

Index