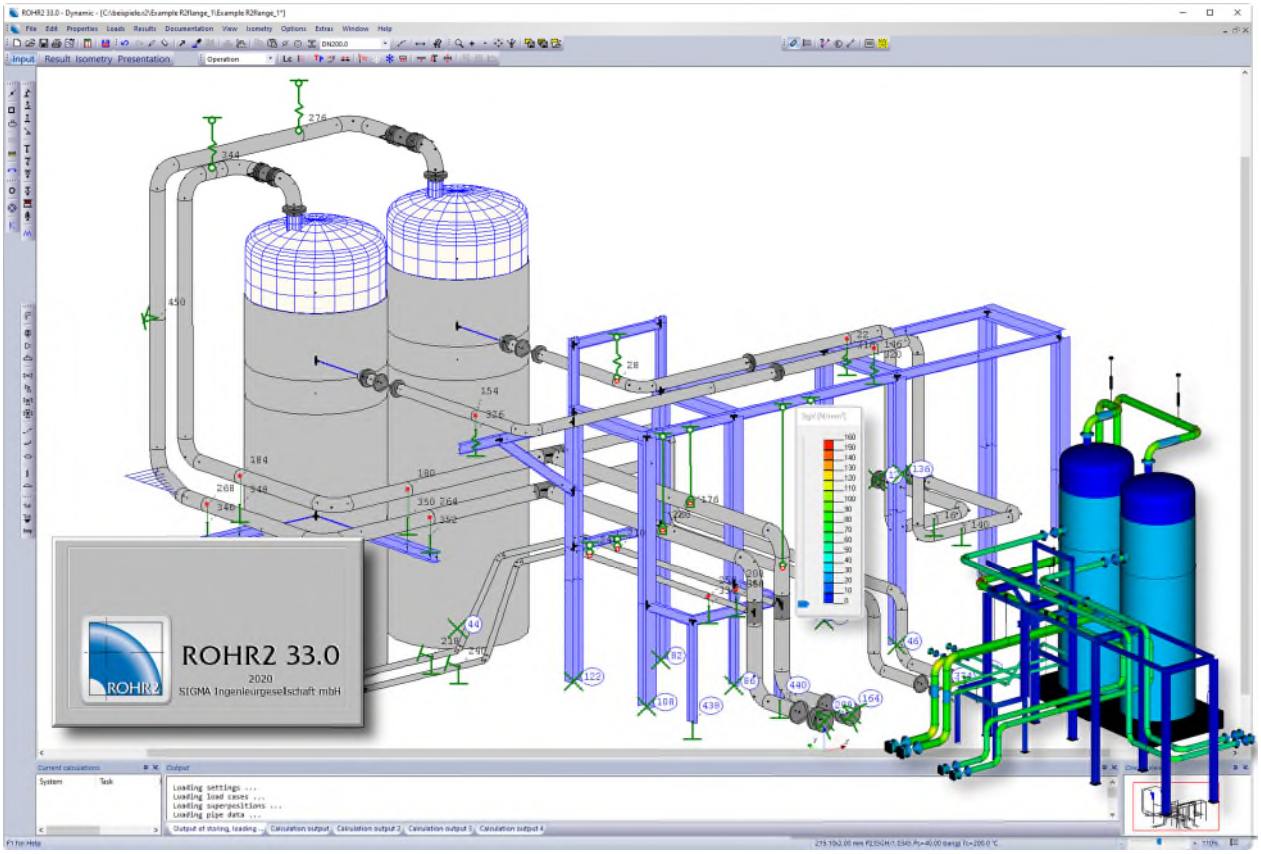
ROHR2

Program System for Static and Dynamic Analysis of Complex Piping and Skeletal Structures



ROHR2tutorial

ROHR2 Trial license
Introduction: Editing a Piping System

Release January 2021

SIGMA Ingenieurgesellschaft mbH

Software-Support, German	++49 (0) 2303 332 33 33	support@rohr2.de
Software-Support, English	++49 (0) 2303 332 33 44	support@rohr2.de

Published by

SIGMA Ingenieurgesellschaft mbH
Bertha-von-Suttner-Allee 19
D-59423 Unna
Germany
Telephone +49 (0)2303 332 33-0
Telefax +49 (0)2303 332 33-50
E-mail: info@rohr2.de
Internet: www.rohr2.de www.rohr2.com

Contents

1 ROHR2 Tutorial

Thank you for reading this document, introducing into the work with the program system ROHR2 and additional modules:

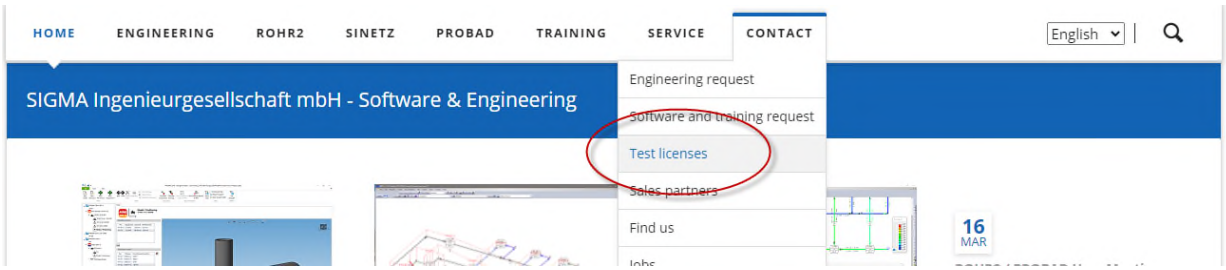
This manual is applicable to the
- ROHR2 full license
- ROHR2 test license

We would be very pleased to provide you with a program license or viewer download. Please contact our sales department in Germany (sales@rohr2.com) or one of the ROHR2 sales partners (see /www.rohr2.com for contact details).

1.1 ROHR2 test license

We would be very pleased to provide you with a full featured test license.

To get a test license please
- Use the registration form on www.rohr2.com in the Service/ test licenses area



- contact our sales department in Germany (sales@rohr2.com) or your sales partner (see www.rohr2.com , area international, for contact details).

1.2 Program start

Start the ROHR2 application by double-clicking on the program icon or start the program manually from the program subdirectory by running \R2WIN\R2win.exe.

1.3 Projects and Examples

For an introduction into pipe stress analysis with ROHR2 we are providing projects and examples to the user.

- sample calculations are stored in the. ../ROHR2/R2BSP/... directory after installation of a full-featured **ROHR2 license**
- project files, explanations and movies of the ROHR2 tutorial examples can be downloaded from the website www.rohr2.com in the *Service* area.
  They can also be accessed by the program function HELP > Training videos.
- **ROHR2 test license**: sample calculations are stored in the. ../ROHR2/R2BSP/... directory

Please refer to topic 3 of this document for a detailed introduction into a calculation example.

Project editing by means of a full-featured test license

Projects, created by the test license program are marked by TEST LICENSE. They can NOT be opened and modified by a commercial program license. For the conversion of projects, made by a trial license please contact the sales team.

1.4 User support, hotline and ROHR2 board

All software commands are documented in the user manual and in the program online help.

Additional information sources are available

as user support providing advice on installation and application of the program (hotline-service) on workdays (Mondays to Fridays) from 9.00 - 16.00 (Central European Time).
in the internet, e.g. ROHR2 Forum incl. ROHR2 FAQ (Frequently asked questions), see *Help menu,*

User support by email

An email function, integrated in to ROHR2win enables to transmit program data directly (see menu *Help |Support request)*.

User support address

SIGMA Ingenieurgesellschaft mbH
Dept. Program-Support
Bertha-von-Suttner-Allee 19
D-59423 Unna
Germany

Telephone and email

Software-Support, German	++49 (0) 2303 332 33 33	support@rohr2.de
Software-Support, English	++49 (0) 2303 332 33 44	support@rohr2.de

Internet

www.rohr2.de www.rohr2.com

2 ROHR2win User Interface - Overview

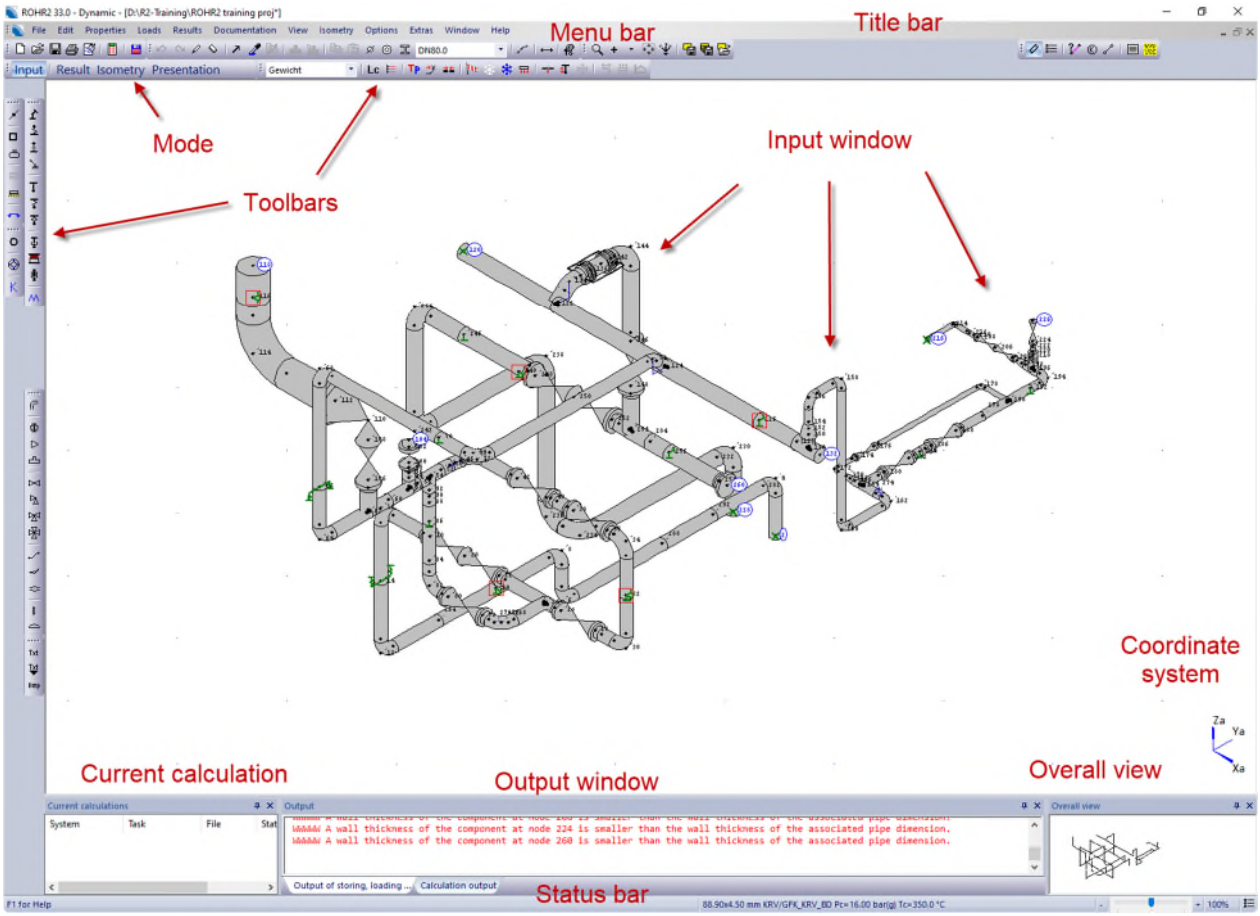
The input window shows the piping system and the drawing created.
All program functions are accessible by menu commands and symbols (icons).
The elements of the user interface can be positioned on the screen be placed free on the screen.

Title bar

The title bar shows the name of the current project including complete path.

Menu bar

The program functions can be accessed by the menu bar.



Mode

Depending on the view mode (see *Toolbar Mode*) ROHR2 appears in different modes.
Activate the mode related commands by switching between the program states in the toolbar mode.

See also *Simplified System Input* Menu *Loads*,

Status bar

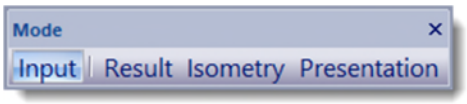
The status bar displays the currently used program command and selected system data

Toolbar Mode

Select between

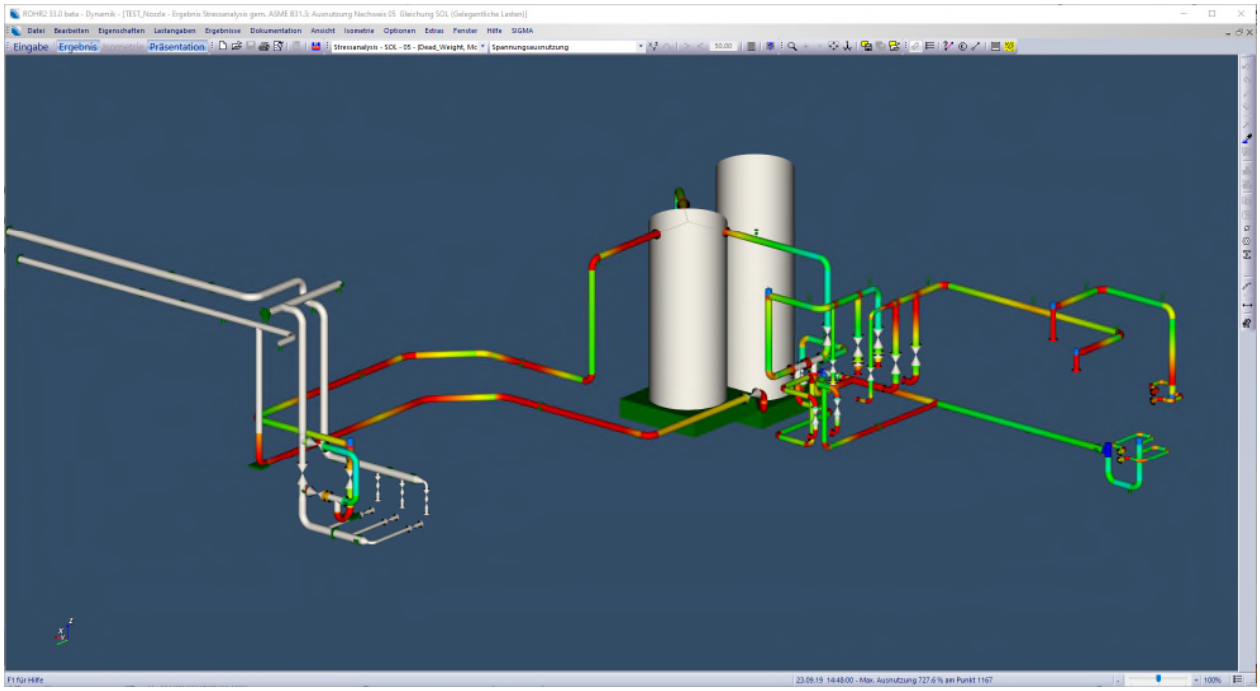
- input mode,
- results mode and
- isometric mode.
- Presentation mode,

The scope of functions is reduced to the necessary commands. See also *Customizing the graphical interface* when ROHR2fesu is installed.



Presentation mode

The presentation mode is an alternative view mode for inputs and results.
Tis mode is used to show the project as a 3D-model
This program mode is a pure presentation mode without any edit functions and reduced to commands for the treatment of views. It is possible to present the piping system in different color modes using the properties of the display using the *Segment properties* funtions.

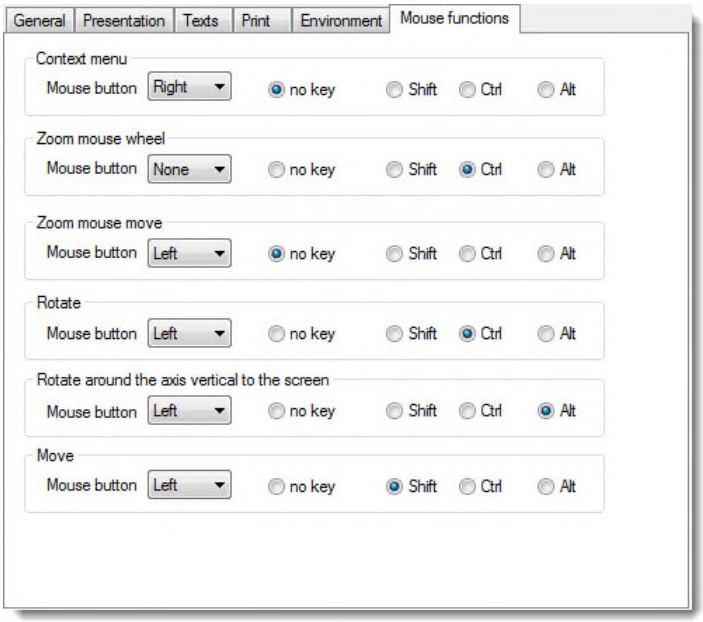


Views can be defined to be used in reports in the presentation mode.

Mouse functions for graphic operations

The key commands of the mouse functions can be adapted here:

- Context menu,
- Zoom,
- Zoom mouse move,
- Rotate,
- Rotate around the axis vertical to the screen,
- Move



The mouse functions are carried out by a mouse button tob e defined here. If required an additional key can be specified.

Mouse button

Select the mouse button which needs to be pushed for this command.

No key, shift, ctrl, alt Taste

Select the key which need to be pushed additionally

3 Definition of a piping model in ROHR2

This chapter is showing the essential steps necessary to define a calculation model.
For details to dialog windows please use the program online-help by pressing F1 or look into the printed manuals.

Additionally the first training example is done by means of the data entered here.

The preparation of the stress calculation at first requires collecting all calculation relevant data.

The following project schedule is showing the information, necessary for a stress calculation in plant/piping construction.

Project schedule

1.	Plant plan with design data	o.k.
2.	**Nominal widths**	pre-dimensioned (check or modification by pressure loss calculation)
3.	Selection of materials and nominal pressures	
4.	Definition of insulation thicknesses	
5.	**Stress analysis** for wall thickness dimensioning **Tip** The stress analysis is done e.g. acc. to AD-2000 or EN 13480. At first the wall thickness for each dimension has to be defined, considering stress due to internal pressure Alternatively there is a pipe class containing the design parameters	
6	Planning of pipe routing / pipe plan / isometrics	incl. temporary definition of support positions and anchor points
7.	Pressure Loss Calculation	modification of nominal widths if required
8	**Pressure instrument codes, Company safety rules or customers specification**	**requires stress calculation**
9.	**Stress analysis using ROHR2**	**if required modification of the pipe routing or support design.**
10.	Final isometrics / parts lists	Order
11.	Assembly	

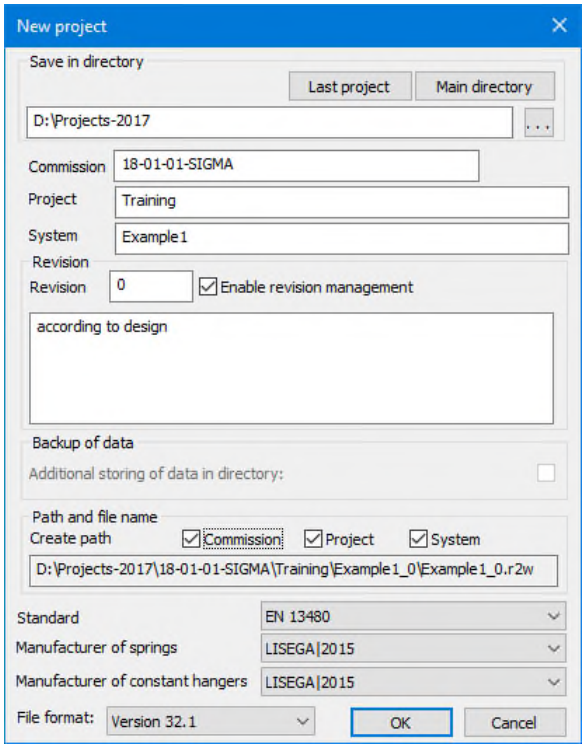
3.1 Creating a new ROHR2 project

- Create a project directory with up to three levels
- Select stress code /stress specification
- Select spring manufacturer for automatics spring design

Training settings1

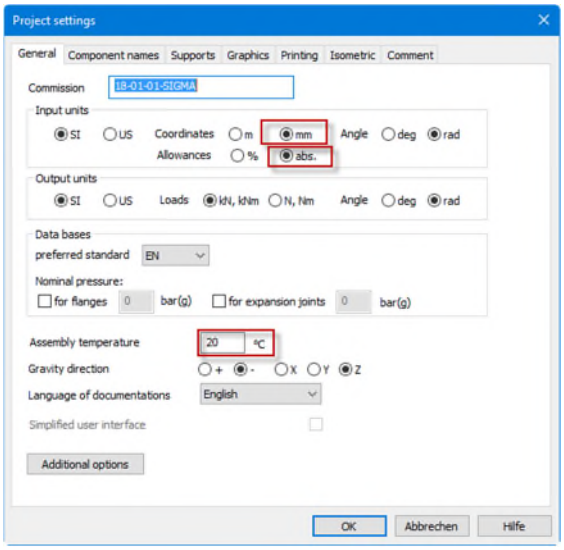
- Commission: SIGMA
- Project: Training
- System: Example1

- Spec. .: EN 13480
- Springs: LISEGA 2015
- Constant hangers LISEGA 2015



Options| Project settings

The project settings can be modified every time. At this moment the training example requires the following entries:



e.g.

- Assembly temp. 20°C
- unit coordinates mm
- Wall thickness tol. mm

Please note!

By the function *Options | Presettings for new models* individual standard settings for piping models can be defined to be used in future projects. Adapting the settings in new piping models will be required any more.

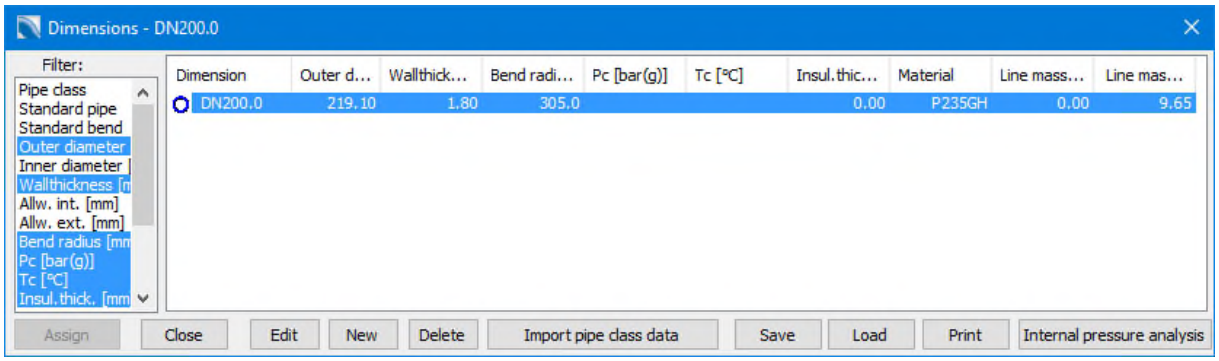
3.2 Piping model generation

3.2.1 Editing pipe dimensions

Menu edit > Pipe dimensions

Dimensions, available in the project

Die dialog window *Dimensions* contains the pipe parameters, available in the project. A new project includes one dimension by default.



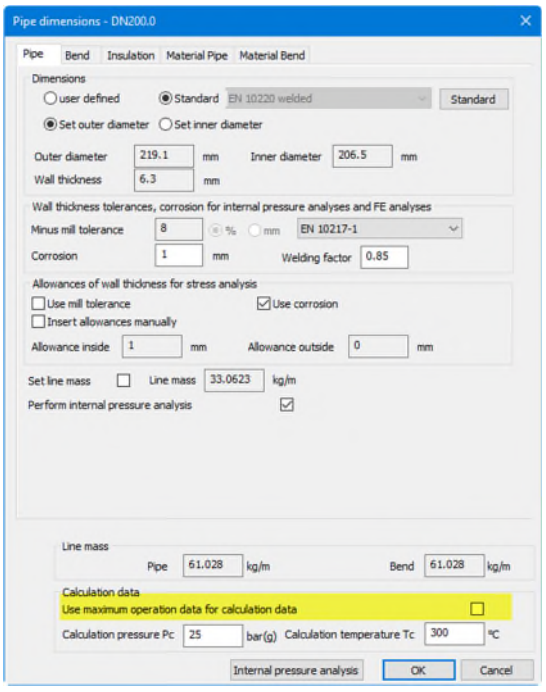
A double-click on the dimension opens the pipe dimensions input window.

All necessary dimensions are defined here

This dialog window offers 5 registers for the input of data belonging to a nominal width.
At first there are the dimensions of the straight pipe (see picture), followed by bend data, insulation and material data. The line masses are determined automatically, if not entered manually.

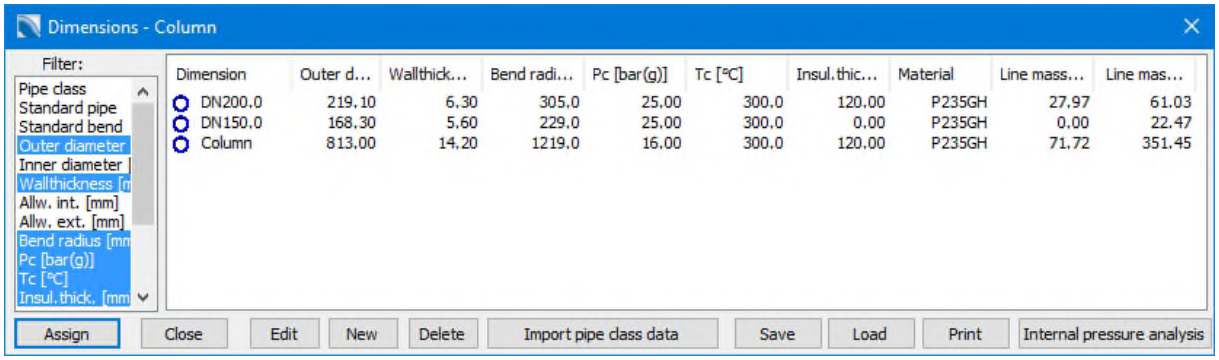
Parameters of the internal pressure definition can be defined in the pipe and bend windows.
Check if tolerances and allowances are to be considered in the stress analysis.

Additionally the parameters of the internal pressure can be inserted. There are nearly similar dialog windows for structural steel sections (beams) and jacket pipes.



Note:

When selecting the material, please note that the characteristic values used in ROHR2 are taken from different standards (e.g.: seamless / welded).
The user has to determine which characteristic values he wants to use.

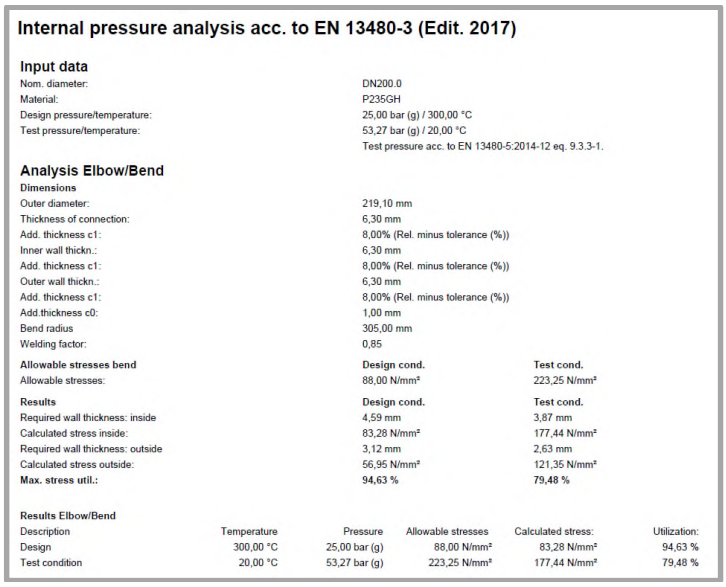


Handling of the Dimensions list

- Use filter functions for the display of desired parameters only:
- Press STRG-key and select the desired parameters in the left column.
- Adaption by table header for several columns.
- Selection by materials file: sort by name/number in the column header.
- If listed by name: use the character keys to jump to the position in the table.

3.2.2 Internal pressure check

When defining the dimensions, care must be taken to select a sufficient wall thickness depending on the internal pressure. If dimensions from an already calculated pipe class are used, this verification is no longer necessary. ROHR2, however, recalculates automatically under consideration of preset additions and factors for the straight pipe and the bend.
If required, the INSIDE PRESSURE CHECK function provides detailed results for pipes and bends.
An internal pressure verification can later be requested for individually inserted components such as T-pieces or reducers.
The results of the internal pressure test can later be transferred to the documentation of the calculation for each dimension or as an overview.



Internal pressure analysis acc. to EN 13480-3 (Edit. 2017)

Input data

Nom. diameter:	DN200.0
Material:	P235GH
Design pressure/temperature:	25,00 bar (g) / 300,00 °C
Test pressure/temperature:	53,27 bar (g) / 20,00 °C
	Test pressure acc. to EN 13480-5:2014-12 eq. 9.3.3-1.

Analysis Elbow/Bend

Dimensions

Outer diameter:	219,10 mm
Thickness of connection:	6,30 mm
Add. thickness c1:	8,00% (Rel. minus tolerance (%))
Inner wall thickn.:	6,30 mm
Add. thickness c1:	8,00% (Rel. minus tolerance (%))
Outer wall thickn.:	6,30 mm
Add. thickness c1:	8,00% (Rel. minus tolerance (%))
Add.thickness c0:	1,00 mm
Bend radius	305,00 mm
Welding factor:	0,85

Allowable stresses bend	**Design cond.**	**Test cond.**
Allowable stresses:	88,00 N/mm²	223,25 N/mm²

Results	**Design cond.**	**Test cond.**
Required wall thickness: inside	4,59 mm	3,87 mm
Calculated stress inside:	83,28 N/mm²	177,44 N/mm²
Required wall thickness: outside	3,12 mm	2,63 mm
Calculated stress outside:	56,95 N/mm²	121,35 N/mm²
Max. stress util.:	**94,63 %**	**79,48 %**

Results Elbow/Bend

Description	Temperature	Pressure	Allowable stresses	Calculated stress:	Utilization:
Design	300,00 °C	25,00 bar (g)	88,00 N/mm²	83,28 N/mm²	94,63 %
Test condition	20,00 °C	53,27 bar (g)	223,25 N/mm²	177,44 N/mm²	79,48 %

For further details on the internal pressure analysis please refer to chapter S*tress Analysis, 3.12.*

3.3 Drawing a system

At first the main lengths of the piping system are drawn. All necessary steps are shown in the following.

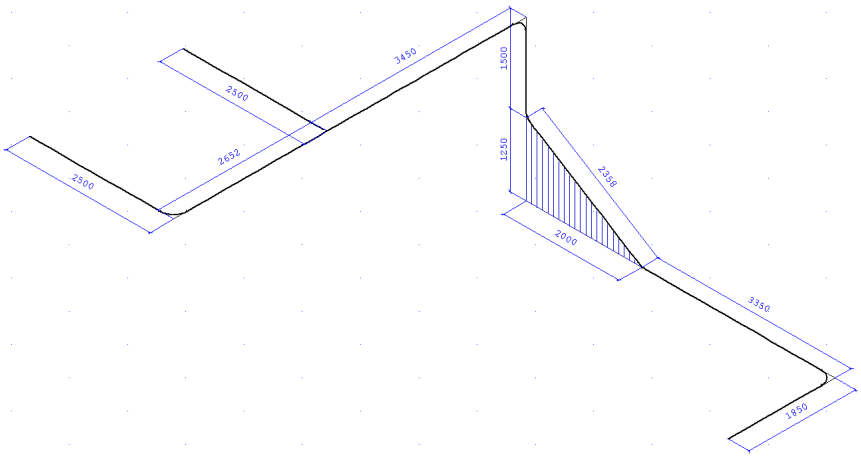
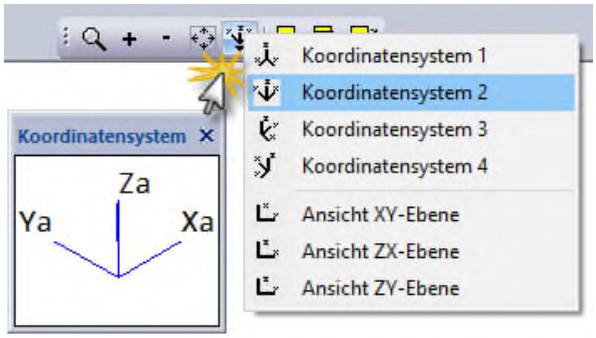


figure: system lines

3.3.1 Select a coordinate system and draw

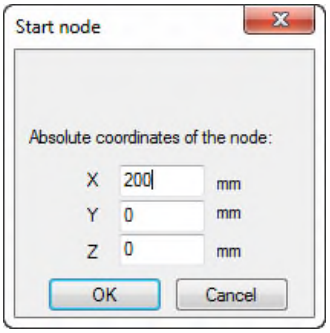
Select a coordinate system from the toolbar *View*



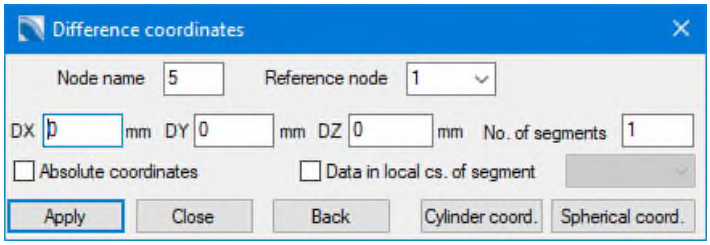
Draw the piping system by means of the *drawing tool* from the toolbar *Edit*

or user the menu command
Menu Edit > Draw

Enter absolute coordinates of the first node



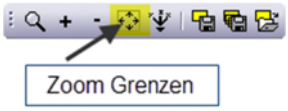
Press **any key** to open the dialog window **difference coordinates**



The piping from C1/N1 up to the anchor point right below is drawn by the nom. width DN150.
The assignment of the dimension is done in the next step.
Click *Close* to exit the window.

Please note!
Activate the input fields in the coordinate window by using the keys X / Y / Z.

For an overview the drawing can be adapted to the maximum screen size by the function *Zoom limits*.



or menu right mouse button

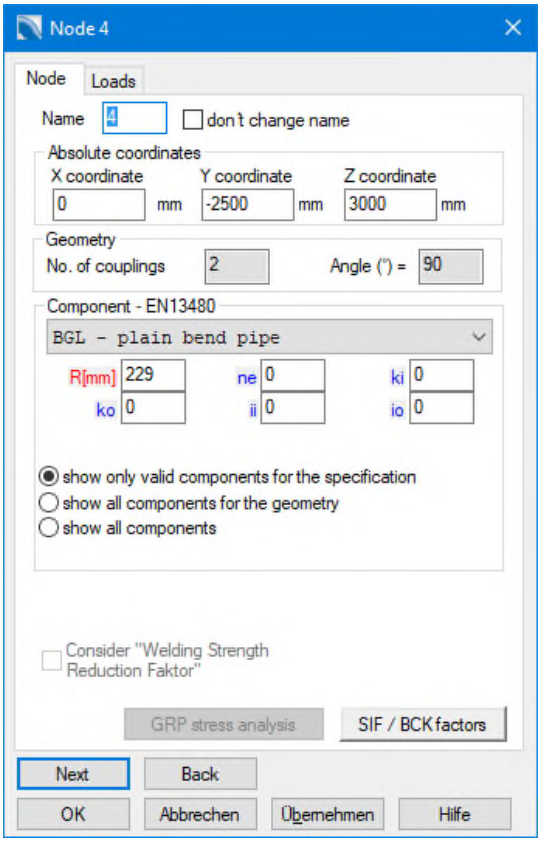
Drawing a branch

- Use Edit| Draw or 
- Get the branching node (highlighted red) and press any key. After that continue drawing as mentioned above.
- If the branching node does not exist click into the segment and enter the distance between branching node end start node/ end node manually.
- Draw a branch with DN150
- „Close“ to terminate the window

3.4 Input data modification and checking

3.4.1 Edit nodes

Properties menu > Data of Nodes



The node, whose properties should be edited, must be left-clicked with the mouse.
The dialog window *node* opens to display all properties in the registers *node* and *loads*.
The properties of the boundary conditions set at the node will be shown in separate registers each (here: spring hangers).

3.4.2 Register Nodes

The register *node* shows node name, coordinates and component name.

Here several changes can be made:

- Change the node name (max. 4 characters). A node name, changed manually, will not be revised even in case of automatic renumbering.
- Change the component name. Depending on the component name the SIF (stress intensification factor) is defined for stress analysis.
  Defining additional parameters is not mandatory at this step.

Beyond it the node coordinates can be checked here. The check of node coordinates normally is done in the graphic by adapting the following segments.

3.4.3 Register Loads

The register loads is used to check the loads assigned to each single load case.
The loads cannot be modified in this dialog window.

3.4.4 Additional registers

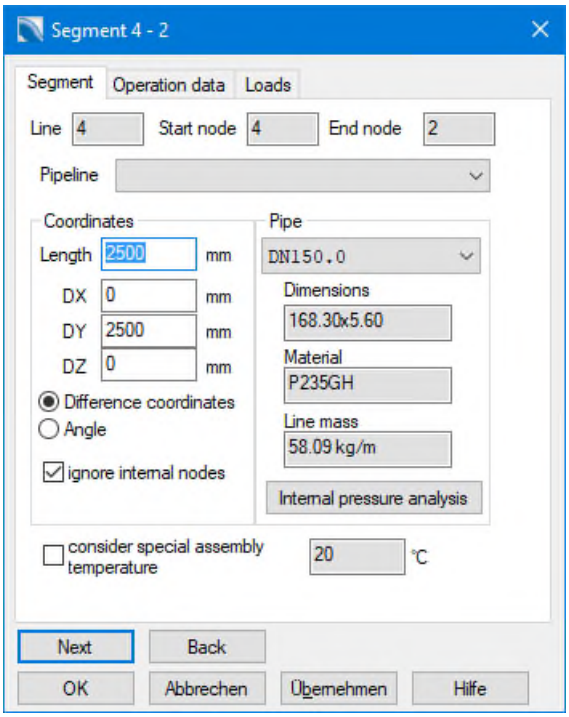
If there are boundary conditions (e.g. like supports) assigned to a node, they are shown in their own registers. Support conditions can be modified.
For more details referring to support types and support conditions see the following paragraphs.

Input record help

Opens the ROHR2 help text referring to the selected input record in a separate window.

3.5 Edit segment

A segment is a part of a system between a start node and an end node.
Select the segment to be modified by the left mouse button.



Register segment

The first register, *segment,* shows the corresponding line name, description of features, start and end node, coordinates, as well as dimensions, material and line mass for the current load case.
Modifications can be done as following:

- Dimension
- Piping assignment
- Segment length

The coordinates of the segment normally are modified by input of new X-, Y, and Z-difference coordinates.
The direction of the segment is remains, if alternatively the length is modified.

Coordinates

It can be selected if the difference coordinates or the angles to the main axes shall be shown for the segment. The angles cannot be changed.
When the difference coordinates are changed the direction of the movement at the connecting segments must be inserted. Input can be made in the dialog window *Move direction*

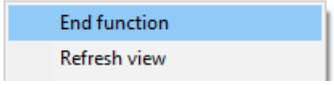
3.6 End function

There are two possibilities to end a program command:
Using the **ESC** key finish the action by the right mouse button/context menu.

Context menu

- terminate the running function. The program command, currently in use is shown in the menu at 2nd position.
- cancel the selection of system parts

Right mouse button



3.7 Select

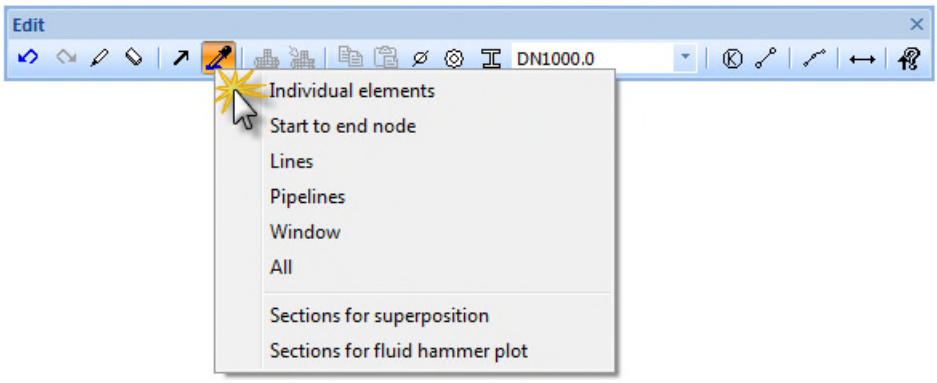
The *Select* function often is used to define parts of the model and add parameters to those in the next step. E.g., new dimensions, a new material or varying operation parameters can be assigned.

In the following different methods are explained to select and highlight parts of the system:

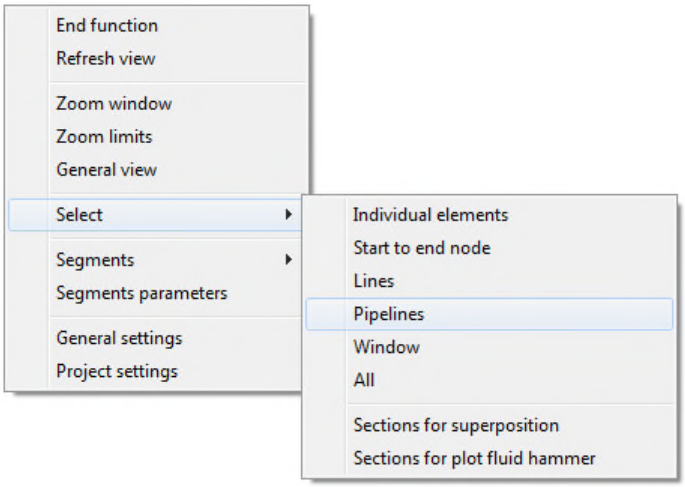
Edit > Select

There are three possibilities to select (highlight) an object in the pipe system:

- Menu Edit - Select
- The button  in the toolbar *Edit*. Activate it and make your selection in the sub-menu:



- the mouse menu (click the right mouse-button over the graphic) see also *Mouse commands*



There are several possibilities to select an object:

3.8 Insert components

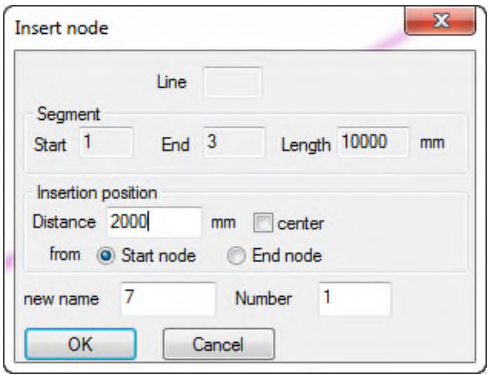
3.8.1 Insert a reducer

Note!
The following instruction assume that the diameter of the created pipe is DN200.
When a reducer is inserted the nominal width of the pipe changes from the reducer position up to a selected end node.

These steps are required:

Edit > Insert component > Reducer

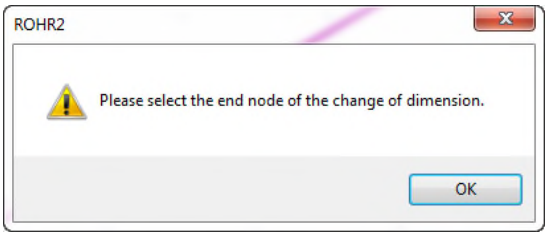
or use  .



- Click into the segment where the reducer is inserted
- enter the distance to the next node: distance 700 mm to the branch

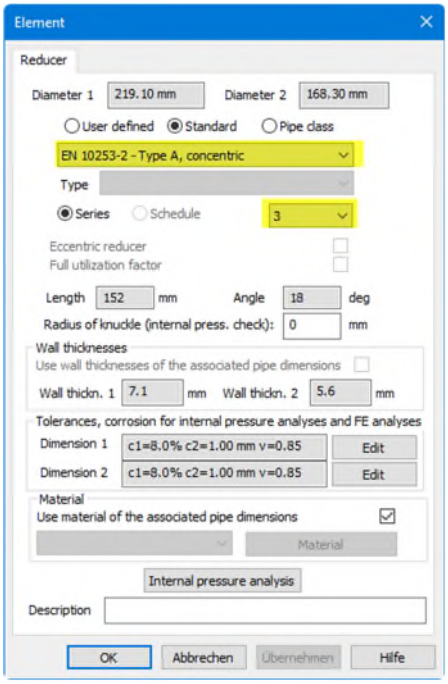
After that a message box occurs: the node representing the end of the dimension change need to be selected.

First confirm this message by OK, then enter the end node of the changed dimension.
The region to be changed will be highlighted in red.



The dialog window *Insert reducer* opens for the next steps:

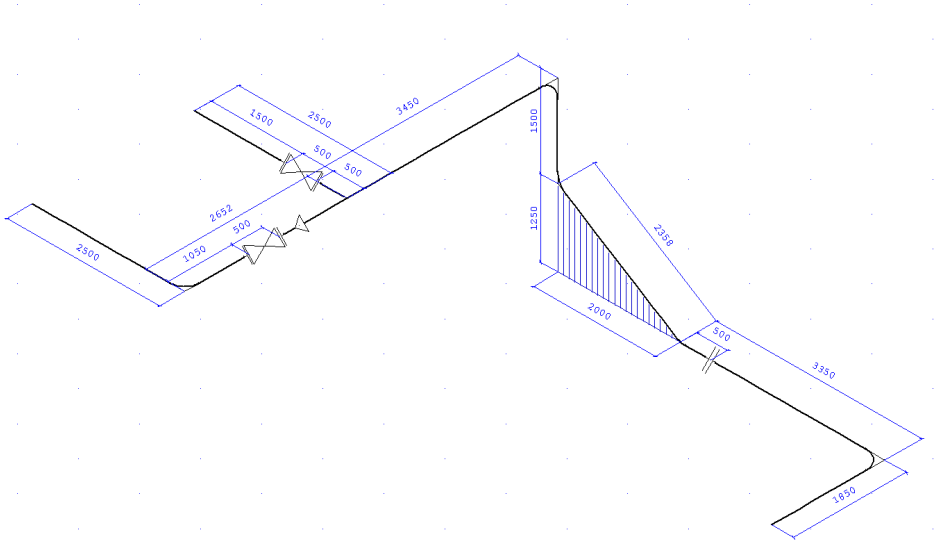
- Select a dimension
  if not existing the dimension can be inserted here.
- Select reducer from the norm DIN 2616, part 2
- Confirm by OK and insert the reducer.



Note:
If the pipe has been drawn in DN 150 please consider:
- *the insert position must be 1800 mm to the next bend*
- *the pipe needs to be expanded to DN200 up to the branch and beyond.*

3.8.2 Insert components

Normally components need to be inserted directly on a pre-defined center line of the pipe.
It is not possible to append a component to a drawn segment.
The component can be modified later on by the segment dialog window.

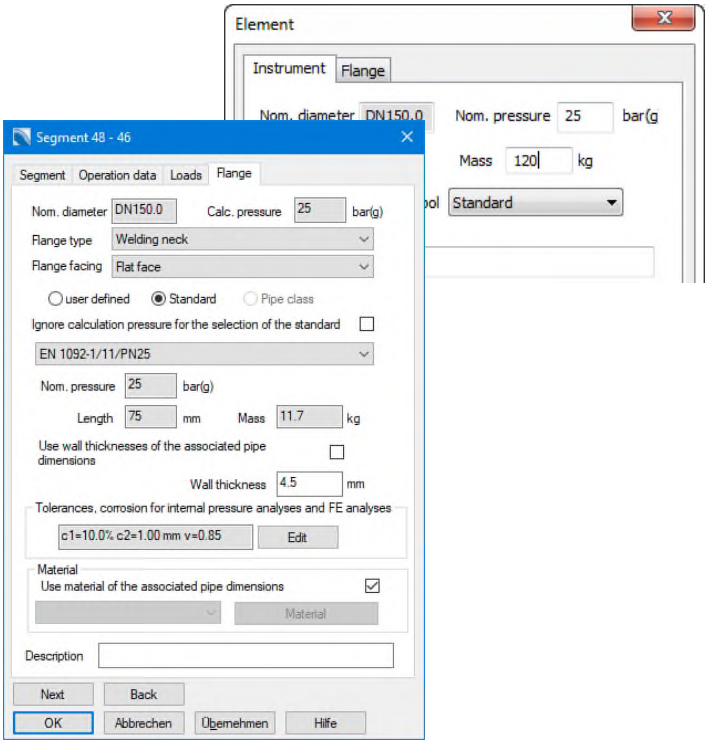


Insert component

- Select the component symbol from the toolbar (toolbar components on the left side)
- select the desired segment by the cursor
- Enter the distance to the next node
- Select/define component

Instrument

- Length seal to seal or weld to weld
- Enter instrument mass
- Select Option *with flange*

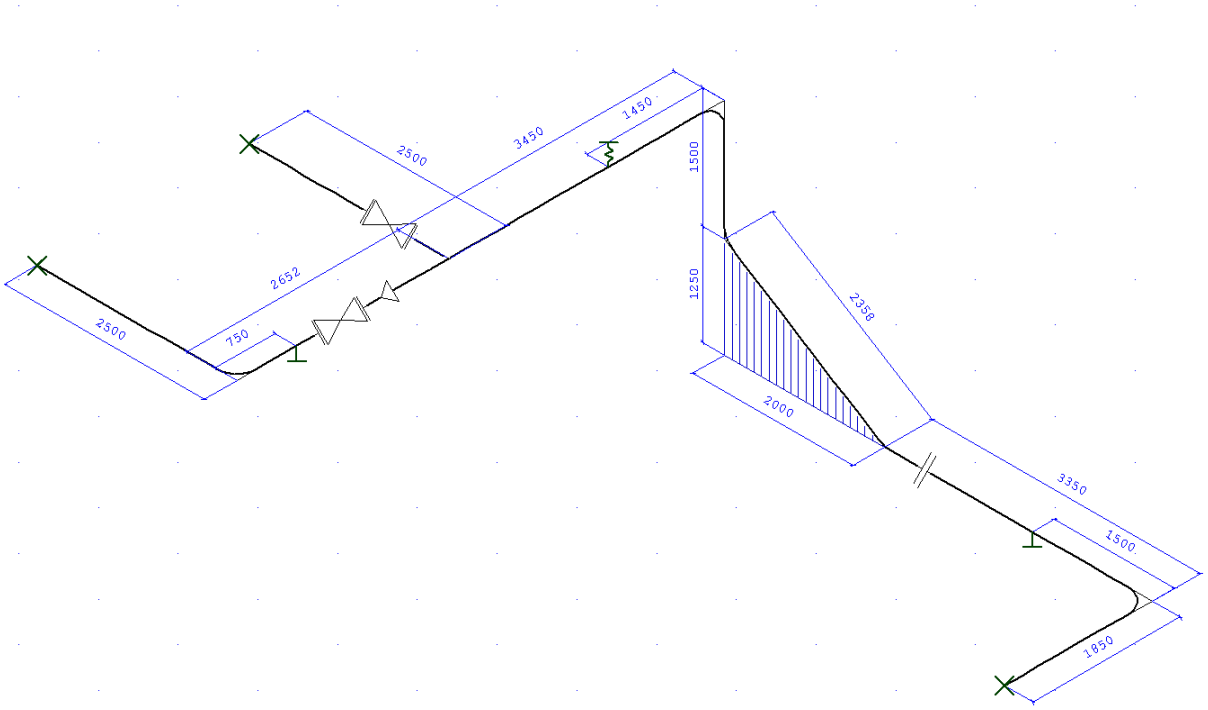


Flange:

- Select flange

3.8.3 Insert supports

Supports may be inserted at existing nodes or segments. If the user places a support into a segment, automatically an intermediate node is created at this place.

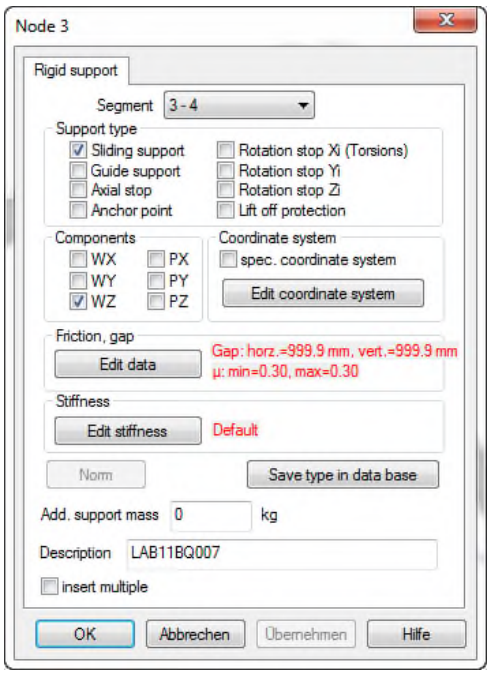


Defining a rigid support

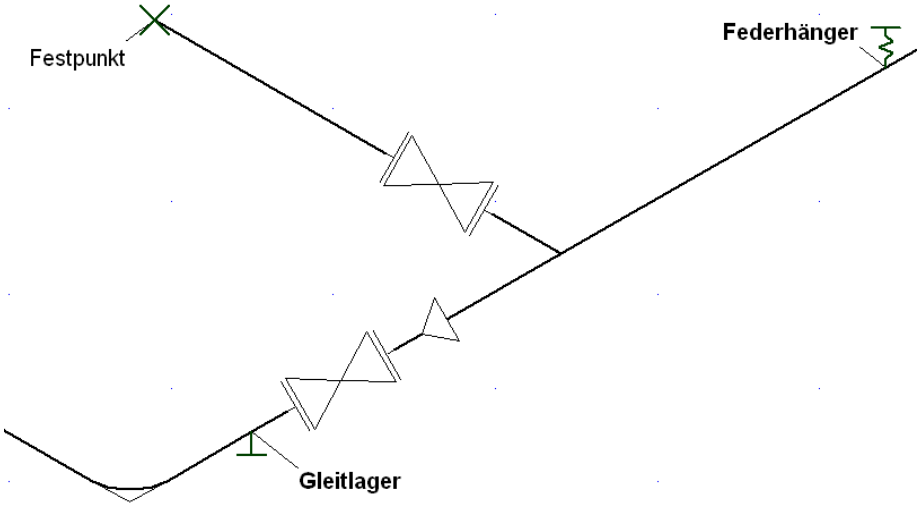
ROHR2 regards sliding supports, bearing supports, axial stops and anchor points as rigid supports.

The way to insert a support:

- select the type of support from the toolbar (toolbar on the left side)
- select the desired node or segment by the cursor
- define support. Alternatively here types of support or components (degrees of freedom) may be assigned.



3.8.4 Support condition symbols

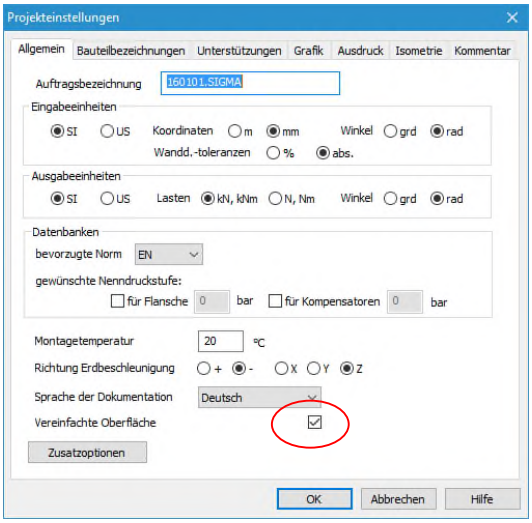


All supports of this example at first are calculated considering the standard values of friction, gap and stiffness.

The design of the spring hanger  shall be carried out by the program (Default-settings, no more inputs required).

3.9 Load case definition

Depending on the project settings ROHR2 can be started using the standard interface or a simplified input.



The difference between those two modes are in the definition of load cases and calculation tasks.

The simplified user interface enables to carry out the calculation of load cases dead weight, operation, shut-down and, optionally wind, earthquake and pressure test basing on standard settings.
Load case superpositions and stress analyses are generated automatically using standard settings.

Settings of the simplified user interface

The use of the simplified user interface can be selected
- for new ROHR2 projects in *Options*| *Presettings for new models*| | *Register Common*

- activate the simplified user interface for the current ROHR2 project by the menu *Options*| *Project settings* | *Register Common*

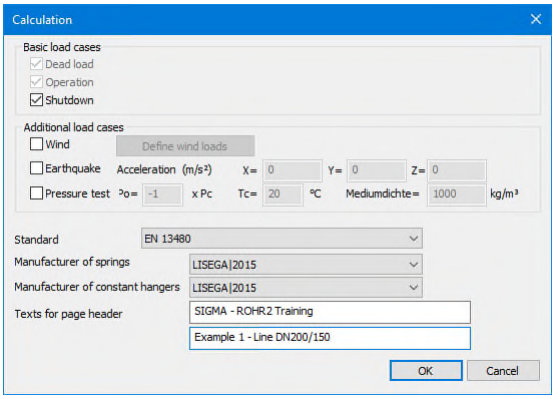


Figure: simplified load input

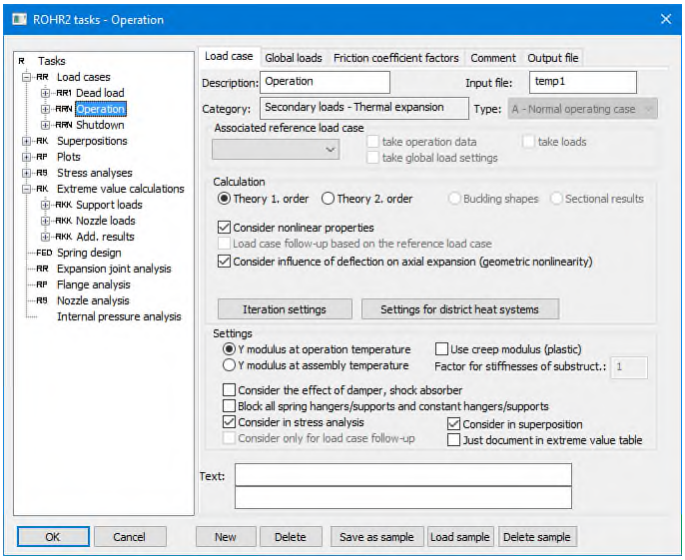
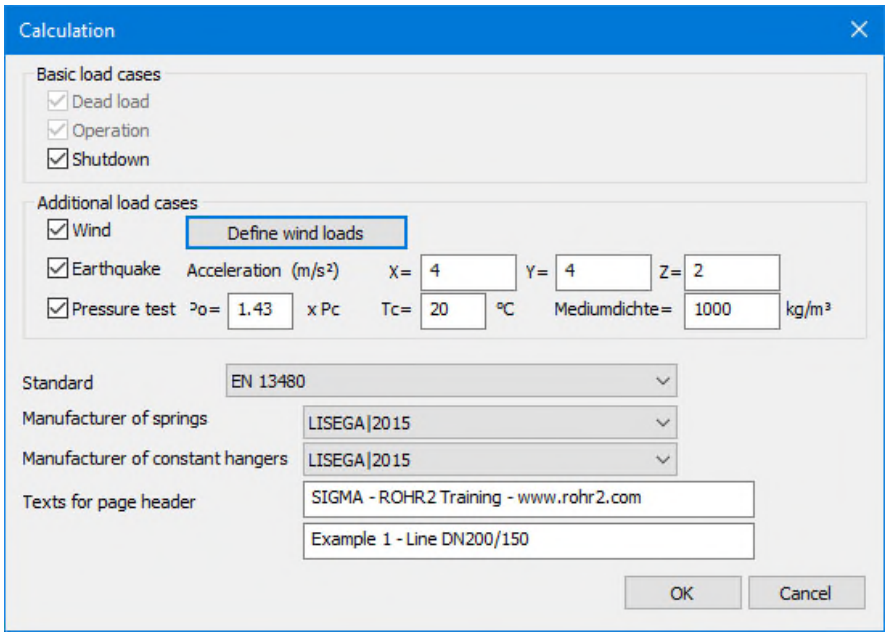


Figure: detailed load input

The following instructions refer to the settings to be made when using the simplified user interface.

3.10 Load case definition using the simplified user interface

The simplified user interface enables to run the calculation with standard settings for the load cases: dead weight, operation, shut down and optionally wind, earthquake and pressure test.
Load case superpositions and stress analyses are generated automatically basing on standard settings.



The load cases weight, operation (weight + thermal expansion) and shut down (with ambient temperature) are predefined using fixed parameters.
The load cases wind, earthquake and pressure test can be selected optionally.

3.10.1 Assign operation data

Operation data is similar with all load cases except of the load case shutdown and pressure test. That means it is not required to select a particular load case when entering the operation data.

Operation data is assigned per segment. At first the segments where need to be highlighted where consistent operation data need to be assigned. It is required to add operation data to all segments.

Select segments

using

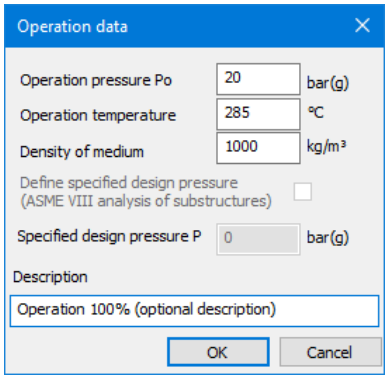
Edit menu| Select| All

to select the entire system.
Selected areas are highlighted red.

Please note:
The *Select* command also can be accessed by the context menu (right mouse button)

Select segments

Using the command
Loads| Operation data| Referring to load cases
or 
opens a dialog window where operation data sets can be defined



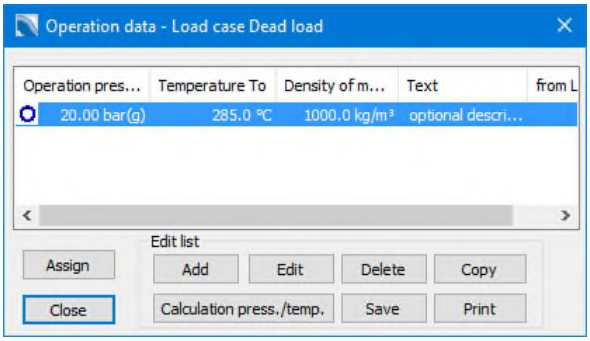
Use the command *Add* to show a new data set.

Operation data, example

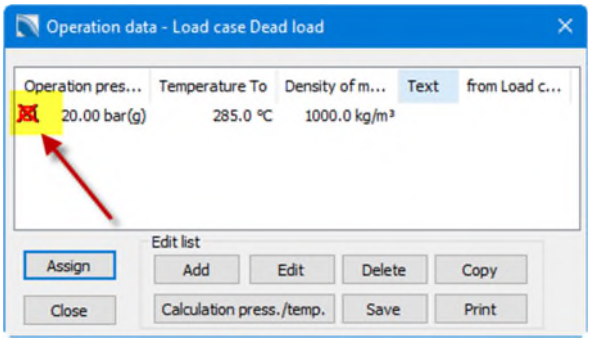
Consistent operation data for the entire system:
- 20 bar / 285°C / 1000 kg/m³

The *Assign* command is used to assign the data to the selected segment.

Optionally you can add descriptions to the data records.



Please note:
The symbol in the first column changes from blue circle to red X when the data have been assigned.



The operation data of lc pressure test are defined globally by the user.

Data of the pressure test:

- 1.43 x Pc / 20°C / 1000 kg/m³

The definition of additional loads is restricted: anchor point movements, wind loads and constant earth quake accelerations can be entered. The assignment of loads to load cases is done automatically. It is not required and not possible to select the current load case.

- Anchor point movements are considered only in lc operation
- Defined wind loads are considered only in lc wind for the entire piping model
- Earthquake accelerations are considered only in lc earthquake

Data to be entered for the example:

- Wind loads: EN 1991, wind zone Germany WZ2, Ground category III
- Earthquake loads: Accelerations in X: 4.0 m/s² / in Y: 4.0 m/s² / in Z: 2.0 m/s²

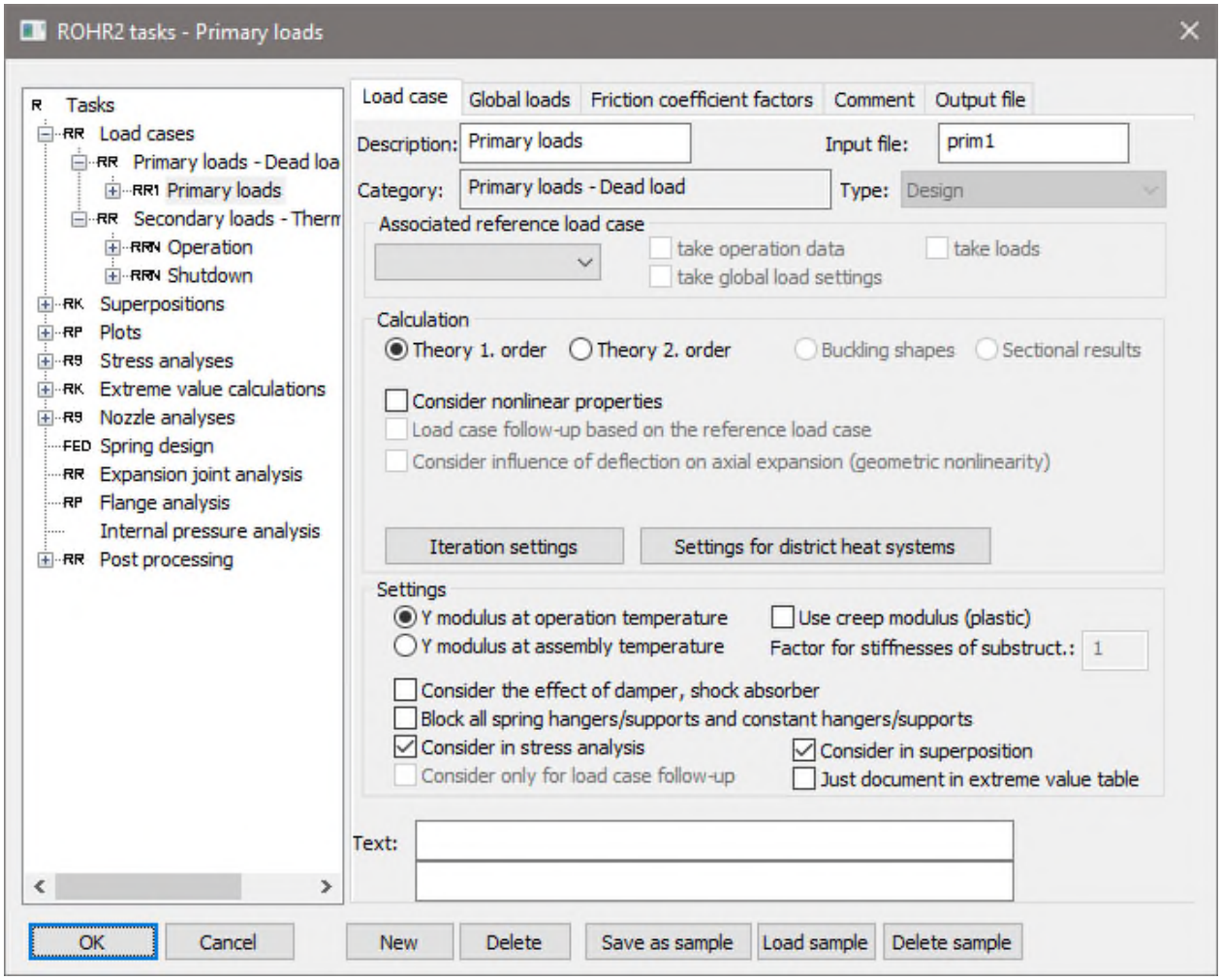
If more detailed inputs or additional load cases are required, the program can be switched to normal mode by *Options*| *Project settings*| *Common*.

In current projects switching between normal mode and simplified user depends on the availability of the input data in the simplified mode.

3.11 Definition of load cases and tasks in ROHR2 normal mode

The load cases *dead weight*, *operation*, (weight + thermal expansion) and *shut-down* are pre-defined. Global loads, already considered in the load cases can be shown in the register *global loads*.

The menu *loads*| *tasks* or LF opens the window for the definition of load cases and ROHR2 tasks like stress analyses or extreme value calculation.



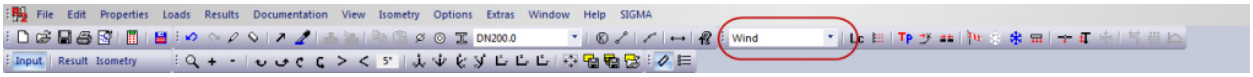
The definition of additional load cases is explained later.
As a first step the calculation includes the pre-defined load cases dead weight, operation and shut-down.

Further information on load case superposition, stress analysis and on possibilities for individual adjustment of the settings can be found in the documents for supplementary training or in the manuals.

3.11.1 Assign operation data per load case

It is important to define operation data per load case for the calculation. Before assigning the operation data at first the load case must be specified.

Select load case



Operation data are assigned by segments. At first highlight the segments where identical operation data will be assigned to. The operation data must be assigned to all segments.

Highlight parts of the system

To select the entire system use

Menu Edit| Select| All,

All selected parts of the system are highlighted red.

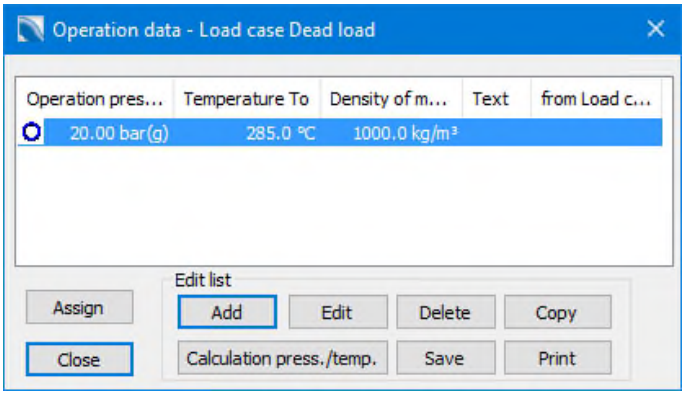
Please note
The Select command can be found in the Context menu (right mouse button), too).

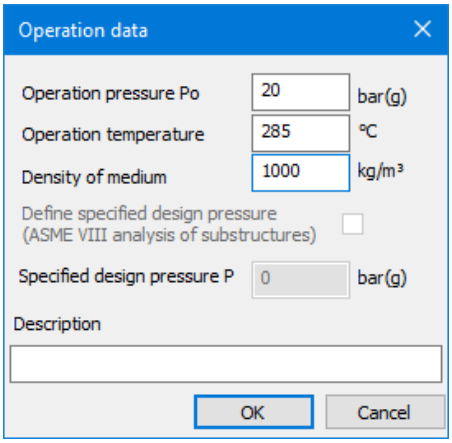
Select parts of the system

Use

Loads| Operation data| Referring to load cases

or  a dialog window opens where record types with operation data can be defined.





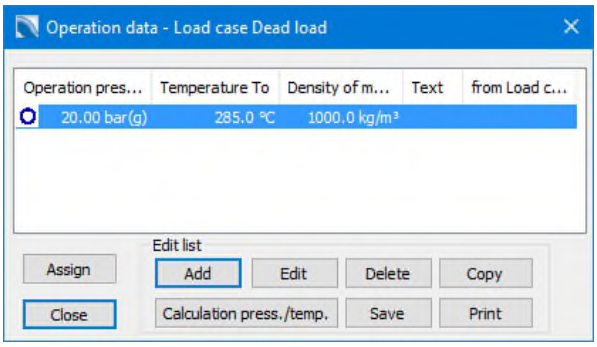
Define a new record by *Add.*

Operation data example

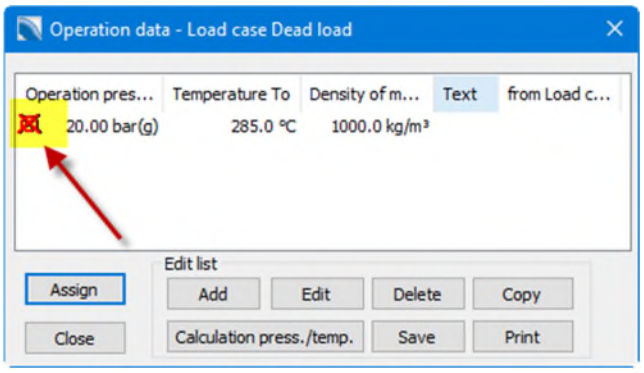
Identical Operation data in the whole piping system:

- 20 bar / 285°C / 1000 kg/m³

Assign data by *Assign* to the selected parts of the system.



Please note:
If the data is assigned, the symbol in the first column of the table changes from blue circle to a red X.



The operating data dialog window stays open for further treatment. Open dialog windows can be identified by their colored headline

Copy operation data

Copy the data from load case *Operation* into the load case *Weight* by

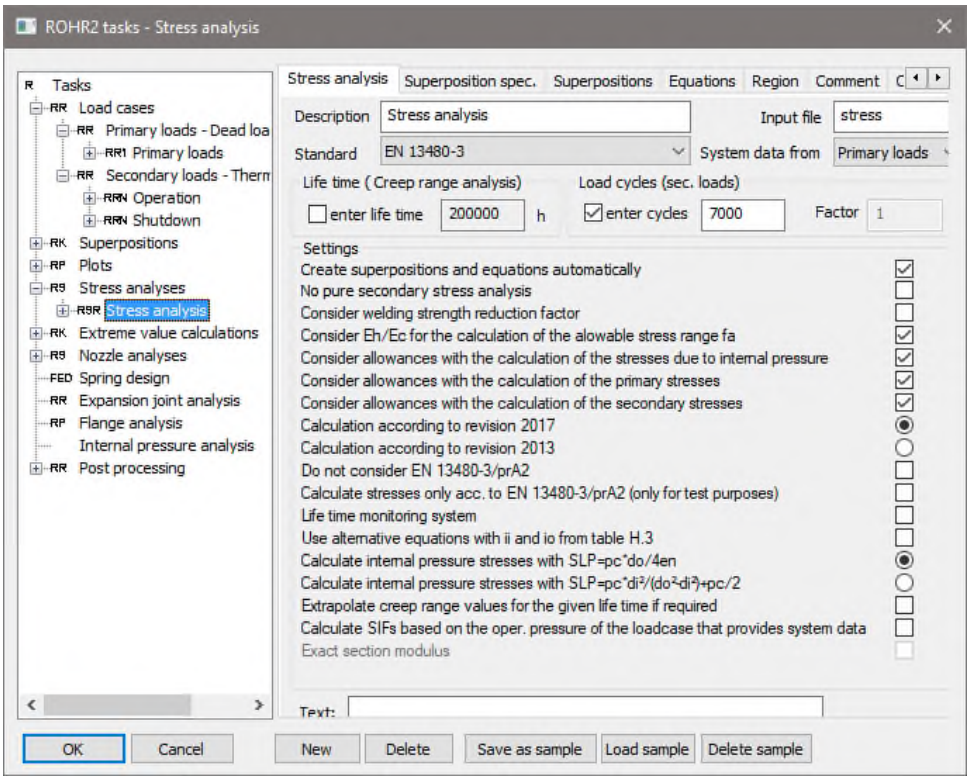
- Select record
- Command *Copy*
- Choose the load case *Weight* in the select box and confirm by *OK*.

3.12 Stress analysis, loads on supports and spring design

Please note:
The settings, explained below are part of the automatic settings of the simplified interface and can´t be modified by the user.

Use *Loads*| *Tasks* or LF to open the dialog window for the definition of load cases and ROHR2-tasks. At first select a stress analysis in the left column.

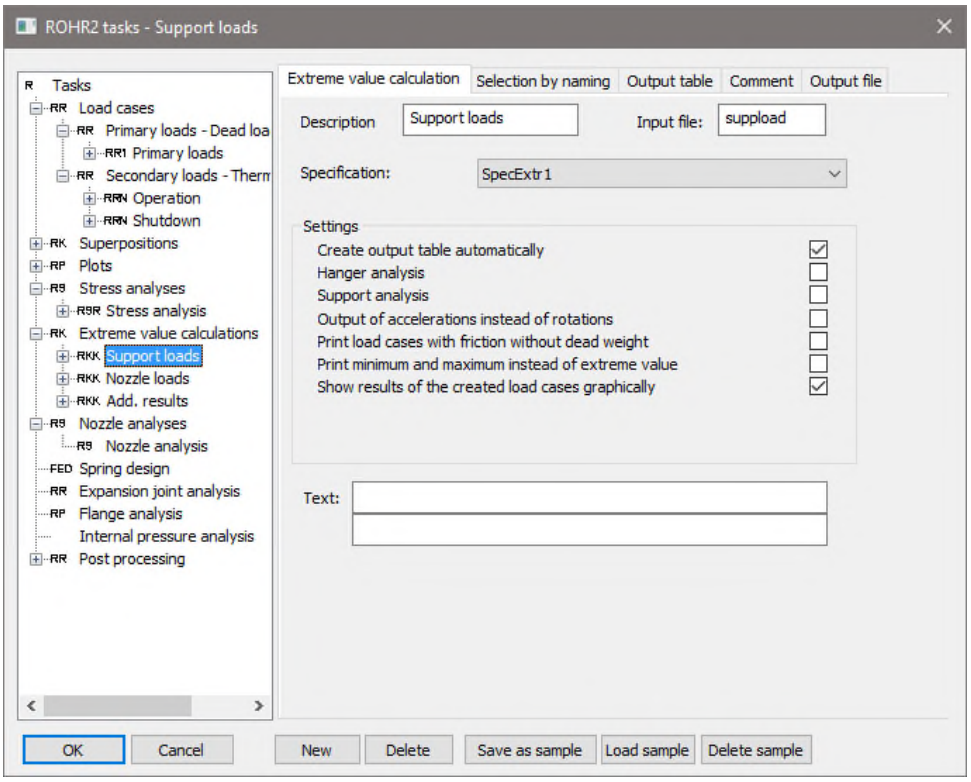
3.12.1 Stress analysis



- Define or change the stress code here to be used for the calculation.
- The load case superposition for this load case is carried out automatically, but can be modified.

3.12.2 Loads on supports

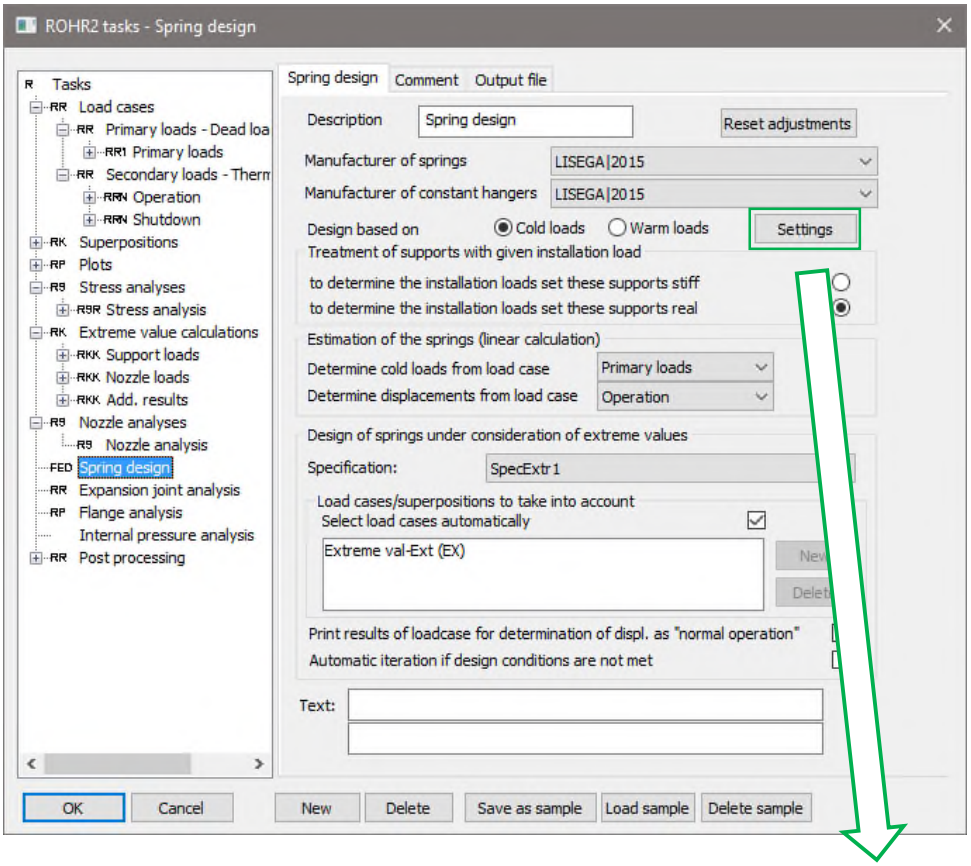
Use Loads| Tasks or LF to open the dialog window for the definition of load cases and ROHR2-tasks. Select an extreme value calculation in the left column.

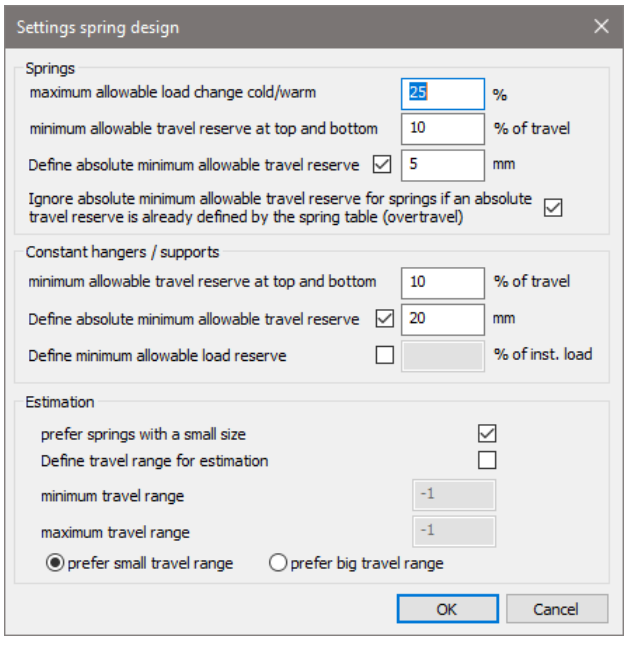


- The load case superposition, used to determine the maximum loads on supports is carried out automatically, but can be modified manually.
- The option „Show results of the created load case graphically“ offers the opportunity for the graphical representation of the results of combined load cases in ROHR2win.
  Activate this option, e.g. for generation of loads overviews basing on the support loads of the extreme value load case.

3.12.3 Spring design

- Carry out the spring design for various spring manufacturers at any time





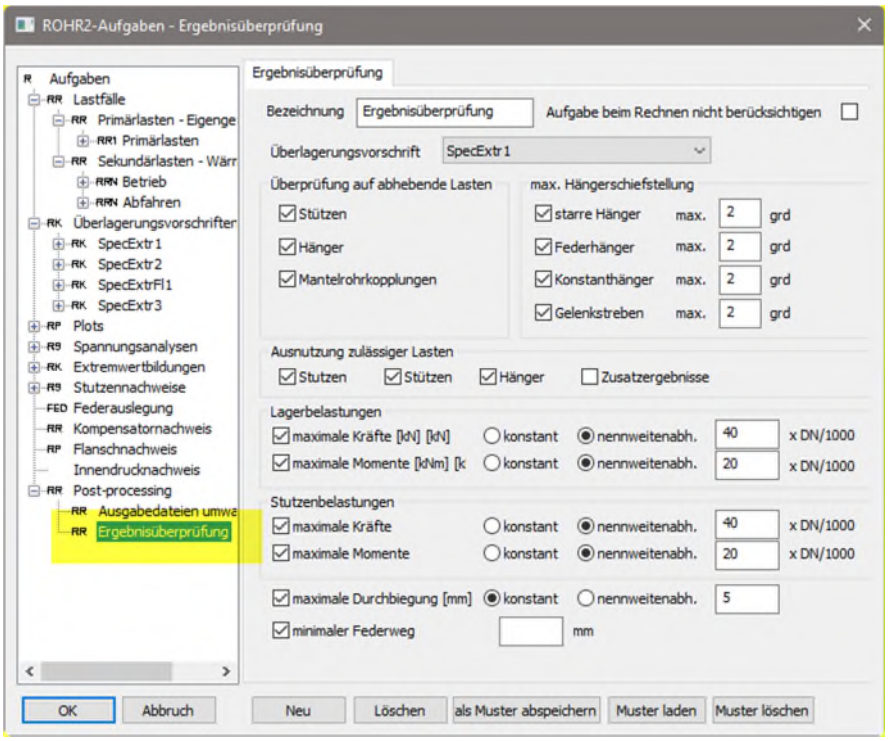
3.12.4 Post-Processing

Post-Processing-tasks are executed after the calculation. They are used. e.g. to run automatic export functions. The post -processing tasks are selected in the File | calculate window.

Available types of Post-Processing-tasks:

Convert output files	Convert output files of load cases, extreme value calculation and stress analyses in the same calculation into RTF or PDF format
Results check	The results of load cases in a calculation process are checked, e.g. for upward loads. A detailed configuration of the test process is made here. Optionally the results of loac case combinations can be checked.
Report generation	Report generation. Appendix optionally
Export CSV	Export in CSV format
Export support results	Export of support data to LICAD, FlexperteE3D/PDMS, …

Select the criteria of the results analysis:

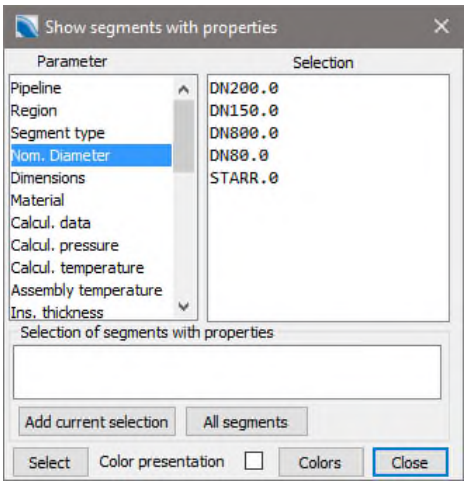


3.13 Checking the input data

After finishing the piping model and defining the first load cases it may be useful to check the input data. This can be done by the function *Segment parameters* and *List data*

3.13.1 Segment parameters

The function *Properties* | *Segment parameters* enables to check the input data by selecting segments by means of specified properties.



Choose the desired parameter.
Several entries can be entered at the same time in the select box.

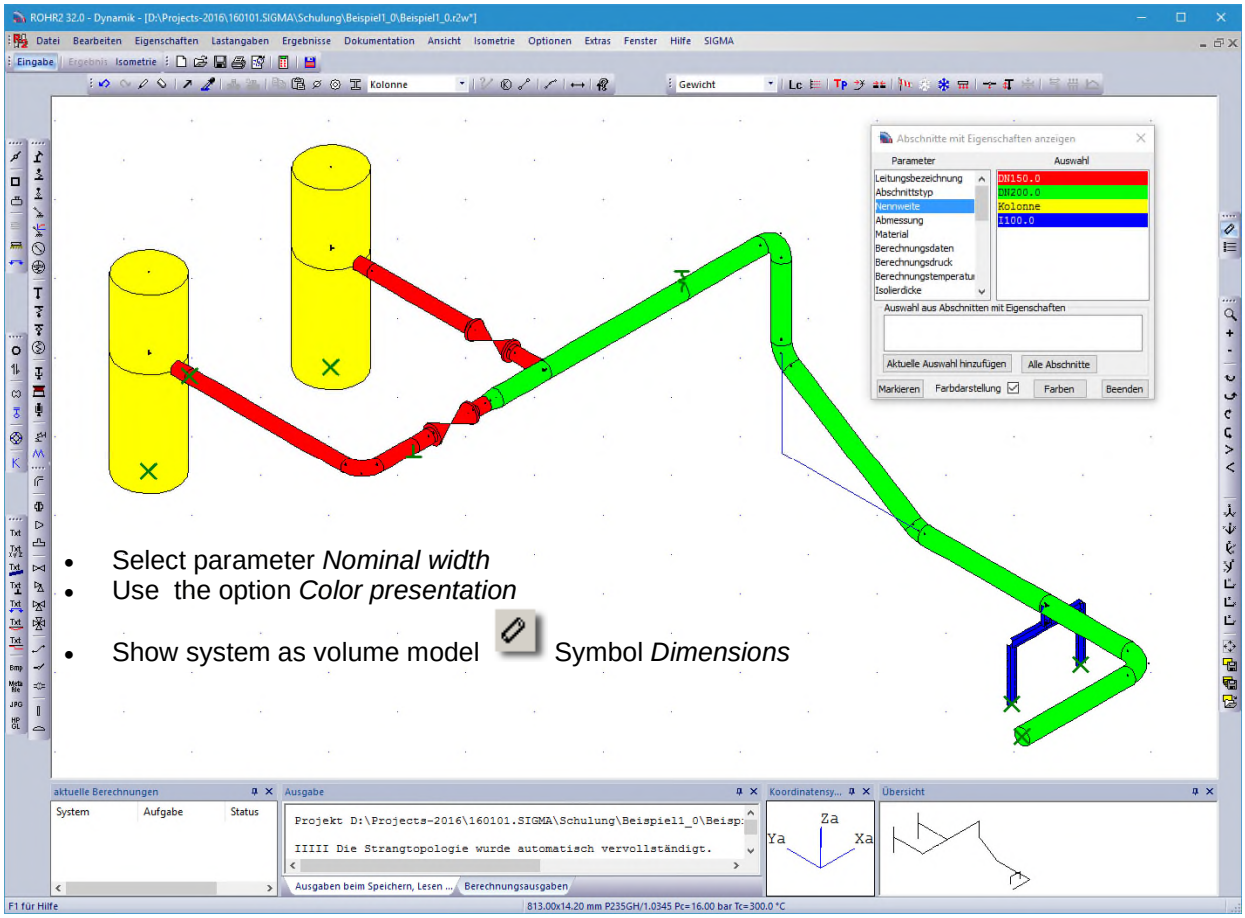
Selection

The segments containing the chosen parameters are selected

Colored display

The properties are shown in different colors.

Example, checking assigned dimensions



- Select parameter *Nominal width*
- Use the option *Color presentation*
- Show system as volume model Symbol *Dimensions*

3.14 Correcting and adapting the input data

This chapter shows some common mistakes when modeling a piping system and their correction.

3.14.1 Geometry

Modifying the segment length

- Double-click on the segment
- Input of a new length with identical orientation or input of the X-, Y-, and Z-coordinate in the global coordinate system
- Displacing the start node or end node

Moving a node

- Function Edit| Move or 
- Click at the node, press any key and enter the displacement vector
- Please note: it is recommended to move an intermediate point only between two neighboring segments.

Moving parts of the system

- Function Edit| Select or 
- At first the required part of the system must be highlighted by the *Select Start to end node* command. After that click at a node in the highlighted region, press any key and enter the displacement vector.
- Please note that moving a region is followed by the automatic modification of the neighboring segments.
- Cancel the selection by *ESC* or by *End function* in the context menu /right mouse button

3.14.2 Dimensions

Change the dimensions of one segment

- Open the dialog window *Data of segments* by a double-click on a segment and choose the right nominal width.

Modifying dimensions for a region/part of the system

- At first select the desired region  . E.g. by highlighting the main part by *Select*| *Start node… end node* and adding segments by *Select*| *Individual elements*.
- After that choose a dimension by *Edit*| *Pipe dimensions* or  and assign by *OK*.
- Cancel the selection by *ESC* or by *End function* in the context menu /right mouse button.

3.14.3 Operation data

Please note that the definition of operation data is always load case dependent
That's why it is required to select the load case at first and then change the data.

- Then select the desired region . E.g. by highlighting the main part by *Select*| *Start node… end node* and adding segments by *Select*| *Individual elements*.

- Use *Loads*|*-Operation data*| *Referring to load cases* or
- Choose operation data or generate a new record
- Assign data to the highlighted region by *OK*.

- If desired the new generated record can be selected and copied into other load cases
- Cancel the selection by *ESC* or by *End function* in the context menu /right mouse button.

3.15 Calculation

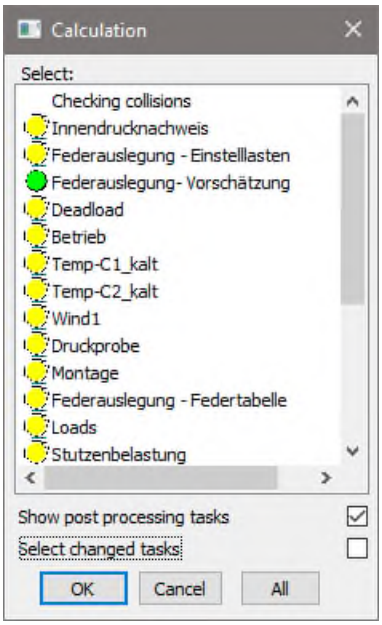
In the program directory there are some examples enabling to check the results and the documentation of ROHR2 analysis as shown below.

Function *File*| *calculate* or

All tasks, shown in the window can be calculated single or, by using the option *All*, the analysis is done for all tasks.

The *internal pressure check* entry offers the opportunity to check components for internal pressure.

Another check is the *collision* test, analyzing if there is a conflict between segments of the piping model.

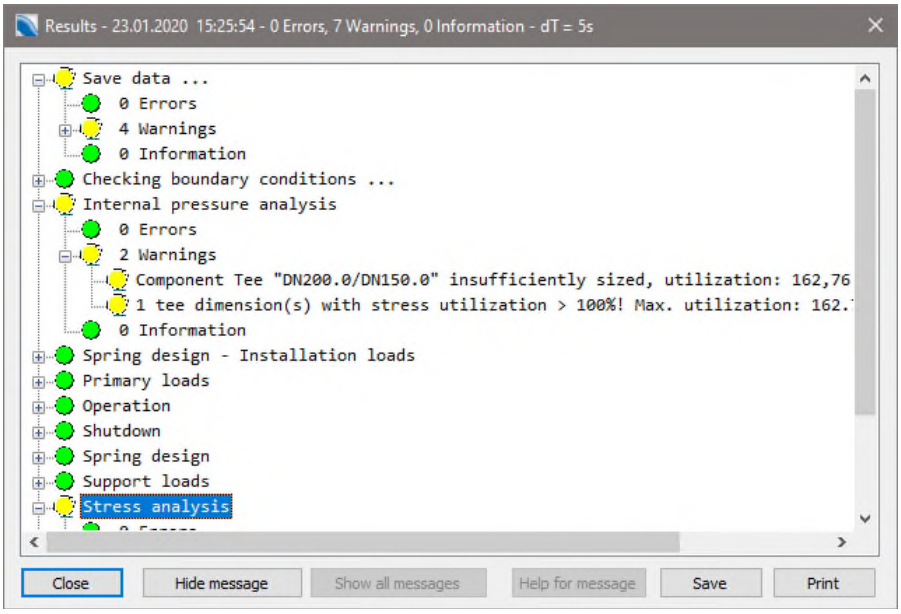


The colored symbols beneath the load cases are showing if the results have been updated.

results updated

the task has been changed after the last calculation and the results are not updated.

After finishing the calculation the results are summarized in an output window.

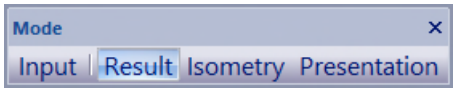


	green	Everything is alright. The calculation runs without any mistakes
	blue	General system information
	red	Error messages. a problem occurs: The calculation result is totally missing or it is not recommended to use the result (e.g. the calculation did not use the required analysis accuracy)
	yellow	Warnings, **Check the results!**

Select one of the messages in the window to reach a help text.

3.16 Results analysis

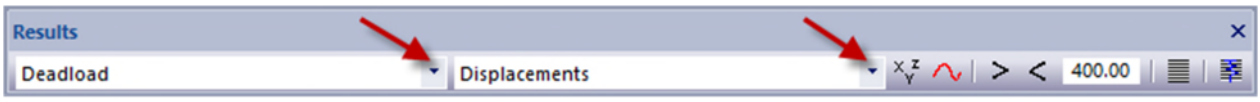
Switch to *Results* mode in the *Mode toolbar* to show the results of the calculation.



Here the load case results as well as the stress analyses with stress utilization can be shown graphically. In the following different opportunities of results analysis in ROHR2 are shown:

3.16.1 Load case results

At first select the load case in the results mode



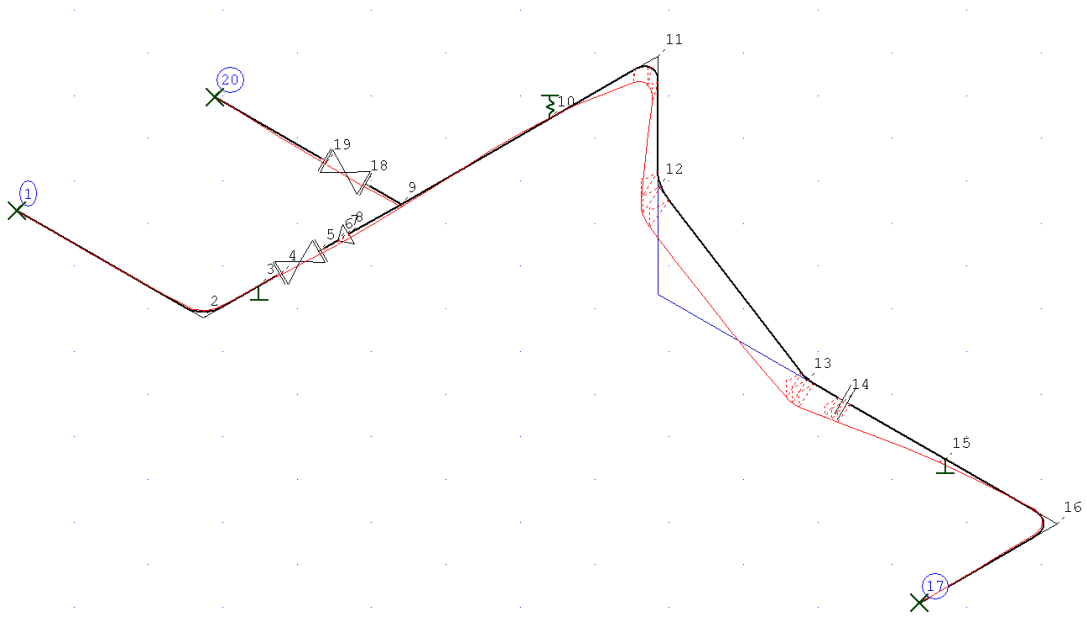


Fig.: deformed structure

Get single results by double-clicking at a node and selecting the register *Single results*

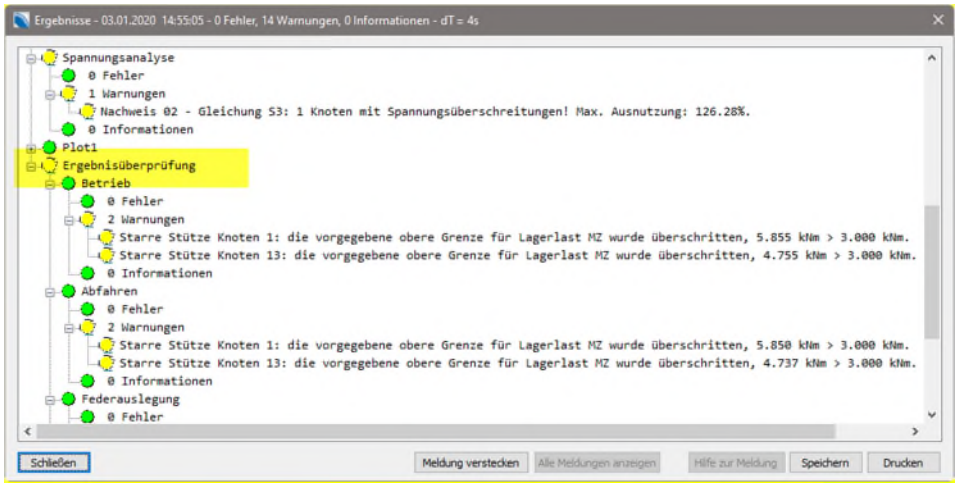
To get an overview in tables, use the symbols

 / Cross section results or  / Loads on supports.

By this way different results can be shown, e.g.:

- max. bending in the load case *Weight*
- Loads on supports at nozzle C1 in the load case *Weight*
- vertical expansion at the spring support in load case *operation*
- the spring type selected by ROHR2

In the results window, the results of the automatic result check are displayed if permissible values are exceeded.



3.16.2 Stress analysis

At first select the required stress analysis in the results mode.

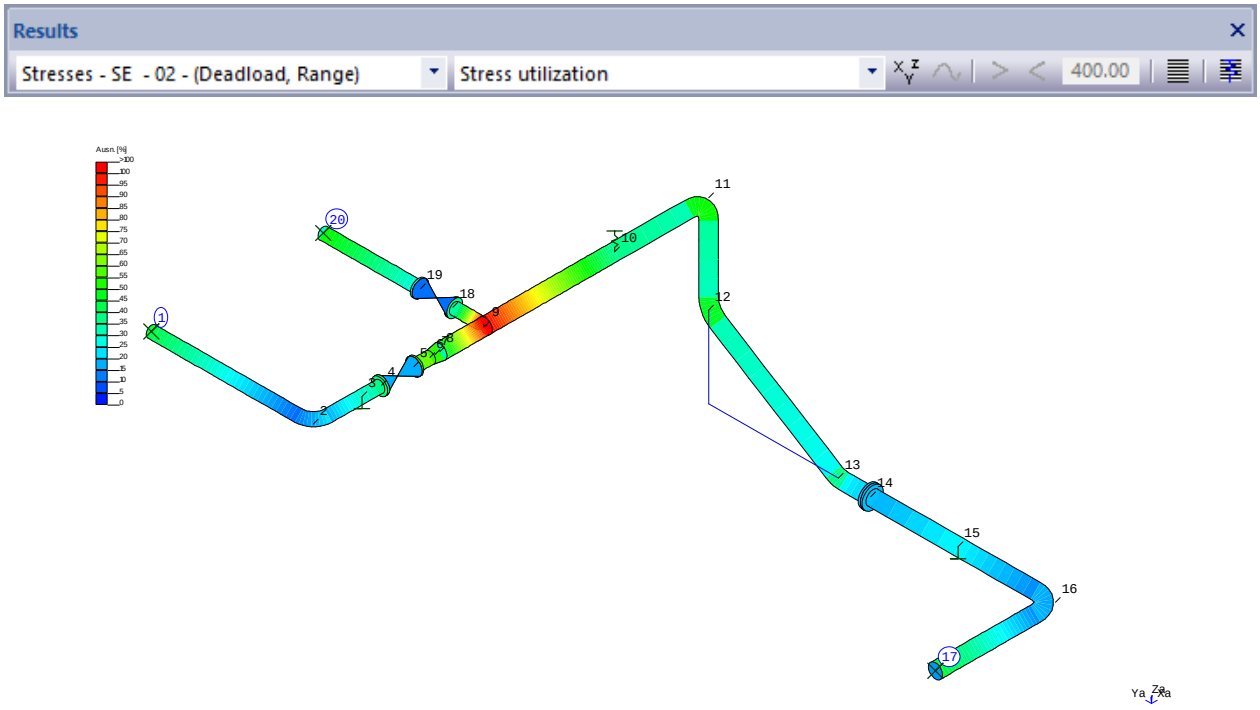


Figure: Stress analysis equation 4 (weight + range)

Get single results by double-clicking at a node and selecting the register *Stress analysis*

To get an overview in tables, use the symbol

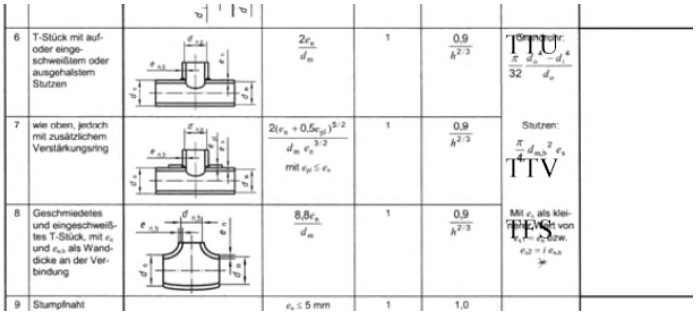
 / cross sectional results.

By this way different results can be shown, e.g.:

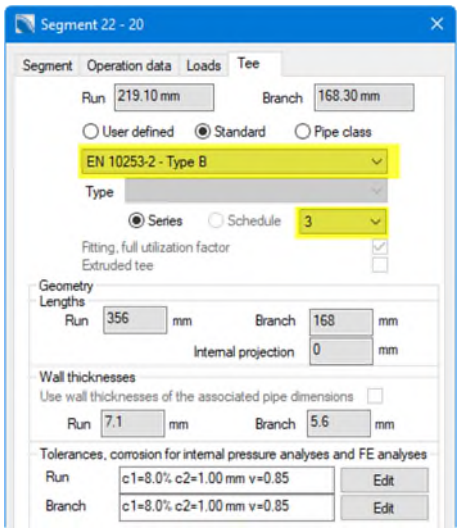
- maximum stress utilization of analysis 1 (weight)
- maximum stress utilization of analysis 1 (weight + range)

3.16.3 Optimization by inserting a tee at a branch

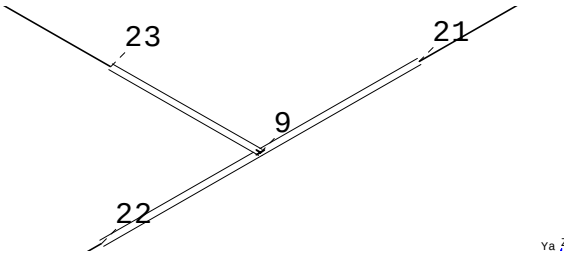
The stress codes (here EN 13480) are making differences at branches like this



Select a tee by Edit| Insert component| tee



Tee acc. to EN 10253-2 Type B, row3



Structure with tee

Tip
The stress intensification factor is modified automatically after selecting the tee.

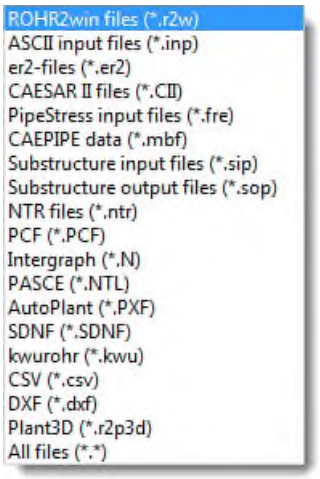
4 Checking CAD/CAE import with ROHR2 Interfaces

The program system ROHR2 offers a wider angle of interfaces to CAD and CAE systems. For details please refer to the ROHR2 Interface feature list.

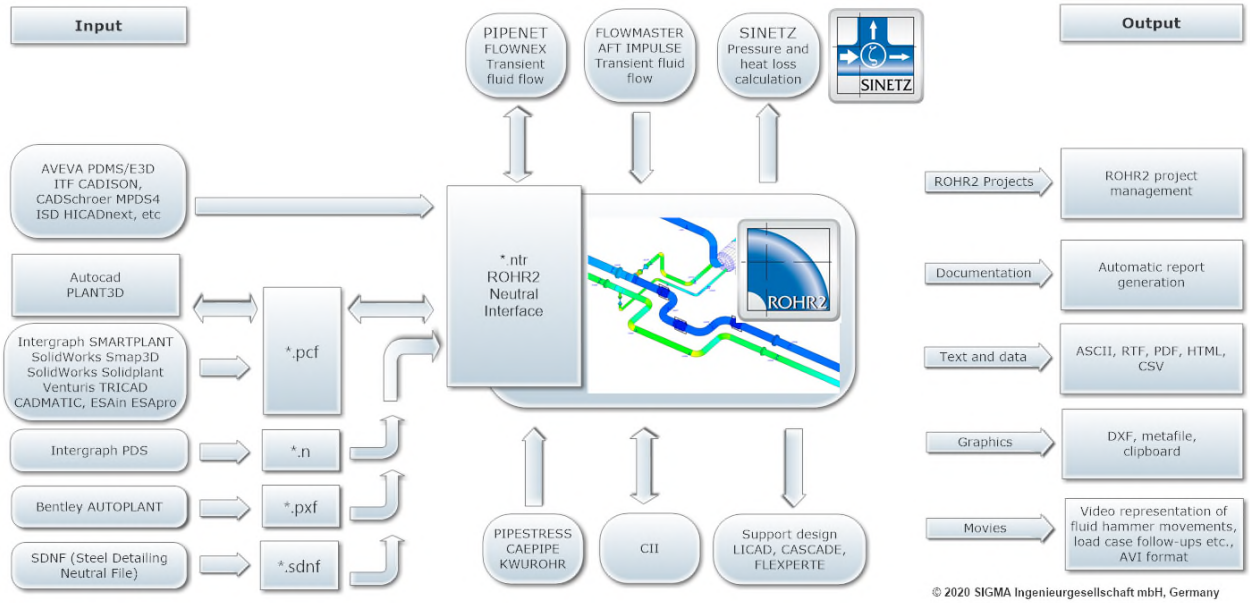
The ROHR2 standard program delivery includes:

- Neutral CAD Interface including Export AVEVA PDMS - ROHR2
- CAESAR II, PIPESTRESS, CAEPIPE Import Interface
- SINETZ, FLOWNEX, PIPENET Export interface
- DXF format, KWUROHR Import Interface
- Export Interface into the Support Design Programs LICAD, FLEXPERTE, CASCADE

Using ROHR2 interfaces with the ROHR2 trial license
All import formats are available in the test license. Please note: some of the interfaces modules are optionally available and not part of the ROHR2 standard configuration.
Load the data by means of ***File| Import*** command and get an overview on the capacity of the import interfaces.



Using ROHR2 interfaces with the ROHR2 viewer
The ROHR2 viewer program (free-of-charge, available on www.rohr2.com) can be used to test some of the ROHR2 interfaces.



5 ROHR2nozzle - Introduction / Tutorial

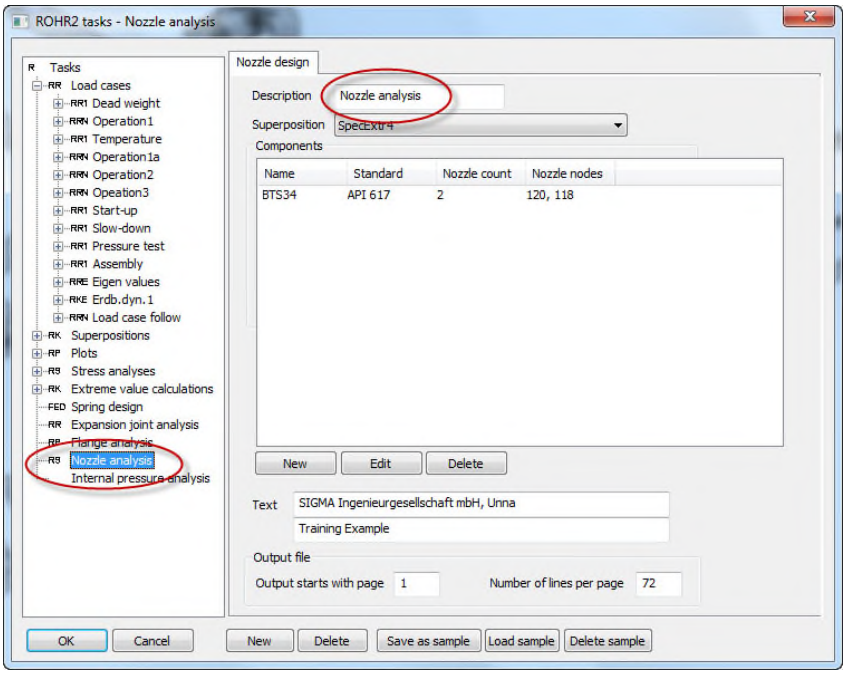
5.1 Program start

The module ROHR2nozzle is integrated into the ROHR2win user interface.

Additionally ROHR2nozzle can be carried out as a stand-alone application. For this purpose start the program from the directory ..\ROHR2\R2nozzle. In this case no data from a ROHR2 project are taken over, but the input data has to be entered manually.

5.2 Introduction into the analysis of nozzles by ROHR2nozzle

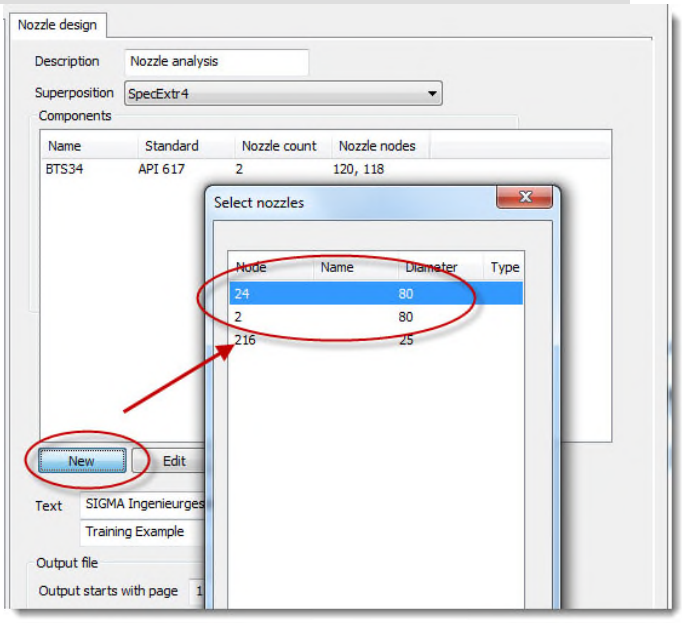
For the analysis of a nozzle a new task has to be defined in ROHR2 by means of the *Loads* | *Tasks* command. The calculation of the nozzles is part of the ROHR2 calculation started by see *file* | *calculate*



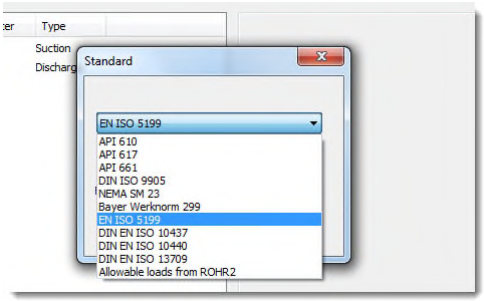
See *ROHR2nozzle commands* for generation and modification of a nozzle and the description of the ROHR2 tasks are in the ROHR2win manual.

Selecting nozzles

Enter the nozzles to be calculated here. Depending on the calculation rule a minimum number or maximum number of nozzles must be entered Additional nozzles can be added later.

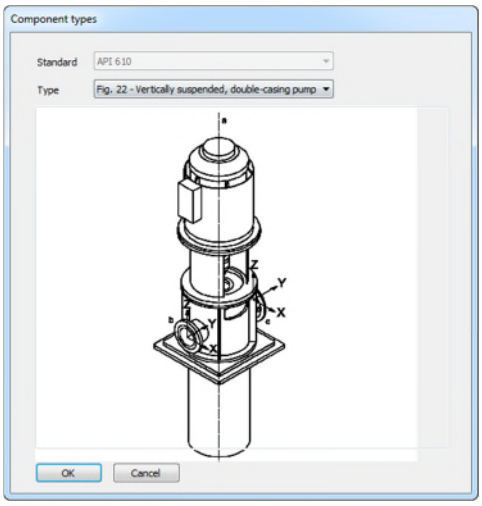


Selecting a standard



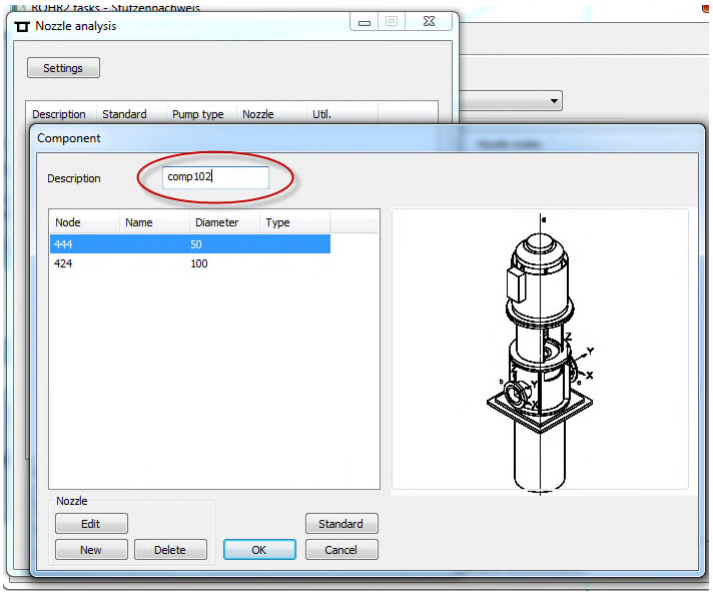
Component type

Depending on the selected calculation standard it is required to choose a component type.



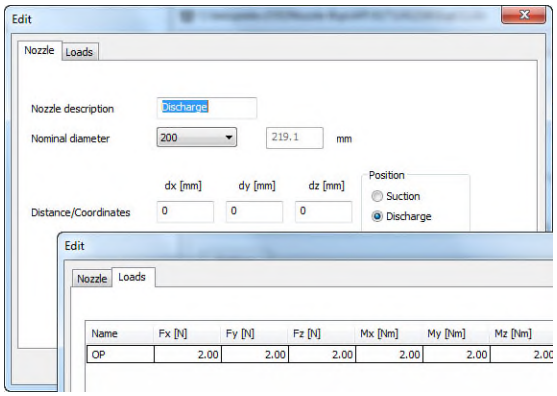
Showing the component

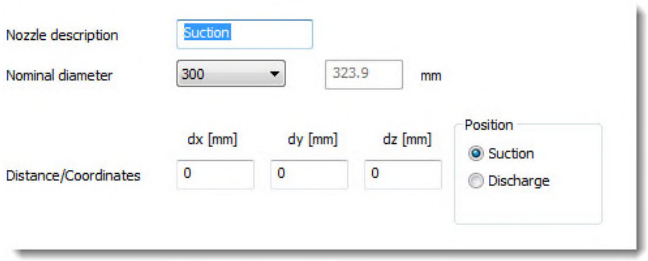
The selected component is shown including a sketch graphic. Insert a description to give the component a unique name. The identifier is also used for the documentation of the project.



Edit component

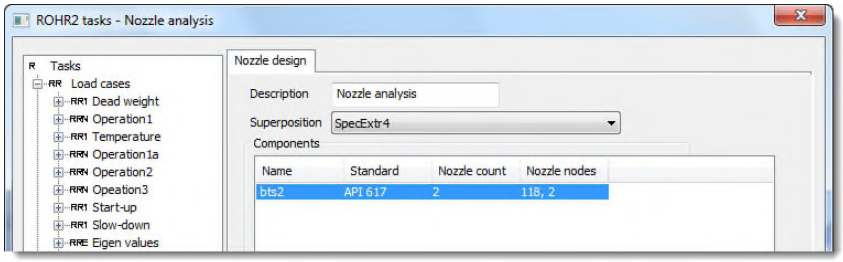
Use the Edit function to define the nozzle in the dialog window *Edit nozzle*
By means of the *Edit* and *Loads* registers the input data of the nozzle can be modified or additional nozzles may be added.





Depending on the selected standard a suction and discharge nozzle need to be defined.

When the nozzle definition is finished the component will be inserted into the nozzle table.



5.3 Calculation

The calculation of nozzles is integrated in the ROHR2 calculation process.
If ROHR2nozzle is used as a stand-alone application the program offers a command for the start of the calculation.

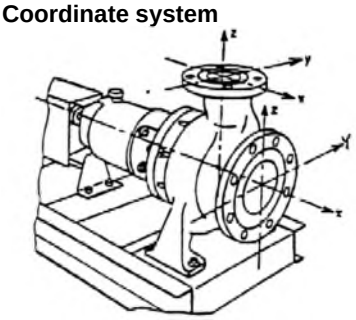
5.4 Results representation

The results of the nozzle analysis are shown by the ROHR2 function *Results* | *View output file* | *Nozzle analysis*.
If ROHR2nozzle is used as a stand-alone application the program offers a command for the report generation.

Example:

EN ISO 5199 - Centrifugal pumps - Class 2
Component: Pump 4711-1

Coordinate system



Nozzle	Node	Diameter NPS	Position	dx [mm]	dy [mm]	dz [mm]
P17S	P1S	80	Suction	2902.0	-3850.0	7670.0
P17D	P1D	80	Discharge	2702.0	-3850.0	7950.0

Analysis

Nozzle		P17S		P17D
Load case		Temp-C2_kal_OC1		Betrieb

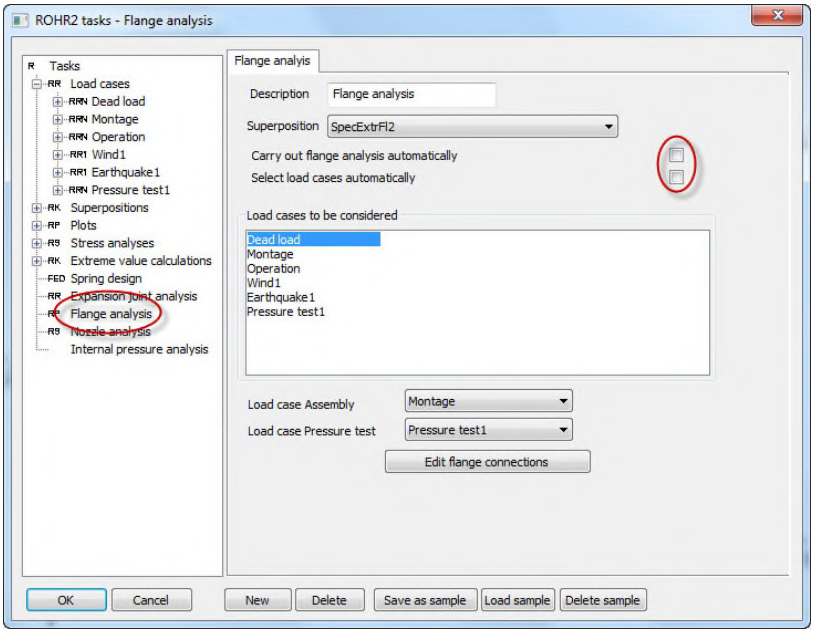
		calc.	allow.	Util. [%]		calc.	allow.	Util. [%]
Fx [N]		973.7	1000.0	97.37		0.0	900.0	0.00
Fy [N]		-414.3	900.0	46.03		-14.5	820.0	1.76
Fz [N]		-184.8	820.0	22.53		-197.9	1000.0	19.79
Mx [Nm]		-166.3	640.0	25.99		-1.3	640.0	0.20
My [Nm]		115.0	460.0	25.01		0.0	460.0	0.00
Mz [Nm]		-225.6	520.0	43.38		-0.0	520.0	0.00

6 ROHR2flange - Introduction / Tutorial

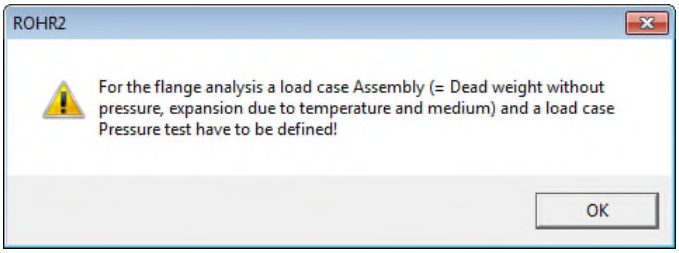
The module ROHR2flange is used to analyze the leak tightness flange connections.
The flange analysis includes the loads at flanges, determined in the stress analysis as well as the assignment to referring load cases.

6.1 ROHR2flange overview

Start ROHR2flange in ROHR2win by *Loads* | *Tasks*| *Flange analysis*.



Using the option *Flange analysis automatically* enables to consider the flange analysis in the ROHR2 project or to treat it as a separate process.



If this warning message occurs, the further processing requires defining the load cases *assembly* and pressure test in *Loads*| *Tasks*.

ROHR2flange icon in the Windows taskbar

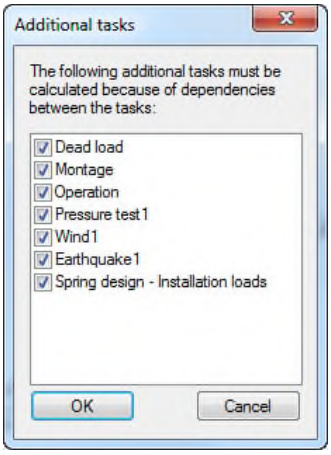
The program module ROHR2flange is represented by a symbol in the windows task bar as an independent process.

6.1.1 Load additional tasks

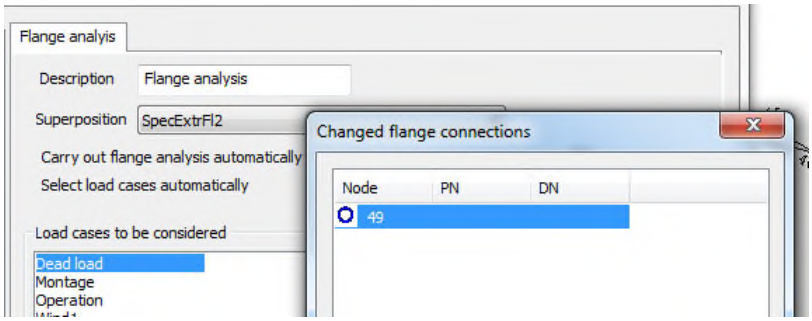
Loads | Tasks > Flange analysis

The program ROHR2flange is starting and loading the file.
If the dialog *Additional tasks* opens there are some load cases missing for the ROHR2 calculation. These load cases need to be calculated before processing the flange analysis.

This guarantees that the current loads are part of the flange calculation.



6.1.1.1 Overview Changed flange connections



All flange connections that have been changed after the last calculation process are listed in the dialog window *Changed flange connection*.
If no modified flanges are available, the dialog window does not open.
All flange connections are listed here at the first program start.

Pre-settings

The dialog window *Pre-settings* opens where the settings for all flanges in the system are to be made, see 6.1.2.

OK

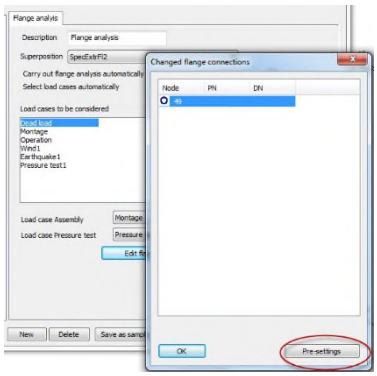
OK closes this window. The dialog *List of flange connections* opens, 6.1.3.

6.1.2 Flange analysis - Basic settings

Flange analysis > Changed flange connections > Pre-settings

Pre-settings

Select the global settings for all flange connections in the calculation system.



Use material from ROHR2-model

If activated, the material of the ROHR2-model will be used. It is not possible to insert another flange material in this case.

Material dialog

Materials opens the ROHR2 material dialog to select a material from the ROHR2 project material database.

Material

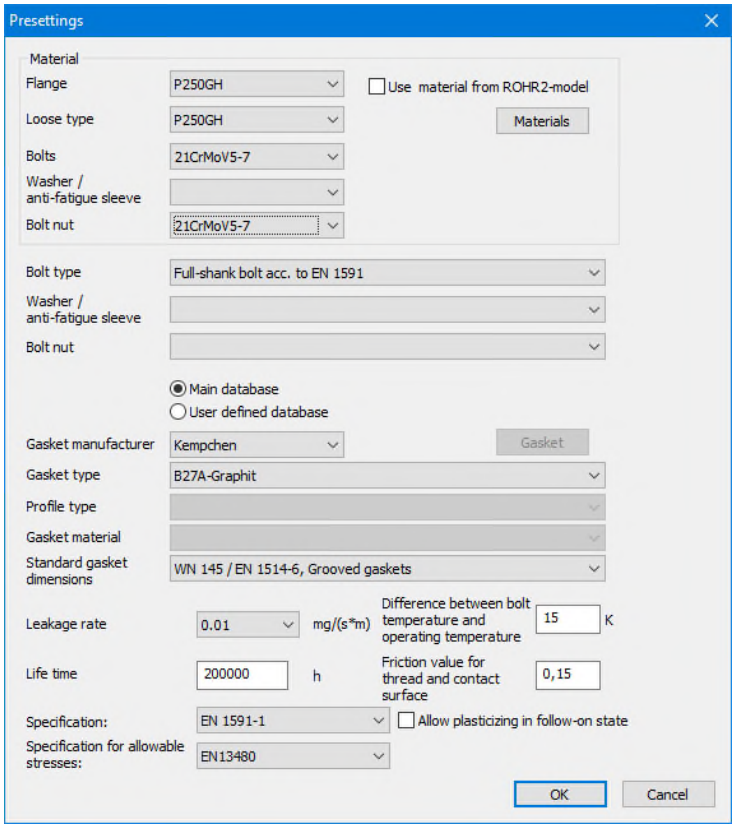
Select the material for

- ***Flange***
- ***Loose type***
- ***Bolts***
- ***Washer/anti-fatigue sleeve***
- ***Bolt nut***

The bolts of the flange connections have the same material.

Parameters at this example:

Flange: P250GH
Bolt: 21CrMoV5-7
Bolt type: EN 1591
Gasket: Kempchen B27A
Gasket dimensions: EN1514-6, Kammprofil



Gaskets

A gasket can be taken from the main database. I f required user gaskets can be defined and stored in a gasket database.

Specification

Select calculation standard. Available are

- EN 1591-1
- ASME VIII Div.1

Specification for allowable stresses

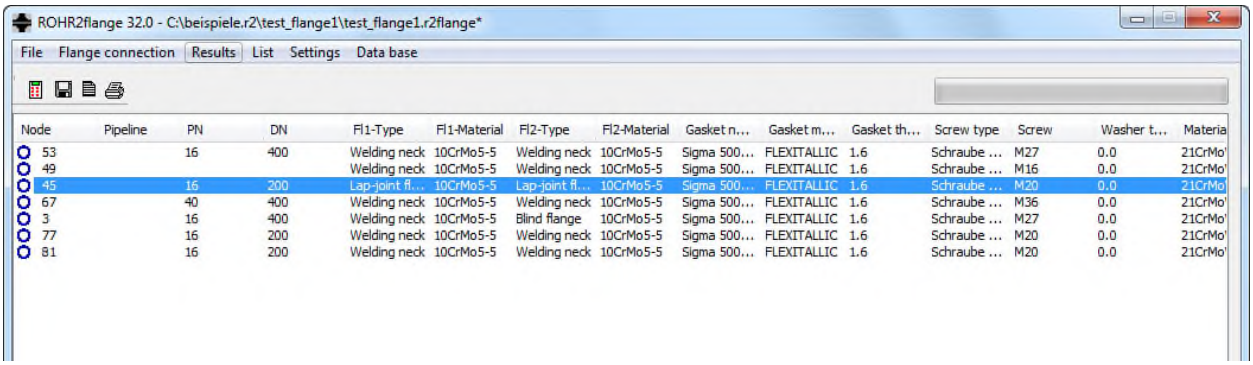
Select a stress code for the determination of the allowable stresses from the number of stress codes included in ROHR2, e.g. EN 13480, ASME B31.1, etc.

OK

After confirmation with OK the values are assigned to any particular flange connection. The dialog window *List of flange connections* opens.

6.1.3 List of flange connections

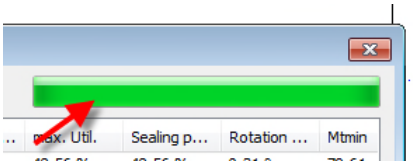
This is the main window in the ROHR2flange analysis.

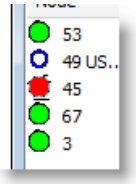


A click on the calculator symbol starts the analysis.

Elements of the dialog window

Calculation status
Indicates the status of the running calculation.





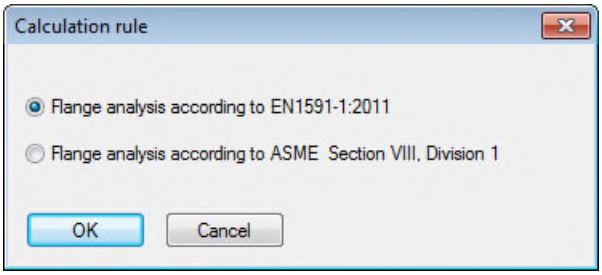
The results additionally are represented by a graphic symbol at each dimension.
It is
Red dot utilization > 100%,
Green dot utilization <=100%.
Blue dot no flange analysis carried out up to now

Change the order of the flange parameters by a click into the table header.
E,g. The minimum, mean and maximum tightening torque now is shown in the list, parameters Mtmin, Mtmax,

Mtmin	Mtmax
475.61	803.27

Calculation rule flange

The command Settings R*ule* in the *List of flange connections*, 6.1.3 opens the dialog window *Calculation rule to* select the flange analysis norm.

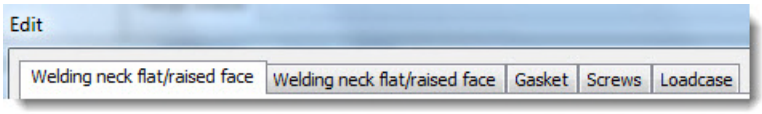


Select between
- EN1591-1:2011
- ASME BPV Section VIII Division 1

6.2 Edit flange connection

Flange connections can be edited by a double-click into the list of flanges.
Editing of the flange connections occurs in the following tabs:

- 1 Input flange 1, 6.2.1
- 2 Input flange 2,
- 3 Gasket,
- 4 Bolts (screws),
- 5 Load case,

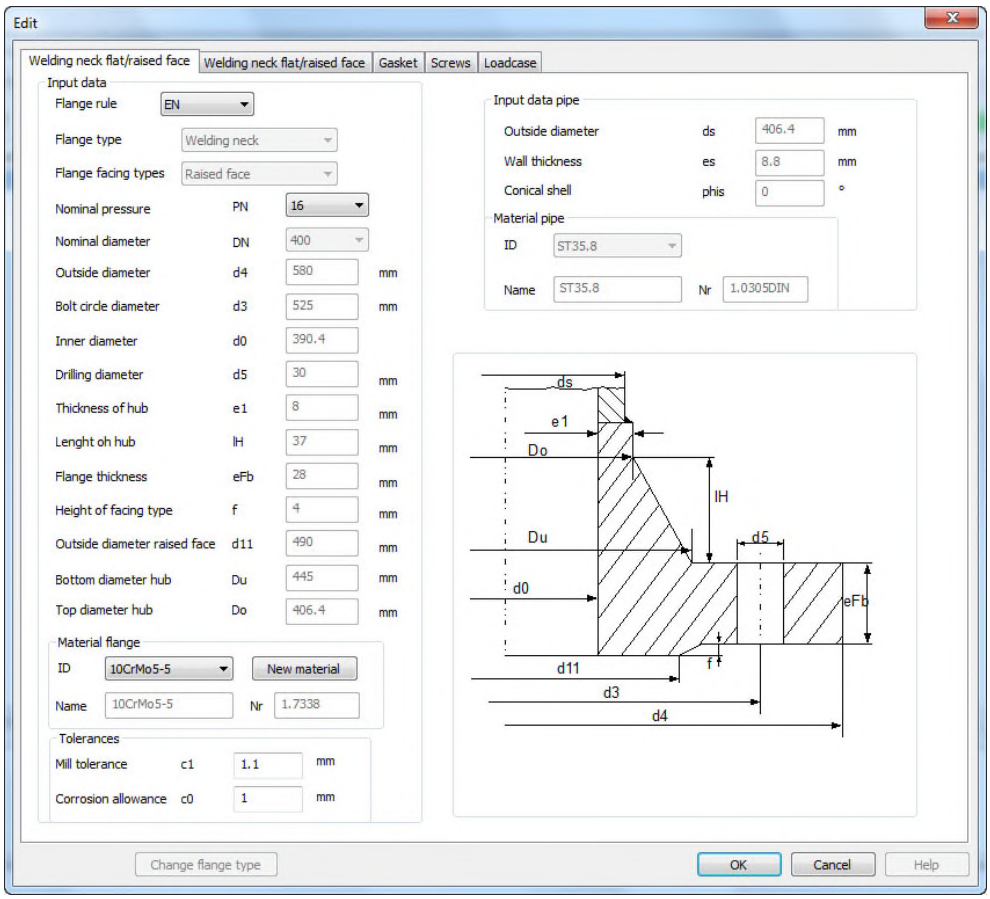


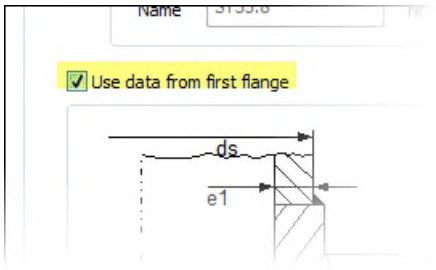
The header shows the currently selected type of flange, here: Welding neck flat.

The geometry of the flange is shown graphically in the dialog.
Basic functions of the edit controls:

6.2.1 Edit flange /gasket/ bolts

Flange 1 / 2

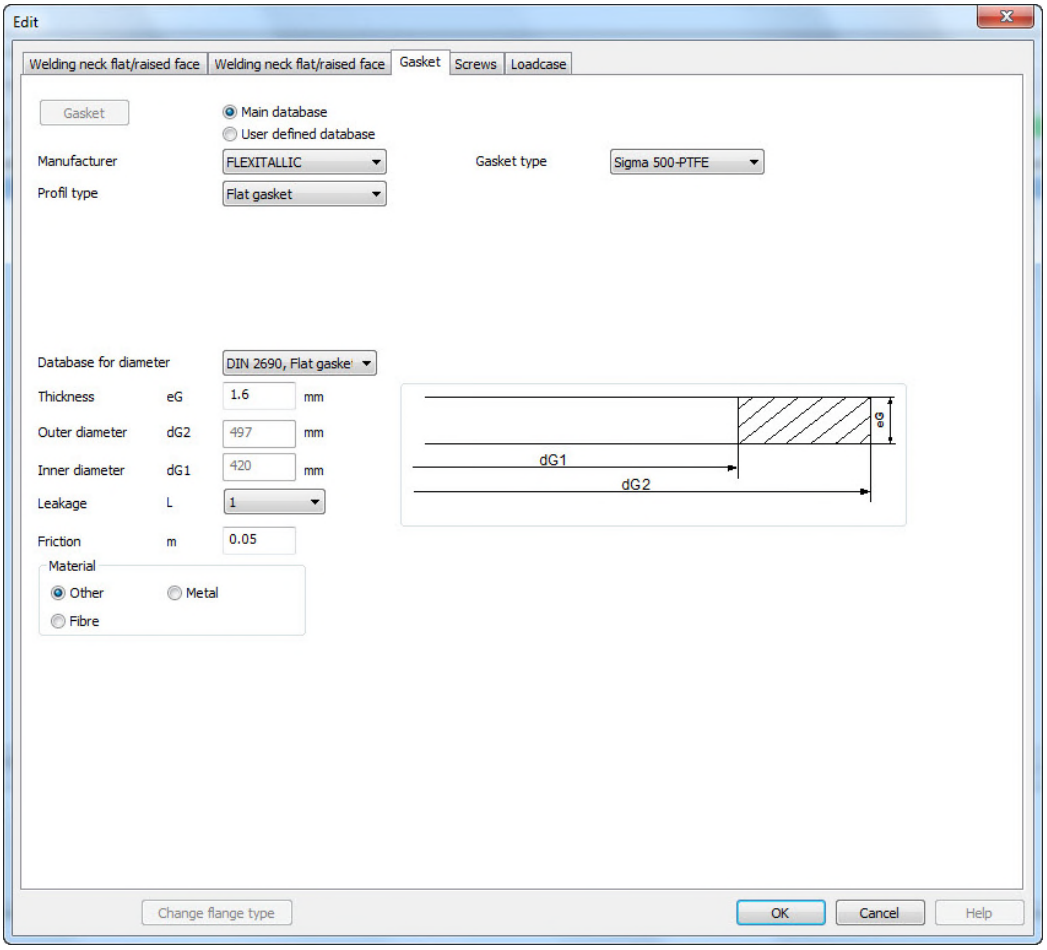




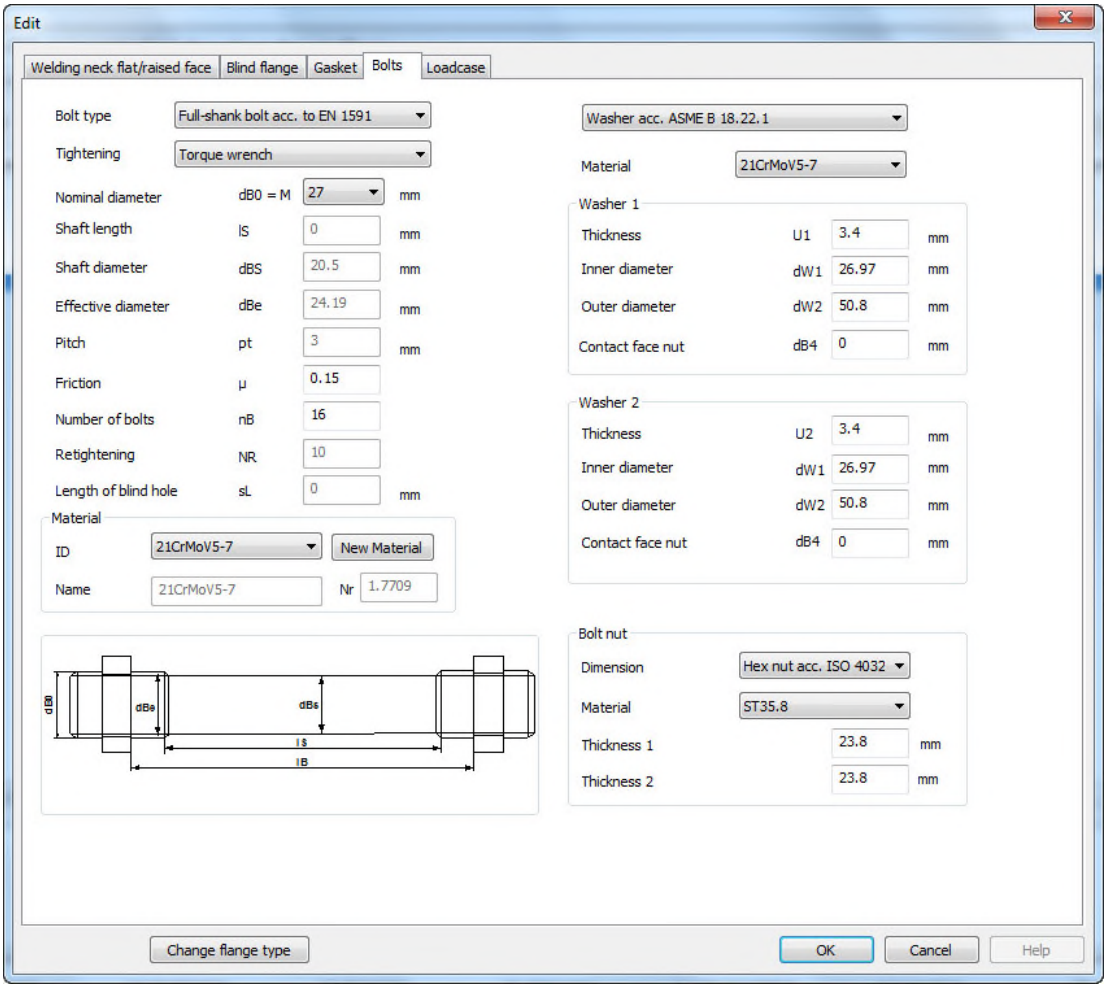
Use data from first flange

The function *Use data from first flange* can be used to copy the entire parameters of the first flange and assign them to the second one.
This function is available at identical flanges and gaskets.

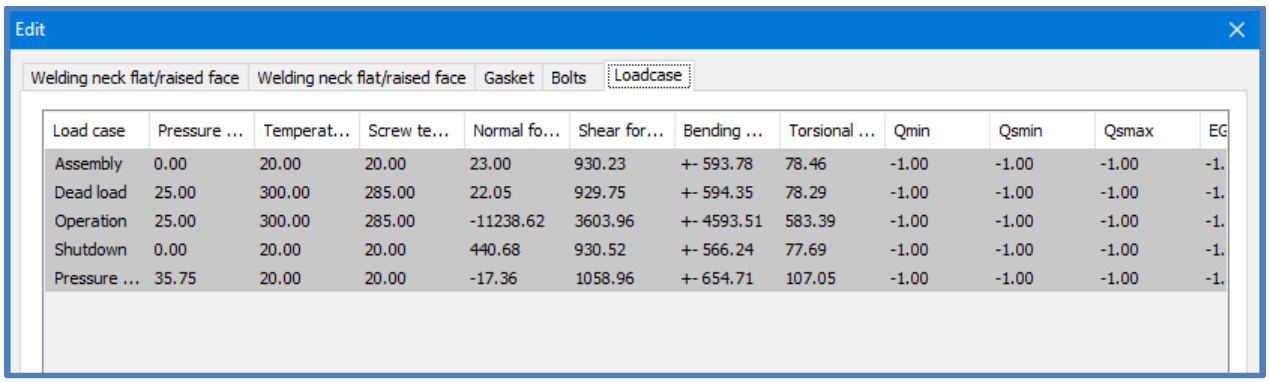
Gasket



Bolts



Loads per load case



Load case	Pressure ...	Temperat...	Screw te...	Normal fo...	Shear for...	Bending ...	Torsional ...	Qmin	Qsmin	Qsmax	EG
Assembly	0.00	20.00	20.00	23.00	930.23	+- 593.78	78.46	-1.00	-1.00	-1.00	-1.
Dead load	25.00	300.00	285.00	22.05	929.75	+- 594.35	78.29	-1.00	-1.00	-1.00	-1.
Operation	25.00	300.00	285.00	-11238.62	3603.96	+- 4593.51	583.39	-1.00	-1.00	-1.00	-1.
Shutdown	0.00	20.00	20.00	440.68	930.52	+- 566.24	77.69	-1.00	-1.00	-1.00	-1.
Pressure ...	35.75	20.00	20.00	-17.36	1058.96	+- 654.71	107.05	-1.00	-1.00	-1.00	-1.

This overview shows the load case which cannot be edited here.

6.3 Report, documentation

Part of the flange analysis is the generation of a calculation report

Use the symbol *Output file* to open the generated output document
Results - - Show overview: Showing the first page of the flange analysis report

SIGMA	SIGMA Ingenieurgesellschaft mbH Bertha-von-Suttner-Allee 19 D 59423 Unna	ROHR2
Comm.: Project:	160101.SIGMA ROHR2-Schulung - Beispiel 1 Leitung DN200/150	Date: 26.01.16

1 - Flange analysis acc. to EN 1591 at node 17 // DN 150 - PN 25

Abstract

Flanges	**Flange 1**	**Flange 2**
Flange type	Welding neck	Welding neck
Standard	EN 1092-1/11/PN25	EN 1092-1/11/PN25
Facing type	Raised face	Raised face
Nominal pres.	PN 25	PN 25
Nominal diam.	DN 150	DN 150
Material	P250GH / 1.0460	P250GH / 1.0460
Description		

Pipes	**Pipe 1**	**Pipe 2**
Outer diam.	168.30 mm	168.30 mm
Wall thick.	5.60 mm	5.60 mm
Material	P235GH / 1.0345	P235GH / 1.0345

Bolts	
Type	Full-shank bolt acc. to EN 1591 (ISO 4014)
Diameter	M 24 x 3.00
Number	8
Friction μ	0.15 (average normal state)
Tightening	Torque wrench
Material	21CrMoV5-7 / 1.7709

Gasket	
Name	B27A-Graphit
Manufacturer	Kempchen
Date of creation	20.1.2014
Type	Flat gasket
Leakage	0.01000000
Dimensions	190.00 x 170.00 x 4.90
acc. to	WN 145 / EN 1514-6, Grooved gaskets

Results

Min. tightening torque
Mtnom 167.47 Nm per bolt
At a max. tightening torque of
MaxMt 260.00 Nm per bolt
the first component to fail is/are the Flange 2

Flange 1	Flange 2	Bolts	Gasket
99.95 %	99.95 %	47.67 %	18.16 %

Average tightening torque 213.73 Nm per bolt is determined

Util.	Bolts	Gasket	Flange 1	Flange 2	Deformation
Assembly	24.38 %	11.83 %	36.43 %	36.43 %	0.19 °
Dead load	28.47 %	6.62 %	63.35 %	63.35 %	0.17 °
Operation	28.72 %	9.37 %	66.60 %	66.60 %	0.21 °
Shutdown	31.10 %	9.72 %	40.97 %	40.97 %	0.15 °
Pressure test	16.82 %	6.14 %	26.54 %	26.54 %	0.18 °

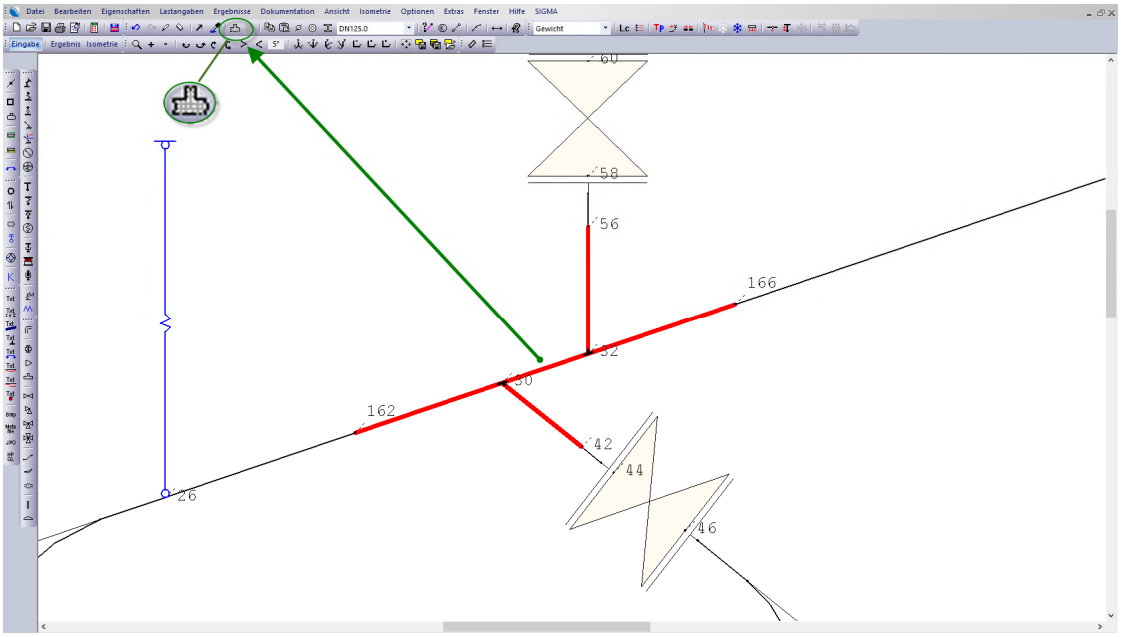
7 ROHR2fesu - Introduction

Program start ROHR2fesu

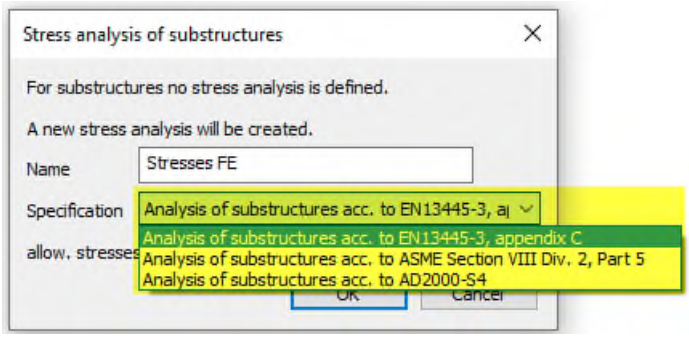
ROHR2fesu is integrated into the graphical user interface of ROHR2 (ROHR2win).
The program module can be carried out by the ROHR2win commands *Edit*| *Substructures*| *Create* or *Edit*| *Substructures*| *Insert*.

7.1 Define the coarse model in ROHR2

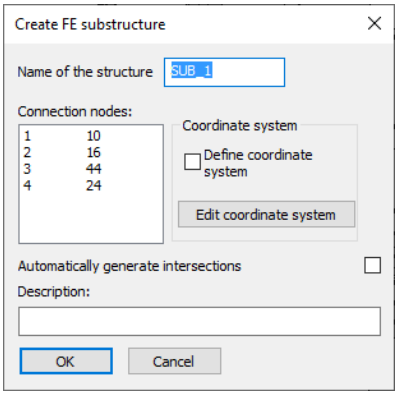
- At first the substructure is modeled as framework including all available properties.
  For eccentric or tangential placed nozzles rigid beams (profile type: RIGID) may be used as auxiliary construction. They will be ignored when creating the ROHR2fesu model. In this way a model will be created by the input of essential, geometric correctly placed beams.
- Next the transition points between framework and substructure are defined.
  Maybe additional nodes in the framework system are required to get useful transition points.
- Supports inside the substructure are not allowed. If required, they must be separated from the substructure by inserting intermediate nodes or other appropriate steps.
- The region to be used for the substructure must be selected (highlighted) in the framework up to the transition points.
- Now from this selected region the ROHR2fesu model is created by means of the framework data.



If not yet available, a new stress analysis for the FE calculation is generated automatically. You can choose between EN, ASME and AD2000:



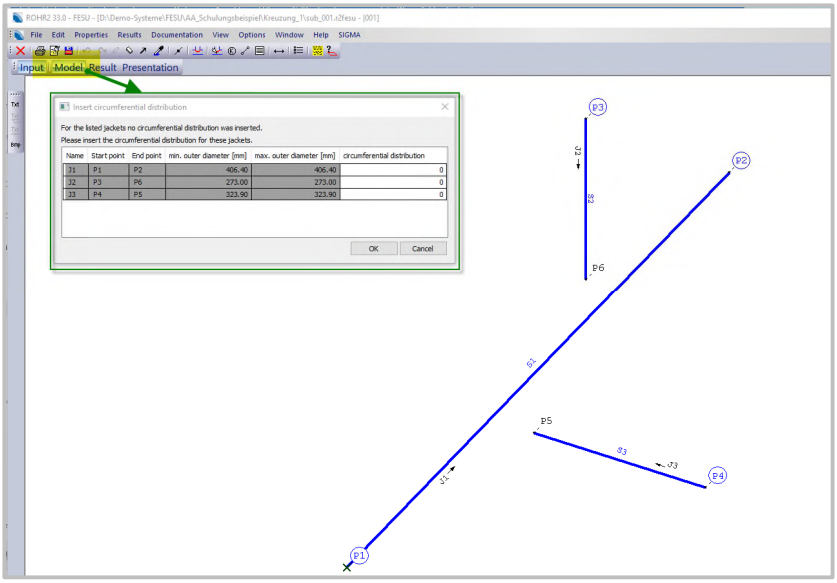
Designations and descriptions can be added or adapted for the substructure:



7.2 Take over coarse model using ROHR2fesu

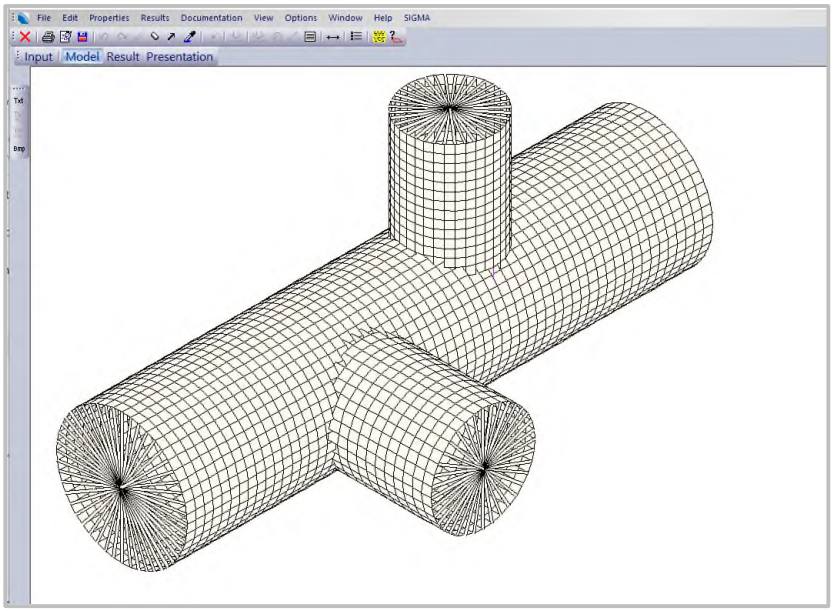
After confirmation via the "OK" button the ROHR2fesu module opens. At first a model similar to that of the framework appears. In this input mode only the center lines of the superelements are displayed.

By clicking on mode *Model* a window opens for the input of the circumferential distribution of elements for the shell model.
It is recommended to use 1/10 to 1/8 of the nominal diameter as a first approximation. However, for small nominal diameters the circumferential pitch should not be smaller than 12 and for large nominal diameters not necessarily larger than 200. Since the given numerical values are a first approximation, the mesh should be checked and, if necessary, adjusted accordingly.

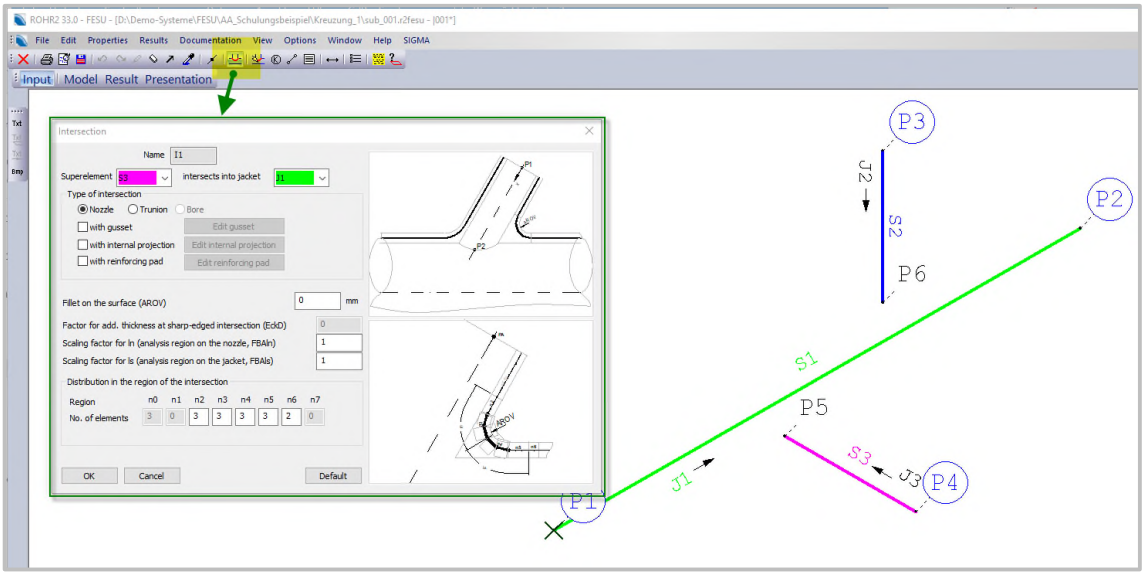


Name	Start point	End point	min. outer diameter [mm]	max. outer diameter...	circumferential distribution
J1	P1	P2	406.40	406.40	48
J2	P3	P6	273.00	273.00	36
J3	P4	P5	323.90	323.90	36

The model is then meshed and displayed.

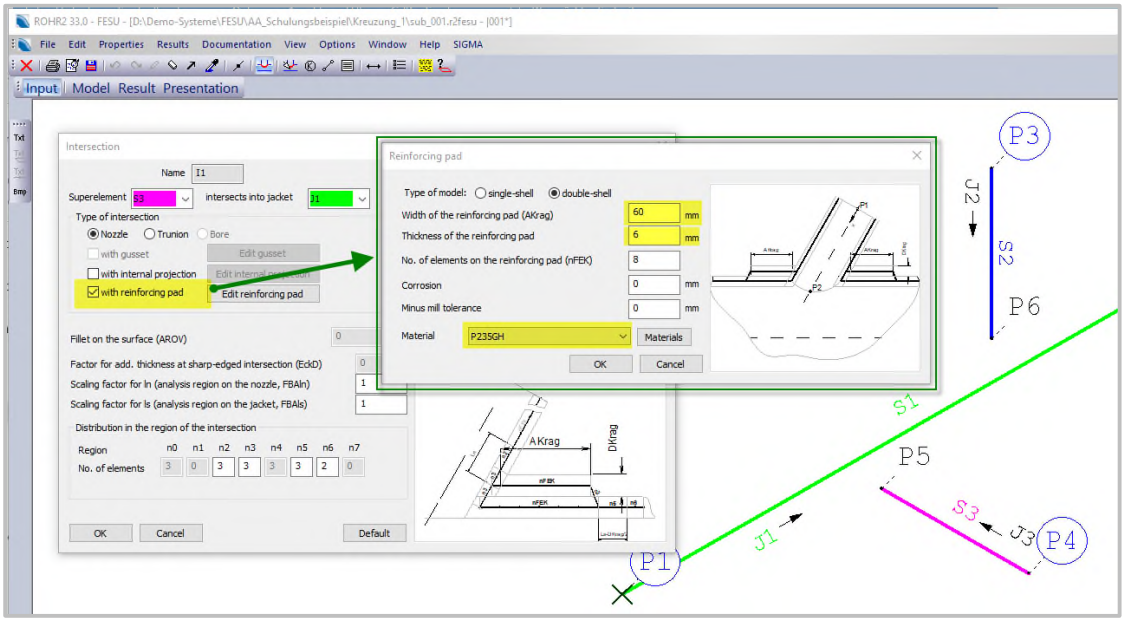


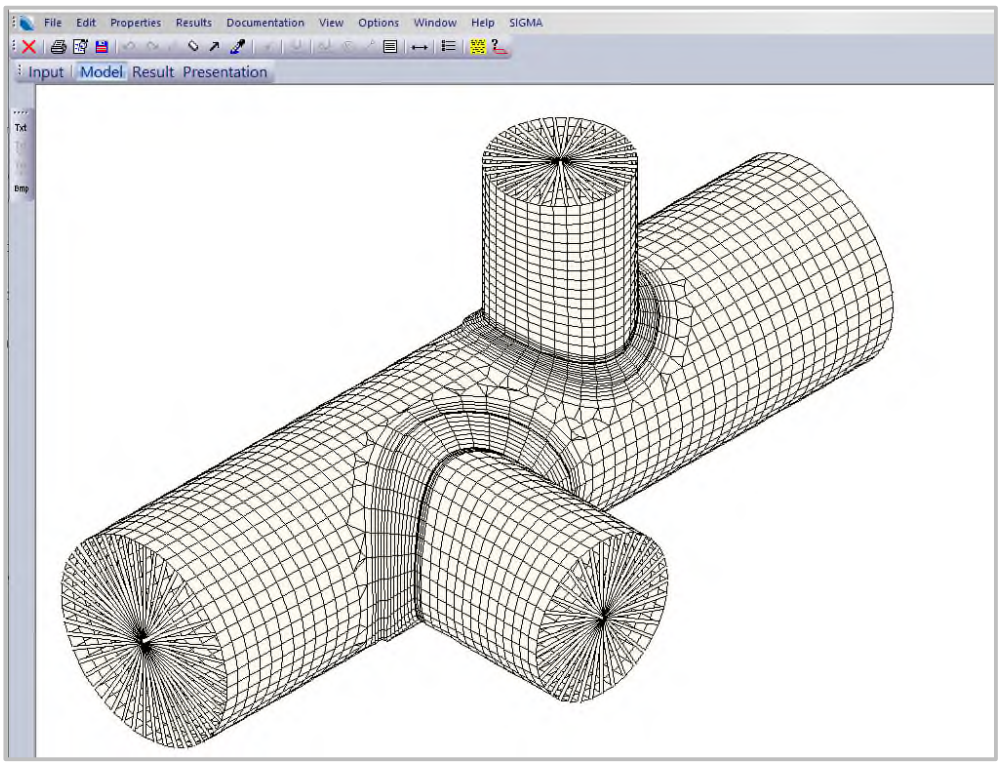
Since the two nozzles are not yet connected to the main pipe, intersections still have to be defined. This is done again in input mode.
You have to decide which superelement is to be intersected with which jacket. The type of intersection (nozzle or trunion) must also be selected. More detailed information about the window "Intersection" can be found in the ROHR2fesu manual.



7.3 Detailing the model in ROHR2fesu

- Intersection must be defined.
- Date of superelements can be detailed by adding parameters of detail drawings possible the structure is completed by additional superelements for the modeling of special transitions, etc.
- The meshing can be optimized, e.g. by progressive element division, to enhance the mesh distribution especially regarding critical areas.





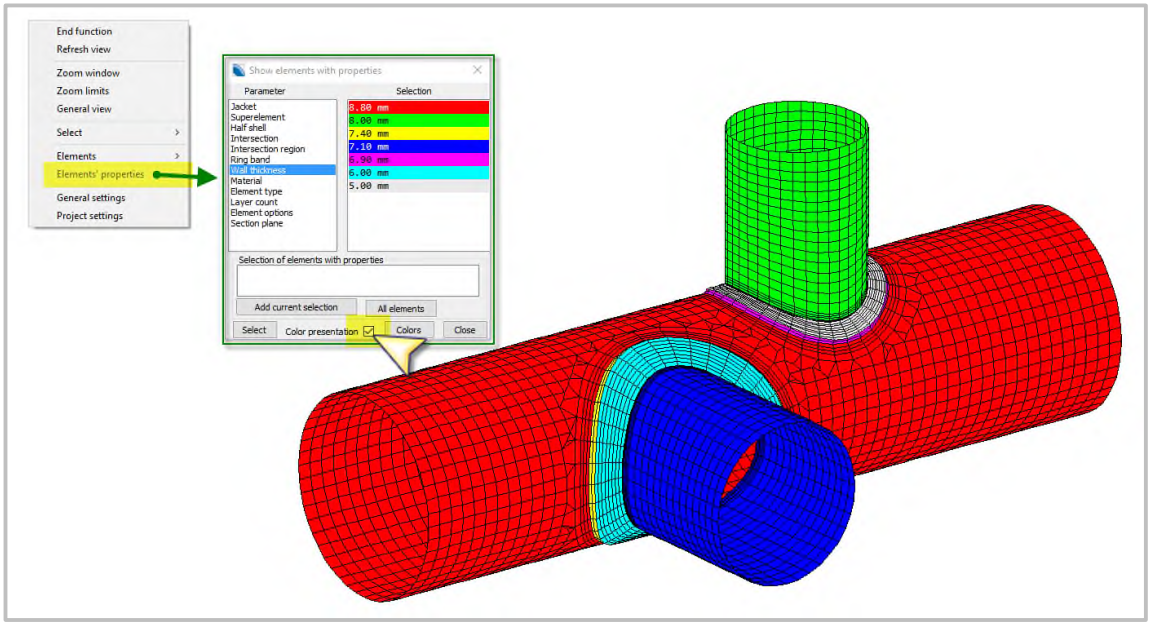
7.4 Checking data

The best way of checking data is the graphical control. Detailed values may be checked by means of super element properties and in the input protocol.
The graphical control of the FESU model at first must include these points::

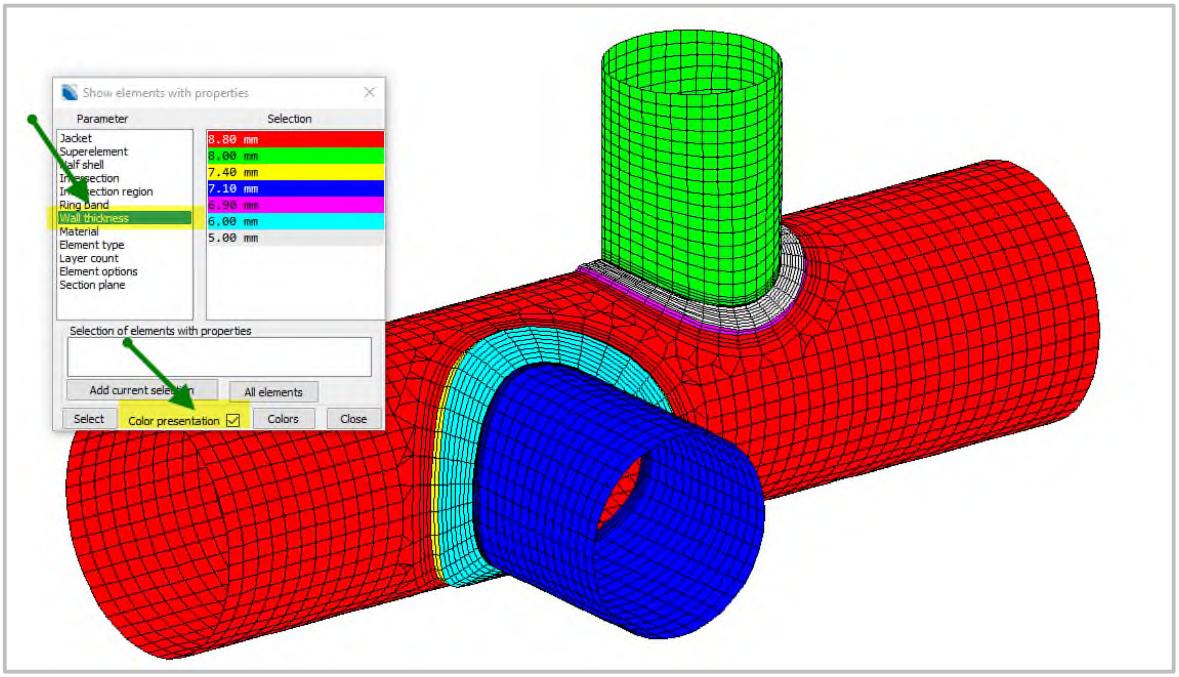
- geometry
- meshing
- net division / mesh size
- intersection zone
- Assignment of wall thickness´
- Assignment of materials

The meshing is supposed to me smoothly and can be influenced by the user just by net division parameters.
A progressive axial pith is recommended with increasing division in direction of expected critical areas is recommended. This will improve the precision and reduces calculation time. The mesh size can be controlled by measuring and by checking the properties of the FE-element. They are displayed when the mouse cursor is hovering above the element.

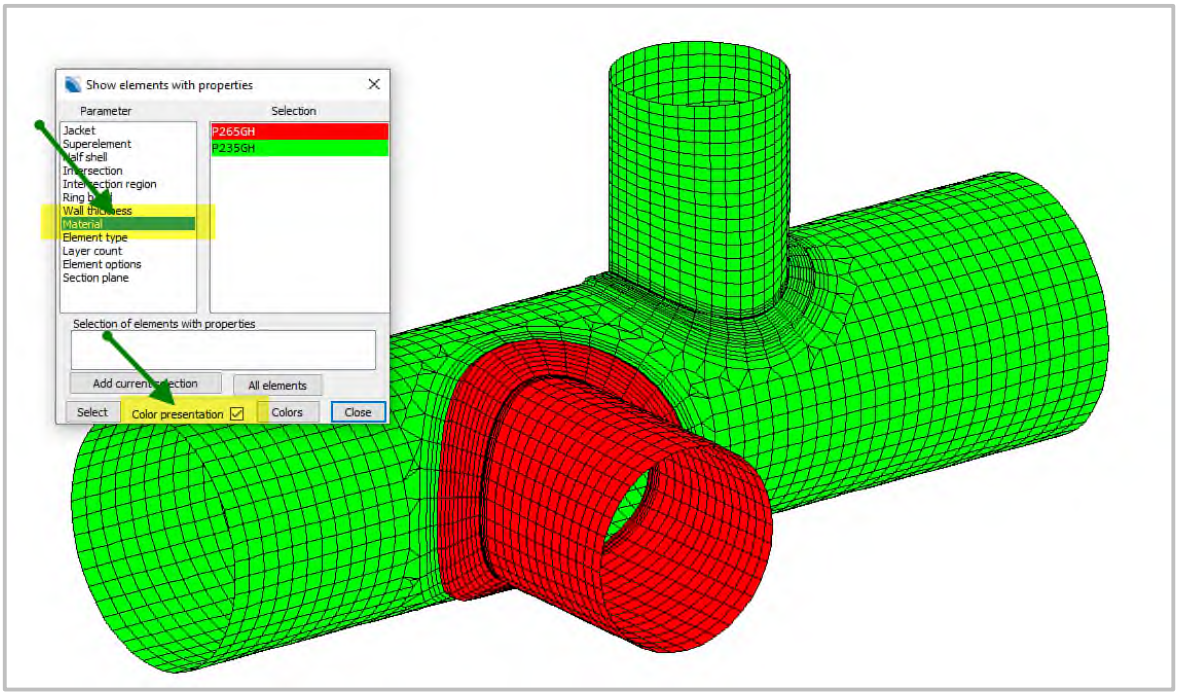
In the model mode most of the data can be checked graphically using the function "Element parameter" (equivalent to "Segment parameter" from ROHR2).



The wall thickness of the elements can be checked most effectively by using the color representation.

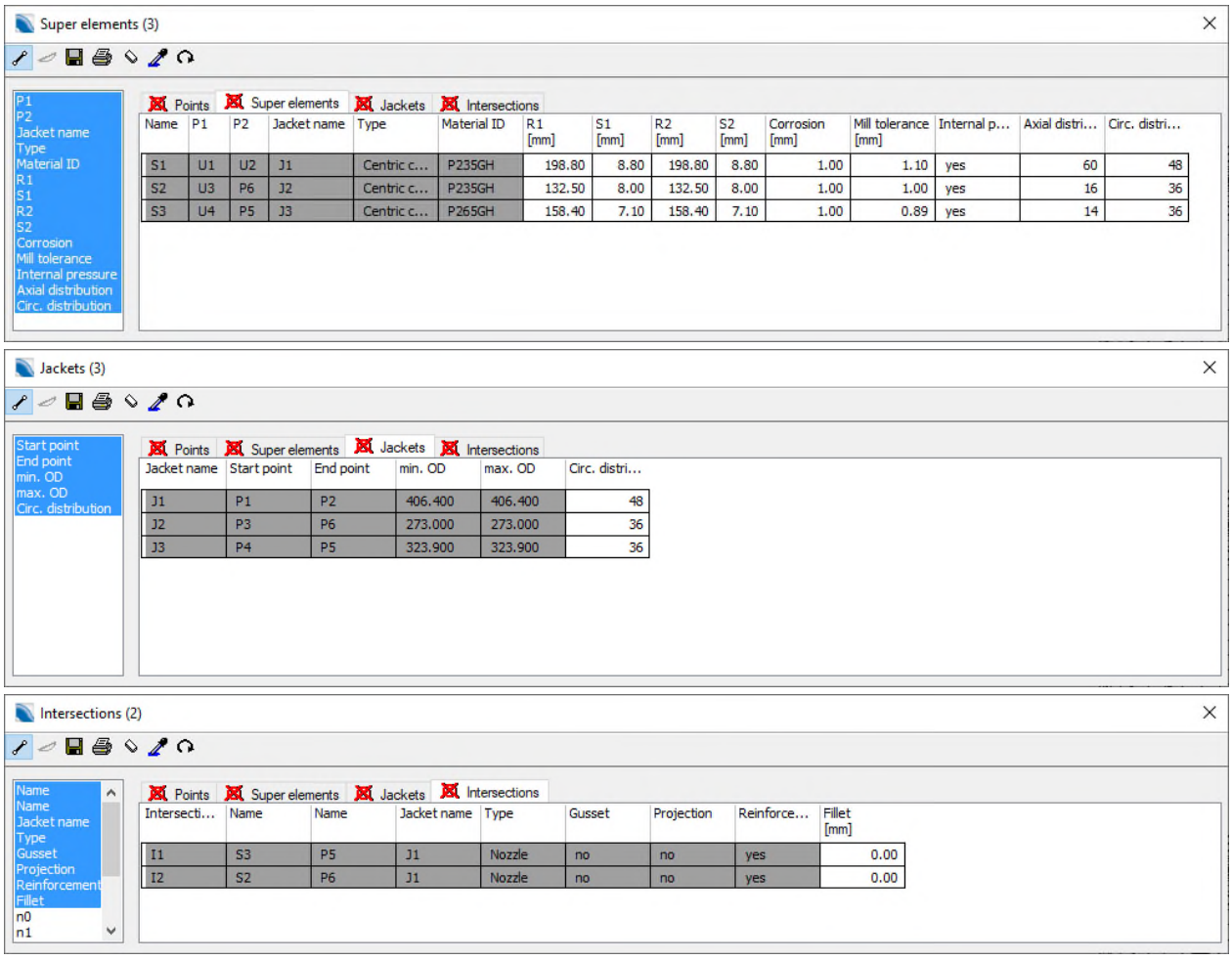


Also the materials of the individual SE can be checked effectively by using the color representation.



Alternatively the graphical user interface offers the opportunity to list the results in different tables, using *Properties – List data*:

- Nodes with coordinates
- Super elements with diameters
- Material data
- Wall thickness



7.5 Calculation

Proceed ROHR2 analysis using stiffness and loads from substructure(s).
If required the substructures will be calculated automatically before the analysis of the framework. results of the calculations are stiffness matrices, loads at nodes at connection points and node deformations at all elements from element loads.

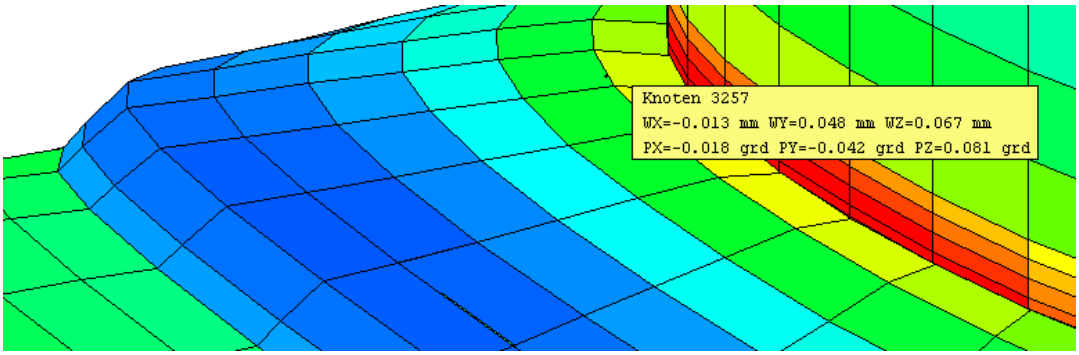
7.6 Results analysis

After the ROHR2 analysis the results from load cases as well as the results of substructures are available. The graphic shows stresses, deformations on shells and others.
It is additionally recommended to examine the results of the ROHR2fesu calculation by graphical control.

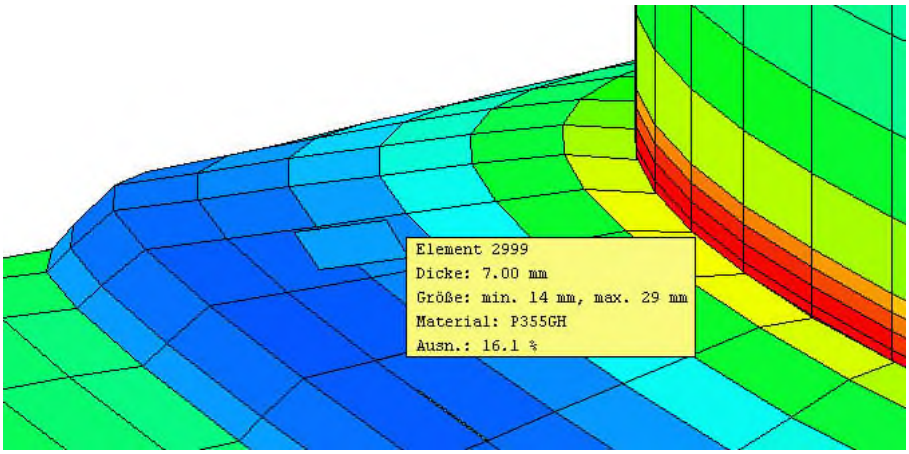
7.6.1 Graphics plausibility check

The results of the FESU calculation should also first be subjected to a graphical check.

- Checking deformation graphics regarding plausibility:
  symmetric systems must show symmetric deformation and stress graphics under symmetric loads.
- Checking stress diagrams regarding plausibility
- detailed results at nodes or elements



Results at nodes



Results at elements

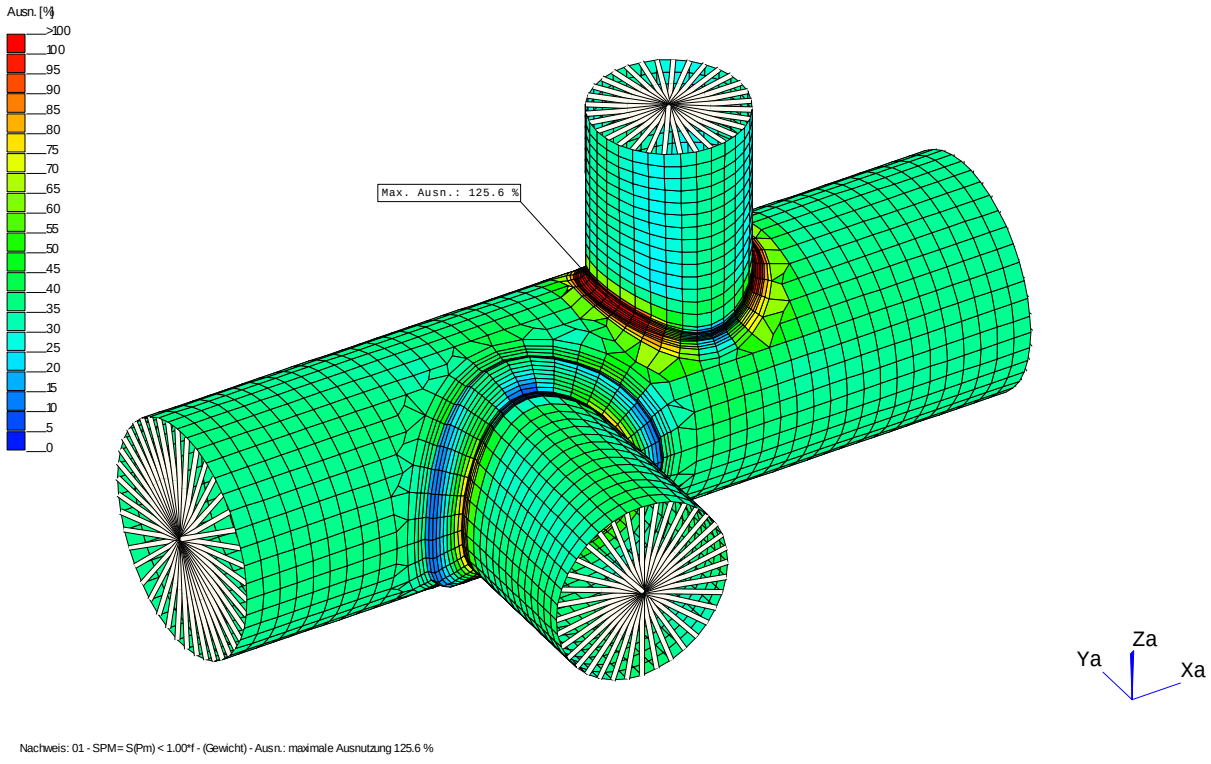
7.6.2 Stress analysis

ROHR2fesu allows an automatic stress analysis according to EN 13445-3, appendix C, ASME Section VIII, Div. 2, Part 5 and AD S4. Appendix C of EN 13445-3 describes a method of stress verification in which the stresses are categorized and verified against allowable limits.
In ASME Section VIII, Div. 2, Part 5 and AD S4 comparable processes are described. Here the introduction into the stress analysis is shown based on the example of EN 13445-3 appendix C

The following analysis are carried out with these load case groups:

- Global primary membrane stresses Pm < 1.0 x Sm
- Local primary membrane stresses Pl < 1.5 x Sm
- 3. Primary membrane and bending stresses (Pl + Pb) < 1.5 x Sm
- 4. Range of primary and secondary stresses (Pl + Pb + Q) < 3 x Sm

Example: Stresses SPM



Equation SPM

7.7 Optimization

Optimization at the ROHR2 model does **not** require the repetition of the substructure calculation.
The optimization of substructures always requires to renew the calculation of the entire structure.

7.8 Documentation

With the relevant input data for the modeling and the results of the stress analyses a report is created automatically.
The report can be adjusted by the user.