

ROHR2

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

ROHR2 31.2a Tutorial

ROHR2 Introduction - Editing a Piping
System

Release 31.2a

January 2014

SIGMA Ingenieurgesellschaft mbH

Contents of this document are subject to change without notice. The manual is protected by copyright. No part of this document may be reproduced or transmitted in any form or by any means, electronic or mechanical, for any purpose, without permission.

Specifications subject to change without notice.

All of the mentioned products and brand names are trademarks or indexed trademarks of the respective manufacturers.

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	1
1.1	Introduction	1
1.2	Projects and Examples	1
2	ROHR2 test license (full featured)	2
3	ROHR2 Demo program (viewer)	2
3.1	Features of the demo program (viewer)	2
3.2	Installation of the demo programs	3
3.3	Running the demo programs	4
3.4	Sending project data, created by the ROHR2 demo program	4
3.5	User support, hotline and ROHR2 board	5
4	ROHR2win User Interface - Overview	6
5	Quick guide	7
5.1	General inputs	7
5.1.1	Creating a new ROHR2 project	7
5.2	Editing pipe dimensions	8
5.3	Drawing a system	10
5.3.1	Select a coordinate system and draw	10
5.3.2	Insert a reducer	12
5.3.3	Insert components	13
5.3.4	Insert supports	14
5.3.5	Support condition symbols	15
5.4	Load case definition	16
5.5	Assign operation data per load case	17
5.6	Stress analysis, loads on supports and spring design	19
5.6.1	Stress analysis	19
5.6.2	Loads on supports	20
5.6.3	Spring design	20
5.7	Checking the input data	21
5.7.1	Segment parameters	21
5.8	Correcting and adapting the input data	22
5.8.1	Geometry	22
5.8.2	Dimensions	22
5.8.3	Operation data	23
5.9	Calculation	23
5.10	Results analysis	25
5.10.1	Load case results	25
5.10.2	Stress analysis	26
6	Checking CAD/CAE import with ROHR2 Interfaces	27

1 ROHR2 Tutorial

1.1 Introduction

Thank you for reading this document, introducing into the work with the program system ROHR2. This manual is applicable to the

- ROHR2 full license
- ROHR2 test license with access to the test server
- ROHR2 demo license (viewer)

We would be very pleased to provide you with a program license or demo 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).

1.2 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.
- demo users: find ROHR2 calculation examples on the Demo-CD or in the **ROHR2 Demo** installation.

Please refer to topic [5](#) of this document for a detailed introduction into a calculation example

Project editing by means of a full-featured test license

Please note when creating projects using the demo program:

Projects, created by the test license program are marked by TEST LICENSE. They can be opened and modified by a full program license.

Projects, created by a full featured ROHR2 license only can be opened and modified by means of a test license.

Projects made by the demo license (viewer)

Please note when creating projects using the demo program:

Projects, created by the demo program cannot be used in the full program licenses.

Projects, created by a full featured ROHR2 license only can be displayed in a Demo version (no editing, no storing).

2 ROHR2 test license (full featured)

The test license is identical to the full license. The test program is provided by the Sales department and can be used under conditions of non-commercial use.

A special installation manual is available for the test license.

The test license enables to use the entire functionality of ROHR2/SINETZ and it's modules and interfaces. The test license includes optional modules and interfaces which may be different from the your inquiry/order and the quoted/delivered program.

3 ROHR2 Demo program (viewer)

The ROHR2 demo program are provided

- on a ROHR2/SINETZ Demo-CD
- as demo application available for download

To receive a demo program, please contact us by email (sales@rohr2.com), use the contact form on www.rohr2.com or call us by phone.

3.1 Features of the demo program (viewer)

The demo programs are showing the entire range of functions of the graphical user interface (Pre- and Post processing), except of the calculation processor. The analysis and, if requested, a check of your input data may be done by our support team after (see [3.4 Sending project data, created by the ROHR2 demo program](#)).

ROHR2 Additional modules

The demo program is not able to carry out the ROHR2 additional programs ROHR2flange, ROHR2iso and ROHR2stoss.

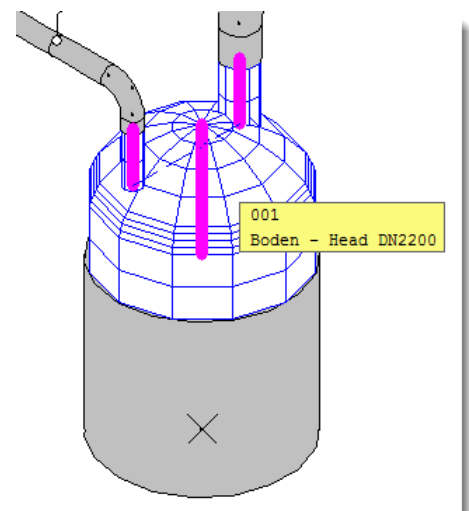
Please refer to the Demo-CD for calculated examples or contact the Program Support or Sales Team.

ROHR2fesu

Projects including ROHR2fesu elements can be opened in the ROHR2 demo program.

ROHR2 Interfaces

The ROHR2 demo program enables to check the ROHR2 interfaces (see [6.](#))



3.2 Installation of the demo programs

The demos are available from Demo-CD or in a zip file by download.

ROHR2 Demo-Application

- download
- unzip into a directory
- run *startdemo.bat* or *..\r2demo_app.exe*



Demo-CD

Insert the CD and follow the menu or select **CD_Start.exe** from the CD.

The programs can be carried out directly from CD, see [3.3](#). Alternatively the programs can be copied to a local harddisk.



License conditions of a demo program

The use of the demo programs is free of charge. Programs are always carried out under the demo software license conditions of SIGMA Ingenieurgesellschaft. Please pay attention to the file *democond_ROHR2_SINETZ.pdf*. The demo conditions are included in the demo-CD, in the attachment of the demo link email or can be downloaded from the internet (www.rohr2.com).



If you not agree to the license conditions, you are not permitted to carry out the programs. Remove the programs in this case.

In case of questions concerning installation or use of the demo programs, please, contact the SIGMA program support.

3.3 Running the demo programs

Starting the ROHR2 demo program from CD

The ROHR2 demo program is running from CD directly. Calculated sample files are included and may be added by the command FILE|OPEN.

Running the ROHR2 demo program by calculation example

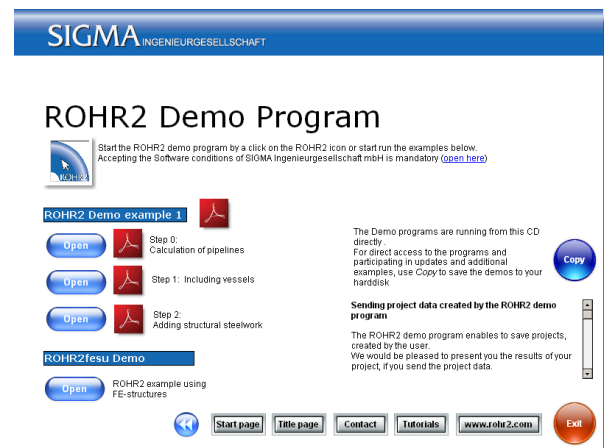
Select one of the calculation examples to start the ROHR2 demo program from CD.

Open the pdf files for an overview on the project and calculation results.

Except of the self-created project files no additional programs are installed on your computer.

Copy the demo program to your PC

Copy the ROHR2 and SINETZ demo programs to your local hard disk. Select the program to be copied in the dialog window.



3.4 Sending project data, created by the ROHR2 demo program

Saving new projects

The ROHR2 demo program enables to save projects, created by the user.

How to get calculation results from the ROHR2 demo application

We would be pleased to present you the results of your project, if you send the project data.

For this purpose, please send the *.r2w file by email. This file contains the ROHR2 project data, stored in the project directory you defined when drawing a new piping system.

Alternatively the function `file|calculate` may be used to start your windows' standard email program and creates a message including the attached project data of the actual piping system.

This email does not transmit any data of your PC, except of the ROHR2 demo project.

3.5 User support, hotline and ROHR2 board

All software commands are documented in the user manual and in the program online help, see *Online help* and *Program documentation*.

Additional information sources are available

- in the internet, e.g. **ROHR2 board** and ROHR2 FAQ (Frequently asked question) see *Help menu*
- the 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).

User support address

SIGMA Ingenieurgesellschaft mbH
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

4 ROHR2win User Interface - Overview

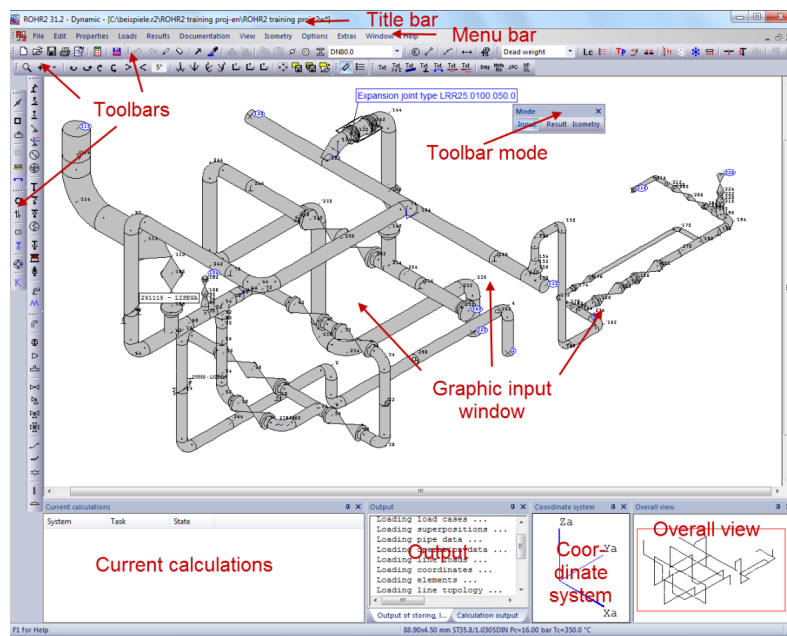
The input window shows the piping system and the drawing created. All program functions are accessible by menu commands and symbols (icons). The elements of the user interface can be positioned on the screen be placed free on the screen.

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*) ROHR2win appears in different modes. Activate the mode related commands by switching between the program states in the toolbar mode.

Status bar

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

Elements in the lower part of the input window

Coordinate system	Showing the <i>Global coordinate system</i>
General Overall window	overview on the entire system with zoom function
Output window	The calculation output text is shown in the output window
Current calculation	The window <i>Current calculations</i> shows the tasks in the calculation queue

5 Quick guide

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.

Open training projects or your own ROHR2 projects by means of the *File| Open* command.

The Open function enables to load ROHR2 projects, as well as CAD data by ROHR2 interface programs. See also *ROHR2 interfaces* [6](#).

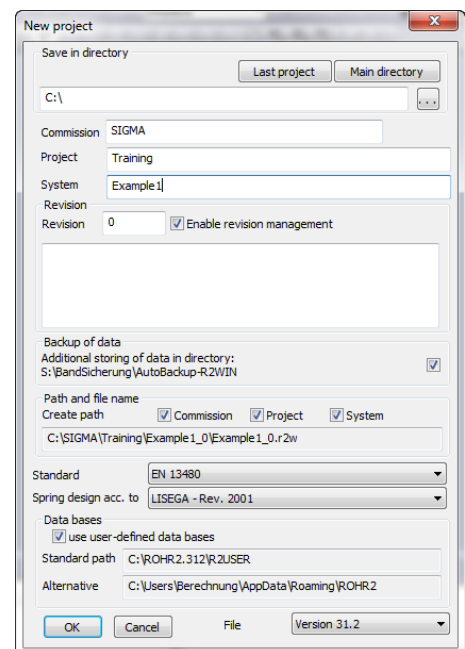
5.1 General inputs

5.1.1 Creating a new ROHR2 project

- Create a project directory with up to 3 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

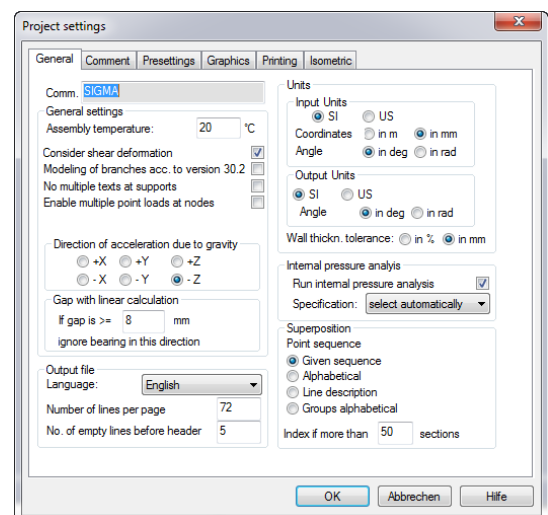


Options| Project settings

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

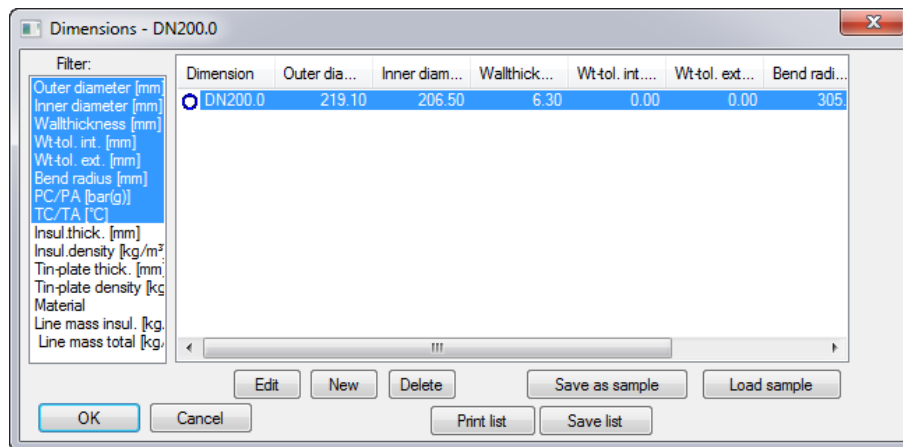


5.2 Editing pipe dimensions

Menu edit > Pipe dimensions

Dimensions, available in the project

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



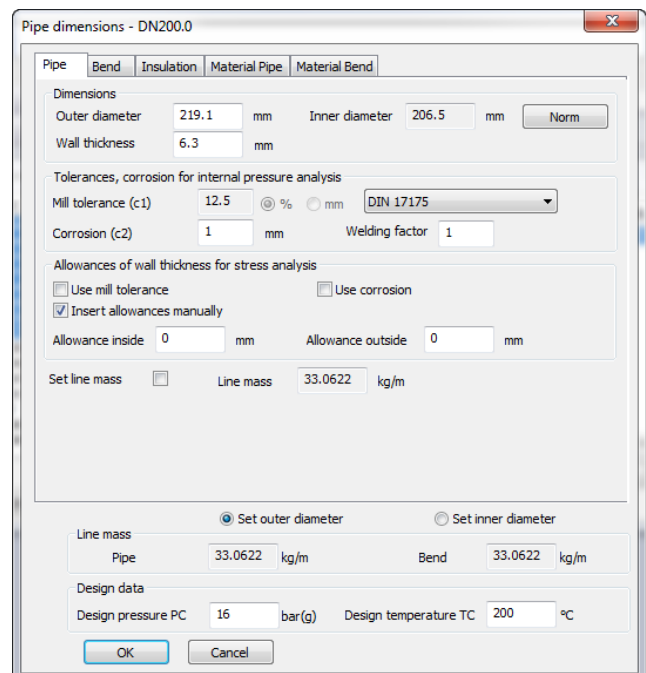
A double-click on the dimension opens the pipe dimensions input window.

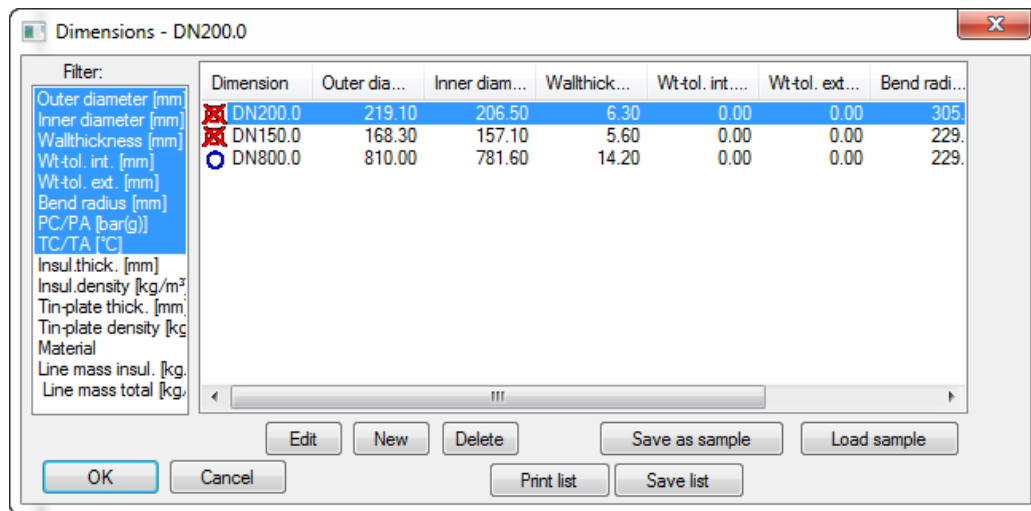
All necessary dimensions are defined here

This dialog window offers 6 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.

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





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.

5.3 Drawing a system

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

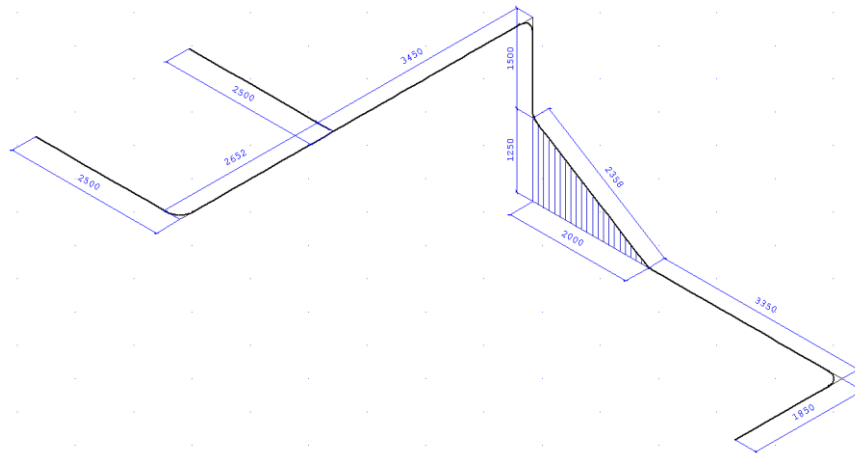
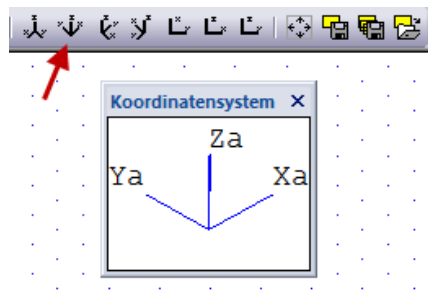


figure: system lines

5.3.1 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*



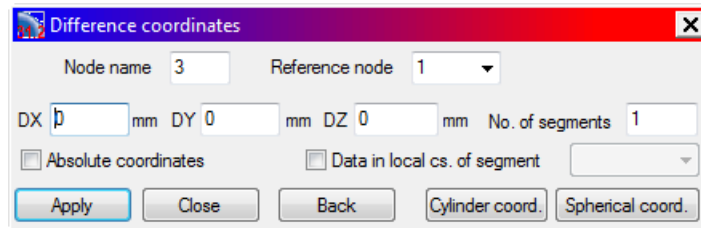
or user the menu command

Menu Edit > Draw

Enter absolute coordinates of the first node

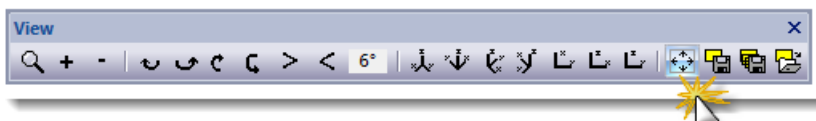
Start node		
Absolute coordinates of the node:		
X	200	mm
Y	0	mm
Z	0	mm
OK		Cancel

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.

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



Oder menu right mouse button

Drawing a branch

- Use Edit| Draw or 
- Get the branching node (highlighted red) and press any key. After that you can continue drawing as mentioned above.
- Draw a branch with DN150
- „Close“ to terminate the window

5 - Quick guide

5.3.2 Insert a reducer

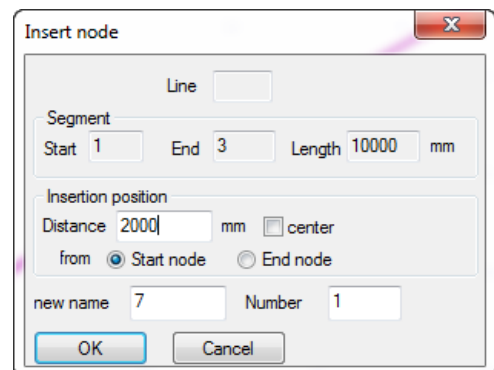
When a reducer is inserted the nominal width of the pipe changes from the reducer position up to a selected end node.

These steps are required.

Edit > Insert component > Reducer

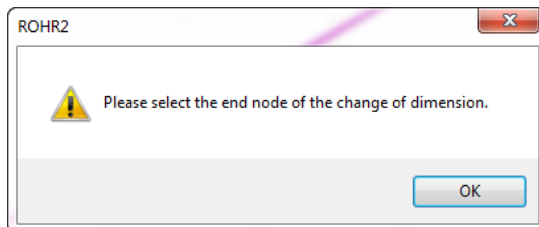
or use .

Click into the segment where the reducer is inserted and enter the distance to the next node.



The 'Insert node' dialog box contains the following fields and options:

- Line:
- Segment: Start End Length mm
- Insertion position: Distance mm center
- from: Start node End node
- new name Number
- Buttons: OK, Cancel

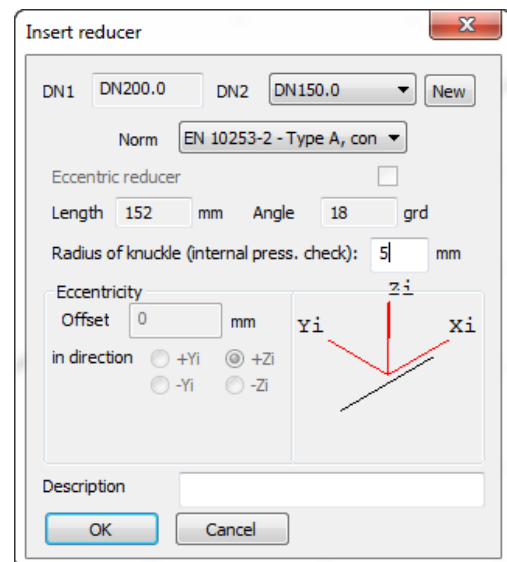


Confirm this message by OK and enter the end node of the changed dimension.

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.



The 'Insert reducer' dialog box contains the following fields and options:

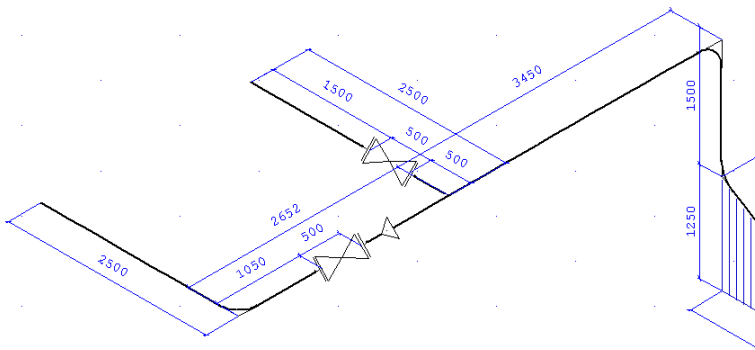
- DN1: DN2:
- Norm:
- Eccentric reducer:
- Length: mm Angle: grd
- Radius of knuckle (internal press. check): mm
- Eccentricity: Offset mm
- in direction: +Yi +Zi -Yi -Zi
- Diagram: A schematic showing a pipe with a reducer. The vertical axis is labeled z_i and the horizontal axis is labeled x_i . The reducer is positioned at the origin of these axes.
- Description:
- Buttons: OK, Cancel

5.3.3 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 in 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:

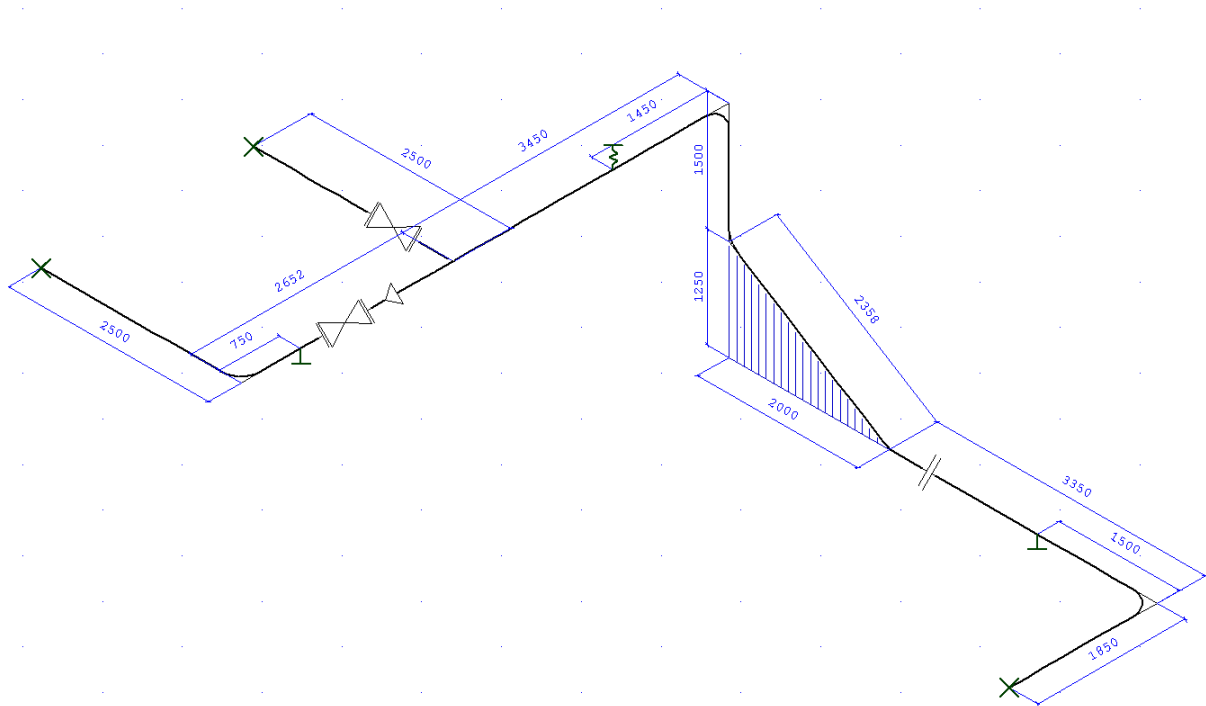
- Length seal to seal or weld to weld
- Enter instrument mass
- Select Option *with flange*

Flange:

- Select flange

5.3.4 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.

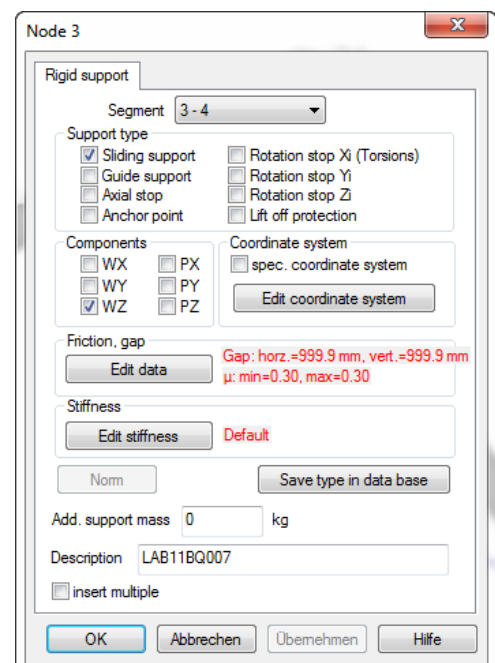


Defining a rigid support

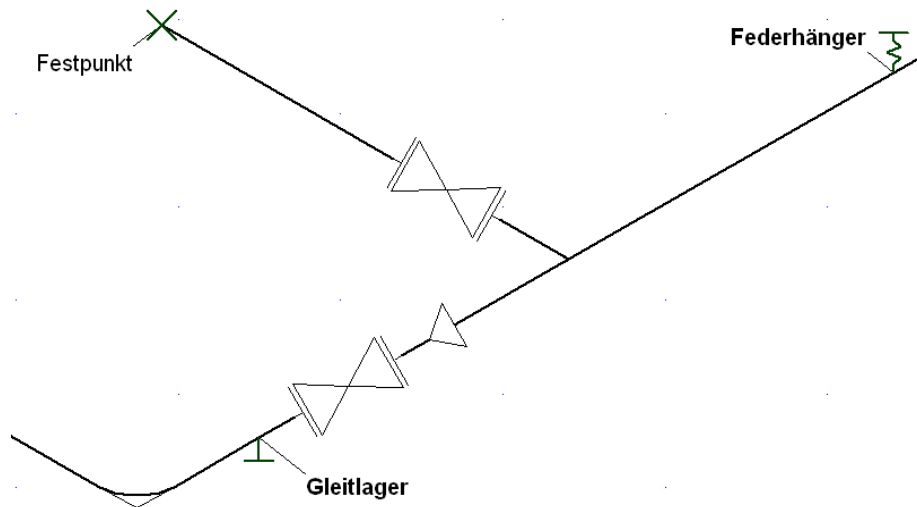
ROHR2 regards sliding supports, bearing supports, axial stops and anchor points as rigid supports.

The way to insert a support:


- select the type of support in 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.



5.3.5 Support condition symbols



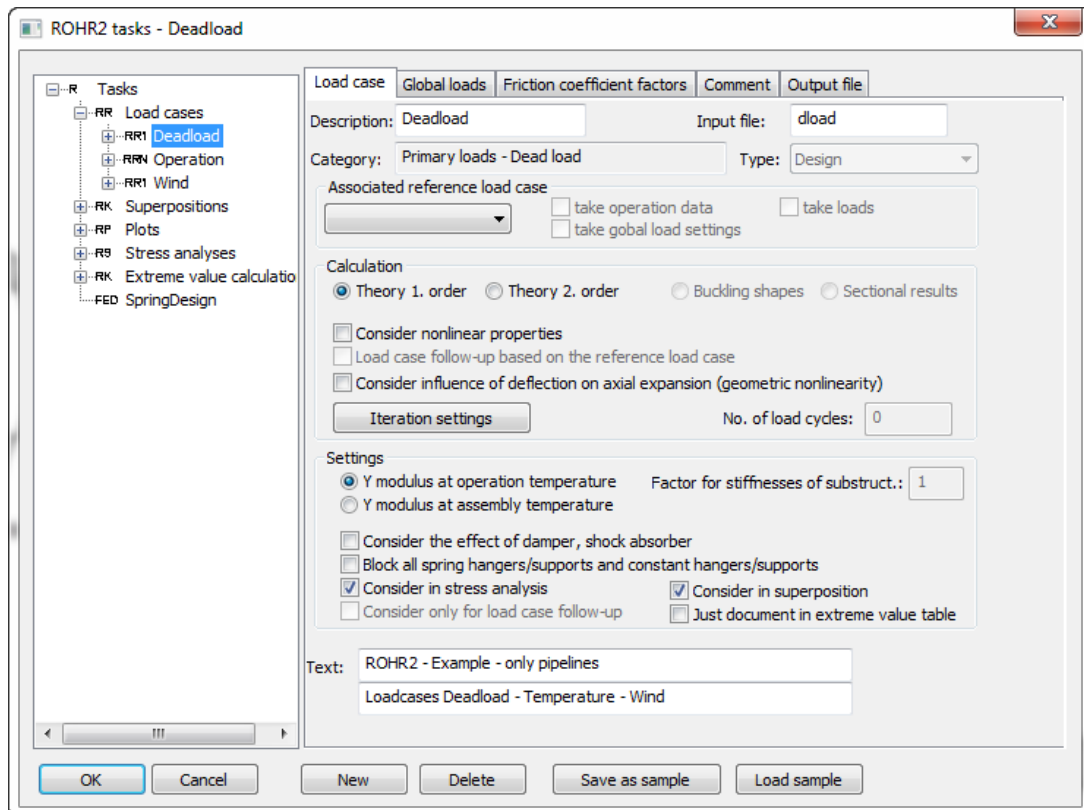
All supports of this example at first are calculated considering the standard values of friction, gap and stiffness.

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

5.4 Load case definition

The load cases Dead weight and operation (Weight + thermal expansion) are pre-defined. View global loads, considered by the load cases in the register *global loads*.

Use Menu *Loads/ Tasks* or **LF** to open the dialog window *Load case definition* and ROHR2 tasks like stress analysis or extreme value calculation



For definition of load cases see below.

The first step just includes the calculation of pre-defined load cases dead weight and operation.

5.5 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.




Tip

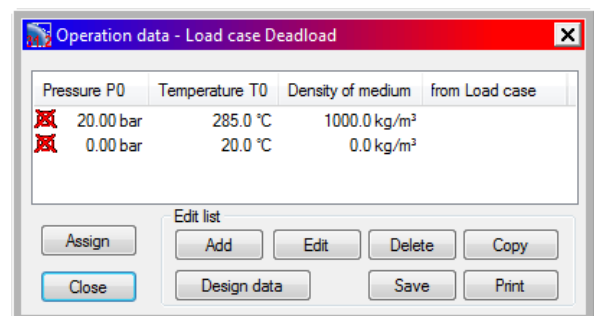
The Select command can be found in the Context menu (right mouse button), too).

Select parts of the system

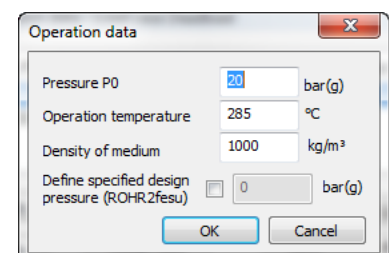
Use

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*.



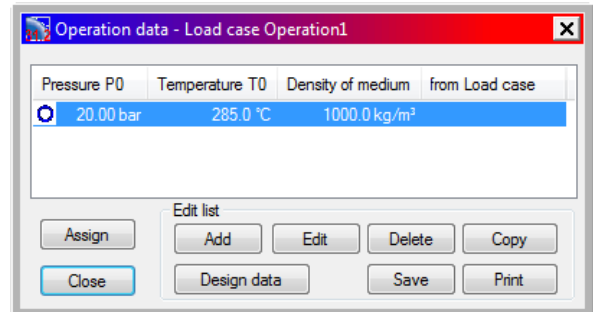
5 - Quick guide

Operation data example

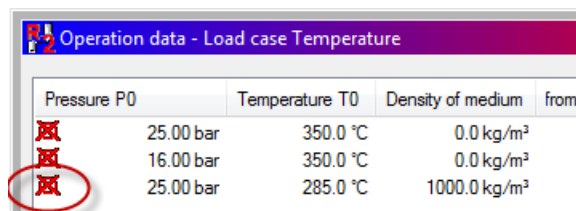
Identical Operation data in the whole piping system:

- 20 bar / 285°C / 1000 kg/m³

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

**Tip**

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

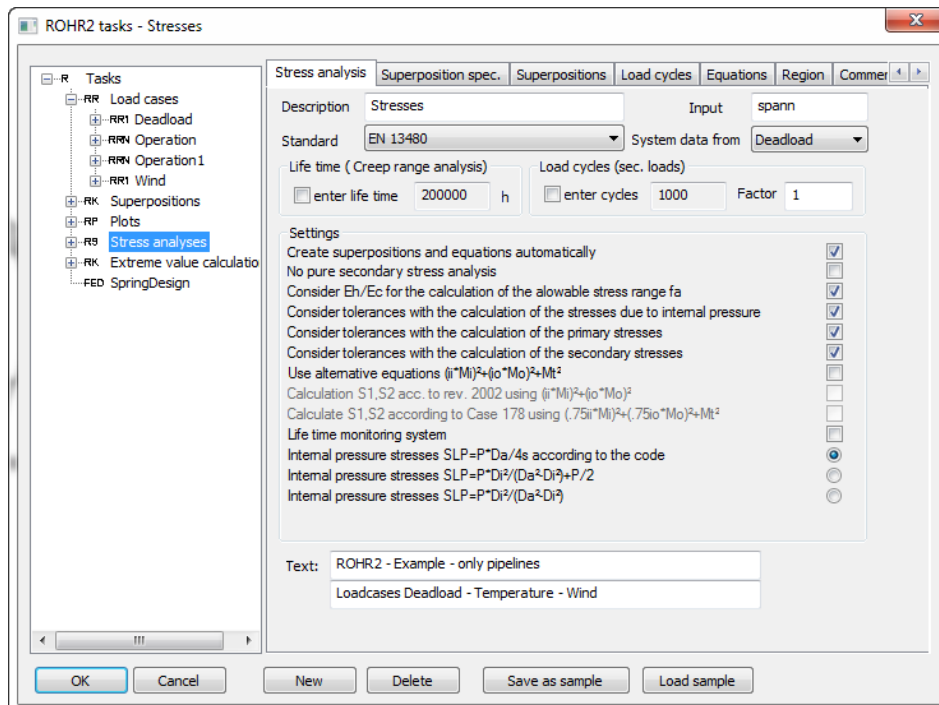
Copy the data from load case *Operation* into the load case *Weight* by

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

5.6 Stress analysis, loads on supports and spring design


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.

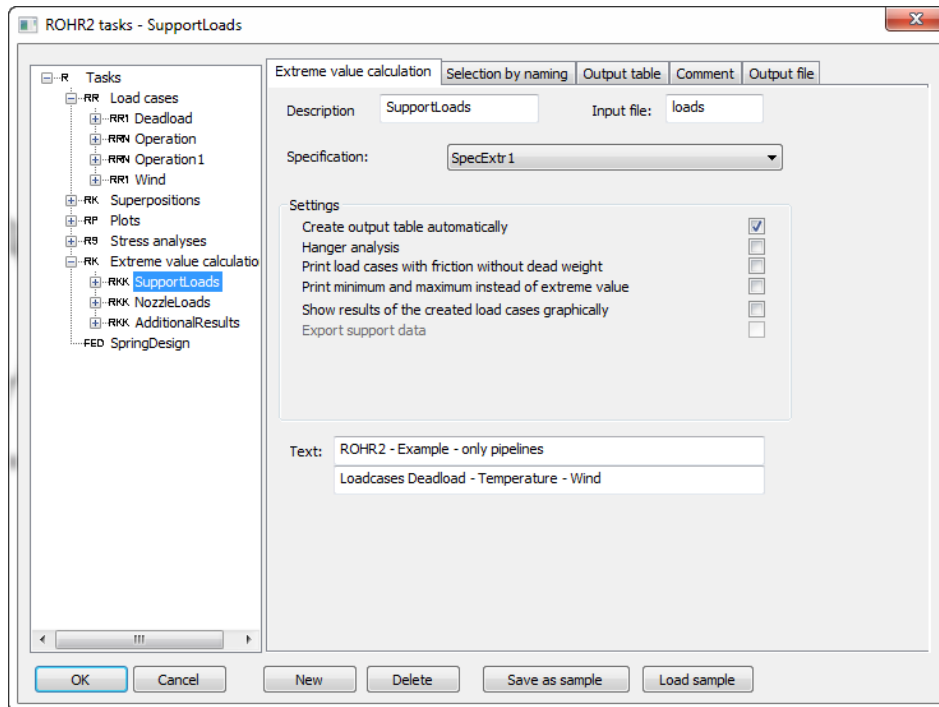
5.6.1 Stress analysis



- Define 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.

5.6.2 Loads on supports

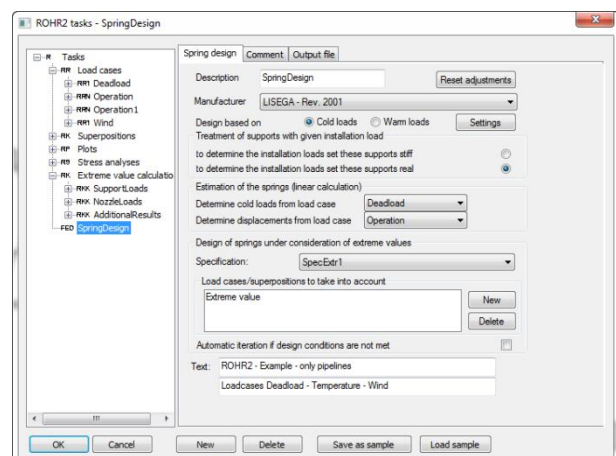
Use Loads| Tasks or  to open the dialog window for the definition of load cases and ROHR2-tasks. Select an extreme value calculation in the left column.



- 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.

5.6.3 Spring design

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



5.7 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*.

5.7.1 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.

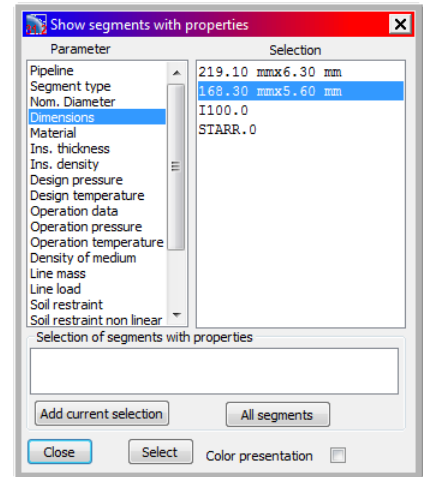
Several entries can be entered at the same time in the select box.

Selection

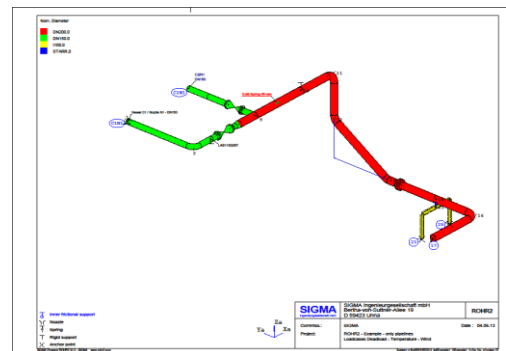
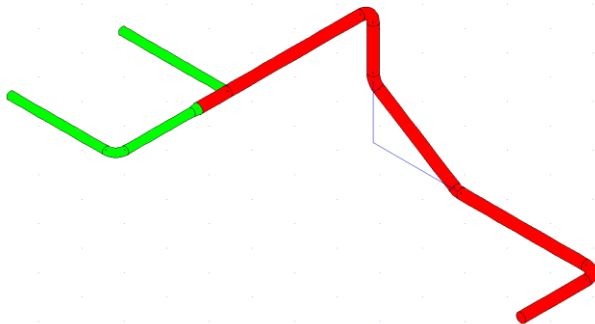
The segments containing the chosen parameters are selected


Colored display

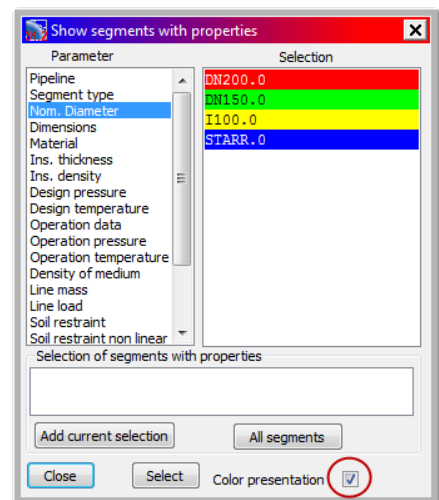
The properties are shown in different colors



Example, checking assigned dimensions



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



5.8 Correcting and adapting the input data


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

5.8.1 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

5.8.2 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.


5.8.3 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.

5.9 Calculation

The calculation capability is not available in the demo program. See [3.4](#) for sending project data.
Additionally on the Demo-CD 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.

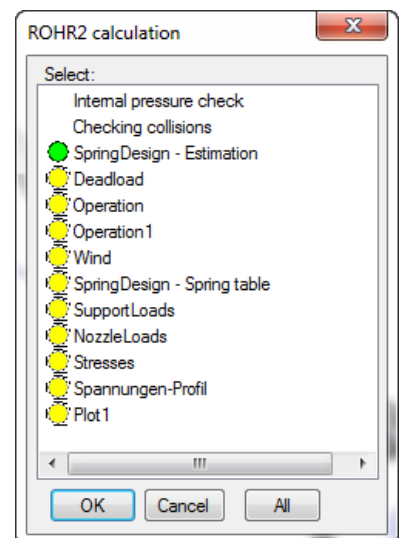
The *internal pressure check* entry offers the opportunity to check components for internal pressure.

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.

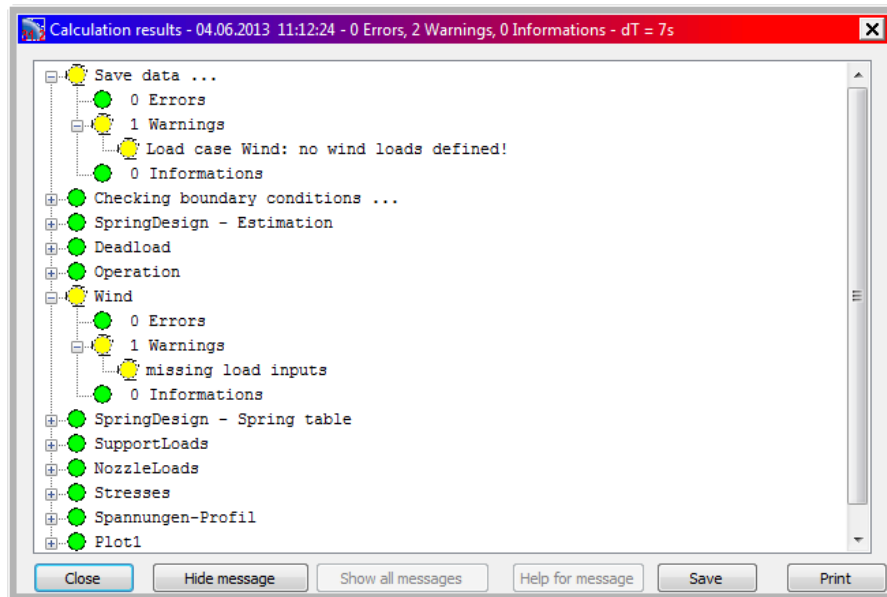
 results updated





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



5 - Quick guide

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

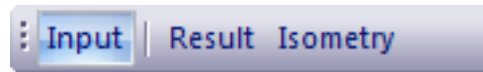


	green	Everything is alright. The calculation runs without any mistakes
	blue	General system information
	red	Error messages. a problem occurs: The calculation result is totally missing or it is not recommended to use the result (e.g. the calculation did not use the required analysis accuracy)
	yellow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

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

5.10 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:

5.10.1 Load case results

At first select the load case in the results mode

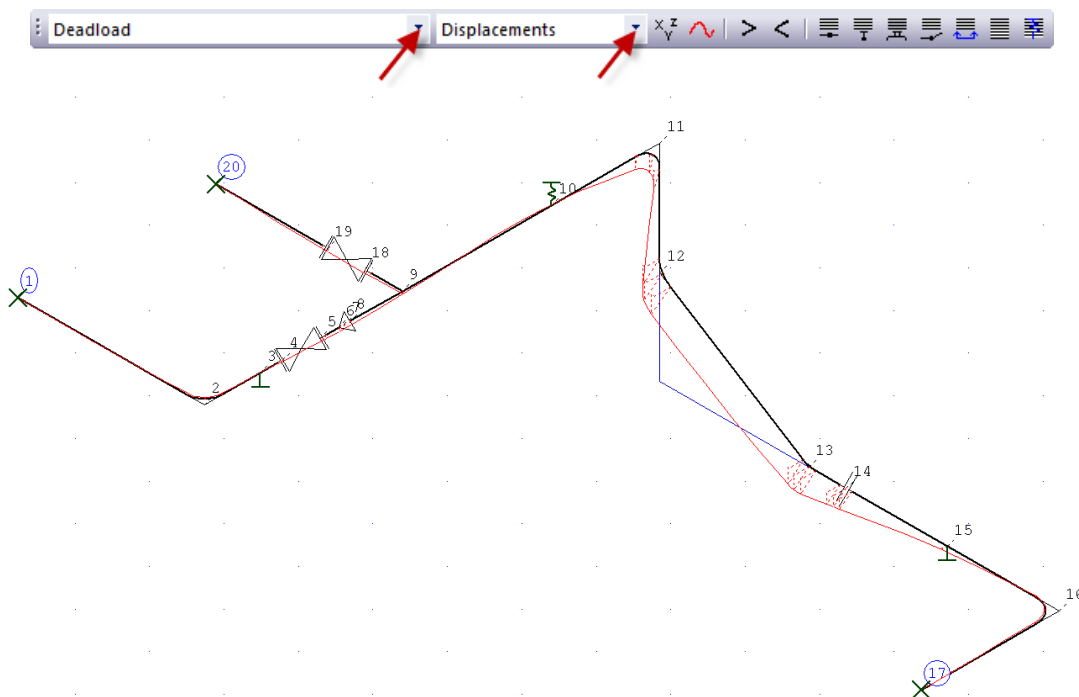




Fig.: deformed structure

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.

5 - Quick guide

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

5.10.2 Stress analysis

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

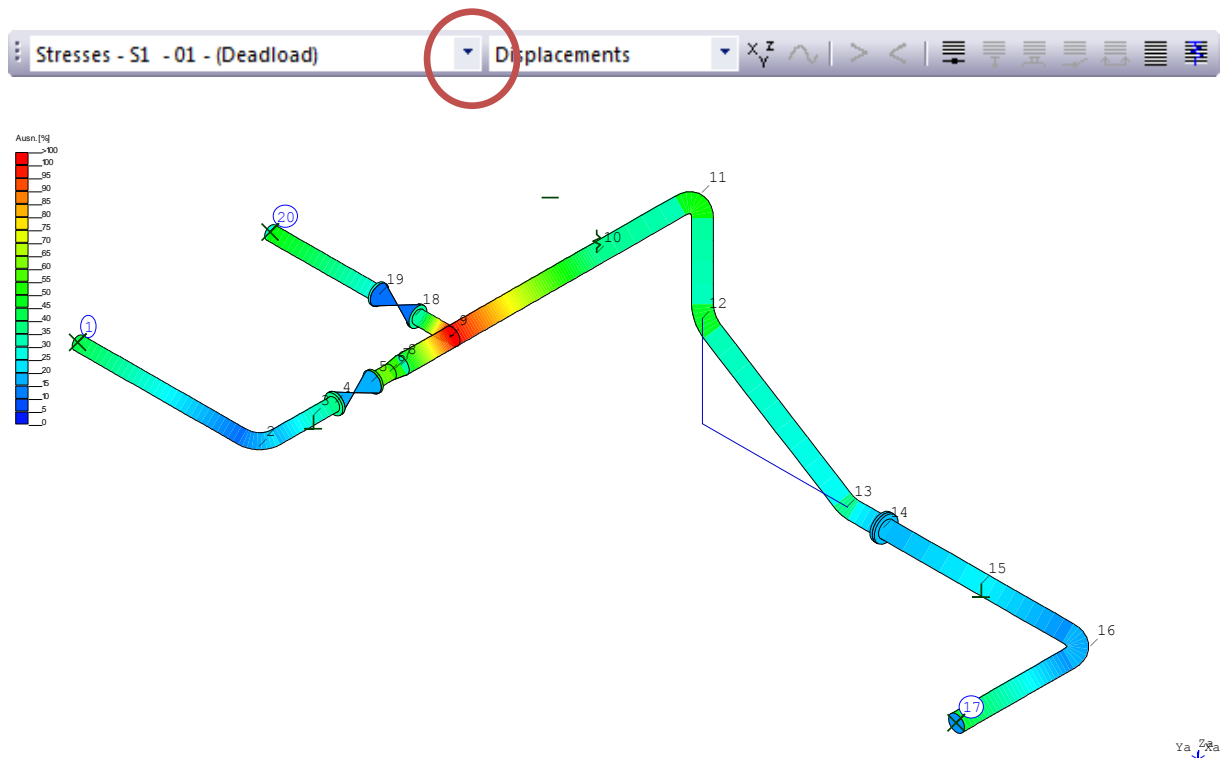


Figure: Stress analysis equation 4 (weight + range)

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)

6 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, enables import from CAD/CAE systems like PDMS, CADISON, RC-Planet, HICADnext, and others
- Export AVEVA PDMS - ROHR2,
- CAESAR II Import Interface
- PIPESTRESS Import Interface
- SINETZ Export interface
- Export Interface into the Support Design Programs LICAD, FLEXPERTE, CASCADE

(For additional details please refer to the ROHR2 program description)

All import formats are available in the demo program *).

Load the data by means of *File/ Open* command and get an overview on the capacity of the import interfaces.

**) some of the interface modules are optionally available products.*

