PROBAD

Code-based Strength Calculations of Pressure Parts

PROBAD tutorial

PROBAD Modeler
Introduction: Editing a Calculation System

Release January 2023

SIGMA Ingenieurgesellschaft mbH

Software-Support, German, English ++49 (0) 2303 332 33 45 support.probad@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 PROBAD Tutorial

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

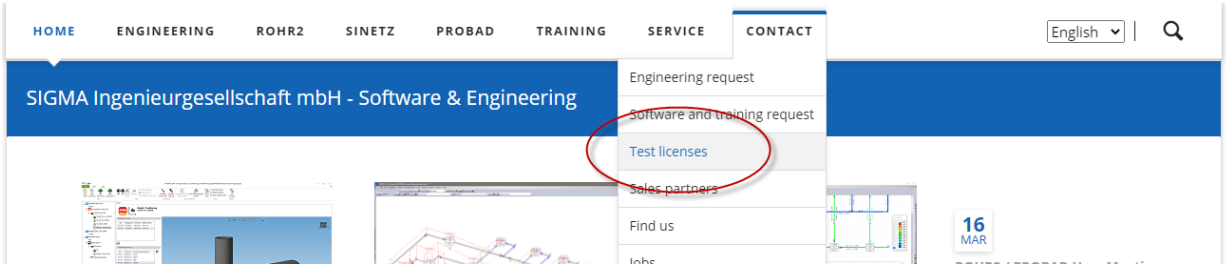
- PROBAD full license
- PROBAD test license

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 PROBAD sales partners (see /www.rohr2.com for contact details).

1.1 PROBAD 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 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 FAQ, see *Help menu,*

User support by email

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

Telephone and email

PROBAD Software-Support ++49 (0) 2303 332 33 45 support.probad@rohr2.de

Internet

www.rohr2.de www.rohr2.com

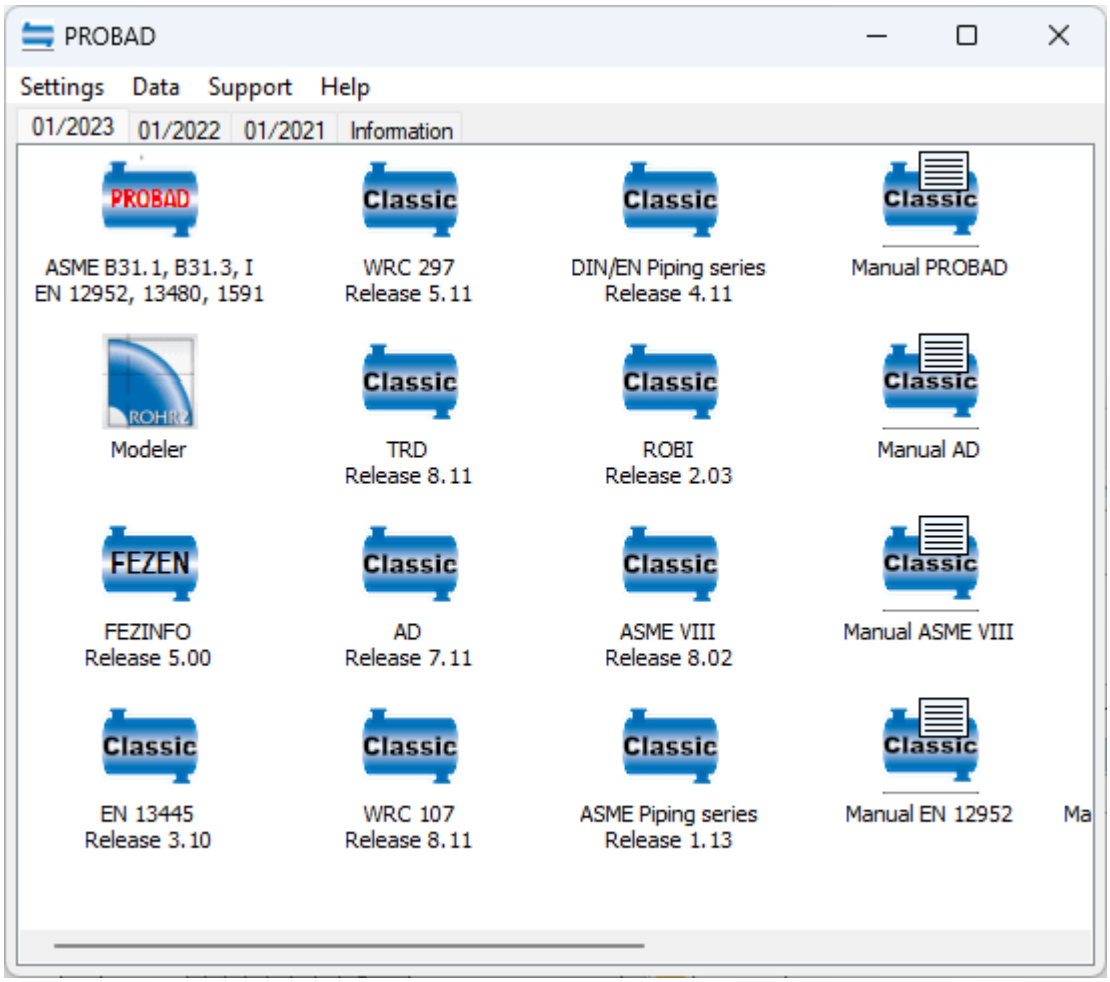
2 Program start

A PROBAD desktop symbol is located in START| PROGRAMS
It allows access to all installed programs. Modules without current license are blocked.

2.1 PROBAD Start Menu

The PROBAD start menu controls the installed PROBAD program modules.
All PROBAD modules of the current program directory are displayed. For each release a tab is created.
The associated PROBAD modules and manuals are displayed in each tab.

Start a program module by a double click on the symbol.



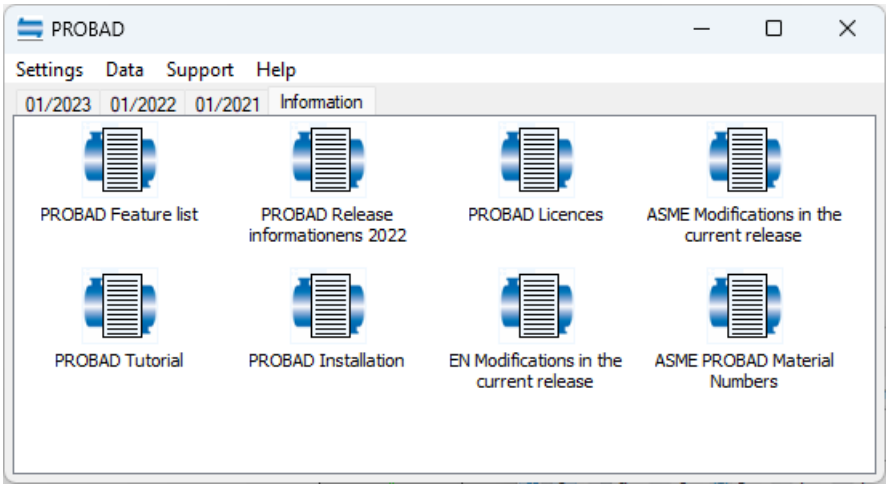
The updated GUI offers the access to:

- ASME B31.1 / ASME B31.3 / ASME I
- EN 12952 / EN 13480 / EN 1591

Further PROBAD modules will be integrated continuously into this new user interface.

The PROBAD Start menu shows all menus, independent of the module architecture and interface.
Previous calculation modules are accessible by registers.

In the tab ***Information*** documents with additional common information are displayed.

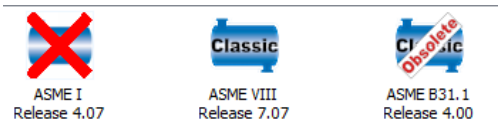


Integration of older PROBAD-Releases

Additional directories with working directories of other PROBAD releases can be defined. The PROBAD modules of these directories are displayed also in separate tabs. This enables the access to commonly used PROBAD modules of different releases via a single user interface.

Display options

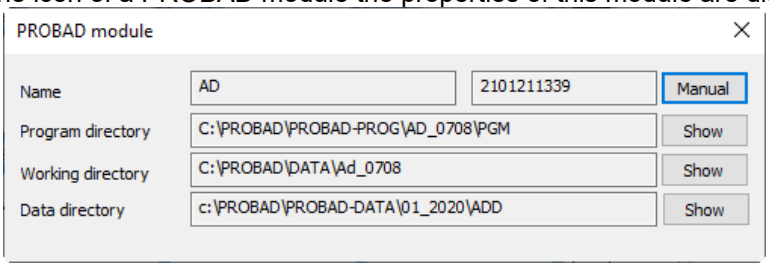
By default only PROBAD modules with existing licenses on all directly connected or via network reachable license keys are displayed. All other modules are hidden.
For PROBAD modules with existing new user interface an icon for the new user interface is displayed. All other modules are displayed separately as ***Classic*** modules.
The settings can be changed so that also not licensed PROBAD modules are displayed and/or additional icons for the old user interface of modules with new user interface are displayed.
Not licensed PROBAD modules are marked with a red cross. Icons for old user interfaces of modules with existing new user interface are marked by ***obsolete***.



Optionally icons for **manuals** can be displayed (see *settings*).

Properties

By right-clicking on the icon of a PROBAD module the properties of this module are displayed.



Manual

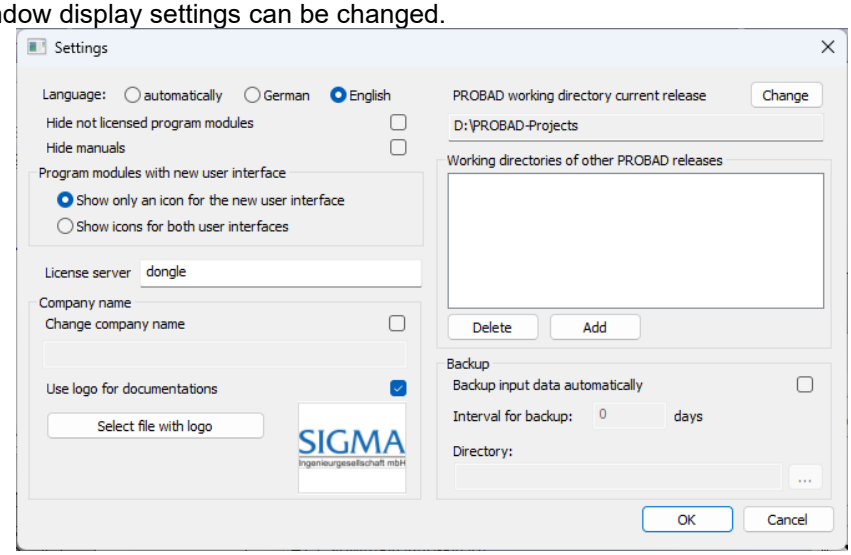
The manual of this PROBAD module is displayed

Program directory/Working directory

Program and working directory of this PROBAD module are displayed. With ***Show*** the directory is shown by the Explorer.

Settings – General Settings

In this dialog window display settings can be changed.



Language

Selection of the language for the start menu. If *automatically* is selected the language is set depending on the language of the operating system. Changing the language setting requires restarting the program.

Hide not licensed program modules

If this option is activated only PROBAD modules with existing licenses on all directly connected or via network reachable license keys are displayed.

Hide manuals

If this option is activated no icons for manuals are displayed.

Program modules with new user interface

This option is used to control if icons for the old user interface shall be displayed at PROBAD modules with new user interface, too.

PROBAD working directory current release

The directory where the working directories of the current PROBAD release are created is displayed. This can be changed with ***Change***. Please note that the working directory of PROBAD must not contain blanks and that the user needs read and write access to this directory!

Working directory of other PROBAD-Releases

Additional directories are listed. The se directories are searched for PROBAD modules of other releases. All found PROBAD modules are displayed in separate tabs.

Delete / Add

Deletes selected directory or add a new directory.

License server

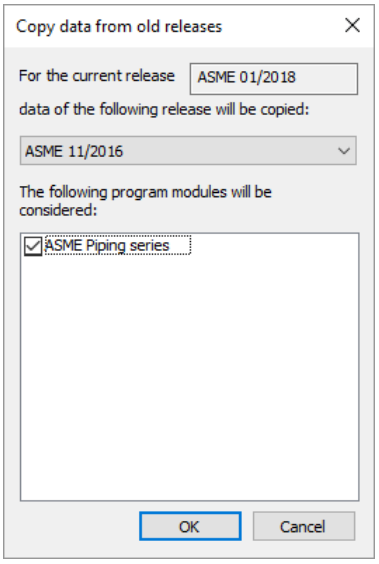
Defines the server names or IP addresses of the computer with network license keys to be accessed by PROBAD. Several names can be inserted separated by commas.
Please note that this setting influences also other SIGMA products (ROHR2, SINETZ) on this computer!

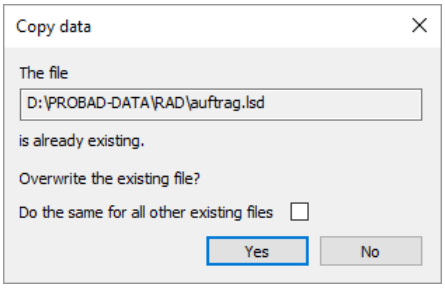
Company name

The company name for all PROBAD modules considered by the start menu can optionally be changed.

Data | Copy data of older releases

Data of older releases can be copied into the data directory of the release of the current tab.
All directories defined in the settings for other PROBAD releases are searched for data directories. All found data directories of older PROBAD releases are displayed.
In the dialog window *Copy data from old releases* the modules and the release of the data to be copied.





If a file is already existing in the data directory of the current release a message is displayed. The user can control how to proceed in this case.

Support | Send working directories to support

In case of trouble-shooting it can be required that the PROBAD support need to check the contents of the PROBAD working directories. With this command a mail is prepared automatically with the working directories as the attachment. This requires an existing and configured email client program on the computer.

Help | Description

Shows this document.

Help | Update license key

Start the program HLSCAN to update the locally connected license key (full version only)

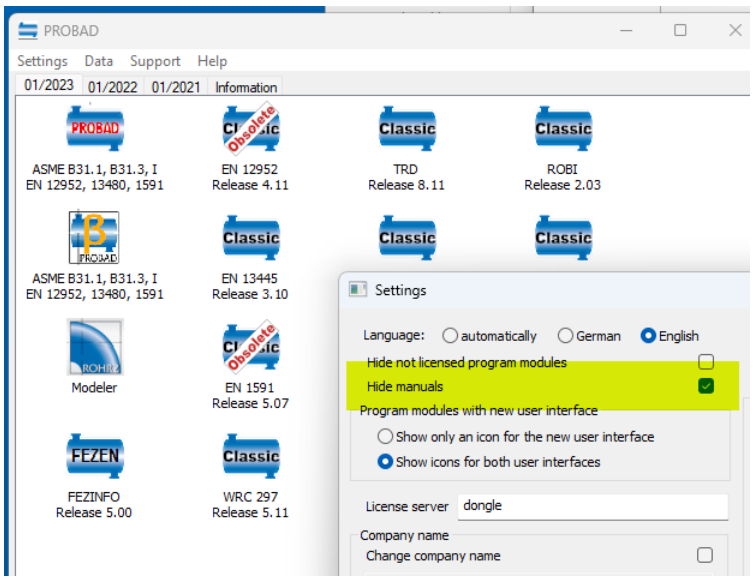
Help | Info

Shows a dialog window with information about PROBAD.

2.2 How to open a PROBAD project

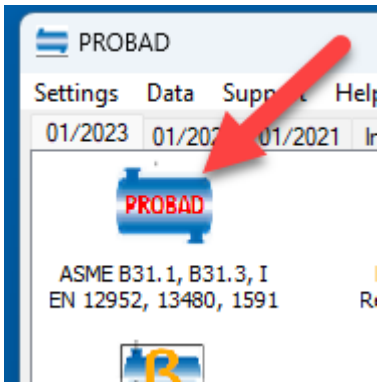
Sample projects/ calculation templates

For a short introduction it is recommend to open one of the available calculations and to start the calculation after the project has been stored by *SaveAs*.



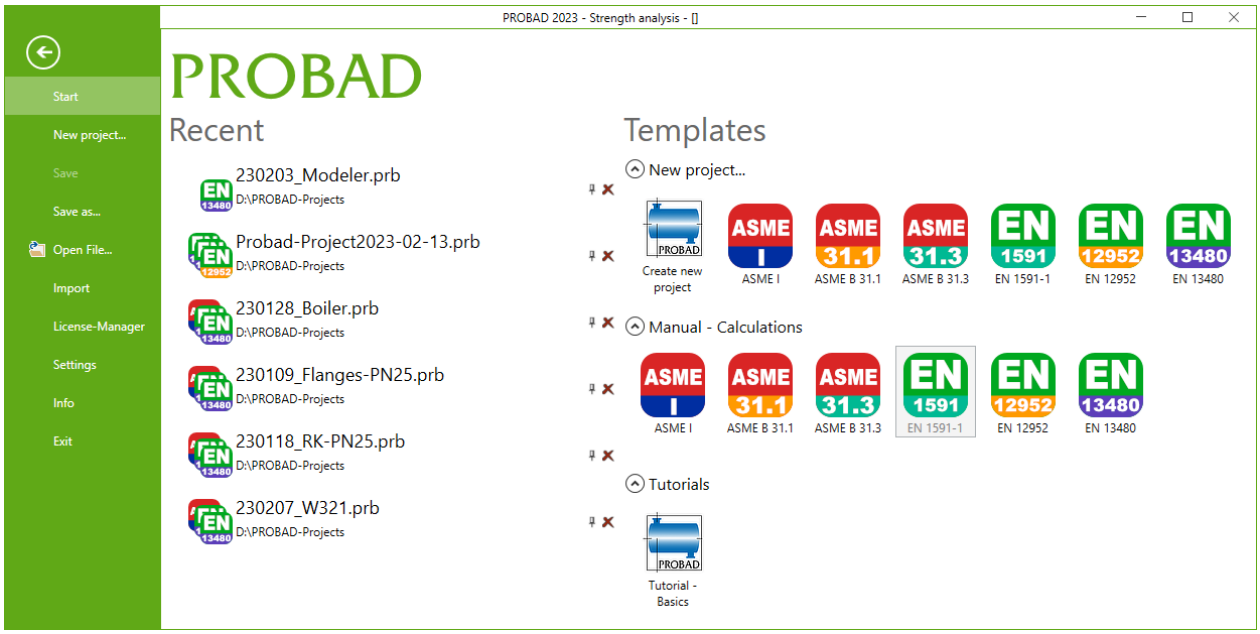
Please make sure that the PROBAD manuals are not hidden (settings | hide manuals)

2.2.1 Calculation with new PROBAD GUI

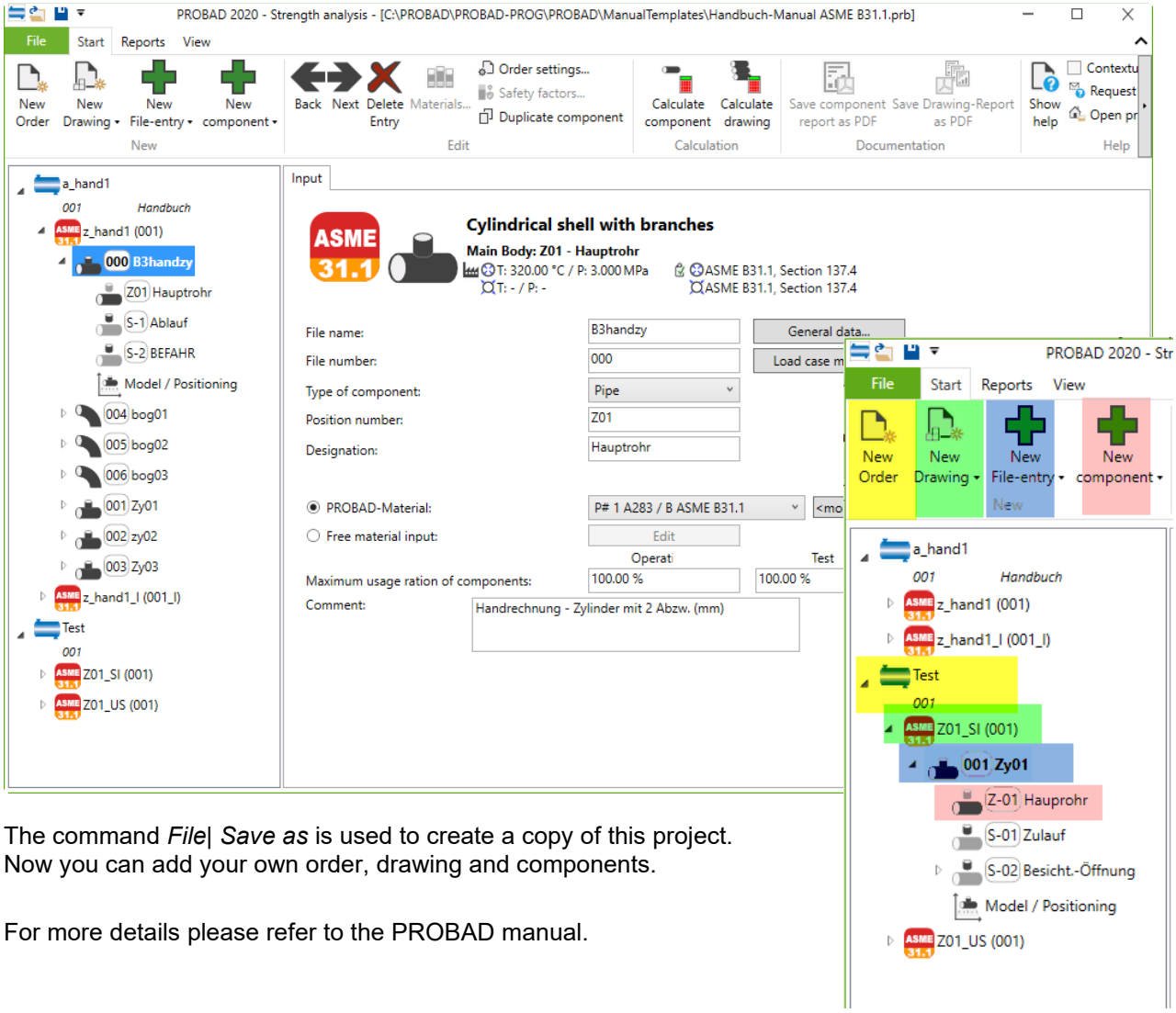


Examples:
Calculation ASME B31.1, similar also for ASME I, B31.3, EN 13480, EN 12952 and EN 1591

- Double-click the ASME 31.1 symbol
- Select templates for ASME B31.1 by mouse click

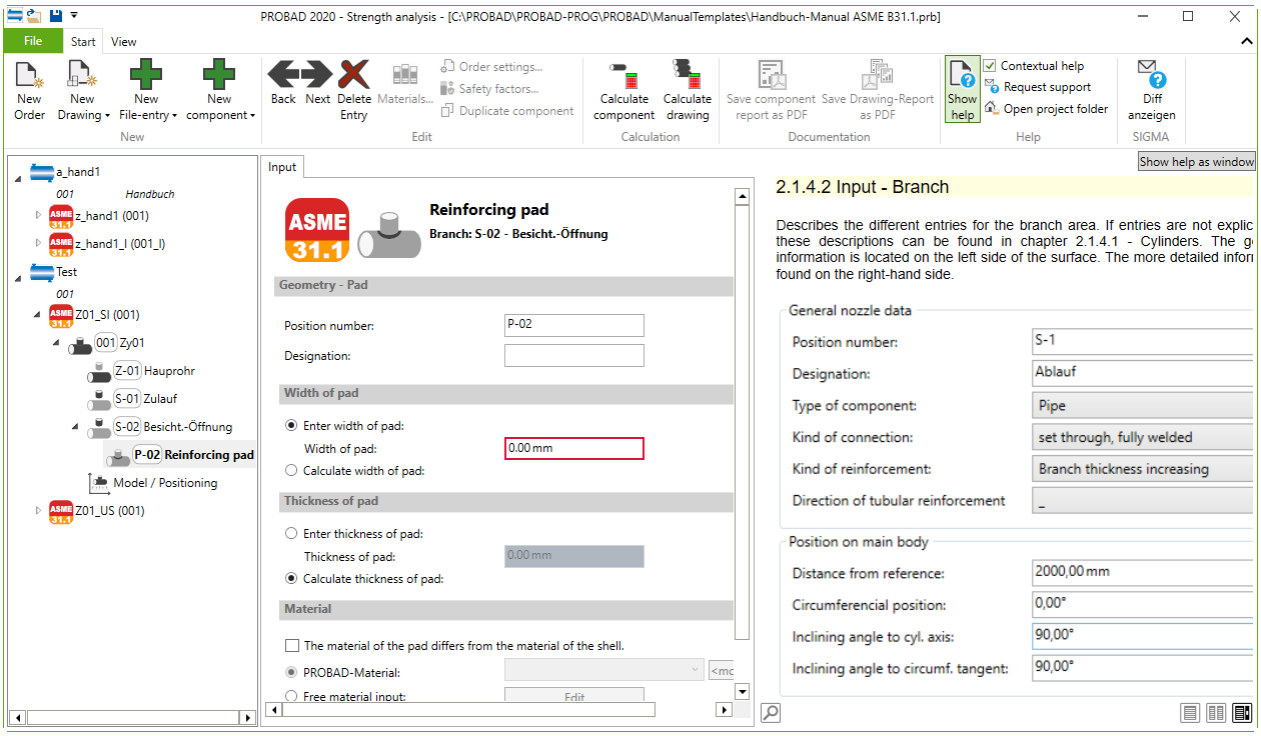


All components which are part of the project are displayed upon mouse click in the project view.

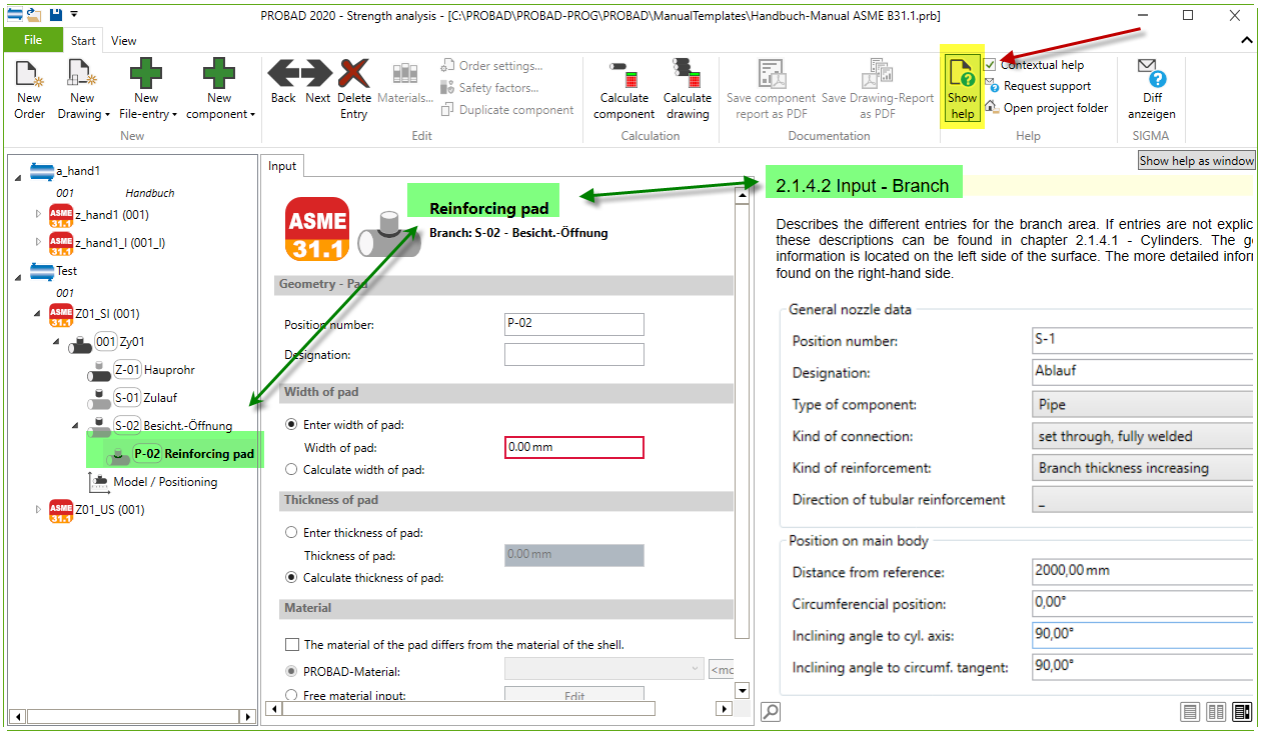


The command *File*| *Save as* is used to create a copy of this project.
Now you can add your own order, drawing and components.

For more details please refer to the PROBAD manual.



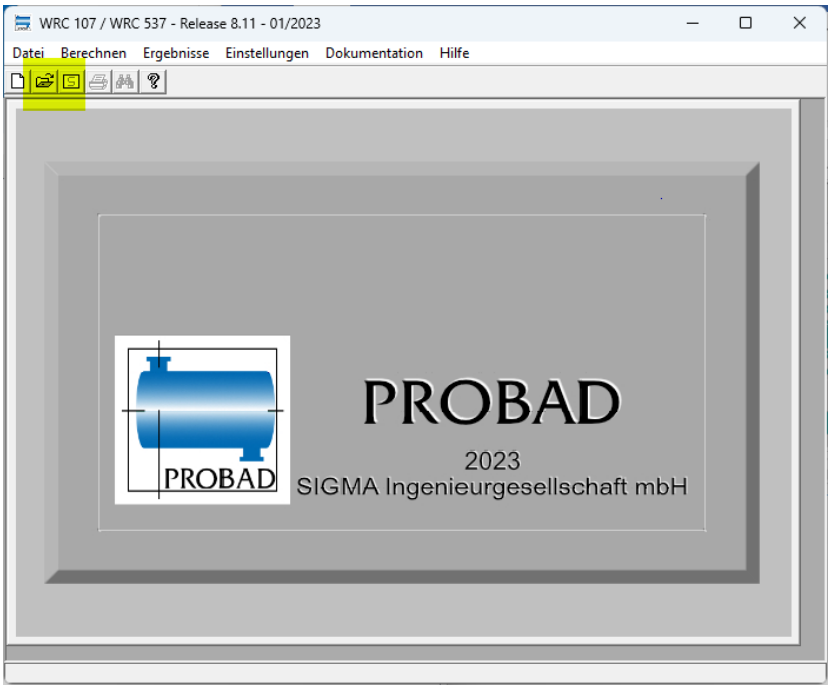
If help is needed, use the contextual help.



2.2.2 Calculation WRC107 (PROBAD old user interface, Type 1)

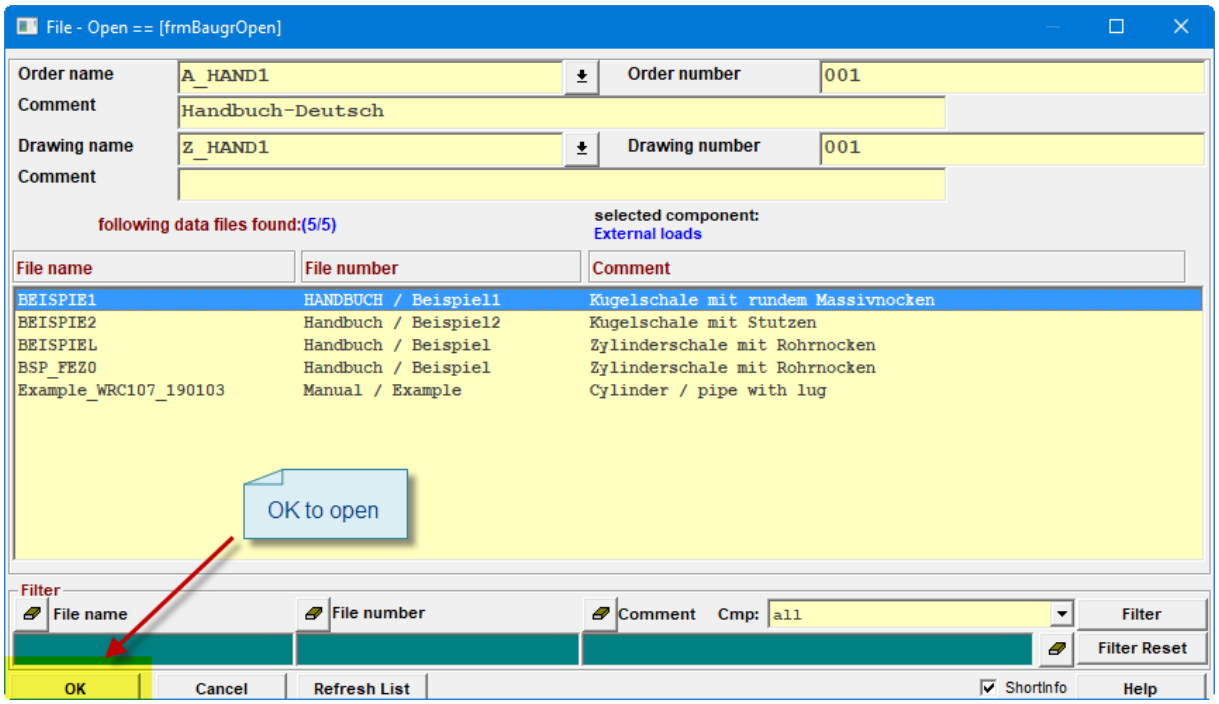
Calculation of WRC107 components (PROBAD old surface type 1).
The calculations of AD 2000 etc. are similar.

Command: double-click on the symbol to starts the calculation

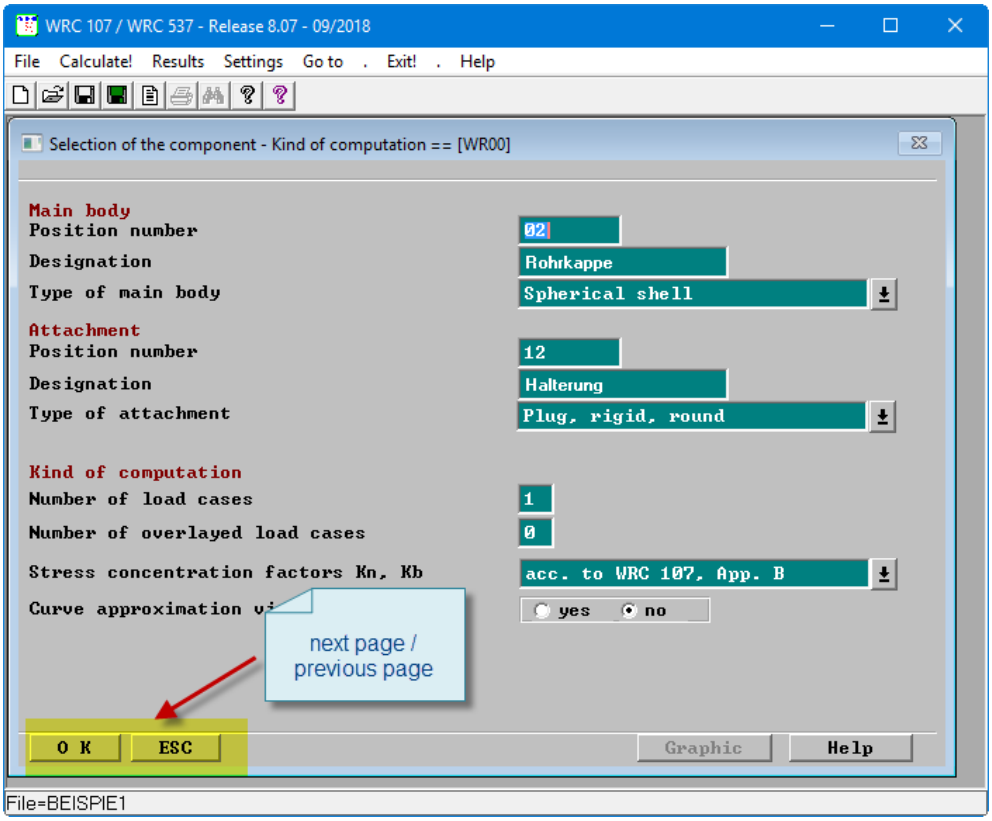


Please ignore a possibly occurring error message concerning a missing directory. The directory wlil be inserted automatically.

Use this function to open the sample calculation available here and select an entry from the list.



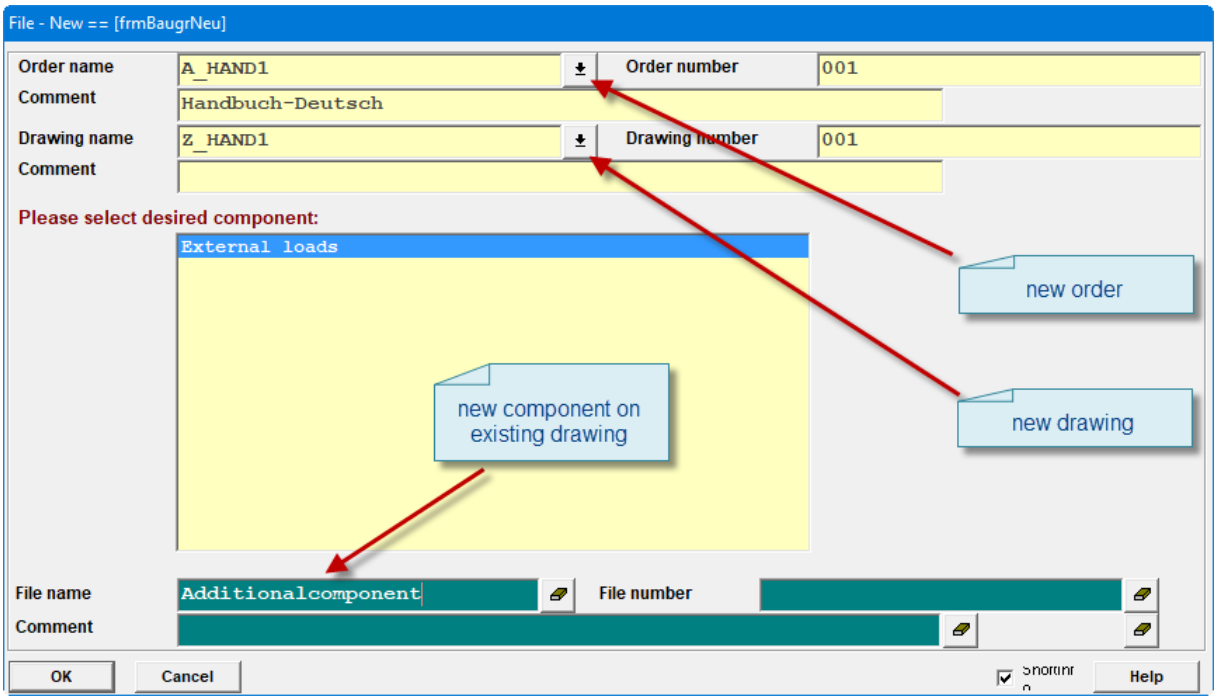
Please note: the project names in the sample projects maybe in German language.
It is recommended to keep the reference example and generate a copy of the project using the SAVE As command.
Next steps:
Modify the green highlighted fields.
Select next or previous pages using OK and ESC.



Creating additional components

Use the command *File*| *New* to define new calculations or components

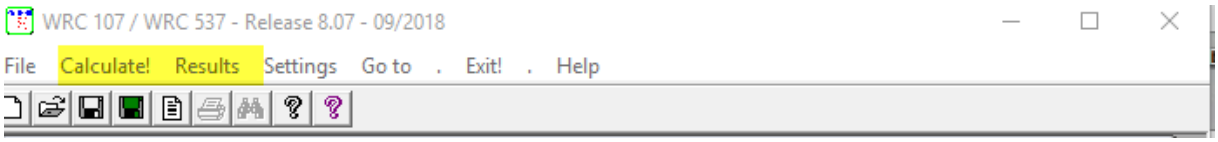
By means of this function it can be generated:
a new order,
new drawing or
a new component on the basis of an available drawing.



For details please refer to the PROBAD help/manual (see 2.1 PROBAD Start Menu)
For additional help in the program please select the particular field and click Help in the right corner of the dialog window.

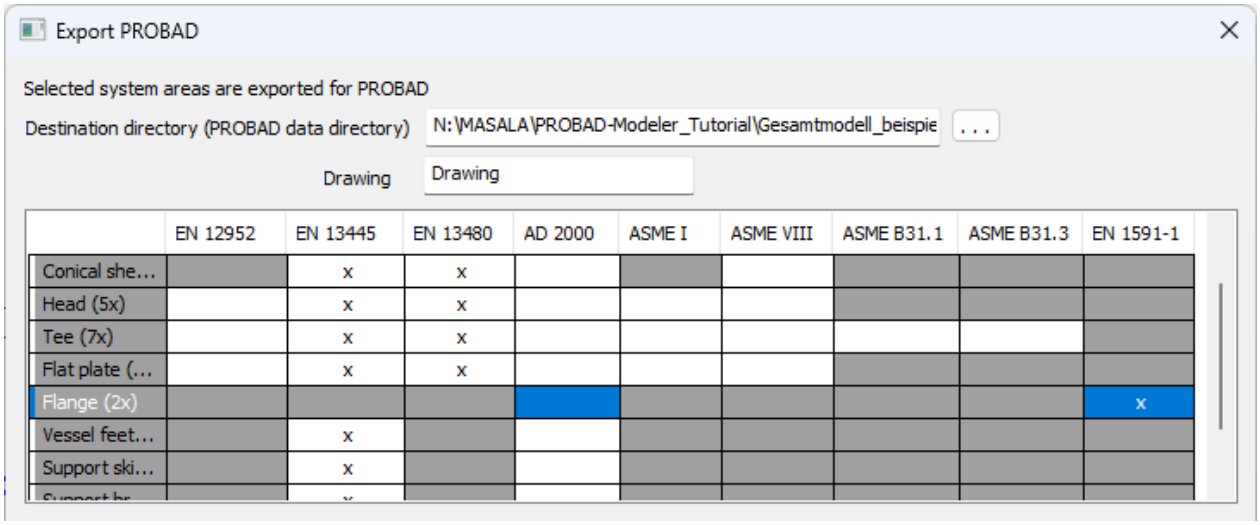
Show results

Start the calculation using the *Calculate* command. Use the *Results* command to show calculation results.



3 PROBAD Modeler

The PROBAD modeler is used to generate complete models quickly and easily based on a center line, irrespective of the set of rules and also across sets of rules.
Generated models can then be transferred to the various PROBAD calculation modules.
Input data for several sets of rules can also be generated in parallel.

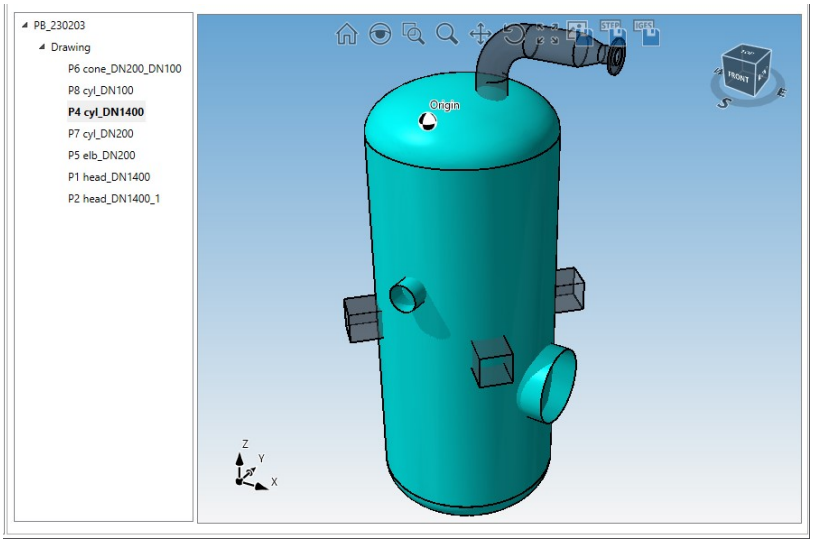


	EN 12952	EN 13445	EN 13480	AD 2000	ASME I	ASME VIII	ASME B31.1	ASME B31.3	EN 1591-1
Conical she...		x	x						
Head (5x)		x	x						
Tee (7x)		x	x						
Flat plate (...		x	x						
Flange (2x)									x
Vessel feet...		x							
Support ski...		x							

In the respective calculation modules, the complete models are divided into subsystems suitable for the calculation (e.g. cylinder with nozzle, head with nozzle or saddle verification).

The Modeler also offers the possibility of determining loads for a stability verification and taking additional loads into account.

Furthermore, the modeler offers the possibility to represent systems as complete systems independent of the required verification.



As the PROBAD Modeller corresponds to a large extent to the user interface of the software ROHR2, every ROHR2 version (from version 34.0) can be used as PROBAD modeler.

3.1 Introduction

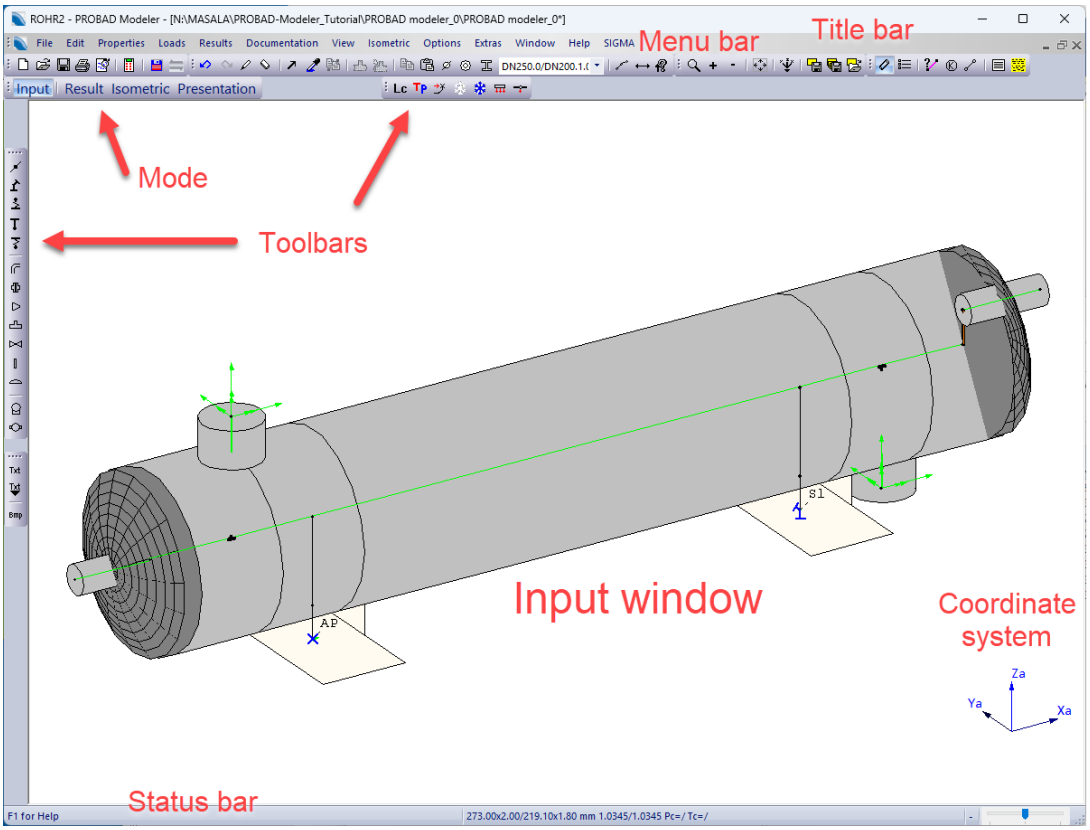
The input window shows the piping system and the drawing created.
All program functions are accessible by menu commands and symbols (icons).
The elements of the user interface can be positioned on the screen be placed free on the screen.

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*,

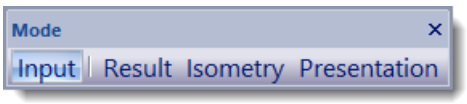
Status bar

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

Toolbar Mode

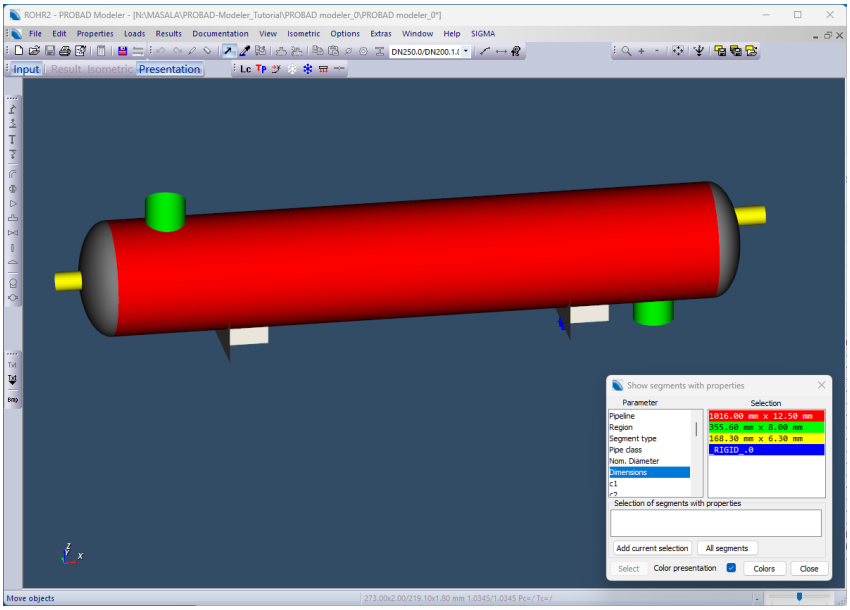
Select between

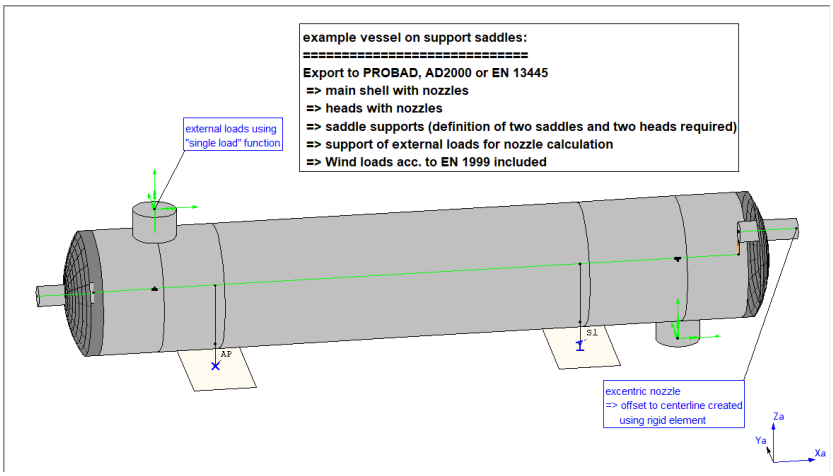
- input mode,
- results mode ~~and~~
- isometric mode.
- Presentation mode,



Presentation mode

The presentation mode is an alternative view mode for inputs and results.
This 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* functions.



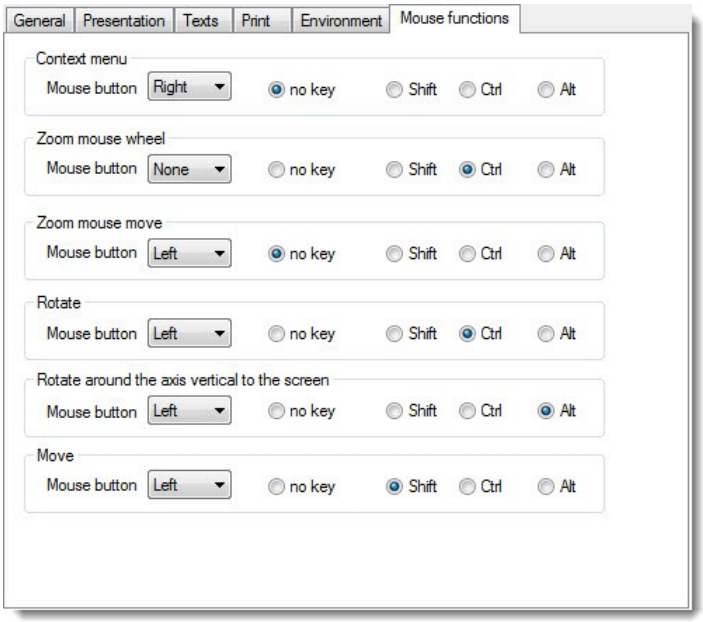


Views can be defined to be used in reports in the presentation mode.

Mouse functions for graphic operations

The key commands of the mouse functions can be adapted in *General settings*:

- 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 to be 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

Select the key which need to be pushed additionally

4 Definition of model in PROBAD

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

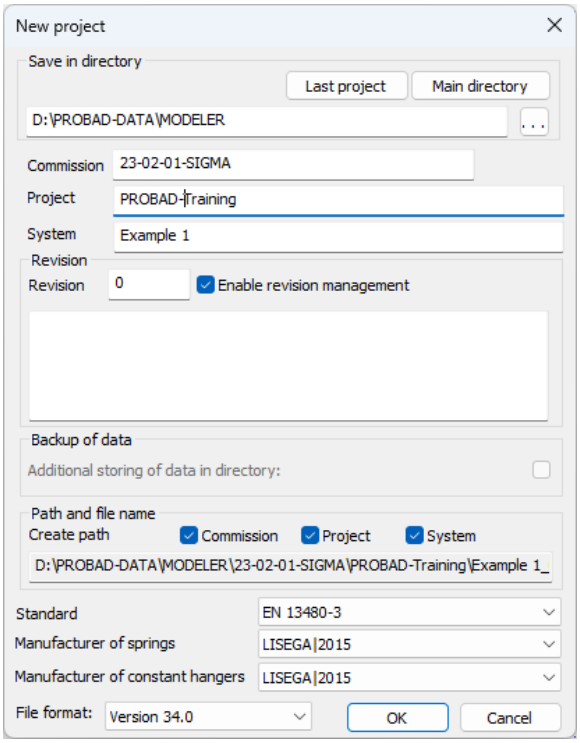
4.1 Creating a new PROBAD project

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

Training settings

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

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

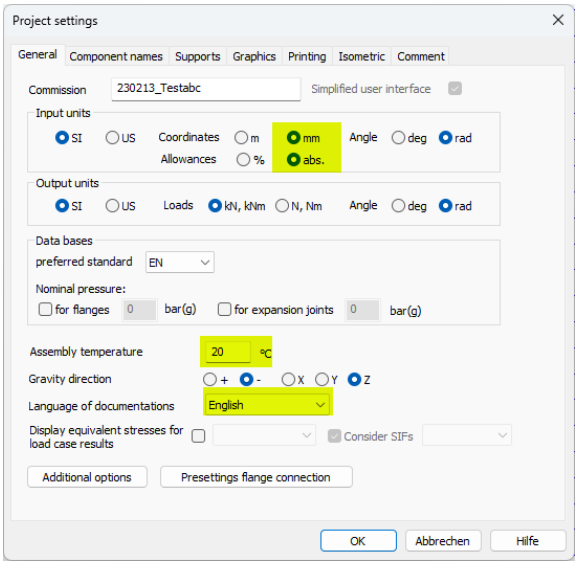


Options| Project settings

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.

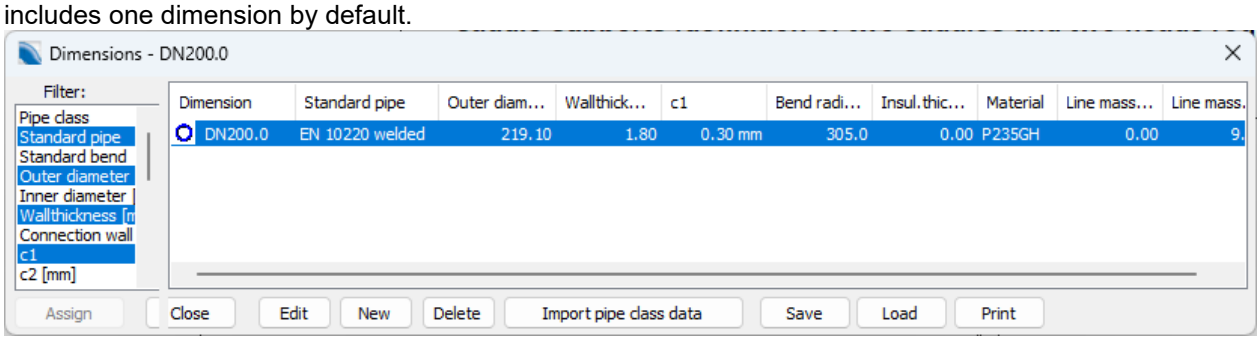
4.2 Model generation

4.2.1 Editing 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.



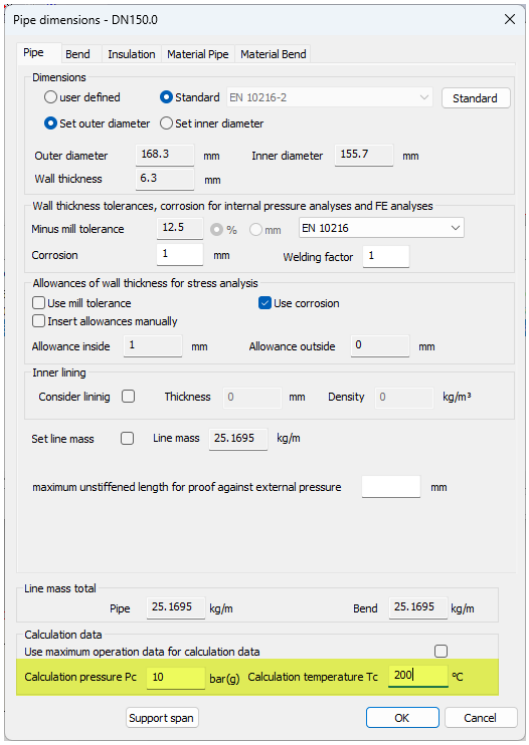
A double-click on the dimension opens the pipe dimensions input window.

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.

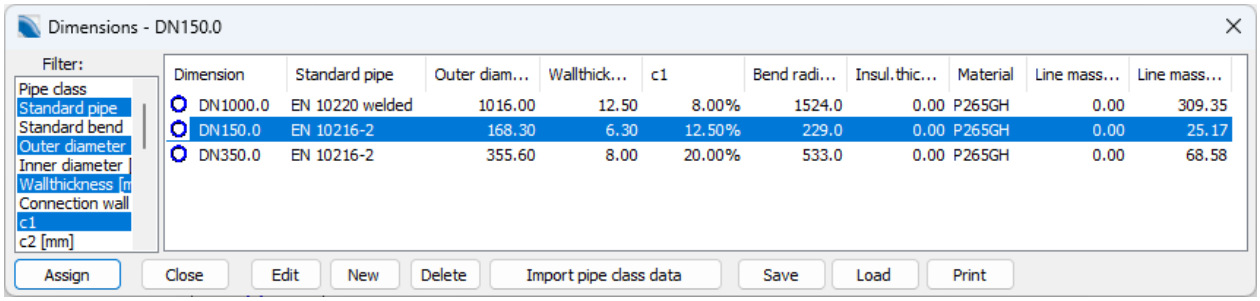
Parameters of the internal pressure definition can be defined in the pipe and bend windows.
Check if tolerances and allowances are to be considered in the stress analysis.

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 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.

4.3 Drawing a system

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

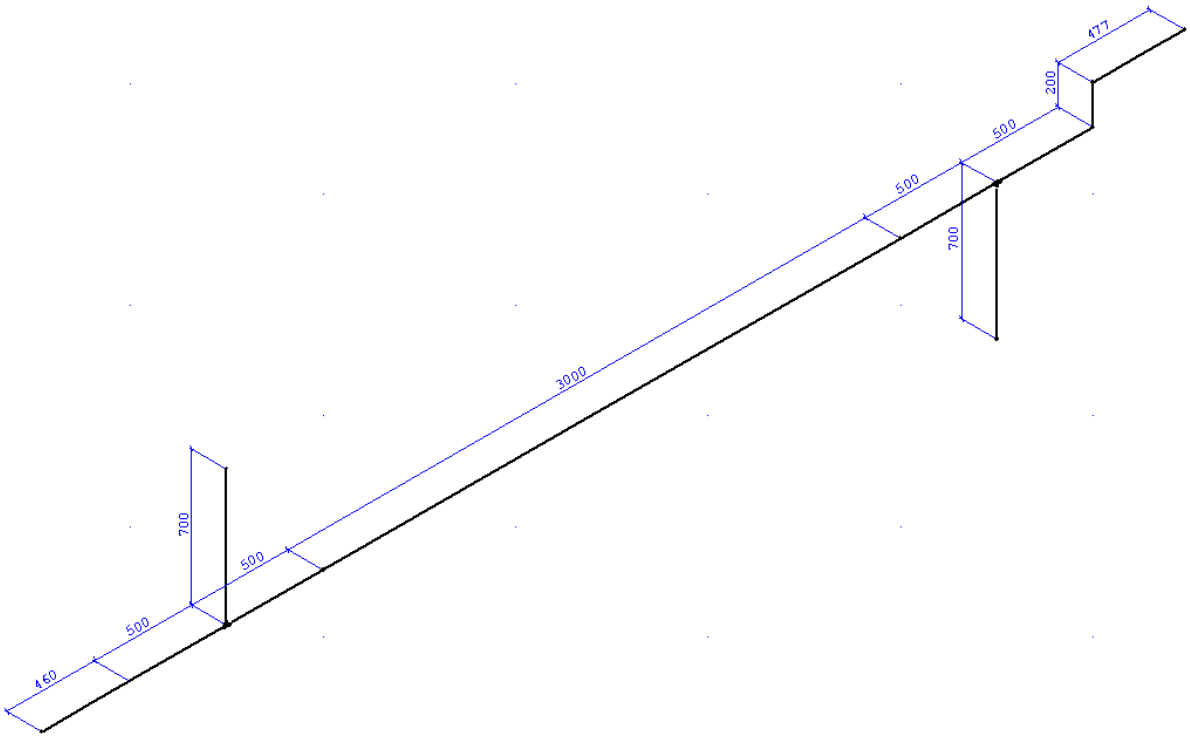
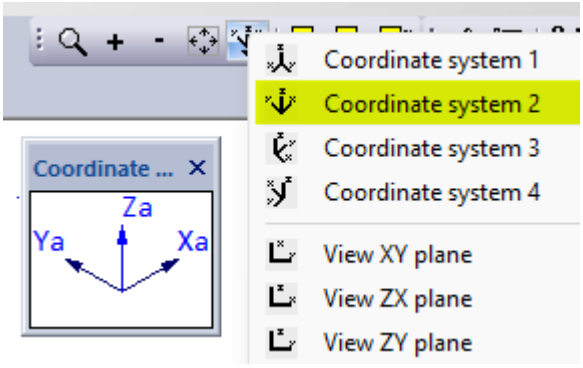


figure: center lines

4.3.1 Select a coordinate system and draw

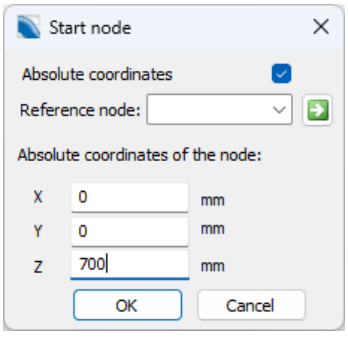
Select a coordinate system from the toolbar *View*



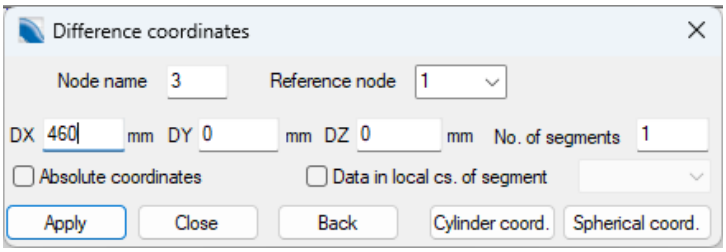
Draw the piping system by means of the *drawing tool* from the toolbar *Edit*

or user the menu command
Menu Edit > Draw

Enter absolute coordinates of the first node



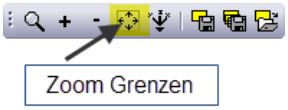
Press any key to open the dialog window difference coordinates



In a first step the whole system 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

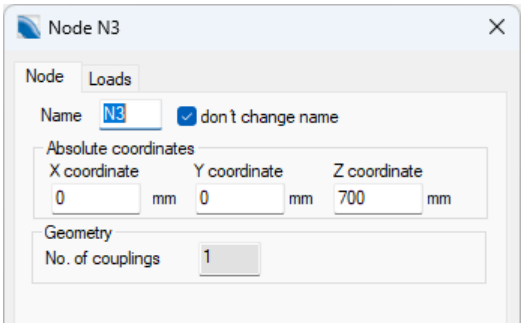
Drawing a branch

- 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 and start node/ end node manually.
- Draw a branch with DN150
- „Close“ to terminate the window

4.4 Input data modification and checking

4.4.1 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).

4.4.2 Register Nodes

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

Here several changes can be made, e. g.:

- Change the node name (max. 4 characters)
- Change of the node coordinates

4.4.3 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.

4.4.4 Additional registers

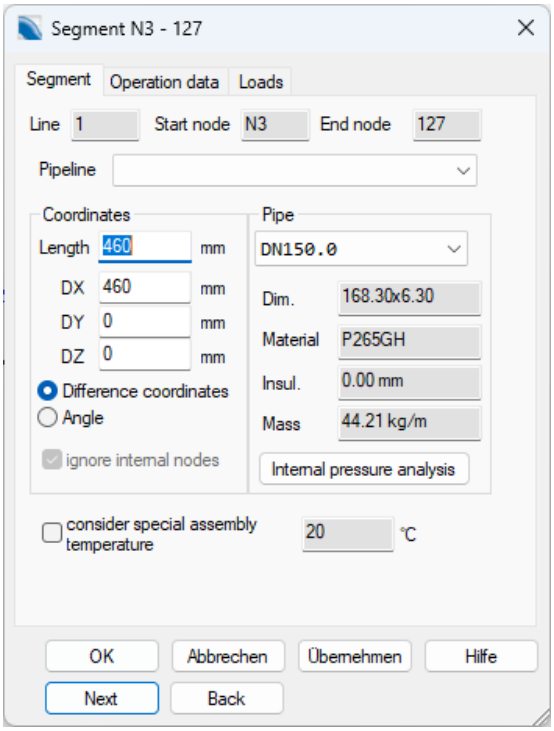
If there are boundary conditions (e.g. like supports) assigned to a node, they are shown in their own registers. Support conditions can be modified.
For more details referring to support types and support conditions see the following paragraphs.

Input record help

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

4.5 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*

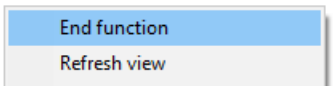
4.6 End function

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



4.7 Select

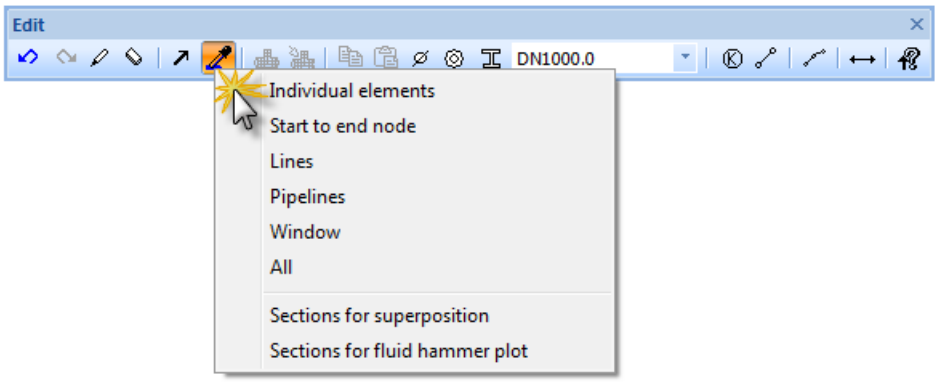
The *Select* function often is used to define parts of the model and add parameters to those in the next step. E.g., new dimensions, a new material or varying operation parameters can be assigned.

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

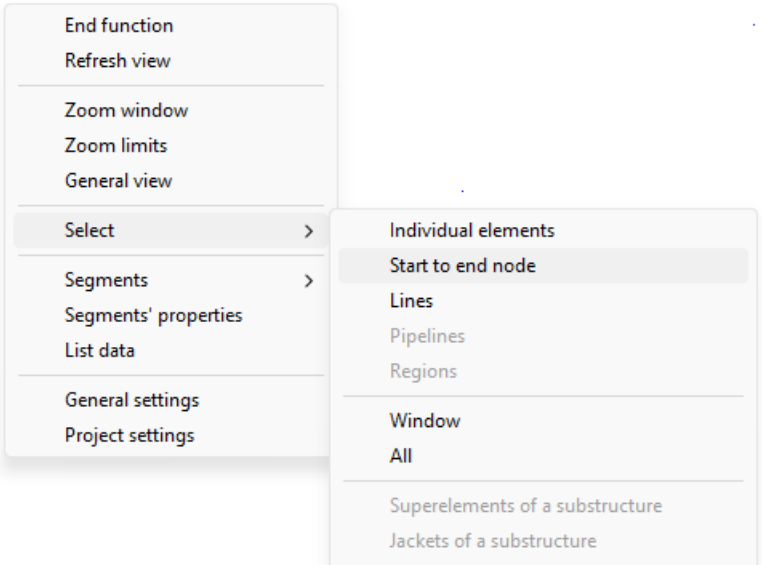
Edit > Select

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:

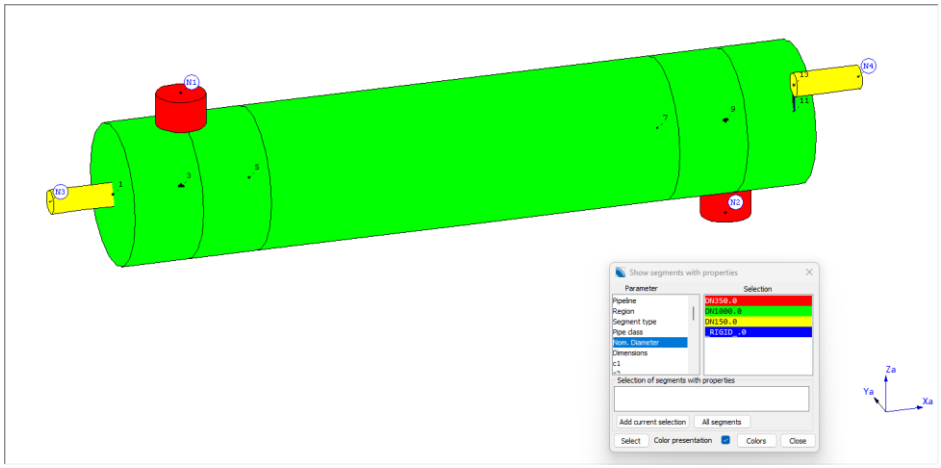


- the mouse menu (click the right mouse-button over the graphic) see also *Mouse commands*



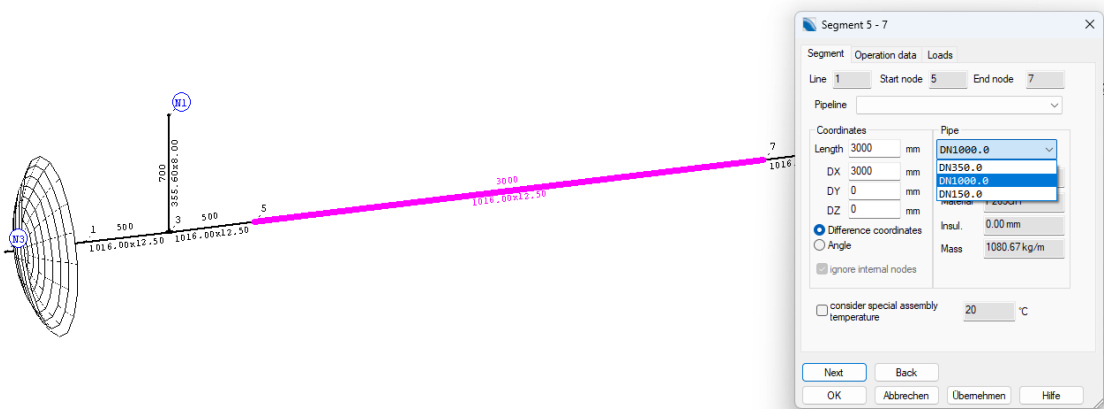
4.8 Assign dimensions

Now all sections can be assigned the corresponding dimensions.



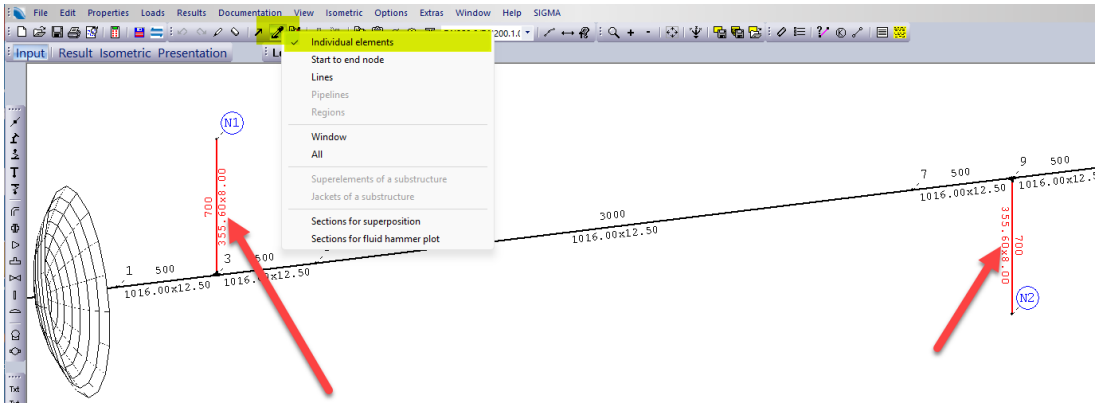
There are essentially two ways to do this:

Double-click a section and assign the correct dimension around the section dialogue.

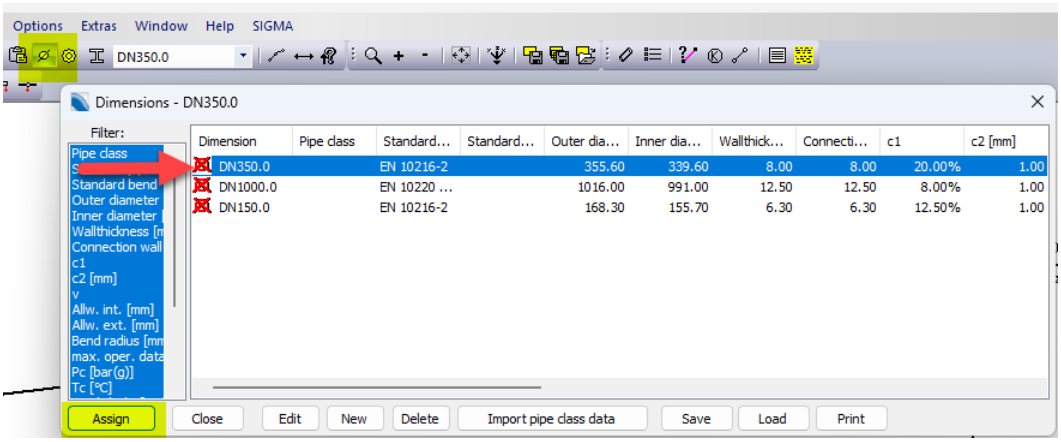


Marking a group of sections and assigning a dimension to the selected sections.

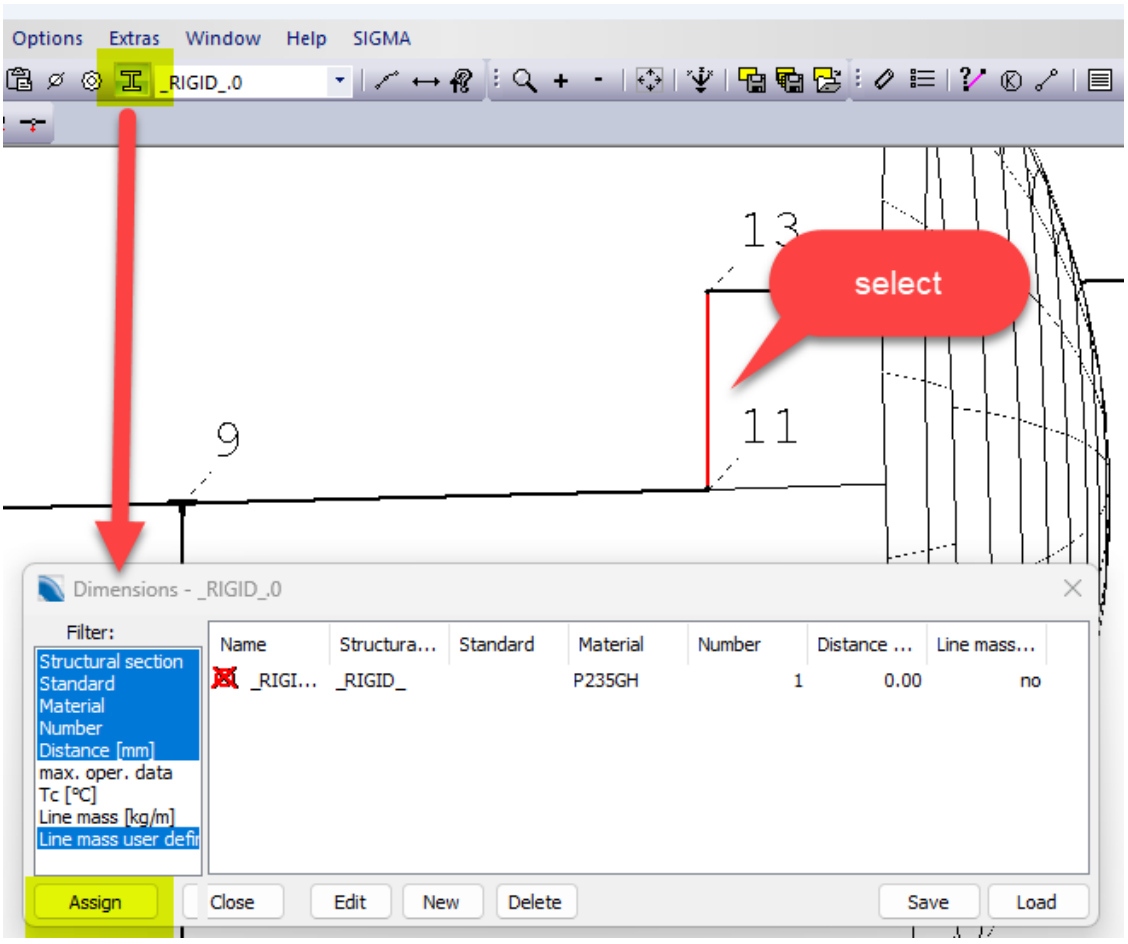
At first you have to select a group of sections:



Open the Dimensions window, select the according dimension and assign it:



Please note!
If you want to model an eccentric nozzle, the connection to the center line must be defined as a rigid profile.

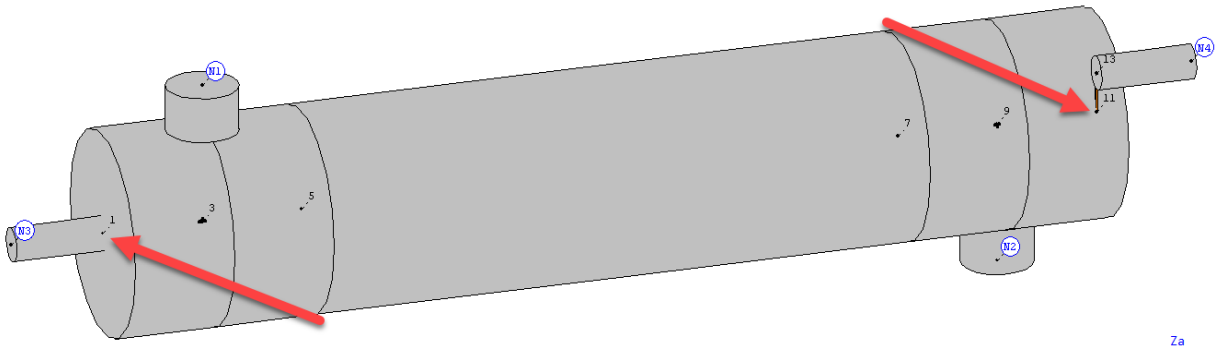


4.9 Insert components

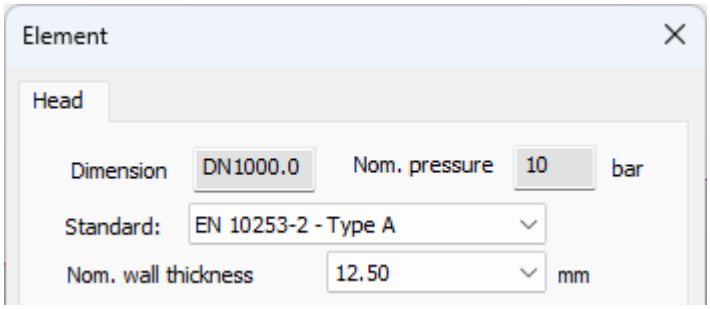
4.9.1 Insert a head

These steps are required to insert a head:
Edit > Insert component > Head
or use  .

- Click the node where the head is to be inserted.



Define the head, in this case acc. to EN 10253-2



Please note!
You can use function “insert multiple” for inserting identical components several times.

4.9.2 Insert a flange or blind flange

Please select the corresponding symbol in the tool bar.

The subsequent steps correspond to the procedure for the heads.

4.9.3 Insert a reducer

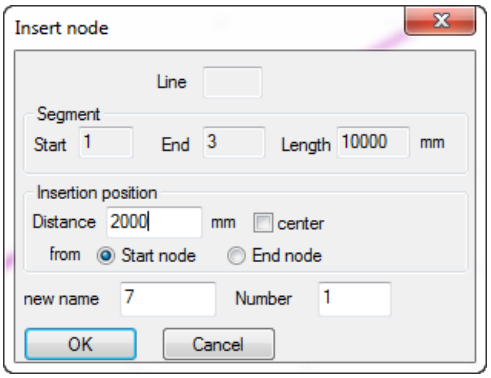
Note!
The following instruction assume that the diameter of the created pipe is DN200.
When a reducer is inserted the nominal width of the pipe changes from the reducer position up to a selected end node.

These steps are required:

Edit > Insert component > Reducer

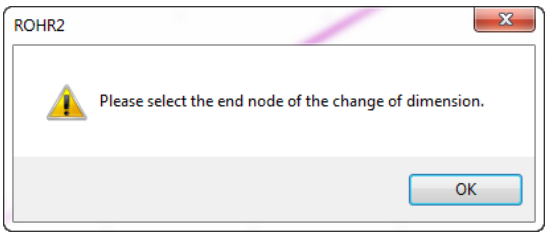
or use  .

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



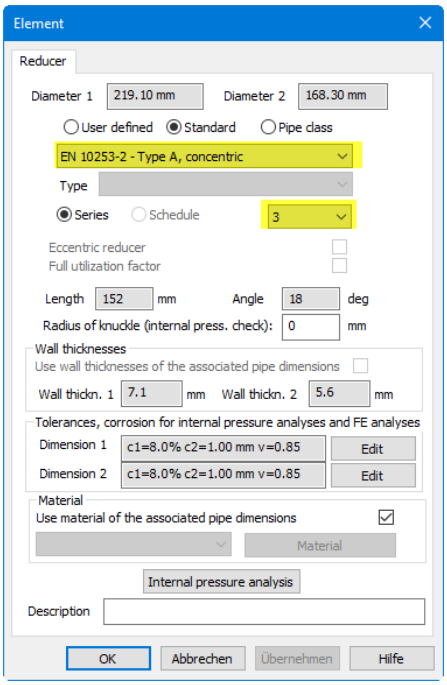
After that a message box occurs: the node representing the end of the dimension change need to be selected.

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:

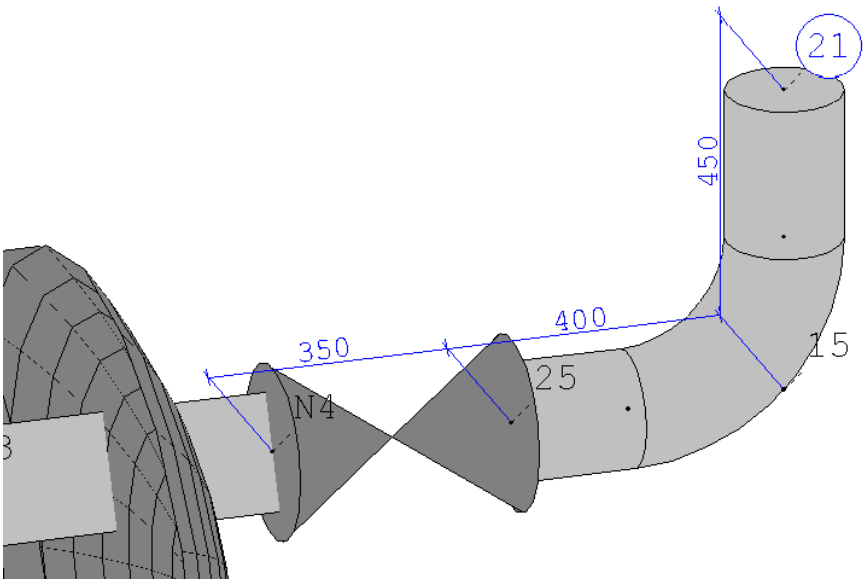
- Select a dimension
  if not existing the dimension can be inserted here.
- Select reducer from the standard DIN 2616, part 2
- Confirm by OK and insert the reducer.



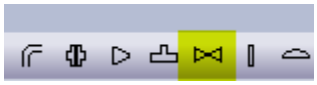
Note:
If the pipe has been drawn in DN 150 please consider:
- *the insert position must be 1800 mm to the next bend*
- *the pipe needs to be expanded to DN200 up to the branch and beyond.*

4.9.4 Insert a valve

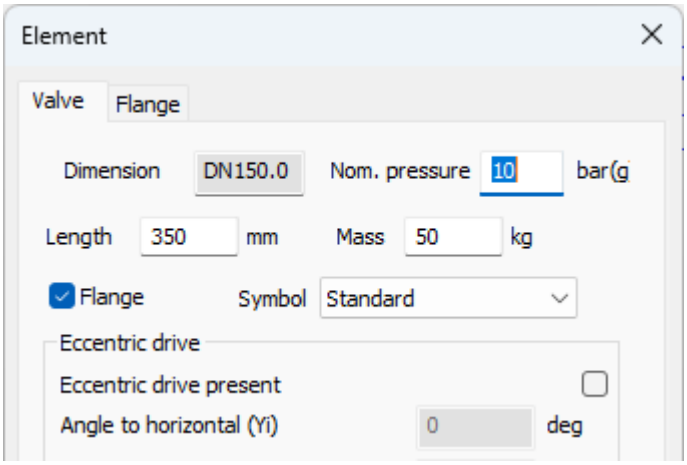
Normally components need to be inserted directly on a pre-defined center line of the pipe.
It is not possible to append a component to a drawn segment.



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

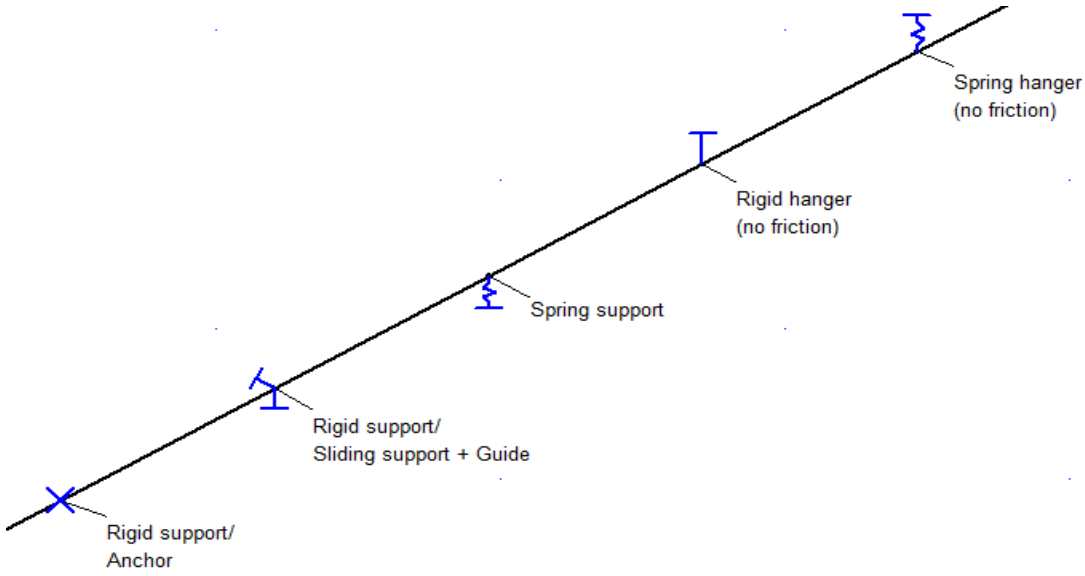
- Length seal to seal or weld to weld
- Enter instrument mass
-
- Select Option *with flange* and define flange if applicable (no flanges in this example)

5 Insert supports

Defining supports is only necessary if you also carry out proof of stability and it's not required if you only carry out internal pressure verifications.

When defining a model, it is also necessary to specify support conditions in order to be able to determine loads for a subsequent verification if required.
The simplest option is to model a support with its functionality only, without modelling the support structure itself in more detail.
The Modeler offers the following functions for this purpose:

5.1 Support condition symbols



Alternatively, it is possible to define supports in detail if a support itself is also to be verified by calculation. This would be the case, e.g. for

- vessels on saddles or ring supports
- vessels on bracket supports

Details on the definition of saddles, brackets and other types of supports are given in the following sections.

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

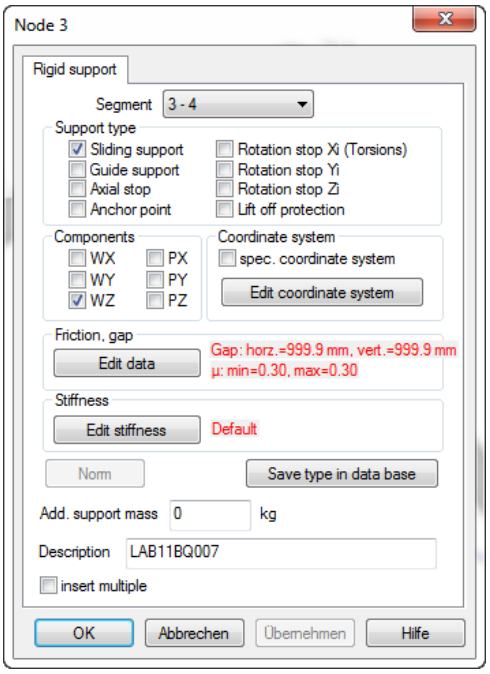
The design of the spring hanger  shall be carried out by the program (Default-settings, no more inputs required).

5.2 Supports

5.2.1 Rigid supports

Defining a rigid support

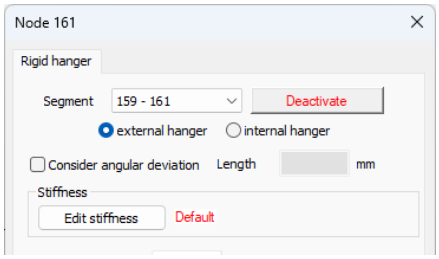
ROHR2 regards sliding supports, guided supports, axial stops and anchor points as rigid supports.



The way to insert a support:

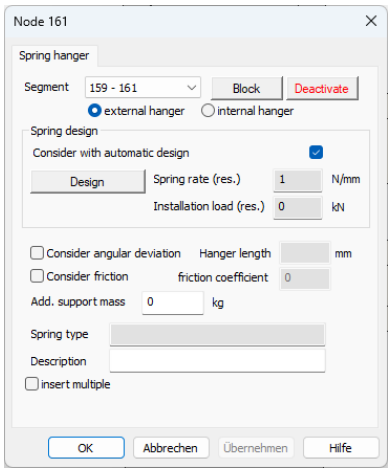
- select the type of support from the toolbar (toolbar on the left side)
- select the desired node or segment by the cursor
- define support. Alternatively here types of support or components (degrees of freedom) may be assigned.

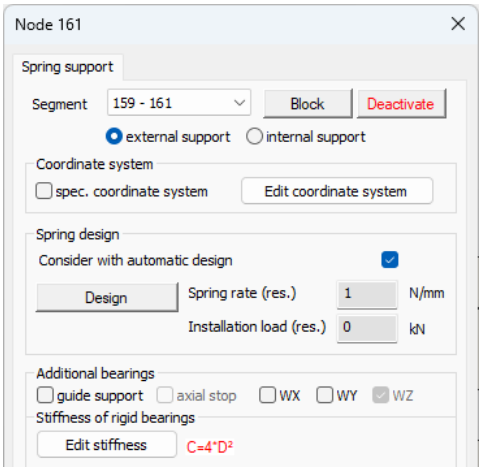
5.2.2 Rigid hanger



Usually no additional input is required. For more details please refer to manual.

5.2.3 Spring hanger and Spring support





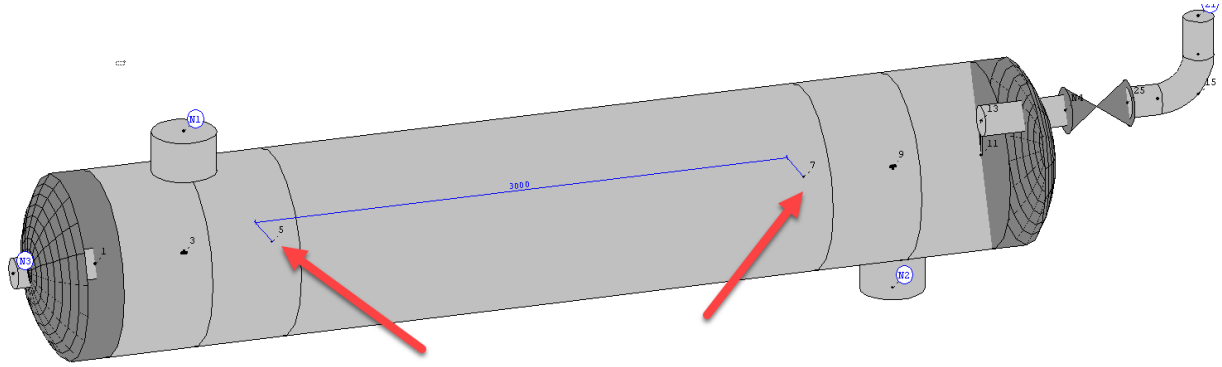
Spring types and required preload are defined by default during calculation. An entry is only necessary if existing spring bearings with a predefined setting load are to be taken into account.

Please note!
The main difference between the definition of a hanger and a support is that no friction is taken into account in the case of a hanger and therefore no horizontal forces occur.

5.3 Supports in detail

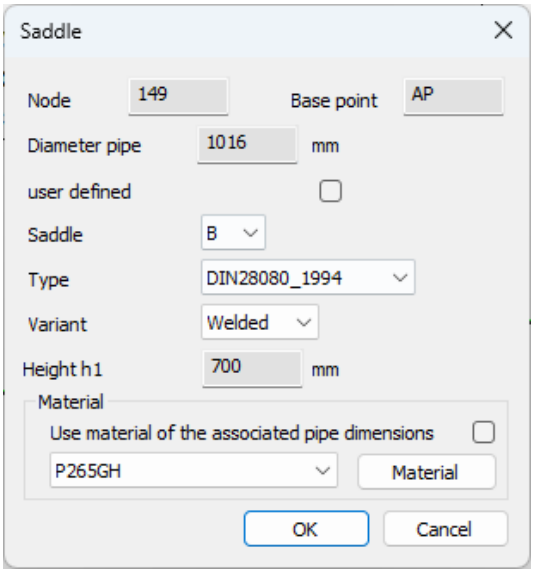
5.3.1 Saddle

In this example we have to define two saddles:

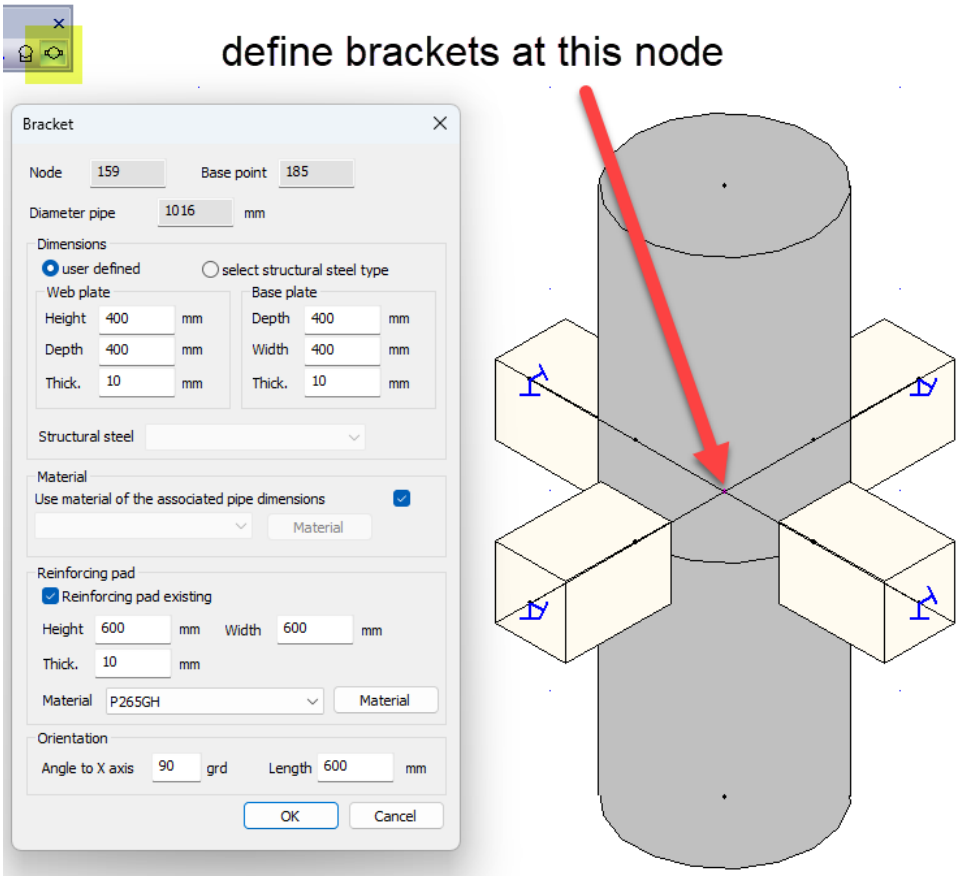


Supports may be inserted at existing nodes or segments. If the user places a support into a segment, automatically an intermediate node is created at this place.

Please note!
If you want to insert a saddle or brackets this is possible only at existing nodes.



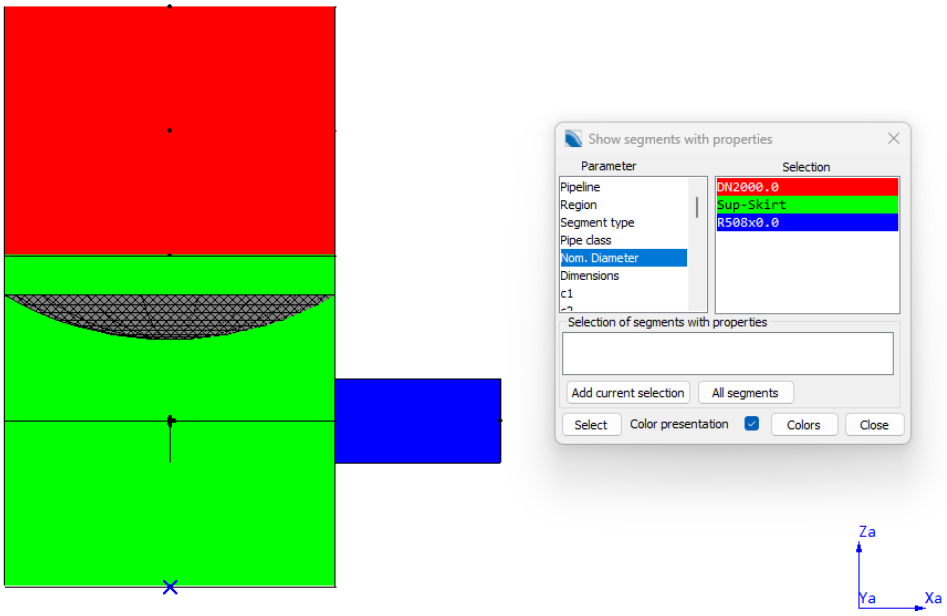
5.3.2 Brackets

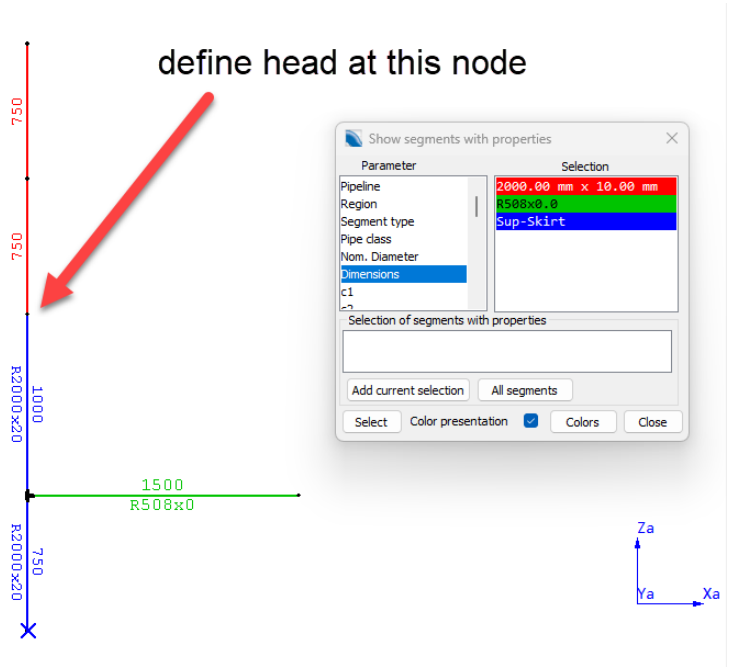


Please note!

- If you want to use a structural steel type instead of user defined dimensions you have to define it at first using the command Edit – Dimensions of structural steel sections or

5.3.3 Support skirts



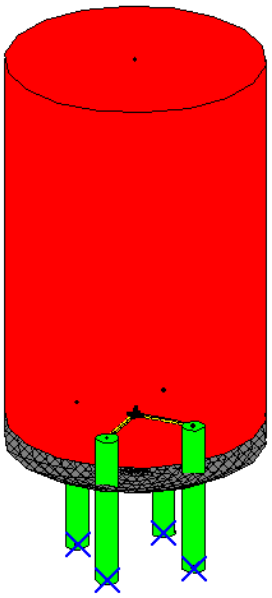


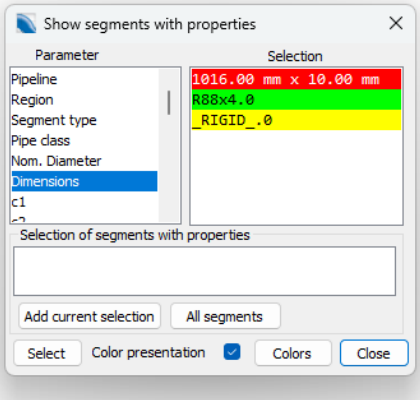
Please note!

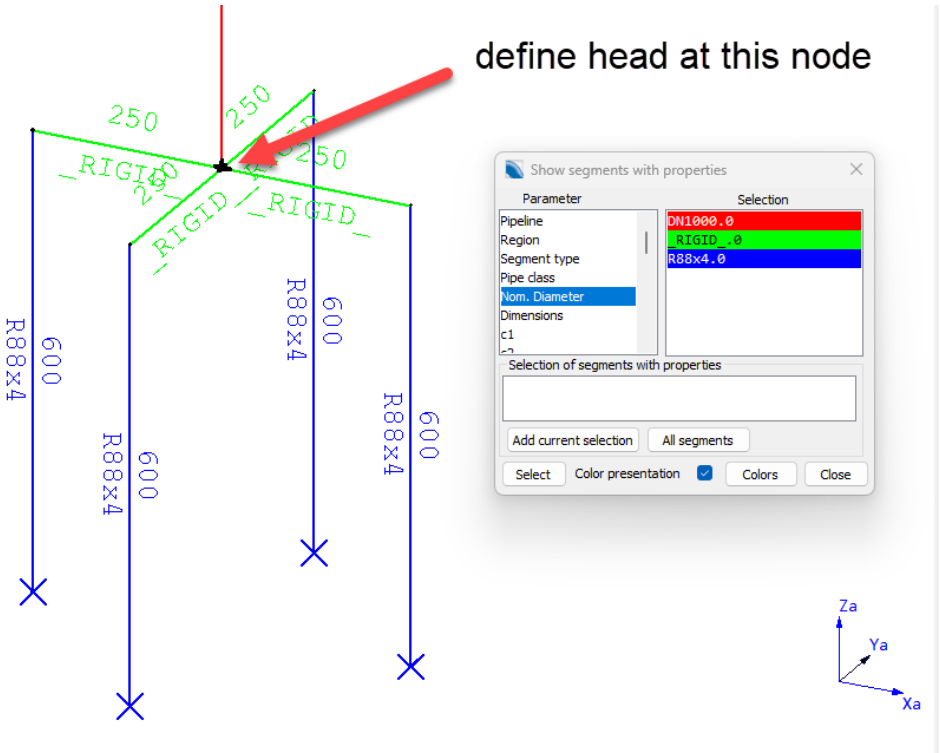
- Define skirt not as a pipe but as a round structural section
- Define the head at the transition point between the vessel and the skirt.

It is also possible to attach a nozzle to the bottom of the vessel. In this case, the center line of the nozzle and the skirt are positioned at the same level. A corresponding warning is to be ignored.

5.3.4 Supporting legs







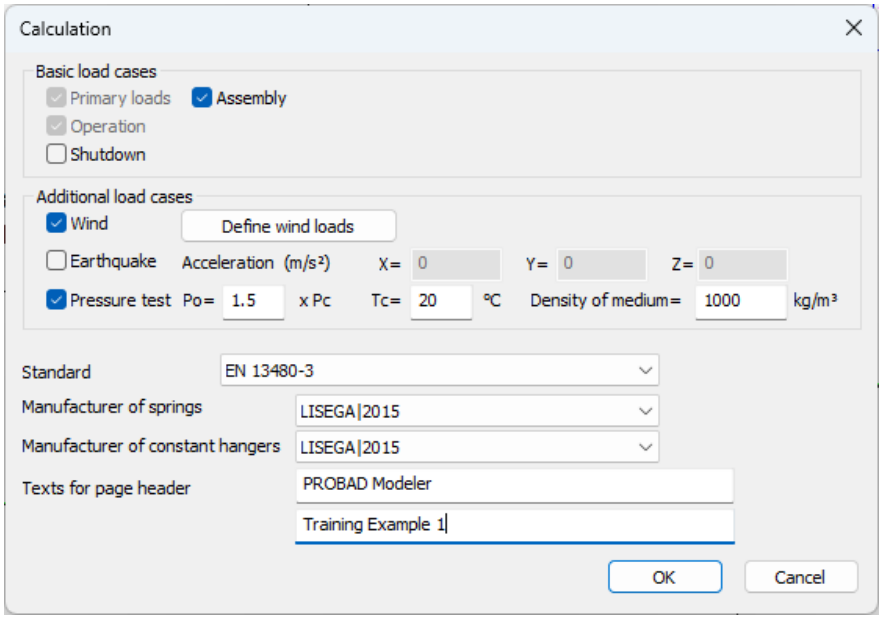
Please note!

- Define the head at the end point of the center line.
- Define supporting legs not as a pipe but as a round structural section
- Define the connection between the center line and the feet as "Rigid".nodes.

6 Load case definition

Defining load cases is only necessary if you also carry out proof of stability and it's not required if you only carry out internal pressure verifications.

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



The load cases primary loads (weight), assembly, 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.
Load case assembly, if selected, is always calculated without medium, without pressure and at assembly temperature.
Load case shutdown, if selected, is always calculated without pressure and at assembly temperature.

6.1 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)

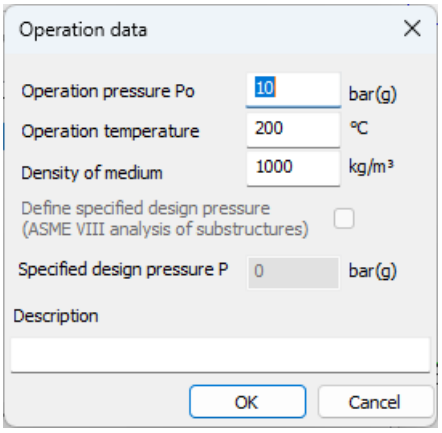
Select segments

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.

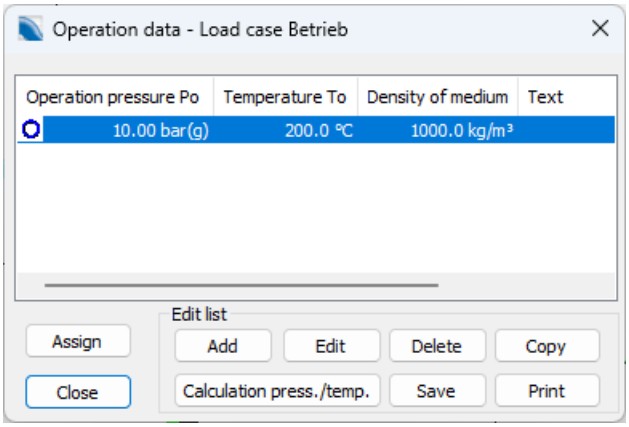
Operation data, example

Consistent operation data for the entire system:

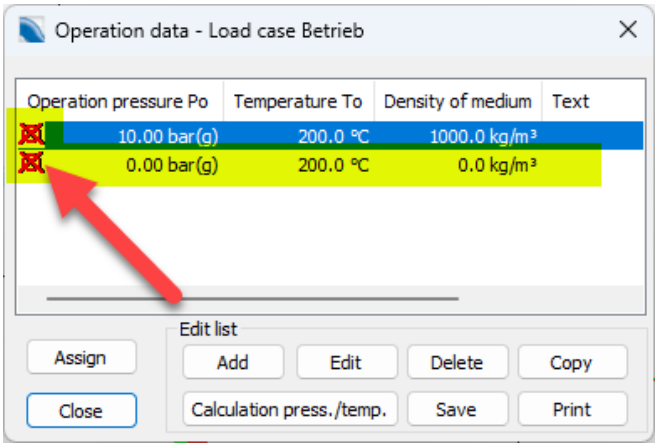
- 20 bar / 285°C / 1000 kg/m³

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.
A dataset with 0 bar/ 0.0 kg/m³ is added automatically.



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

6.2 Additional loads

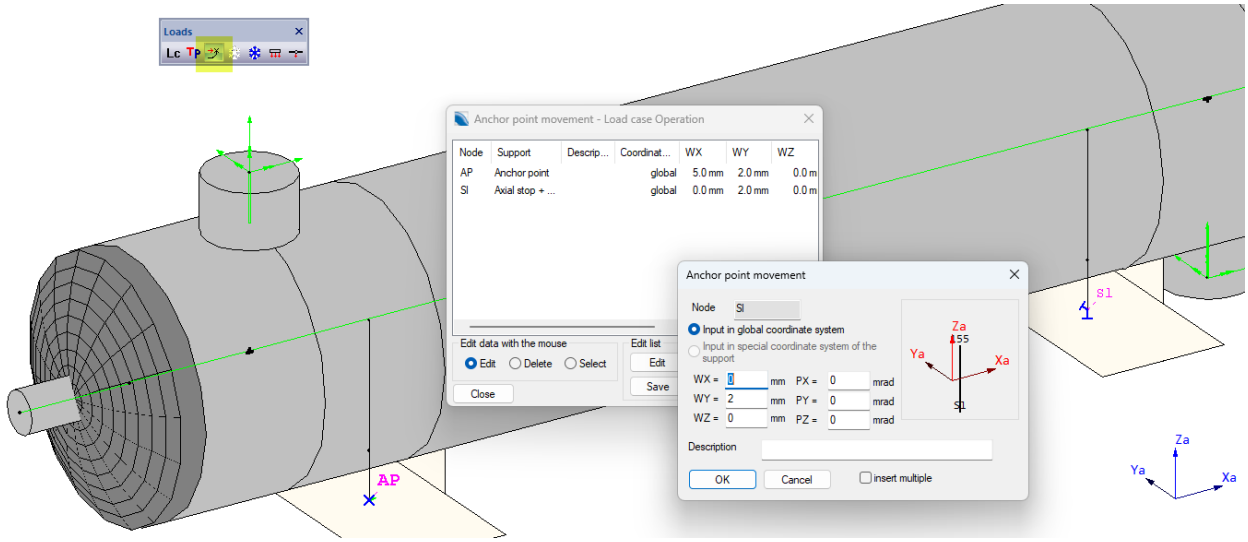
The definition of additional loads is restricted:
Wind loads and constant earth quake accelerations can be entered directly in the load case dialogue. The assignment of this loads to load cases is done automatically. It is not required and not possible to select the current load case.

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

Besides that, additional loads can be defined as follows:

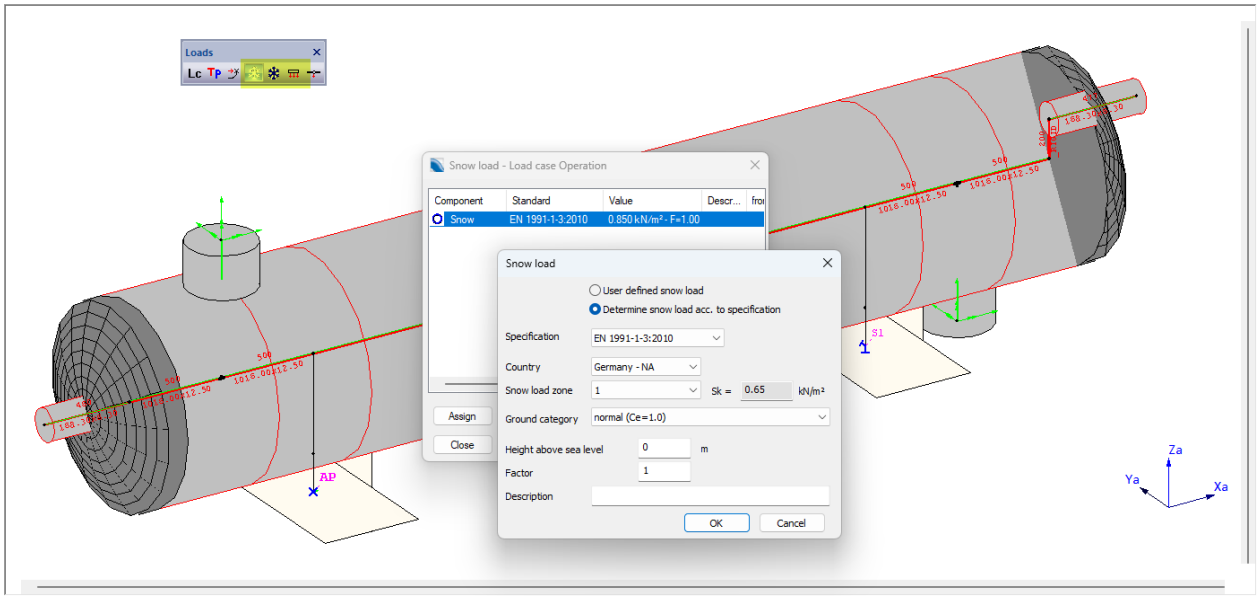
Click the according symbol from the toolbar loads and then click the node in the model where to assign the load.

6.2.1 Anchor point movements



Anchor point movements can be defined only at nodes with support.
They are considered only in lc operation

6.2.2 Snow, Ice and line loads



Snow and Ice loads are considered always with load case primary loads.

Please note:
You have to select the whole system (or a part of it) to assign the defined loads.
Finally, the defined loads must always be assigned with ASIGN.

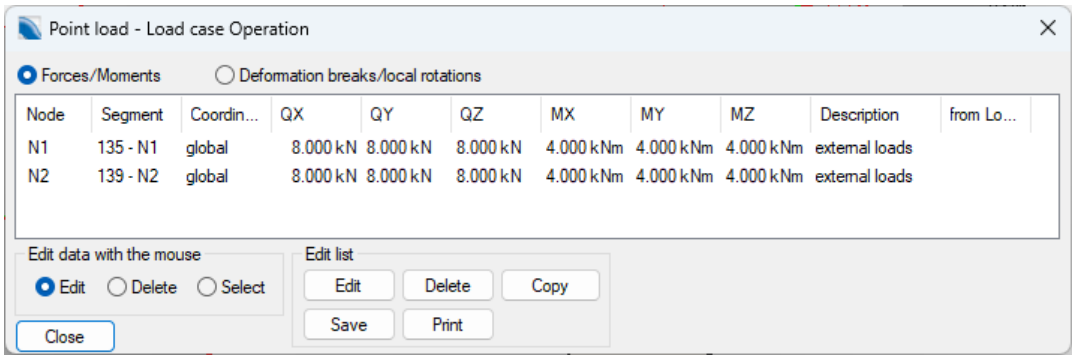
6.2.3 Point loads

Point loads are usually specified at nozzles to take into consideration external loads due to connecting piping or components.

6.3 Loads for this example

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
- Point loads acc. To table
-

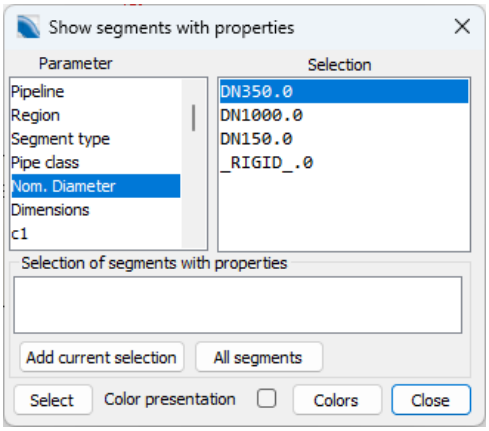


7 Checking the input data

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* and *List data*

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.

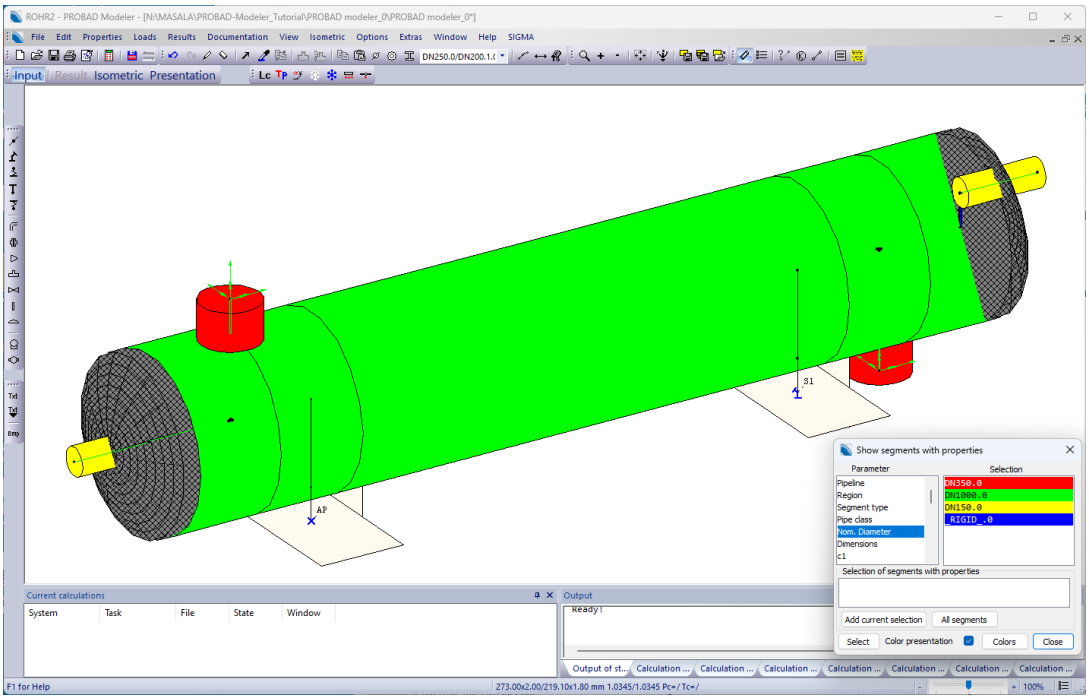
Selection

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*

7.2 Correcting and adapting the input data

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

7.2.1 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

7.2.2 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.

7.3 Calculation

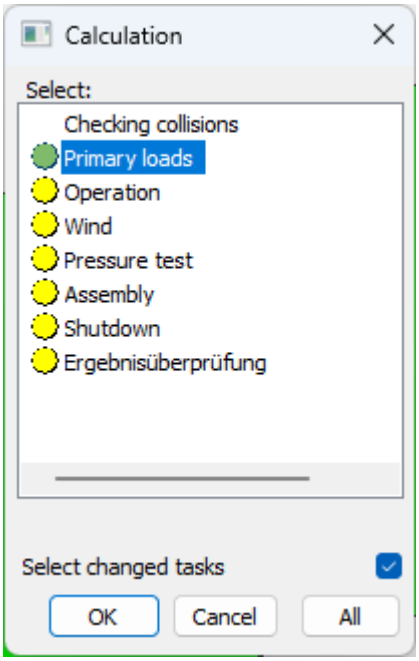
Defining load cases is only necessary if you also carry out proof of stability and it's not required if you only carry out internal pressure verifications.

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.

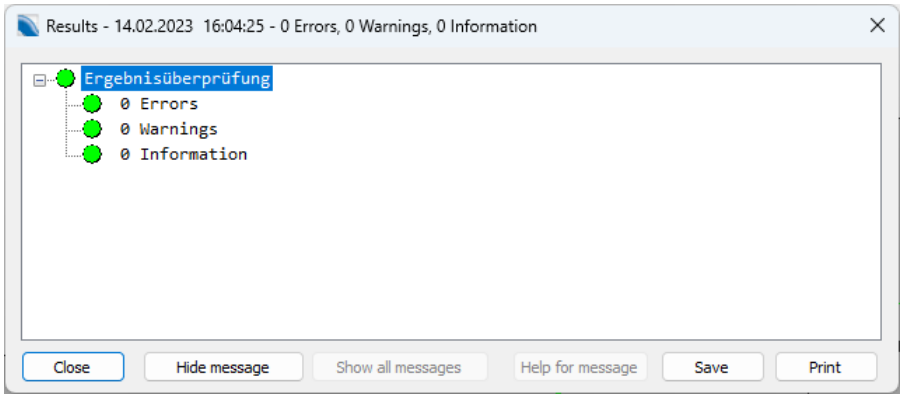


The colored symbols beneath the load cases are showing if the results have been updated.

 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.



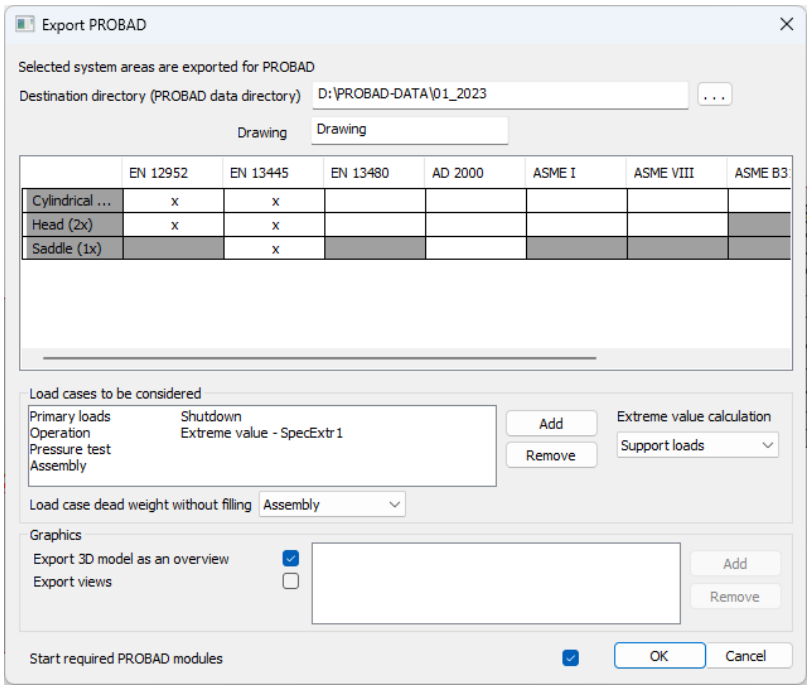
	green	Everything is alright. The calculation runs without any mistakes
	blue	General system information
	red	Error messages. a problem occurs: The calculation result is totally missing or it is not recommended to use the result (e.g. the calculation did not use the required analysis accuracy)
	yellow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

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

8 Export to different calculation modules

8.1 Export from PROBAD modeler

As soon as the system is completely defined, the model data can be exported to the corresponding PROBAD modules for calculation.
To do this, the model is first completely marked and then exported:



Input data for several sets of rules can be generated in parallel.
Different components can be assigned to different sets of rules.
In the respective calculation modules, the complete models are automatically divided into subsystems suitable for the calculation (e.g. cylinder with nozzle, head with nozzle or saddle verification).

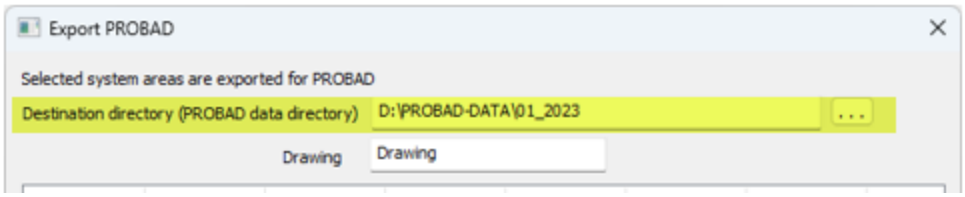
The §D model is taken over as an overview.

The required PROBAD calculation modules are started automatically (new GUI and Classic modules).

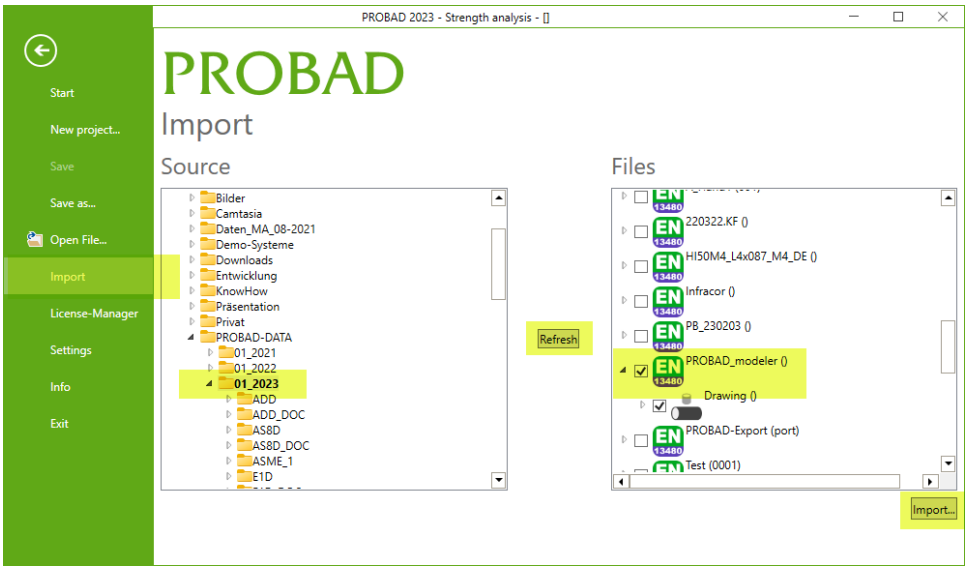
8.2 Import to calculation modules

8.2.1 New GUI

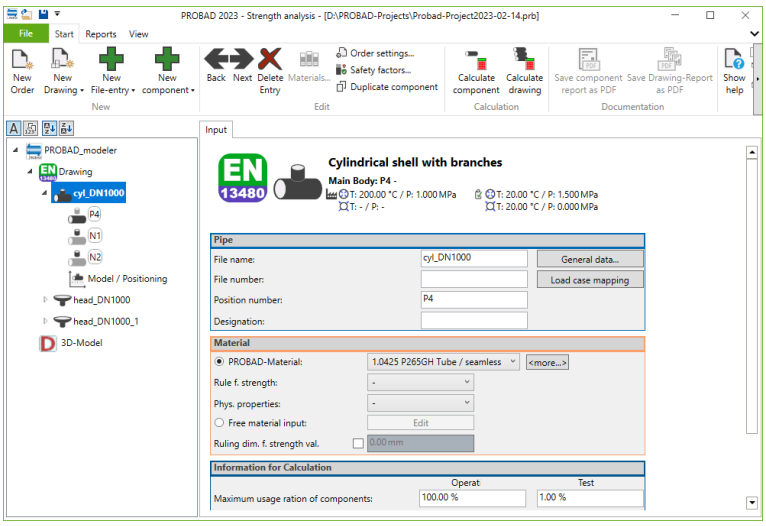
Use File-Import command and select PROBAD data directory, that was defined during export:



PROBAD GUI is started.
Use File-Import and select file.



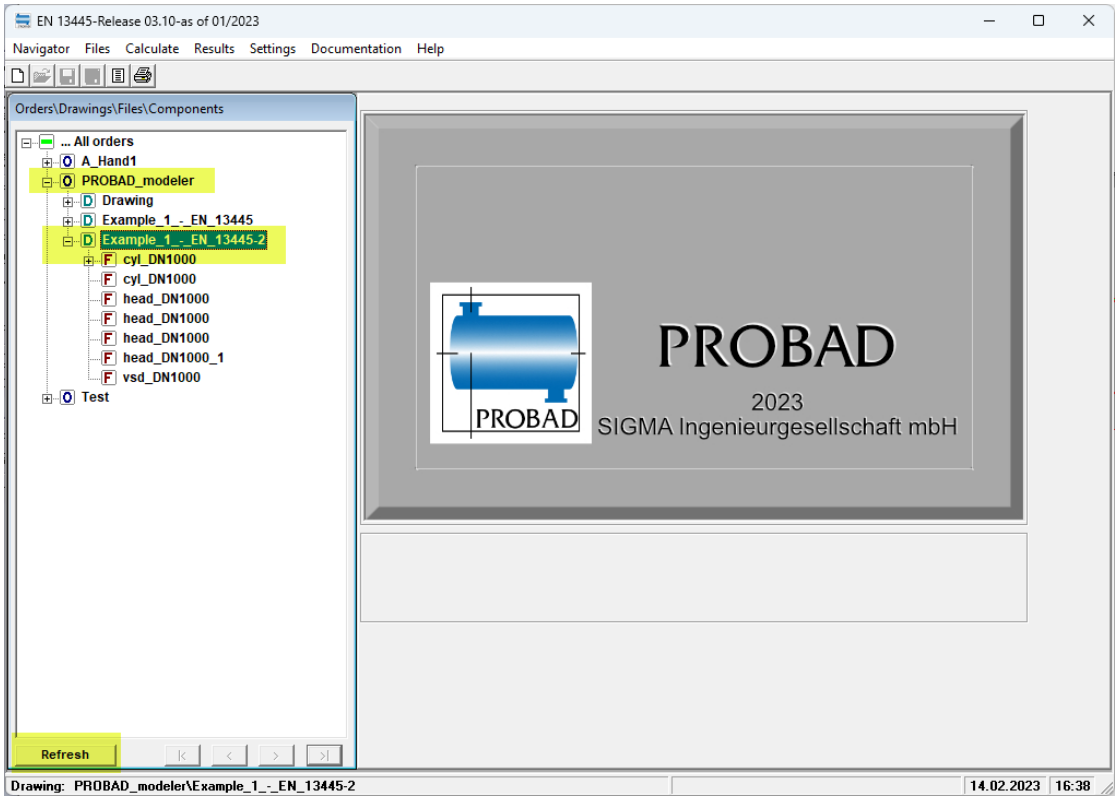
Click Import to open:



Now data is available for further action.

8.2.2 Classic modules

The PROBAD classic module is started and the new project can be found in the list of orders:

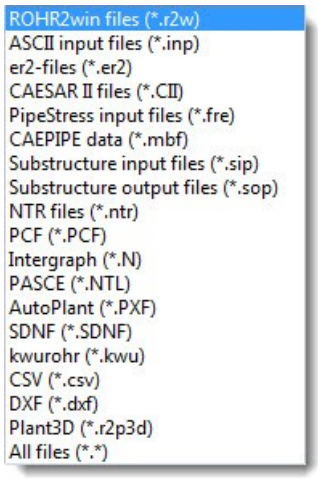


9 Checking CAD/CAE import with ROHR2 Interfaces

The PROBAD Modeler offers a wide range 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 interface 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.