ROHR2

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

ROHR2 33.0
January 2020
New Features and Improvements

The program system ROHR2 with its additional modules is checked and modified continuously within the scope of the maintenance agreement.
The program release ROHR2 33.0 replaces the ROHR2 32.1.
This document shows the improvements and enhancements of the program release ROHR2 33.0:
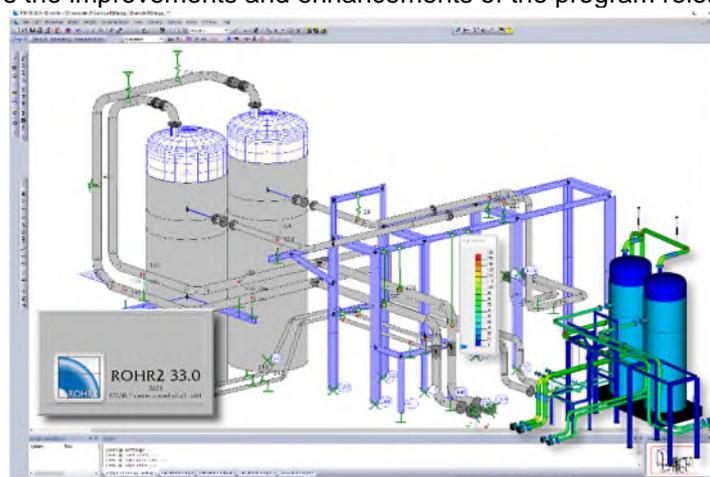
Overview

- ROHR2 now has a **presentation mode** (OpenGL display).
  This mode is used for alternative presentation of the model and the results.
- The **graphical user interface was revised**. Dialogues have been unified and similar functions in the toolbars have been combined in a meaningful way.
- With the help of a new **list function**, inputs and results can now be centrally listed, documented and, where appropriate, adapted.
- Additional **post-processing tasks** make it possible to automate documentation and checking of the results.
- The sets of **rules and databases** contained in ROHR2 have been updated and extended.

Contakt:
SIGMA Ingenieurgesellschaft mbH
Bertha-von-Suttner-Allee 19
D-59423 Unna
Germany
Tel   +49 2303 33233-0
Email   info@rohr2.de

www.rohr2.de

World wide Sales partners

www.rohr2.com

ROHR2 33.0, Modifications and improvements, detailed

Updated Stress Codes

Titel	Inhalt, Stichwort und Quelle	Stand
EN 13480 [1])	DIN EN 13480-3 Metallic industrial piping - Part 3: Design and calculation; German Version EN 13480-3:2012	2017
EN 13941	Design and installation of preinsulated bonded pipe systems for district heating	2019
ASME B31.1	Power Piping ASME Code for Pressure Piping, The American Society of Mechanical Engineers, New York	2018
ASME B31.3	Chemical Plant and Petroleum Refinery Piping ASME Code for Pressure Piping The American Society of Mechanical Engineers, New York	2018
ASME B31.4	Liquid Transportation Systems Piping ASME Code for Pressure Piping The American Society of Mechanical Engineers, New York	2019
ASME B31.8	Gas Transmission and Distribution Piping Systems ASME Code for Pressure Piping The American Society of Mechanical Engineers, New York	2018
ISO 14692	Stress analysis for GRP pipes acc. to DIN EN ISO 14692-3	2017
ASME CL1 [2])	NB-3600 "PIPING DESIGN" in ASME-BOILER AND PRESSURE VESSEL CODE SECTION III SUBSECTION NB CLASS 1	2019
ASME CL2 [2])	NC-3600 "PIPING DESIGN" in ASME-BOILER AND PRESSURE VESSEL CODE SECTION III SUBSECTION NC CLASS 2	2019
ASME CL3 [2])	NC-3600 "PIPING DESIGN" in ASME-BOILER AND PRESSURE VESSEL CODE SECTION III SUBSECTION ND CLASS 3	2019

ROHR2fesu

EN 13445-3	Stress Analysis of FESU structures acc. to EN 13445-3 Appendix C	2018
ASME VIII Div 2	Stress Analysis of FESU structures acc. to ASME VIII Div 2 Part 5 (Elastic Stress Analysis Method)	2019

General

- **License file (*.lic):** This file is now created in the directory %APPDATA%/ROHR2 to enable the license activation of ROHR2 even without administrator rights.

Presentation mode

- **Presentation mode (OpenGL) added**
  This mode is used for alternative presentation of the model and results.
  In presentation mode, views can be created as usual, these can be used e. g. for reports.
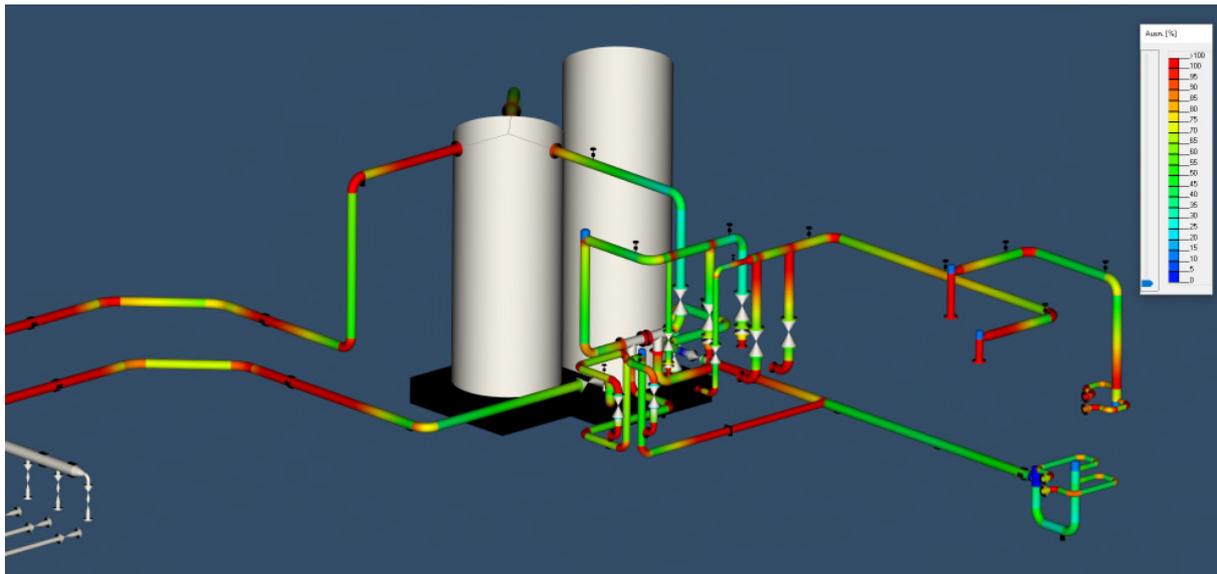
List features

- **List features:** The list features have been completely revised and integrated into a dialog.
  The requested data can be selected via toolbar and tabs.

The data can be sorted in ascending or descending order (clicking on the header again changes the sort order).

- When **listing parts** the **ASME Schedule** is now also displayed for ASME parts, this can be changed for multiple entries in only one step.
- When **listing texts**, the **color** is also displayed and can be changed using the right mouse button.
- **Selected entries can be edited directly in the list function** now**.**

- **Inner supports/ hangers** are now integrated into dialog for standard supports/ hangers:
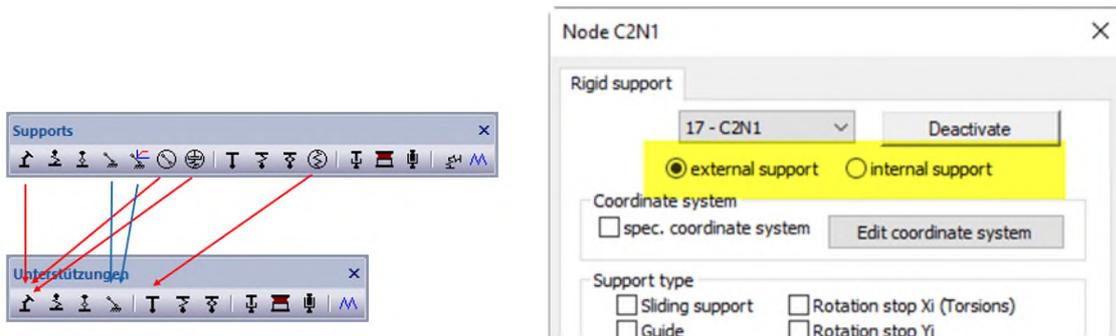
All **coupling options** (Hinge, internal spring, frictional joint) are now combined and all properties can be specified via options:
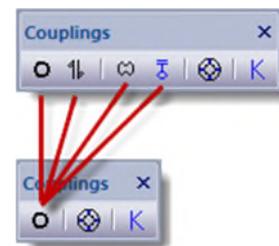
- The support type "**Angulating support with characteristic curve**" is now integrated into the default angulating support, i.e. there is no longer a separate support type.

GUI / Handling

- A **background color** can be selected.
- The optional display of a **north arrow** has been added (see *Project settings-Graphic*).

- A new function for **connecting two sections** (with gap) has been added (Edit-Connect segments). This allows e.g. gaps to be closed easily after importing data via an interface.

- The **tooltip control** for the display of properties in the model has been completely revised. The data to be displayed can now be freely selected.
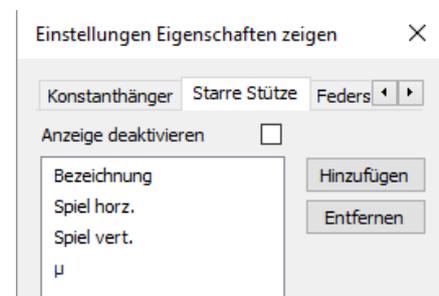
- Texts: Texts with macros can be stored as patterns.
- A zoom function for additional texts, node names, section labels, symbols for the screen view has been added.
  The objects to be considered can be selected:
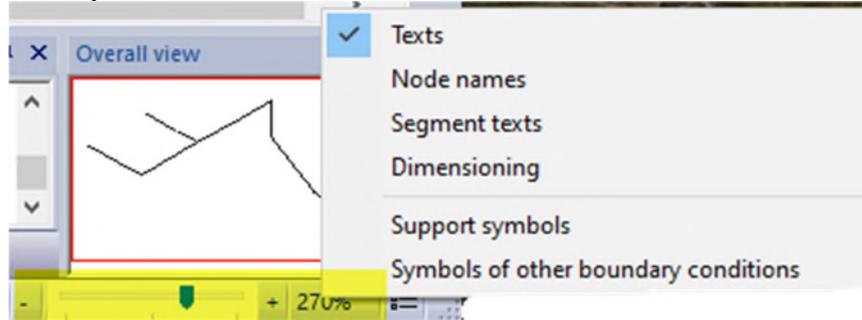
- The selection of the **reference node** for a **starting point** of a new line can now be done with the mouse and not only via the node name.
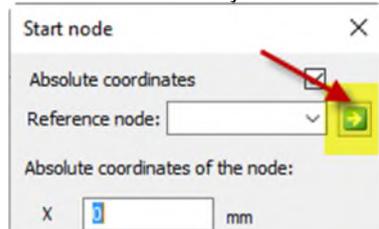
- **Insert boundary conditions:** If system parts are selected and a node in the selected area is clicked, then optionally several boundary conditions can be inserted in the selected area at the specified distance (beyond the boundaries of sections).
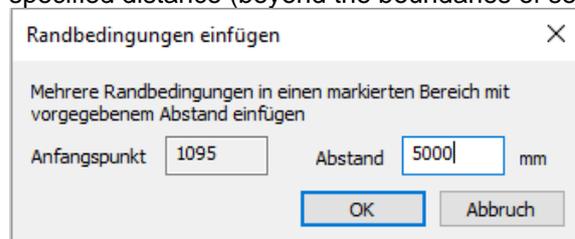
- **Delete boundary conditions**: If system parts are selected, then optionally all boundary conditions in the selected area of the selected type(s) can be deleted automatically.
- **Dialog box Tasks:**
  An additional level (load category) has been added to the load case dialogue.
  You can change the order of the tasks (within a category) using Drag&Drop.

- **Supports can be deactivated depending on the load case**, e.g. to calculate a primary case without supports that have lifting loads during operation.
- Dimensions, Insulation: A selection box has been added for selecting the **material of the sheet metal shell.**

- - For pipes an inner lining can be considered:

The additional linear mass of the pipe is automatically determined, the linear mass of the medium is determined with the correspondingly reduced inner diameter.

- The assignment of the tolerance standard to component standard has been revised.

- The type of **Plastic jacket pipe (**KMR) is now determined by the pipe dimensions, the properties of PE and PUR are also specified here.

- The **Rotate** function has been extended. Optionally, a copy of the data can be created at the original position, this operation can also be performed automatically multiple times to create multiple copies at different positions.

Rotation angle	90	deg		
Create a copy of the selected data at the origin position				☑
Execute function	3	times	OK	Cancel
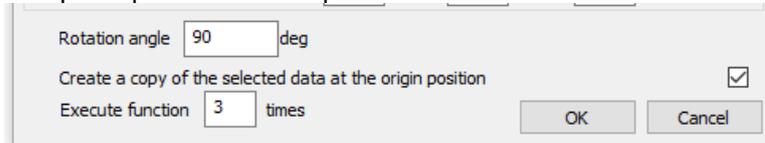
- The function and dialog box **Circle** has been revised:
  o Definition of the level analogous to the Mirror function
  o Optional arcs with the radius of the ring (i.e. 'round' ring)
  o Optionally a gradient can be specified -> 'Spiral

- **Dimension selection:** There is now an additional entry in the toolbar to select a section whose dimensions are to be set as the current dimensions.

DN200.0
<-select segment->
DN200.0
DN150.0
RIGID.0
DN250.0/DN200.1.0
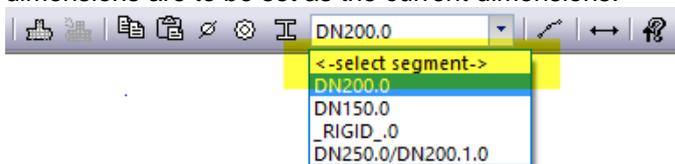
- **Insert multiple** is now available for Blind flanges and Heads
- **Insert multiple** is now available for all texts
- **Segment data** (loads, line names, ...) can alternatively be **assigned to previously marked areas** by copying them from a clicked segment.
- **Renumber nodes**
  Added an option to include only nodes whose names are currently shown in the graph. This allows you to use the filter function of the display to renumber only selected nodes.
- **Structural sections**: The relevant thickness for determining the allowable stresses is now calculated based on the profile dimensions (max(Ts, Tg) or can be specified by the user.

Underground piping

- **Expansion cushions:** Stiffnesses are now also determined alternatively according to EN 13941:2019
- **Soil restraint**: the "vertical stability" according to EN 13941 as well as the max. permissible buckling length according to Euler for unburied areas is checked and documented.
- **Dialog Soil data:** In addition to the name, other significant parameters are now displayed in the list.

Bettungsdaten

Name	Norm	Überdeck...	Dehnpols...	
D1	EN 13941	1000 mm	40.00 mm	Hinzufügen
				Bearbeiten
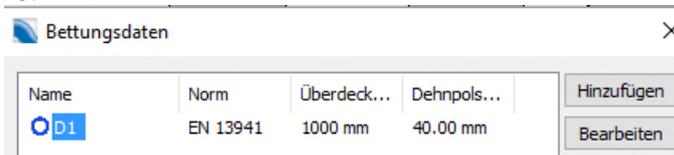
Loads

- **Loads** can now also be **edited directly in the node dialog**. A double-click on the list entry opens the corresponding dialog box for editing the data.
- **Line load Heating up** (ADT, ATX, ATY, ATZ): As an alternative to the direct specification of the expansion, the delta T to the operating temperature of the load case can now also be specified. The differential expansion is then determined based on the material properties.
- The input data for determining the individual loads for **static fluid hammer loads (**Joukowsky) are now managed like line loads. This allows the fluid hammer loads to be updated automatically after the model has been changed (cross-sections, lengths).
- **Wind/snow load**: Extension by further wind/snow zones of other countries
- **Wind, earthquake, modal analysis (earthquake dynamic):** For each load direction, additional loads can be optionally specified (e.g. building displacements). For each load direction, an additional load case is generated and the loads are assigned to these load cases.
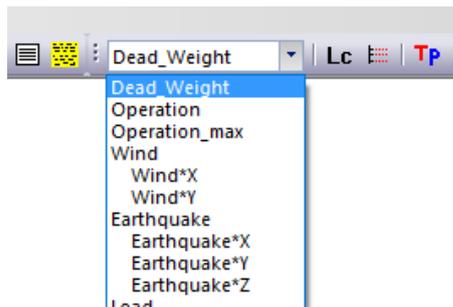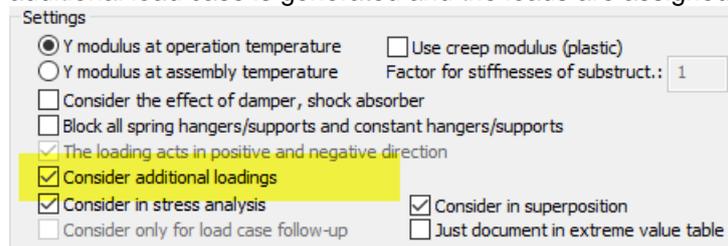
The load case superposition combines the wind/earthquake loads with the additional loads automatically.
- Wind loads DIN 1055, EN 1991, **gust effect** :
The length / width used to determine the gust effect is now always the maximum expansion of the entire construction in one spatial direction.

Spring- / Constant hanger design

- The restrictions regarding load cases with blocked springs during extreme value calculation are no longer applicable

Calculation

- **Changed tasks** are **automatically selected** for calculation in the dialog box.

- **Line topology**: The topology can optionally be completely regenerated automatically before calculation to avoid problems with 'fragmented' topology.

Postprocessing

- post-**processing tasks** can now be defined. The following are currently possible
  - Report generation
  - Conversion of output files into RTF or PDF format based on templates
  - Export support data LICAD/FLEXPERTE/PDMS or *.sup
  - Export of results in *.csv
  - Check the results, e.g. for lifting bearings, permissible loads etc.
- The post-processing tasks are selected in the calculation dialog, so they are automatically executed after the calculation.
- Additionally, there is an automatically generated post-processing task "Convert output files to RTF". This converts the output files of all previously calculated tasks into RTF format using the template for general output.

Represent and check results

- **Display Result:** The scaling factor for the deformations is now displayed in the toolbar and can be changed directly:

- **Results check**
  There is a post-processing task for checking the load case results, optionally also the results of load case combinations / extreme values.
  - The scope of the check can be configured.
  - These settings can also be defined as default for new models.
- **Allowable loads** can be specified for supports and additional results.
- A **warning** is issued **if** allowable loads are **exceeded**.

Documentation

- **User-defined expansion joints and flanges**: The manufacturer can now be specified manually (for documentation).
- Added function **Print regions** analogue to Print lines
- When creating new models or opening a model, **standard report templates** are now automatically copied (if they do not yet exist for the model). This means that a report or other documentation can be generated immediately.
  - The standard templates can be selected for new models via Options-*Presettings*.
- An additional **RTF template for general output** is used for the documentation of ROHR2flange, ROHR2nozzle, expansion joint analysis and spring design to adjust the format of all outputs.
- Report: Added **additional table** with **maximum utilizations of substructures** existing in the overall system.
  This table is also available for the list function in the Results mode.
- **Additional tables** are provided for the **report**:
  - description of stress equations calculated during stress analysis.
    (This table is also available for the list function in the results mode).)
  - Description of the stress equations calculated in the stress analysis.
- **Max. utilizations from R2Flange** as well as the **calculated tightening torques** can be listed and requested as macro in the texts.

ROHR2fesu

- For new substructures, the **intersections can optionally be generated automatically.**
- Full **jacket pipe couplings** at the end of the jacket area (transition to the normal pipe) are now automatically translated into FE models. The following situations are recognized:
  - Simple coupling at the end node of the jacket area -> Coupling via plane plate in the FE model
  - Coupling and component "Head" at the node -> coupling as intersection of the inner pipe with the head (with straight projection).
- Any **sectional planes** can be defined. The function Element properties can be used to display these sectional planes (elements on the positive or negative side of the plane)
- Pl results are now always displayed as extreme values, so that the maximum results are visible at all disturbance points and it can be decided **at a glance** which area is to be considered local. Internal pressure check
- An **internal pressure check** is now also possible at branch nodes (node dialog). It's not required any longer to define a specific component at this node.

Interfaces

- The **PCF export** is now available also for partial areas (e.g. marked area, only displayed area), optionally a separate PCF file can be created for each line.
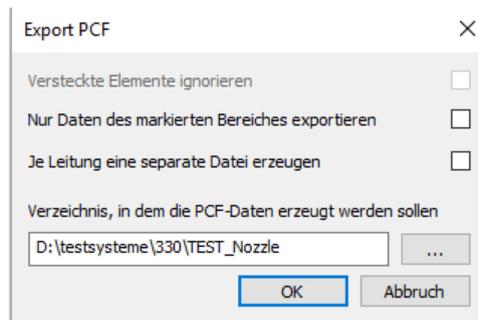
- An existing **model can be extended with new CAD data** (PCF, NTR, ...)

Databases

- Added **database import in CSV format**

Software Development, Sales and Support

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

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