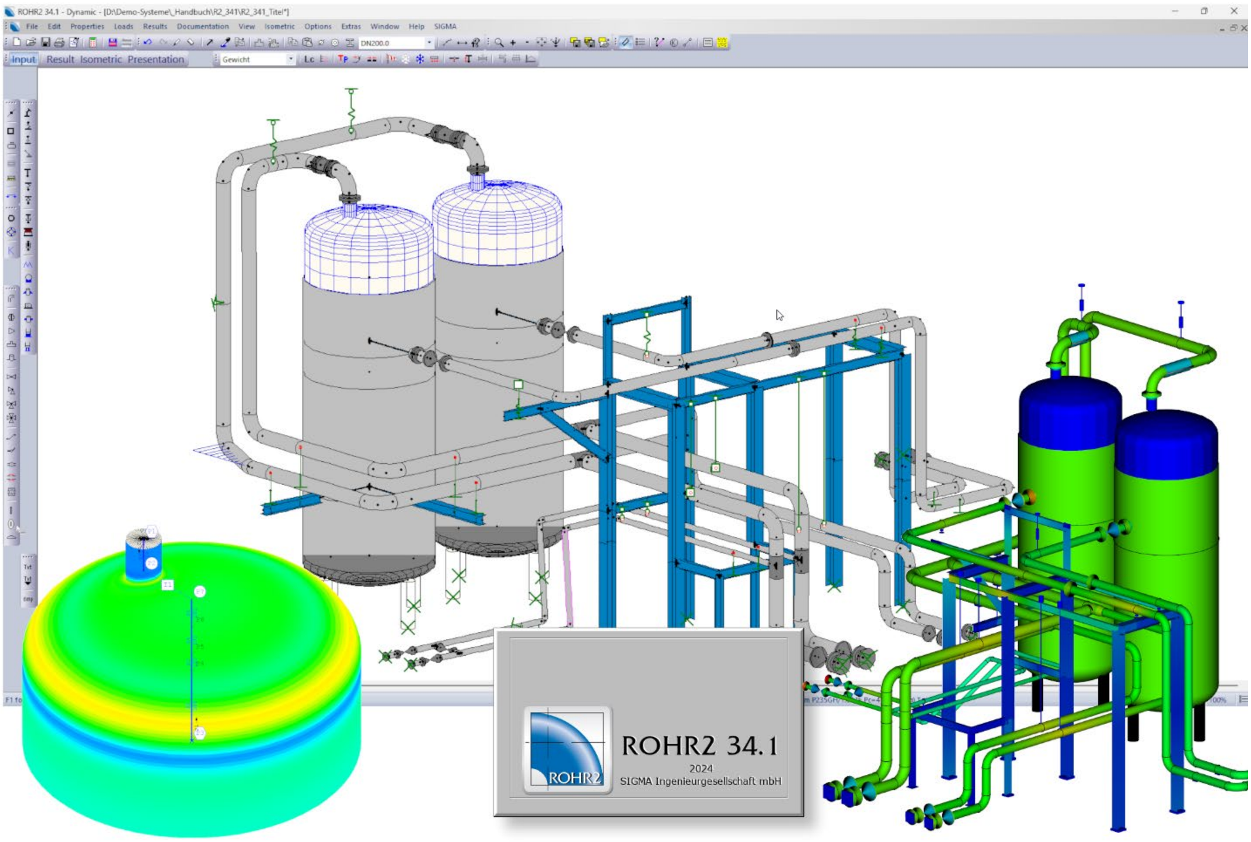


FE-Analysis by the additional module

ROHR2fesu

Program System ROHR2

SIGMA Ingenieurgesellschaft mbH

Content

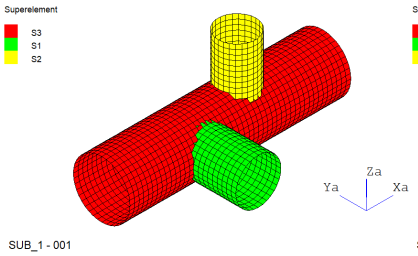
Release 25.06

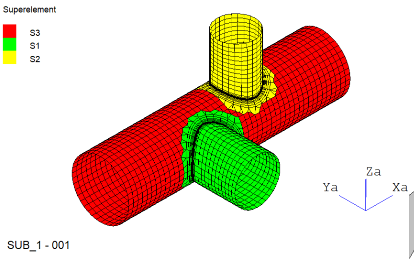
1 ROHR2fesu Overview

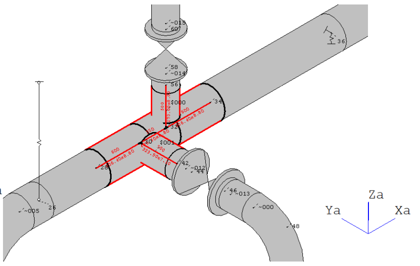
ROHR2fesu is an additional module in the program system ROHR2 for detailed analysis of local segments in pipes and vessels.
ROHR2fesu offers the easy-to-use modeling of sub-structures by nearly any circle and elliptic geometry using shell elements , fully integrated in the ROHR2 framework. This enables to carry out detailed analysis of critical segments inside the framework of the entire model. The shell element sub-structure analysis is carried out using FE-method.
The mesh generator of ROHR2fesu automatically integrates intersections of branches, trunions, and nozzles with and without reinforcement. ROHR2fesu allows controlling the mesh resolution in a simple way.
ROHR2fesu has been verified extensively by comparison against reference solutions of standard problems.

ROHR2fesu offers:

- Complete integration of the FE structure(s) into the connecting frame work
- easy-to-use parameter controlled model generation and meshing
- short calculation time
- automatic stress analysis and documentation
- stress analyses following EN 13445, Appendix C, ASME Section VIII, Div. 2, Part 5 and AD S4







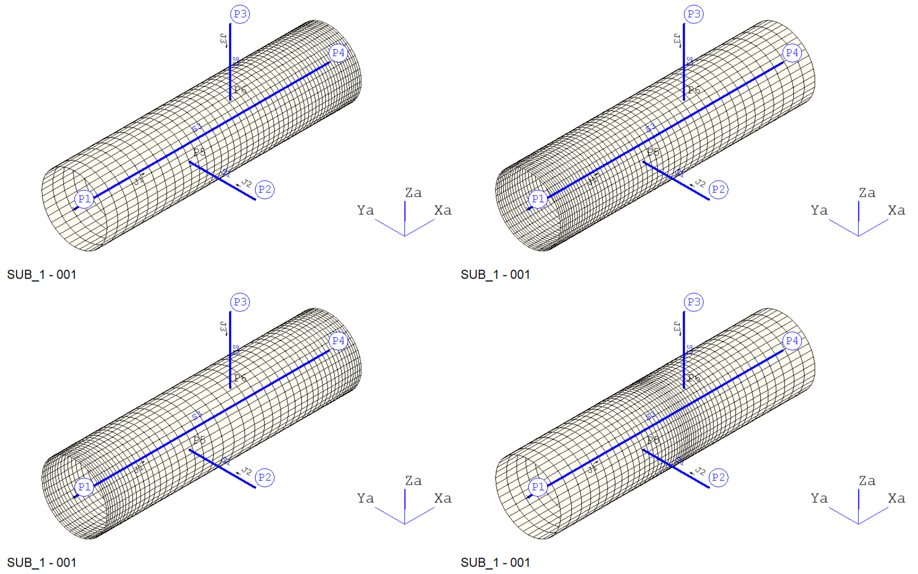
2 Task

The task of ROHR2fesu is the modeling and calculation of shell-sub-structures for the detailed analysis of stiffness and stresses in concentrated segments of the piping structure.
The determination of the substructure in the framework is made on the basis of a stiffness matrix.
The results of a ROHR2fesu calculation are a stiffness matrix and loads at nodes from element loads and extra loads (e.g. loads concentrated on the sub structure) at connection points.
The load cases and load case combinations of the ROHR2 framework model used to determine the element stresses within the shell structure.

The main focus of the application of ROHR2fesu is problems in piping and vessel models where stiffness and/or stress analysis needs to be performed like for:

- Intersections of piping elements generally
- Special components for which internal pressure design stress codes are available but no calculation norms for external forces and moments
- Components where the standards of k- and i-factors are unsatisfactory
- Special components without k- and i-factors defined in the stress codes
- Analysis of local stress utilization
- Optimization of details in pipe and vessel constructions
- Nozzle and cams at vessel with or without reinforcement
- Big diameter tubes with miter bends
- Calculations with to measured pipe segments (as-built), e.g. bends
- Damage analysis at corroded or eroded pipes
- Detail analysis of bend deformation due to internal pressure or bending in oval bends

Of course the standard pipe components (e.g. acc. to EN 13480) can be modeled and calculated in ROHR2fesu.
Stress analyses following EN 13445, Appendix C, ASME Section VIII, Div. 2, Part 5 and AD S4 can be carried out.



3 Features

Integration into ROHR2

- Full integration of the shell model as substructure into the framework of the entire piping system
- The stiffness of the shell model is considered in the framework by its stiffness matrix
- Consideration of the loads at the connection to the framework, which result from the substructure
- Easy-to-use modeling of the shell model by combinations of simple basic elements (superelements)
- The coarse model is generated automatically based on the framework
- Automatic generation of connecting elements (interface elements) at the coupling nodes between pipe element based framework and shell element based sub-structures

Superelements/Components

The structure is made of superelements.
The super-elements (SE) are combined in groups in ROHR2fesu:

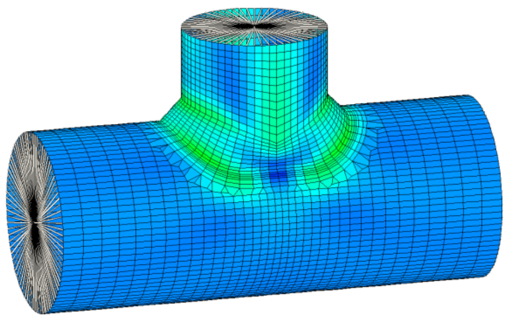
• SE-components pipe	Cone: centric, eccentric, elliptical Bend, circular bend, miter bend, measured bend, creased bend
• SE-component	rectangular duct, saddle
• SE- components head	Spherical slice toroidal slice - concave, convex elliptic slice
• SE- components point	Transition element

Superelements/Jackets

- The superelements are defined by a system center line using the start and end node and dimensions at the end node or by cross section areas inside the superelement
- Easy-to use extension and modification of the model by inserting conical or elliptical transitions
- Variable wall thickness across the circumference and elliptical cross sections can be modeled
- Superelements in sequential order are combined in a “jacket”

Intersections

- Intersections can be placed at all superelements except at the connection to the framework
- Fillets at transitions are considered
- Intersections for the modeling of nozzles and trunions with or without reinforcing and internal projection, gusset, rectangular lugs
- Modeling of inserted (block flange) and welded reinforcing pads

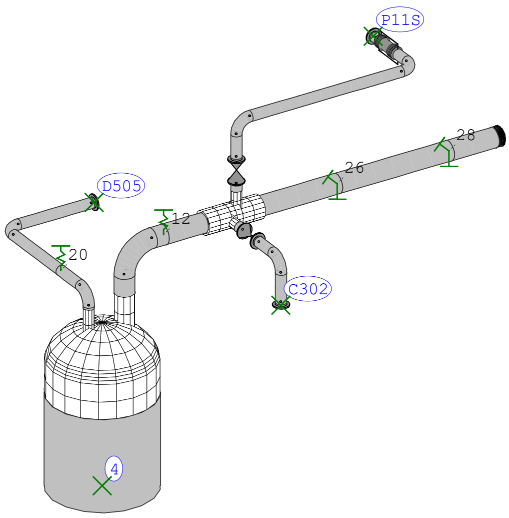


Meshing

- Simple meshing of the shell model by defining the axial and circumferential subdivision
- Optimization of model size and calculation time by progressive or regressive division in axial direction
- Individual pre-definition of the mesh of system parts in the intersection segments
- The mesh division can be modified easily

Loads

- Automatic conversion of loads of load cases and load case combination from the framework at the connections between shell model and framework model
- Assignment of operation data (pressure, temperature, medium density) at each load case in the framework analogue to the ROHR2 framework
- Consideration of wind, snow- and ice loads of each load case in the framework
- Consideration of various additional masses, forces and area loads on the sub-structure of each load case in the ROHR2 framework
- Consideration of the axial force from internal pressure at the transition nodes
- ROHR2fesu offers an easy-to-use way to check allowable stresses, e.g. at nozzles



Stress codes

- Stress analysis acc. to EN 13445-3, appendix C due to the method of stress categories
- Stress analysis acc. to ASME Section VIII, Div. 2, Part 5
- Stress analysis acc. to AD S4
- A local area acc. to EN 13445 can be defined easily and the membrane stresses can be regarded as locally which results in a definition as Pl

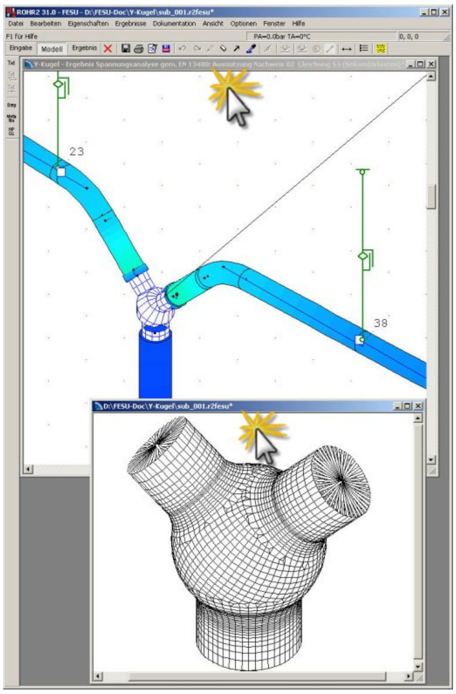
Optimization

- Simply modification of the system in pipe segments, e.g. fillets, transitions
- Substructures can be optimized independent from each other
- Modifications at a sub-structure does not require the calculation of other sub-structures

Documentation

- Colored representation of element parameters like wall thickness or material
- Graphic representation of utilization and calculated stresses
- Graphic representation of deformations
- Automatic report generation, to be modified by the user

3.1 Integration into ROHR2



ROHR2fesu is fully integrated into the ROHR2 user interface. Select the segment, to be analyzed detailed, in the framework. From framework details (geometry, dimensions, component type like bend or head) a coarse model of the sub-structure is generated from the selected segment. This model can be completed in ROHR2fesu by details which are missing in the framework. The sub-structure is shown in the containing piping framework using the system lines or as simple shell model. By a double-click on the sub-structure in the framework, ROHR2fesu is launched to change the model or for analysis or documentation of the sub-structure.

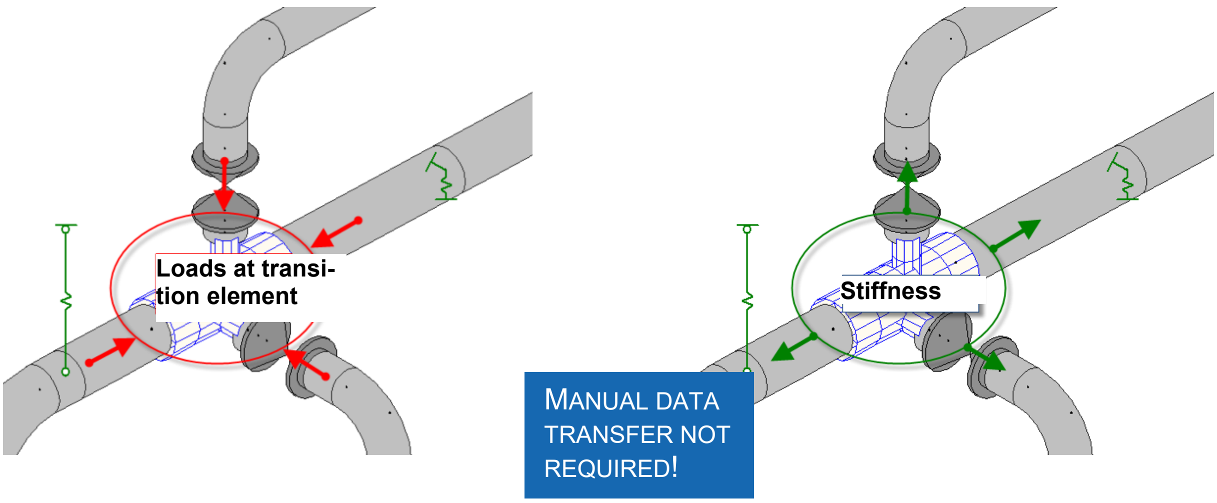
The assignment of operation data (pressure, temperature, medium density) at each load case in the framework is done analogue to the ROHR2win framework

ROHR2 calculation of the connecting framework

The stiffness of sub-structure is considered in the framework by a stiffness matrix resulting from the FE-analysis.

ROHR2fesu calculation of the substructure

Loads at the connection points in the framework are automatically taken over into the calculation of the sub-structure.
Loads from the sub-structure are taken over in the framework analysis.

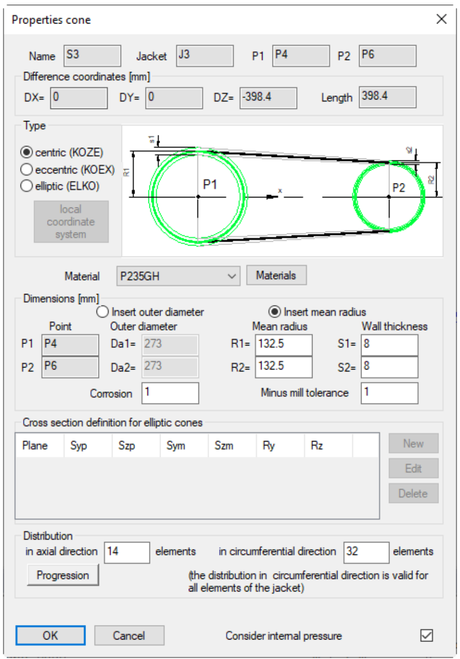


4 Available library of Superelements

4.1 Overview Superelements

The FE structures can be created by nearly any combination and intersection of basic elements (superelements). These superelements are available:

Cone, centric, eccentric, or elliptical / Cylinder (cone special shape)



The definition of the cone can include the parameters:

- Type
- Material
- Outer diameter / radius at start/ end node
- Wall thickness
- Minus-mill tolerance
- Element division in axial and circumferential direction

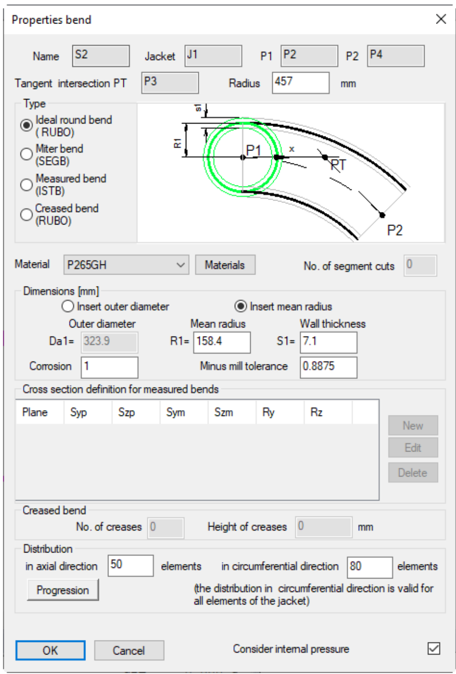
The superelement cone also can be used for the modeling of a nearly flat plate.

Pipe bend as Ideal sound bend, miter bend or measured bend

The definition of the cone can include the parameters:

- Type
- Material
- Outer diameter / radius
- Wall thickness
- Minus-mill tolerance
- Element division in axial and circumferential direction

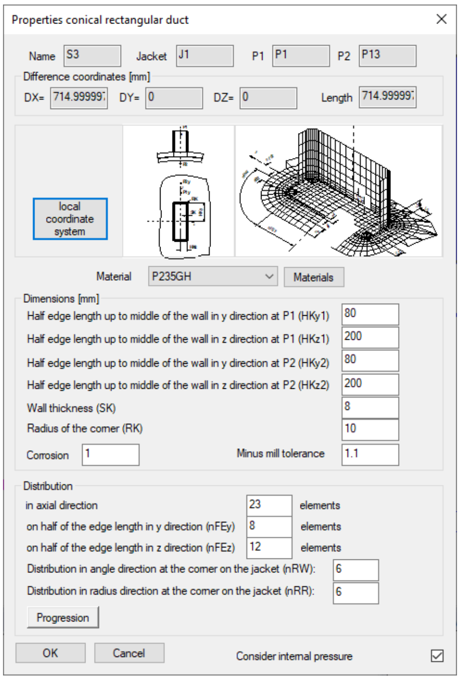
At measured bends additionally the radii and wall thickness´ can be defined for each cross section around the circumference.



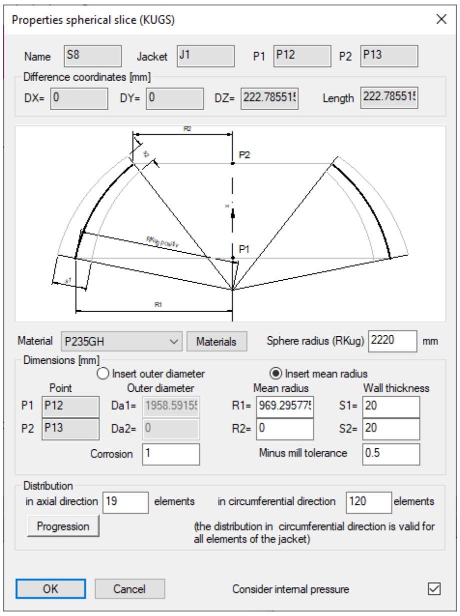
Rectangular duct

The definition of the rectangular duct can include the parameters:

- type
- material
- length/width, radius of the corner
- wall thickness
- minus-mill tolerance
- element division in axial direction and length/width



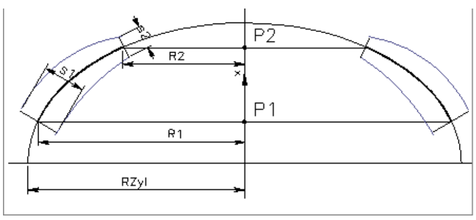
Spherical slice or ellipsoid slice



The definition of the spherical slice can include the parameters:

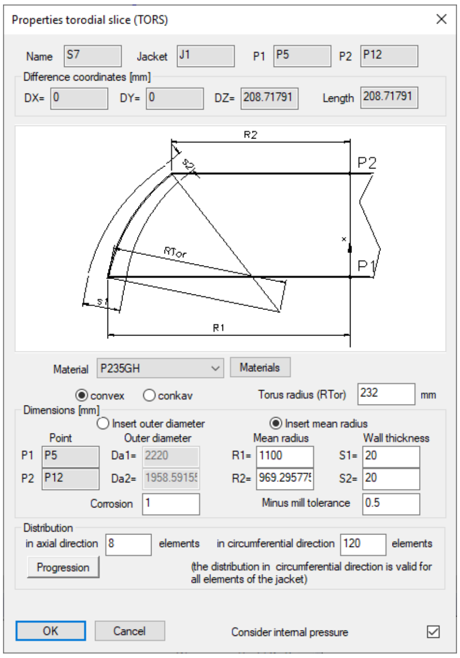
- Material
- Outer diameter / radius at start/ end node
- Wall thickness
- Corrosion allowance
- Minus-mill tolerance
- Element division in axial and circumferential direction

Alternatively an ellipsoidal slice can be defined in ROHR2fesu.



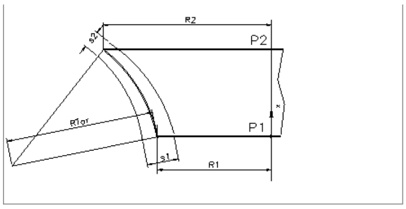
Using these superelements nearly all components used in practice can be modeled.

Torus, convex or concave

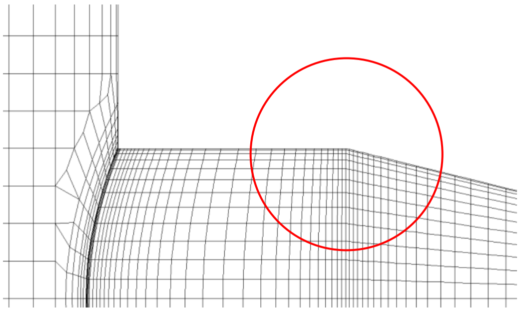


The definition of the torus can include the parameters:

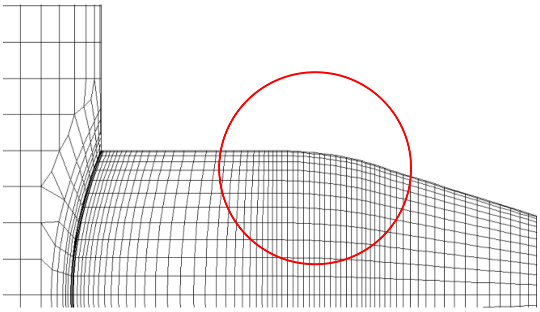
- Material
- Type convex / concave
- Outer diameter / radius at start/ end node
- Wall thickness
- Corrosion allowance
- Minus-mill tolerance
- Element division in axial and circumferential direction



Torus concave as shown in the dialog window



Without fillet

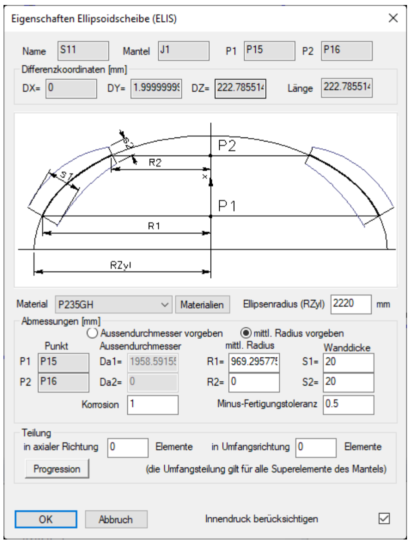


with fillet

Elliptic slice

The definition of the elliptic slice can include the parameters:

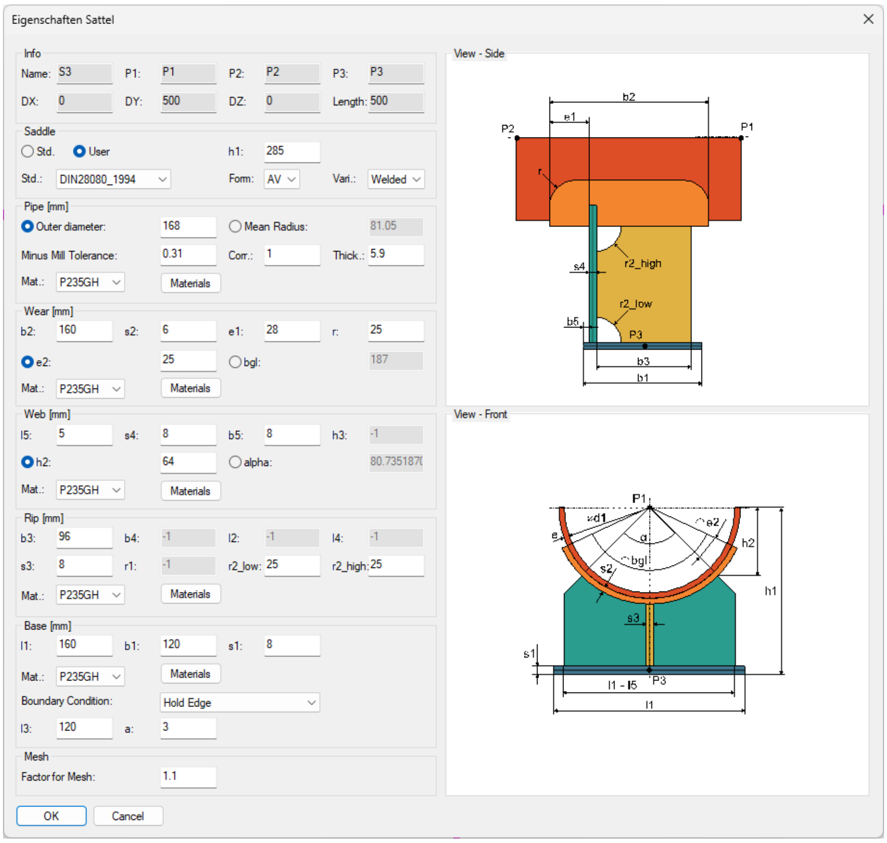
- Material
- Type convex / concave
- Outer diameter / radius at start/ end node
- Wall thickness
- Corrosion allowance
- Minus-mill tolerance
- Element division in axial and circumferential direction



Saddle

Saddle forms based on DIN EN 28080:1994-07 and DIN EN 28080:2015-06 are implemented.

This includes form A, AV, B, BV, C, D, E and F for the revision 1994 in the variants welded and bended and for the revision 2015 the forms AV, BV, CV and D. The values of input parameters are taken from the standard if possible or are individually chosen by the user.

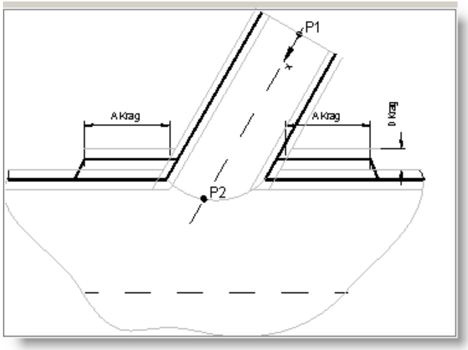


4.2 Connections to the framework

Transition elements are inserted the connection to the framework (transition elements). A transition element is the connection between a shell and a beam. The transition element consists of a short hub (centered pipe stump) with rigid spokes, leading to the shell points at the circumference. According to the Bernoulli Hypothesis of linear statics it guarantees the flatness of the cross section at the limit of the shell model.
In special cases the degrees of freedom of the couplings at the shell can be modified.

4.3 Intersections

The program allows inserting intersections for the modeling of nozzles and trunions with or without reinforcement and internal projection. The model of the reinforcing pad may be entered as single shell or double shell with material selectable for the reinforcing pad.

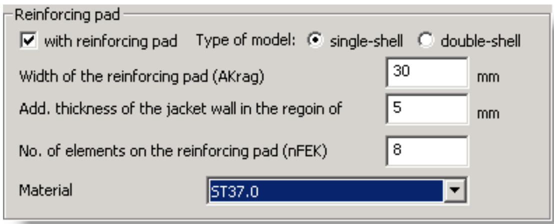


The meshing is done for partial regions inside the intersection area.

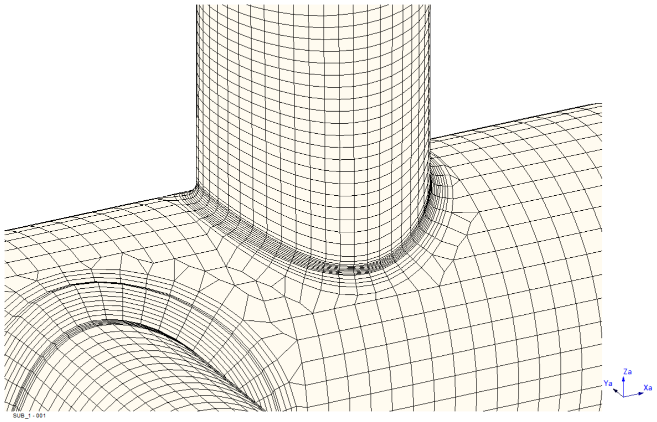
Inserted values are explained by expandable detail drawings.

The intersection includes the parameters

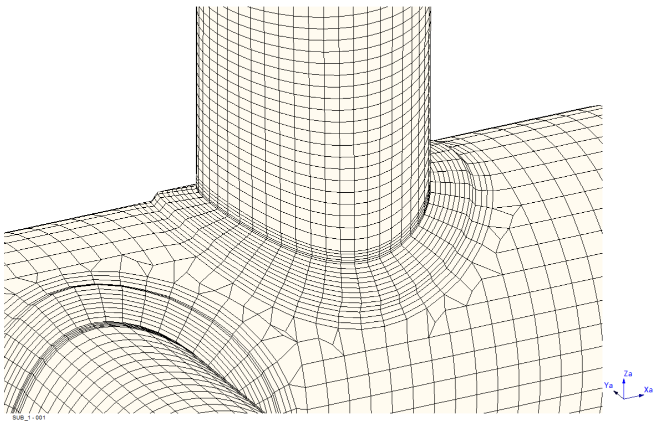
- Type: Nozzle or trunion
- Fillet radius on the surface
- Element distribution
- Internal projection
- Reinforcing pad with details:
  - single shell/double shell
  - width
  - thickness
  - material



Examples: fillet or reinforcing slice



Intersection with fillet



Intersection with reinforcing slice

4.4 Loads

The connecting framework influences the sub-structure by its loads. The loads are transmitted into the sub-structure at transition nodes. This does not require special load inputs in the FE model. Loads from load cases and load case combinations in the framework are taken over into the FE model.

Loads, occurring in the sub-structure are assigned in the framework model, too. Among them:

- Operation data (pressure, temperature, medium density)
- Wind loads (determined automatically or manually entered)
- Snow loads
- Ice loads
- additional masses
- any additional forces
- any loads on areas

The insulation parameters of automatic determined loads form snow, ice and wind are taken from the following segments in the framework. They are considered approximately when calculating the loads in the sub-structure.

Loads at the entire system (framework model) occurring at transition nodes, which are resulting from the sub-structure analysis, are considered in the framework model.

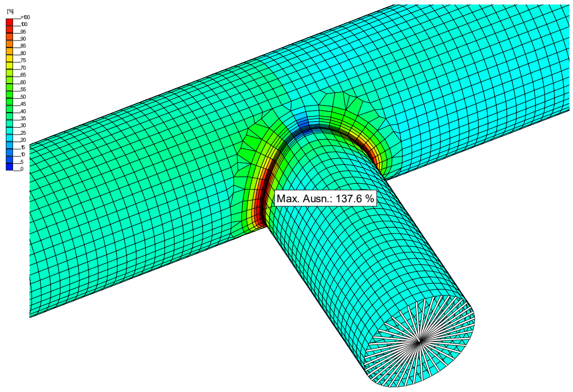
4.5 Model generation restrictions

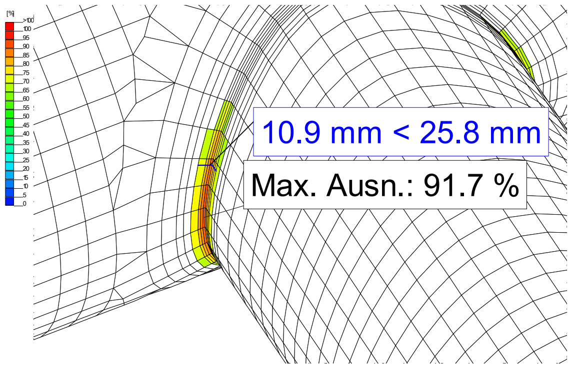
Even though the generation of the shell sub-structure from the piping system is very generic a few limitation apply in order to allow a smooth integrated meshing and calculation: but the intersection of super elements causes a few limitations.

- Assignment of materials can be done only for each superelement
- Opposite nozzles may not though themselves
- Only one superelement can be active in an intersection
- The intersection always needs to go through the entire cross-section of the nozzle
- An intersection at connection points of several jackets (e.g. branching of pipe jackets a skirt support) or transition element are not allowed
- At one end of the superelement an intersection is allowed. At the other end a cross section vertical to the axis is required. If necessary the superelement can be divided by an intermediate node.
- Intersections cannot be intersected again

4.6 Stress analysis

A stress analysis is carried out for the sub-structure following EN 13445, Appendix C, ASME Section VIII, Div. 2, Part 5 and AD S4.
. The generation of load case superpositions and required analyses is done in ROHR2win using the defined load cases or it can be entered manually. Load cases and stress utilization for each analysis at the shell elements are shown in the graphic.



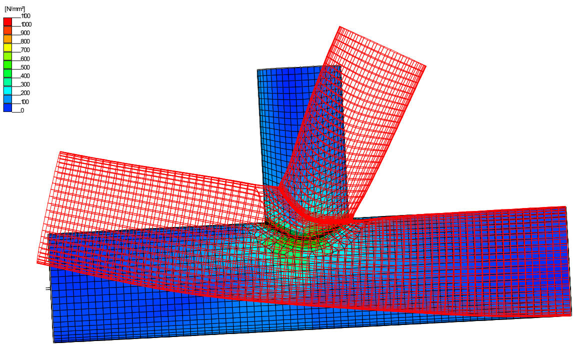


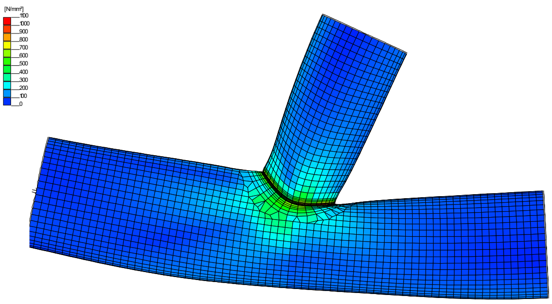
In the analysis of the local membrane stresses (Pl) only the segment is shown, where the calculated stress
is > 1.0 x f which means that they must be characterized as local. An easy-to –handle measuring instrument can be used to verify if the selected part is really a local area according to EN 13445. In the intersection area the measured width in meridional direction is automatically compared to the maximum value ls+ln of EN13445.

Max Ausn. = Max Stress Utilization

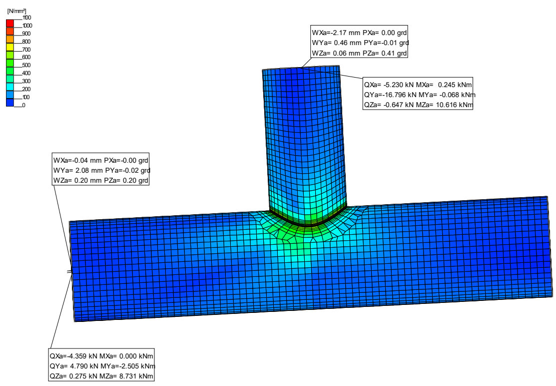
4.7 Results representation

At each analysis the model can be shown as completely deformed shape or the deformations are shown in an additional wire frame model.





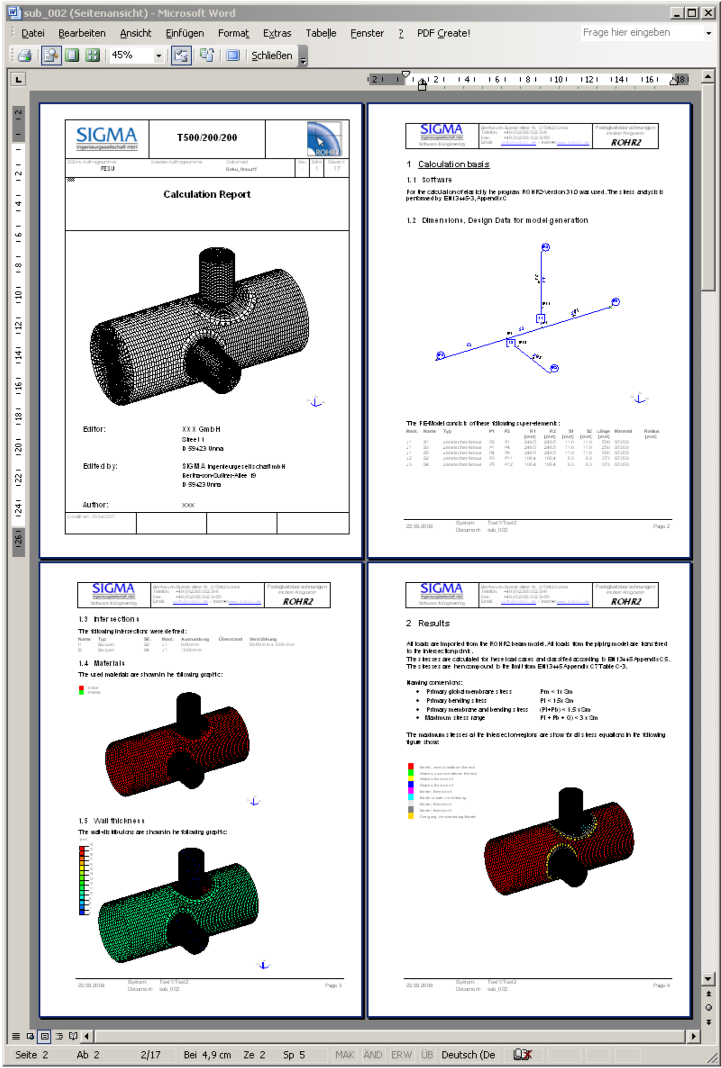
Connection loads and movements at transition elements can be displayed.



4.8 Results documentation

ROHR2fesu automatically generates a report in rtf format after the calculation showing essential results. The report generation is carried out using a report template. Predefined templates are available in German and English. They can be modified by the user.
The reports are updated automatically after re-calculation.

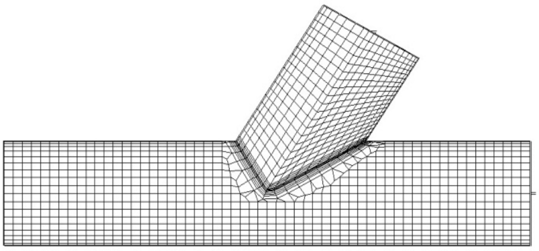
Sample ROHR2fesu - report template

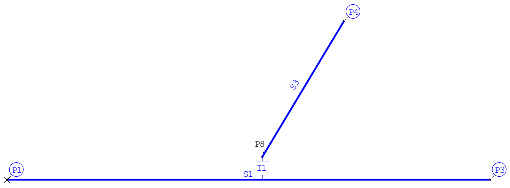


5 Examples

The ROHR2fesu examples listed here are shown in a reduced size.
For bigger drawings and additional comments please go to ROHR2fesu at www.rohr2.com.

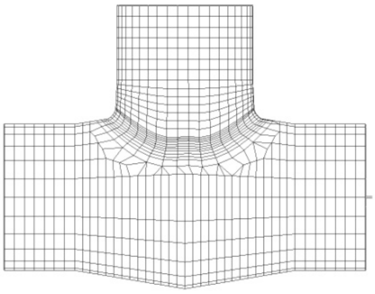
Simple sloping branch

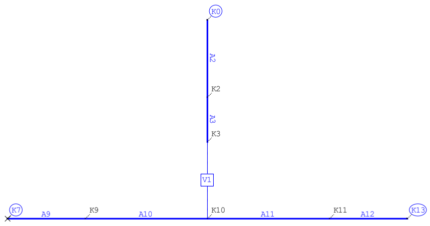




A simple (sloping) branch is created by two superelements of the type *Centric Cone*. For the intersection it must be defined properties like rounding radius, projection or reinforcement.

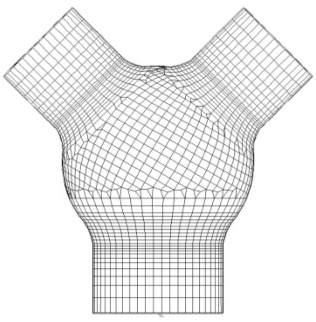
Norm-Tee with conical transitions

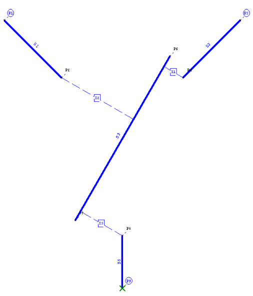




The cylindrical bases of the norm tee are created by superelements type *Cylinder*. The conical transitions in the center of the tee are superelements of the *Centric cone* with different dimensions at the beginning and at the end. The intersection requires the input of a fillet.

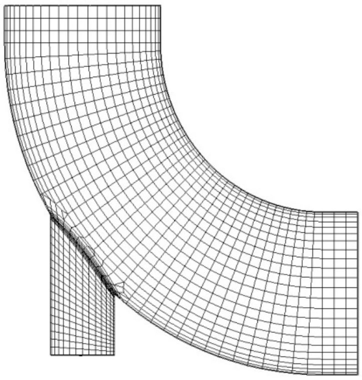
Spherical fitting

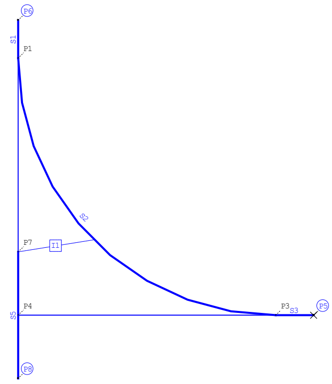




The cylindrical bases as well as the conical transitions are made by superelements of the type *Centric Cone*. The sphere is created by the superelement *Spherical Slice* (in the special form complete sphere).

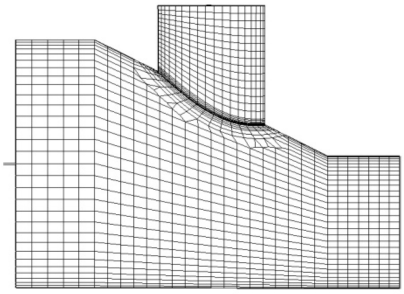
Support at bend

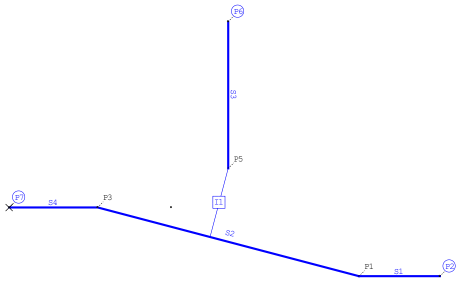




The support and the cylindrical bases are created by the superelement of the type *Centric Cone*.
The bend is created by a superelement of the type *bend* (here round bend).

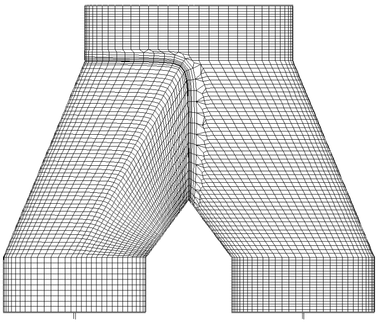
Branch in eccentric reducer

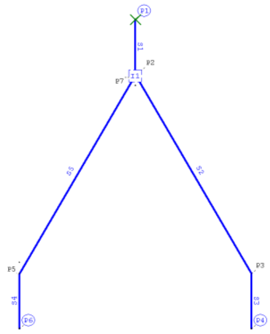




The nozzle and the cylindrical bases are created by a superelement of the type *Centric cone (cylinder)*.
The reducer is created by a superelement of the type *Eccentric Cone*.

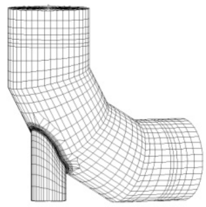
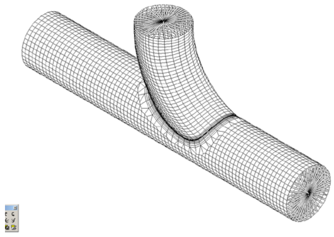
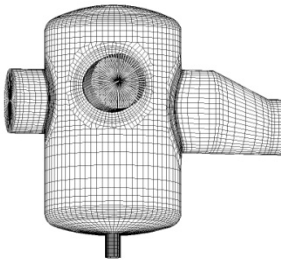
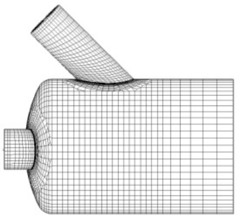
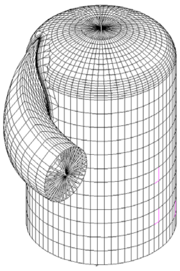
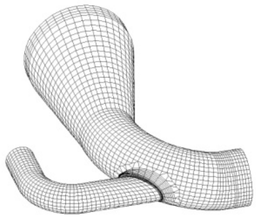
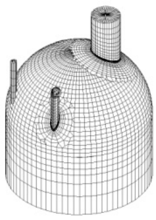
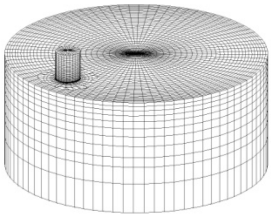
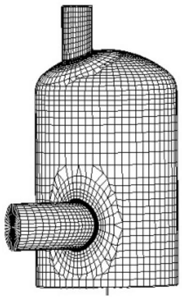
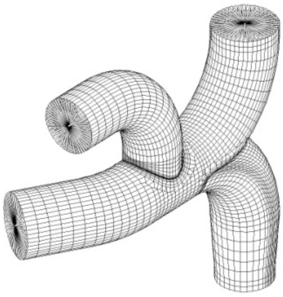
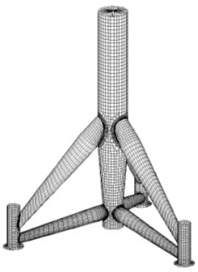
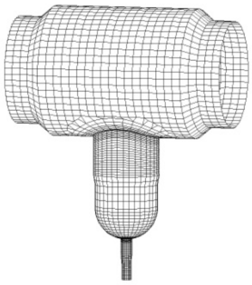
Y-piece



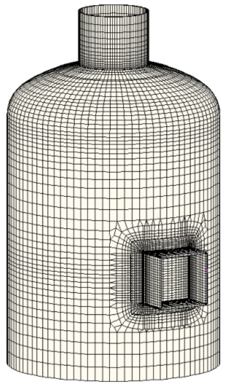
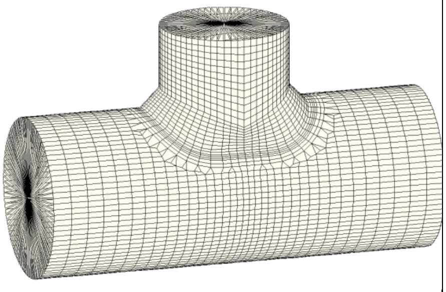
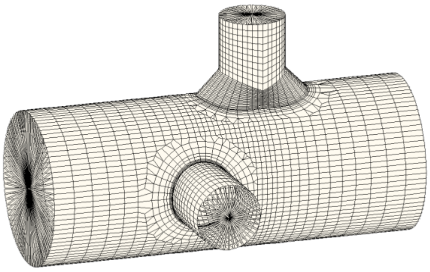


The cylindrical bases are created by superelements of the type Centric Cone (cylinder).
The two “reducers” are made by superelements type *Eccentric Cone*

Complex systems

		
Support in Miter bend	*welded-in bend segment*	*Vessel with nozzle*
		
Head with 2 branches	*Cyclone*	*Branch at bend*
		
Hemispherical head	*Flat plate*	*Torispherical head*
		
Cross-sectioned bends	*Tripod*	*Drainage*

Rectangular lug and Gusset

		
Rectangular lug	*Gusset - model*	*Gusset and tee*

6 Program license and system requirements

Program version, network license

ROHR2fesu is an optional available module in the program system ROHR2.
It can be part of the ROHR2 single user license and ROHR2 network license. In the ROHR2 network license the number of the users of an optional module can be similar or lower than the number of ROHR2 network seats.
For system requirements and program features see ROHR2 Specification.
Running ROHR2fesu requires the installation of **ROHR2.**

Scope of delivery and license key

The programs´ scope of delivery contains
- the program data
- the program documentation in html format
- unlocking the module on the ROHR2 license key (USB, dongle).
The software does not run without the license key. In case of updates/upgrades the license key will be replaced or updated.

Documentation /User manual

The functions of ROHR2fesu are part of the ROHR2 online help.

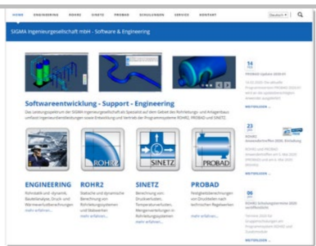
Maintenance and user support

Advice about installation and application is done by the ROHR2 user support (hotline). The hotline is part of the included service after purchase, during time limited licensing (rent) and as a part of a maintenance agreement.
Interfaces and additional programs are integrated into ROHR2. Maintenance of additional programs and interfaces is mandatory in this case.

7 Software Development, Sales and Support

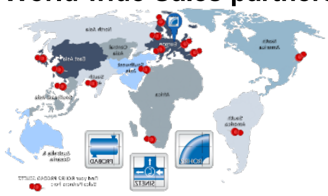
SIGMA as one of the leading engineering specialists in the Pipe Stress Business offers ROHR2, PROBAD and SINETZ: field tested software products, strongly adapted to the user´s needs.
SIGMA engineering division is known as a partner of choice offering consulting services for plant engineering, chemical industry, engineering companies, energy suppliers and technical security boards.

Contact:
SIGMA Ingenieurgesellschaft mbH
Bertha-von-Suttner-Allee 19
D-59423 Unna
Germany
Tel +49 2303 33233-0
sales@rohr2.com



www.rohr2.de

World wide Sales partners



www.rohr2.com