Version
July 2019

Add-on Modules

RF-FORM-FINDING
RF-CUTTING-PATTERN

Form-Finding and Cutting Patterns of Membrane and Cable Structures

Program Description

Am Zellweg 2
93464 Tiefenbach
Germany

Tel.: +49 9673 9203-0
Fax: +49 9673 9203-51
E-mail: info@dlubal.com
Web: www.dlubal.com

Contents

1 Introduction

Membrane structures represent attractive alternatives for the roofing of small and large objects: they are lightweight, elegant, and effective at the same time. However, designing membrane structures requires a specific procedure. Generally, the used materials do not feature any effective bending stiffness. They can only absorb tension forces. Therefore, it is crucial of the design to determine the initial equilibrium state. The shape of membrane structures is not freely selectable but has to be found on the basis of boundary conditions, loading prestresses, and overpressures, if applicable.

When planning membrane structures, the determination of the shape cannot be separated from the prestress design (as it is possible for bending-resistant structures). The shape has to be generated. This will not at all restrict your creativity of design but will provide you with a new dimension. A variety of shapes can be achieved by adjusting the boundary conditions or prestress. The ideal prestress distribution is always individual; it results from the demands of the model.

The first chapter of this manual gives you an introduction on the topic of *form finding*, explaining the key features of the calculation methods implemented in **RF-FORM-FINDING**. The second chapter describes the individual dialog boxes and functions. The third chapter presents some examples for learning how to work with the program. Another chapter with important notes is followed by the final description of the add-on module **RF-CUTTING-PATTERN**.

1.1 Add-on Module RF-FORM-FINDING

The add-on module RF-FORM-FINDING of the main program RFEM helps you to find prestressed initial shapes of membrane and cable structures as well as of models featuring "ordinary" beams and surfaces with tensile or compressive forces applied. Those prestressed shapes can then be analyzed by RFEM.

1.2 Essential Form Finding Process

In general, the materials used for membrane structures feature tensile capacities only. All loads are transferred solely by tension. In order to ensure a sufficient resistance and shape consistency of the membranes, an adequate prestress must be provided.

Due to the zero stiffness of the materials, it is not possible to separate the shape layout from the prestress design – the shape is defined by the prestress. Each spatial equilibrium system of forces (that is the equilibrium of prestressing forces) clearly determines the spatial shape of a membrane structure. By defining boundary conditions and prestress forces in the equilibrium state, the actual shape of the membrane model is set. Searching for the shape of a membrane structure as a function of the defined prestress is called "form finding".

There are two different approaches to the form finding process:

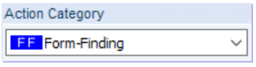
1. Definition of boundary conditions and loading prestress
   The corresponding equilibrium shape is sought.
2. Definition of boundary conditions and shape (deformation) of the membrane
   The equilibrium prestress is sought.

Both methods are valid and have their advantages. However, the interaction between shape and loading prestress must never be neglected. With the second approach, you can affect the final shape more easily, while the first one facilitates the design of the final prestress. As the loading prestress is crucial for the bearing resistance and durability of the model, RF-FORM-FINDING provides this method of form finding as well.

1.3 Key Features of Form Finding Process

There is a direct correlation between the loading prestress and the shape of membrane structures. The properties of the used textiles are irrelevant: the form finding process is independent of the material.

In addition to boundary conditions and loading prestresses, the shape of a membrane model can be also affected by loads. Thus, for pneumatically prestressed membranes, the program seeks the equilibrium shape for the defined prestress and interior pressure. The latter can be applied as a static load acting perpendicularly to the surface, or as a gas solid (ideal gas with isothermal properties of state), see article 001507 of the Knowledge Base at our website.



The load due to self-weight may also affect the form finding process. It is possible to search for a shape that corresponds to the specified prestress, self-weight (as well as other types of loads contained in the *Form-Finding* category load case), and interior pressure, if applicable. In most cases, the self-weight applied in the form finding process has only little effect on the final shape and prestress as the self-weight of the textile is light.

1.4 Calculation Methods in RF-FORM-FINDING

The shape of a membrane structure is clearly defined by its boundary conditions and equilibrium prestress, or by the equilibrium between prestress and loading (interior pressure, self-weight as well as other loads of the *Form-Finding* load case). However, the definition of the equilibrium prestress presents a problem, i.e. how to find the spatial equilibrium system of forces.

When planning a membrane structure, isotropic prestress is usually the only equilibrium prestress that can be defined in advance. But such a prestress is not always suitable – in terms of both the actual shape and the following nonlinear analysis. Moreover, it may occur that a shape that conforms to the isotropic prestress is physically not possible due to specific boundary conditions.

Hence, it is necessary to specify an orthotropic prestress in the form finding process. A constant orthotropic prestress in the equilibrium state is only possible if the Gaussian curve of the respective surface is equal to zero (e.g. for plane or cylindrical surfaces). Double-curved membrane structures are not characterized by this property. It would be necessary to apply a general spatial orthotropic prestress to those membrane models. This would be unrealistic, however, and it would require a tool that is able to find, in addition to the equilibrium shape, the equilibrium prestress by specifying the prestresses in two directions (warp and weft directions).

In RF-FORM-FINDING, two calculation methods are implemented that can be used to find equilibrium shapes and prestresses – the *projection method* and the *tension method*. Both methods are based on the familiar *Updated Reference Strategy* (URS) form finding method published by K.U. Bletzinger and E. Ramm in 1999.

In general, the projection method is advantageous for high conical shapes, while the tension method is more appropriate for membranes supported by points and arches or for pneumatically stabilized membranes. The tension method is defined as *Standard method* in RF-FORM-FINDING.

1.4.1 Projection Method

As already mentioned, it is virtually impossible to define a general equilibrium prestress in space. Yet it is possible in a plane where a constant orthotropic prestress (orthogonally aligned prestress) may exist besides isotropic prestress. Furthermore, when defining a loading prestress in the radial direction, it is also possible to determine the prestresses in the tangential direction for all adjacent points on the basis of a certain equilibrium condition. Thus, it is possible to clearly define an equilibrium system of forces in a plane.

These conditions are used by the projection method. It is based on the "projection" of the prestress defined in the global XY plane into the actual position of the membrane structure (or the surface othogonal to the directive axis of a radial coordinate system). If the inclination of the membrane against the projection plane is equal to zero, the prestress in the membrane corresponds to the specified values. If its inclination against the projection plane is unequal to zero, the prestress in the fall line direction increases while the prestress in the contour line direction decreases. If the inclination against the projection plane is almost perpendicular, the prestress in the fall line direction increases significantly while the prestress in the contour line direction is close to zero.

The projection method allows the prestress equilibrium to be preserved in the directions of the global axes X′ and Y′ within the projection plane. In the form finding process, the program then seeks the layout of the membrane structure in space that retains the equilibrium prestress also in the direction of the global axis Z′ perpendicularly to the plane. The projection method – which determines the equilibrium prestress in the projection – leads to the definition of the equilibrium prestress in space and, thus, to finding a clear spatial shape of the membrane structure.

To find the equilibrium state, the *Updated Reference Strategy* (URS) method is used. Therefore, the form finding process represents a nonlinear problem.

1.4.2 Tension Method

The tension method is set as *Standard* method in RF-FORM-FINDING.

This method is very different from the projection method: the defined prestress is not modified, i.e. stabilized. When determining the equilibrium shape, the tension method applies the two set values that have been defined for the prestresses in the warp and weft directions.

In most cases, a prestress that is physically out of equilibrium (except isotropic prestress) is defined. Therefore, when applying an orthotropic prestress (e.g. 2.0 kN in the warp and 1.0 kN in the weft directions), it is unlikely that a shape of the membrane structure is reached with a loading prestress that exactly meets these requirements. Since the requirement for a constant orthotropic prestress usually reflects no adequate solution, the shape would not converge to the equilibrium position when using this prestress repeatedly in the iteration process. For this reason, the selected prestress in the membrane model is applied only in a user-defined number of iterations. After that, a stabilization is applied.

The tension method uses the fact that, when defining a loading presstress out of equilibrium for the membrane structure, the deformations perpendicularly to the plane of the membrane are more frequent than the deformations within the plane. As soon as the defined number of iterations for applying the specified prestress has been reached, the structure is stabilized. The resulting prestress then comes usually very close to the specified values. The tension method is also based on the *Updated Reference Strategy* (URS) method of form finding.

1.5 Form Finding for Combined Structures

While the shape of membranes in RF-FORM-FINDING is determined by means of the defined prestress values, it is possible for cables to define, in addition to prestresses, geometrical requirements, such as the final rise or length. Same applies to "ordinary" beams and surfaces featuring tensile and compressive attributes.

Membranes are often parts of a structure that contains many bending-resistant elements (beams, plates, shells, etc.) During the form finding process, an equilibrium shape of the entire model is sought. The prestress in the membrane and cables has an active effect on the rigid elements of the structure which have to counteract that prestress. If you do not want to consider the influence of rigid elements in the form finding process, you can supply them with specific temporary form finding supports to fix the objects for the form finding analysis. It is at your individual discretion which variant of the construction assembly is the most appropriate for the structure.

1.6 Add-on Module RF-CUTTING-PATTERN

The RF-CUTTING-PATTERN module represents the second component in the RFEM product family that is used for membrane structures. It generates and organizes cutting patterns for membranes that are available as results of the form finding process. Alternatively, RF-CUTTING-PATTERN can be used for self-defined spatial membrane systems.

If the division of the membrane surfaces is too large for cutting, it is possible to divide the surface into partial strips by cutting lines. The boundary conditions of cutting patterns on the curved geometry can be determined by means of boundary lines or by independent planar or geodesic cutting lines. The flattening process is performed according to the theory of minimum energy.

Prestress and stresses from flattening are applied by compensations. In addition, allowances for welds and boundary connections can be defined separately for each cutting pattern.

Chapter 5 gives you a detailed description of the RF-CUTTING-PATTERN add-on module.

2 RF-FORM-FINDING

This chapter describes the dialog boxes and functions of the RF-FORM-FINDING add-on module.

2.1 Open Add-on Module

To work with RF-FORM-FINDING, it is necessary to activate the add-on module first: Select the **RF-FORM-FINDING** check box which you can access in the *Options* tab of the *New Model - General Data* or *Edit Model - General Data* dialog boxes.

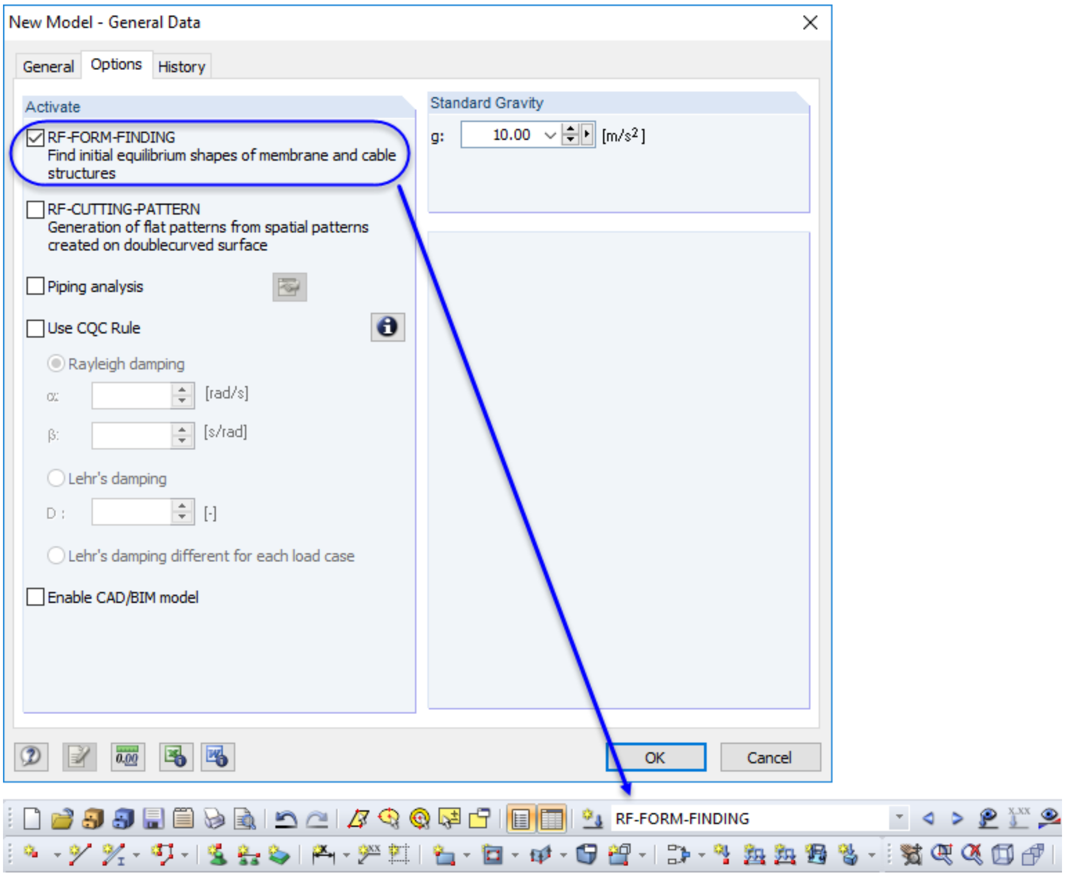


Figure 2.1: Activating RF-FORM-FINDING in *New Model - General Data* dialog box

A new load case with the description *RF-FORM-FINDING* will be created automatically when the add-on module is activated while defining the general data of a new model. This load case manages specific parameters and the results of the form finding process.

2.2 Dialog Boxes of Add-on Module

The activated RF-FORM-FINDING add-on module provides the following dialog boxes and features in the user interface of the main program RFEM.

2.2.1 Calculation Parameters for Form Finding

The *Form-Finding* tab of the *Calculation Parameters* dialog box manages basic options concerning the the form finding. To access this dialog box, click the toolbar button shown on the left.

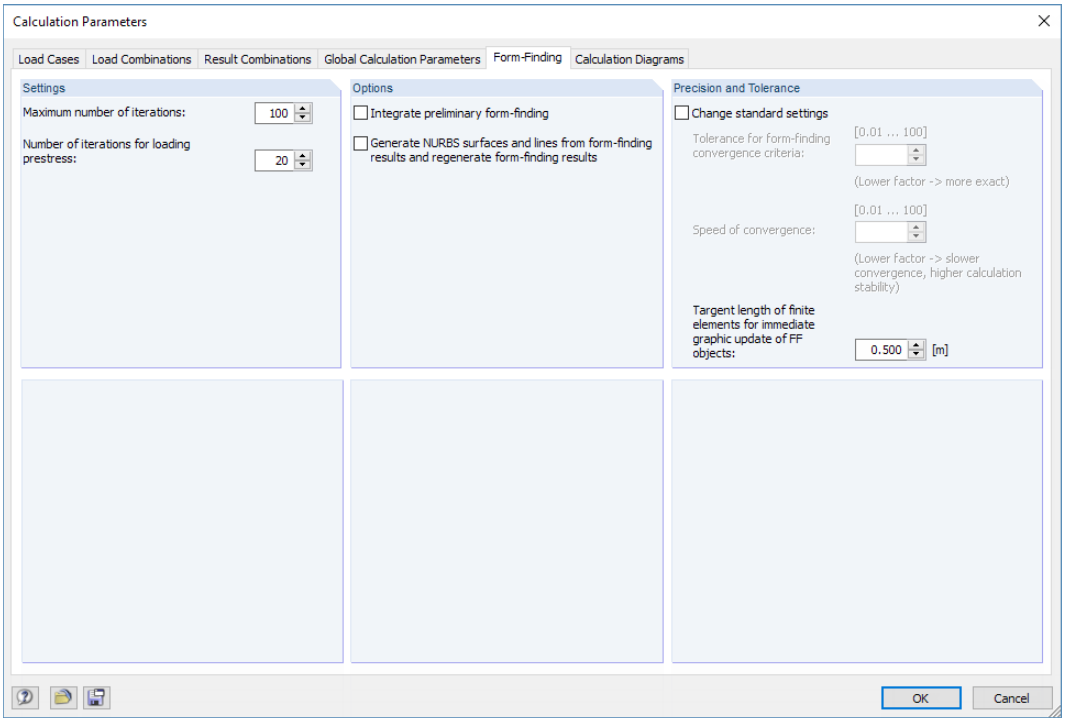


Figure 2.2: Dialog box *Calculation Parameters*, tab *Form-Finding*

Settings

This section of the tab controls the *Maximum number of iterations* for the nonlinear calculation as well as the *Number of iterations for loading prestress*.

Options

With the *preliminary form-finding* you can accelerate the calculation: prior to the actual form finding process, the mesh nodes will be moved to a position that is close to the target geometry taking solely into account the membrane and cable elements. Based on this approximation, the form finding is carried out taking into account all effects from the entire structural system.

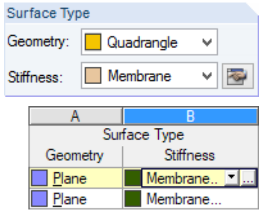
There is also an option to *Generate NURBS surfaces and lines* from the results of the form finding process. The generation of NURBS surfaces is only possible for surfaces that are defined by three or four lines, however. After this transformation, it is recommended to deactivate the option *Hide Membranes and Cables* in the *Display* navigator.

Precision and Tolerance

This section controls the tolerance of the convergence criteria for the form finding process. In general, it is not required to modify the default values.

The *Target length of finite elements for immediate graphic update of FF elements* controls the mesh settings used to calculate the approximate shape of member elements (see Chapter 2.2.3, page 11).

2.2.2 Surfaces of Types 'Membrane' or 'Standard'



The calculation method as well as the prestress parameters for *Standard* or *Membrane* surfaces can be defined in a specific dialog box. It appears when clicking the button next to the list of surface types in the *New Surface* (or *Edit Surface*) dialog box or the button in Table *1.4 Surfaces*. Select the *Standard*, *Membrane* or *Membrane - Orthotropic* stiffness type in the drop-down list first.

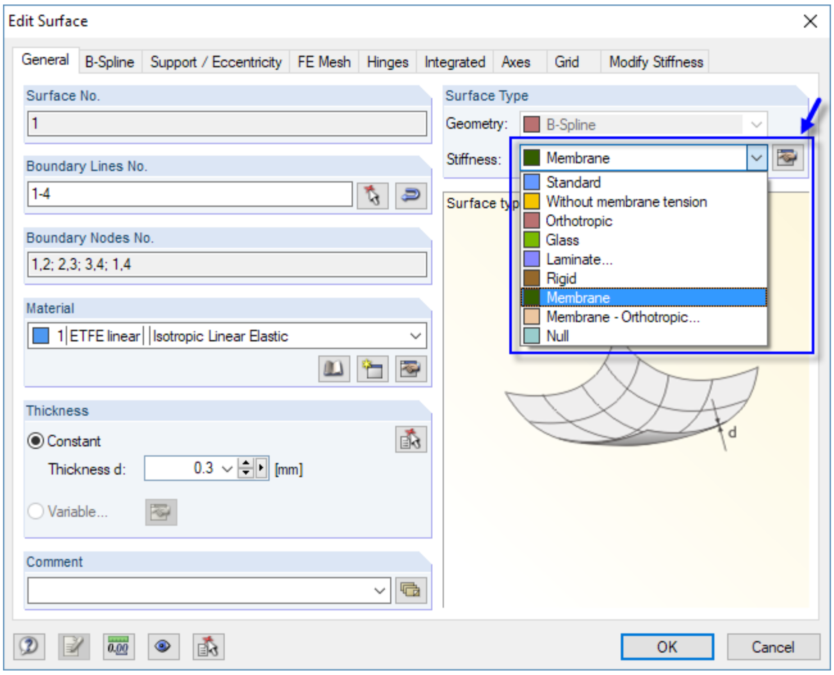


Figure 2.3: Selecting stiffness type and [Edit] button

In the *Edit Surface Stiffness* dialog box, you decide whether the *Standard* tension method or the *Projection* method is to be applied for the calculation. Both methods are described in Chapter 1.4.

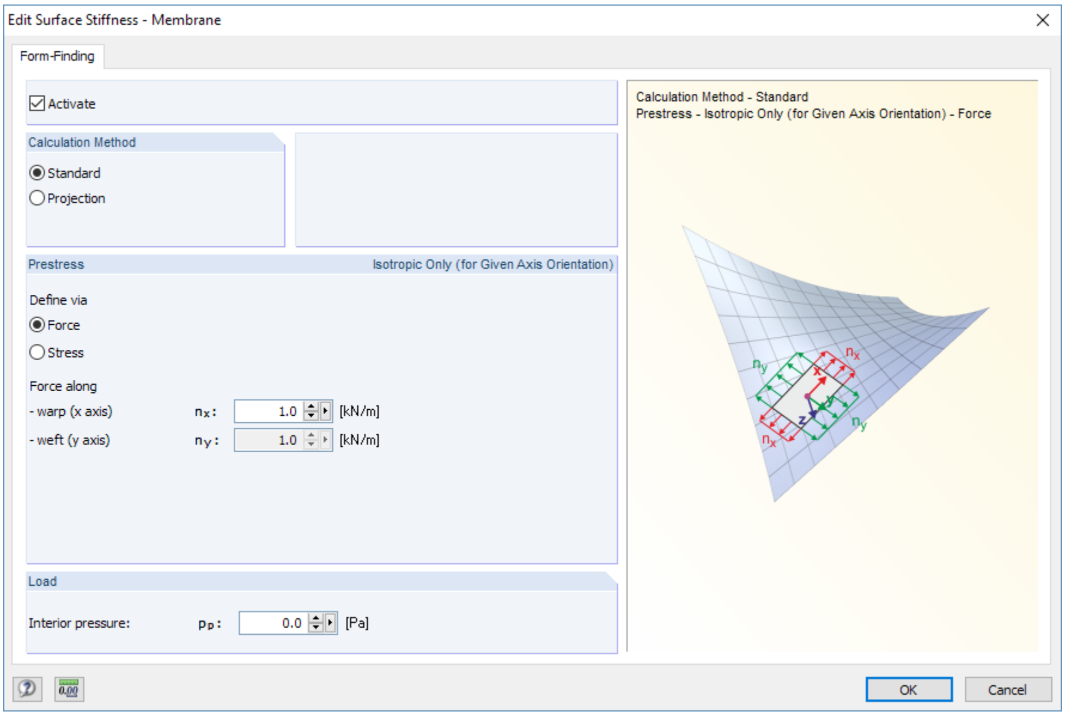
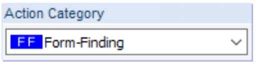


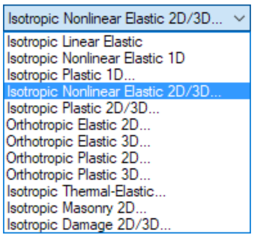
Figure 2.4: Dialog box *Edit Surface Stiffness* for membrane surface

The *Prestress* can be defined as *Force* or *Stress*. The warp and weft directions are linked to the axes of the surface (see *Axes* tab of *Edit Surface* dialog box). In the default setting, you can apply only an isotropic prestress. If the axes are aligned, orthotropic or radial prestress is also available.



For pneumatic membranes, you can specify the *Interior pressure*, p_p, in the *Load* section. Alternatively, the load can be defined in a load case of the *Form-Finding* action category.

Nonlinear materials



If the add-on module **RF-MAT NL** is available, nonlinear materials can be used for the surfaces to apply their properties to the form finding analysis. The features of the material can be defined in the dialog boxes of the selected *Material Model*.

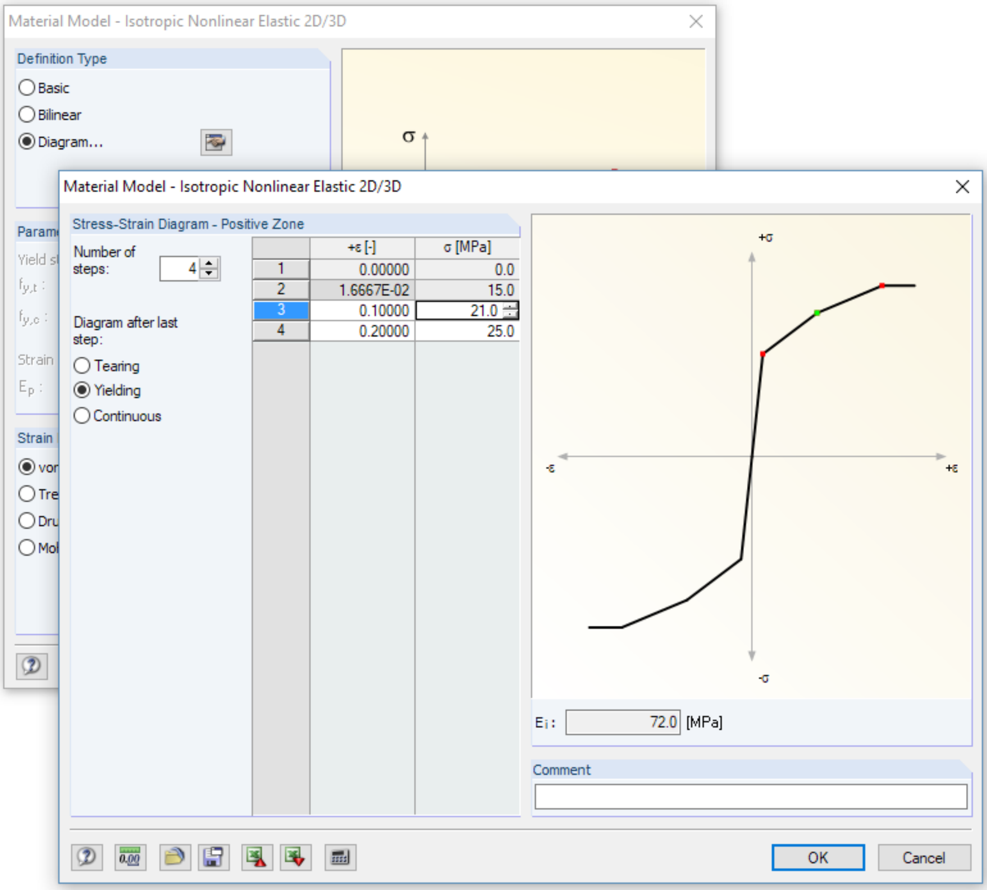


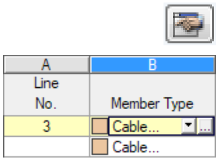
Figure 2.5: Dialog boxes *Material Model - Isotropic Nonlinear Elastic 2D/3D*

For the form finding process, these nonlinear material models are applicable:

- Isotropic nonlinear elastic 2D/3D
- Isotropic plastic 2D/3D

The nonlinear materials are described in the RFEM manual, Chapter *4.3 Materials*.

2.2.3 Members of Types 'Beam' or 'Cable'



To define the parameters for the form finding process of cables (tensile forces) and beams (tensile and compressive forces), open the *New Member* or *Edit Member* dialog box or Table *1.17 Members*. Select the *Cable* or *Beam* member type in the drop-down list. Then click the [button] or [...] button.

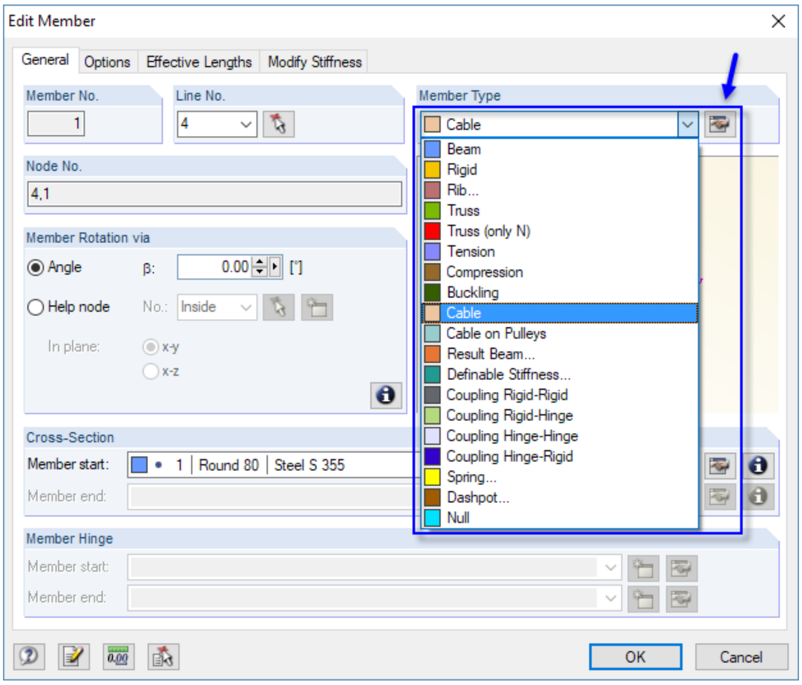


Figure 2.6: Selecting member type and [Edit] button

In the *Edit Parameters* dialog box, you can set the geometrical conditions or forces of the member.

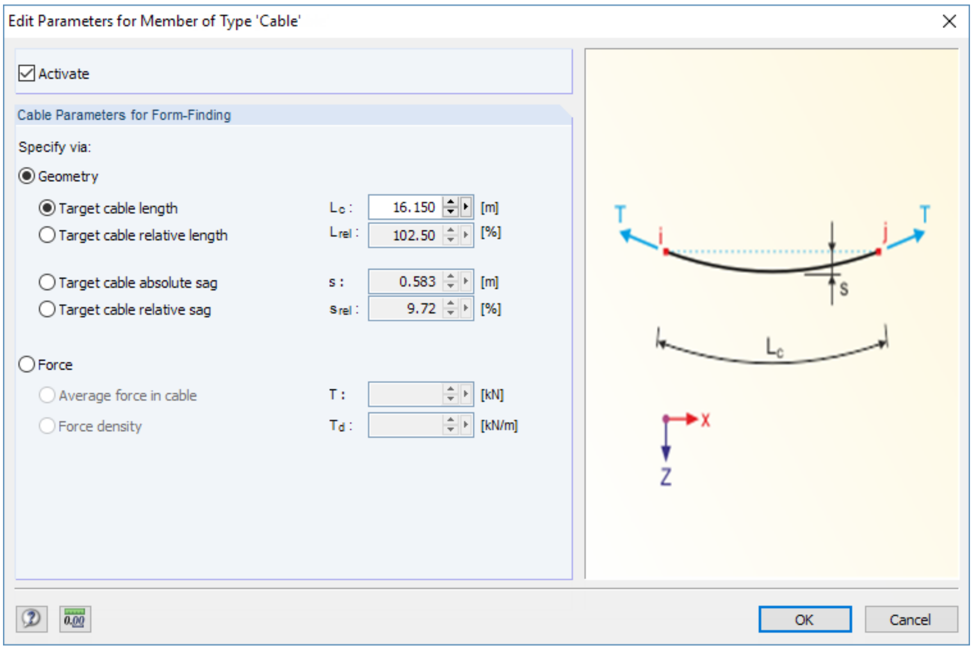


Figure 2.7: Dialog box *Edit Parameters for Member of Type 'Cable'*

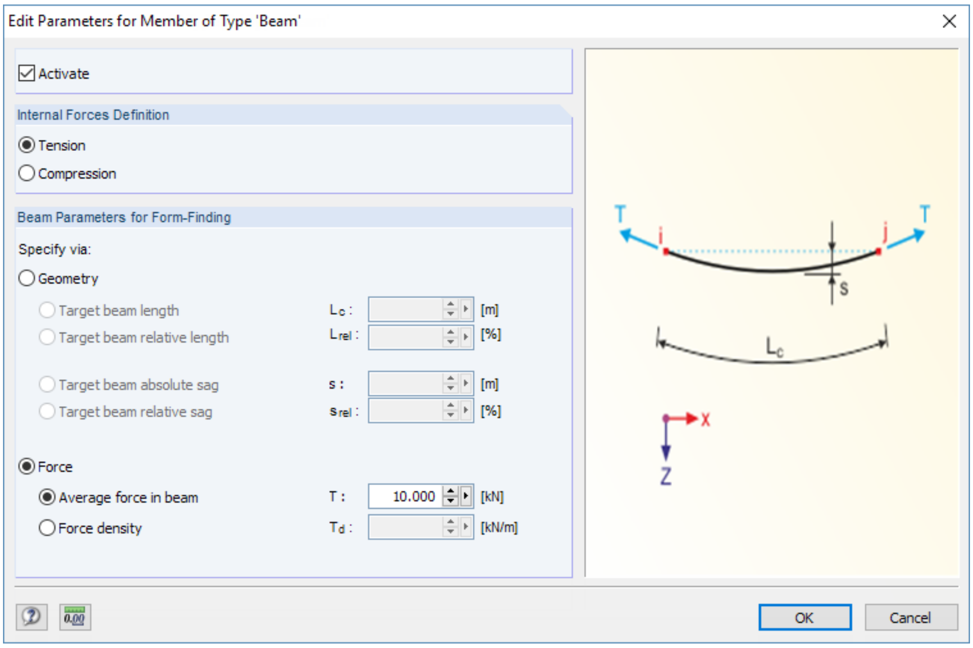


Figure 2.8: Dialog box *Edit Parameters for Member of Type 'Beam'*

Activate the check box to access the parameters of the member.

For beam members, the specific parameters can be set for *Tension* or *Compression*. Cable members only absorb tensile forces.

If you select the parameters of *Geometry*, you can define either the *Target cable/beam length (absolute/relative)* or the *Target cable/beam sag (absolute/relative)*. This option is relevant for cable or beam members at the borders of membrane structures.

If you select the parameter of *Force*, you can enter the *Average force in cable/beam* or the *Force density*. This option is suited to analyze cable webs or to determine the adequate inclination of column beams, for example.

When you activate the *Show Preliminary Form-Finding* option available in the general shortcut menu, the approximate shape of the model is determined according to the force density method. That "estimated" shape will then react to any modification of the topology as well as of the loading. Note that this option is only effective for *Force* type parameters (see above) of the cable or beam.

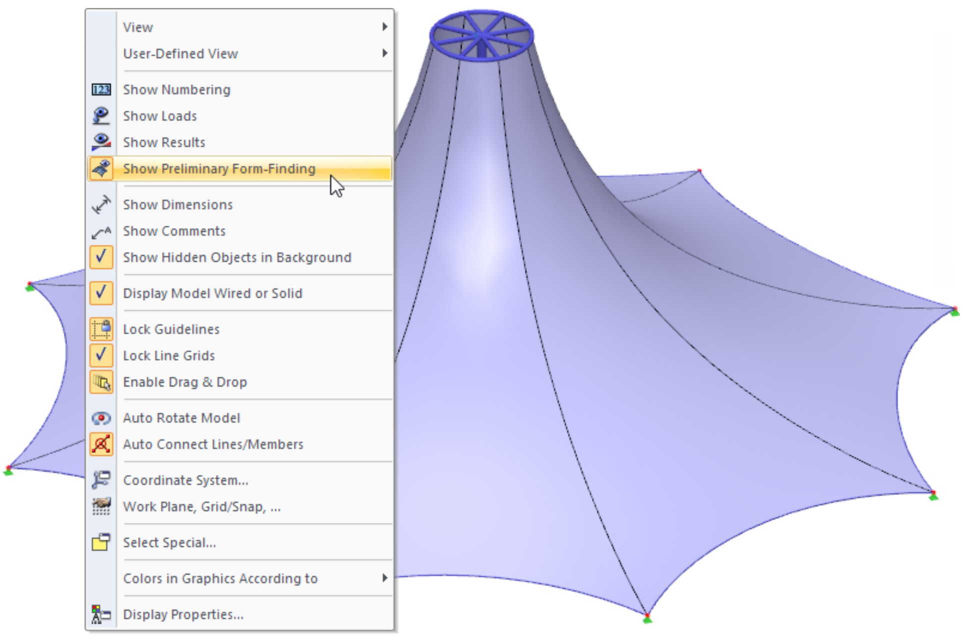


Figure 2.9: Shortcut menu item *Show Preliminary Form-Finding* to calculate approximate shape

2.2.4 Temporary Form Finding Supports

For the RF-FORM-FINDING module, a specific type of support *Nonlinearity* is available for nodes, lines, and surfaces. It can be set for supports that are only effective during the form finding process.

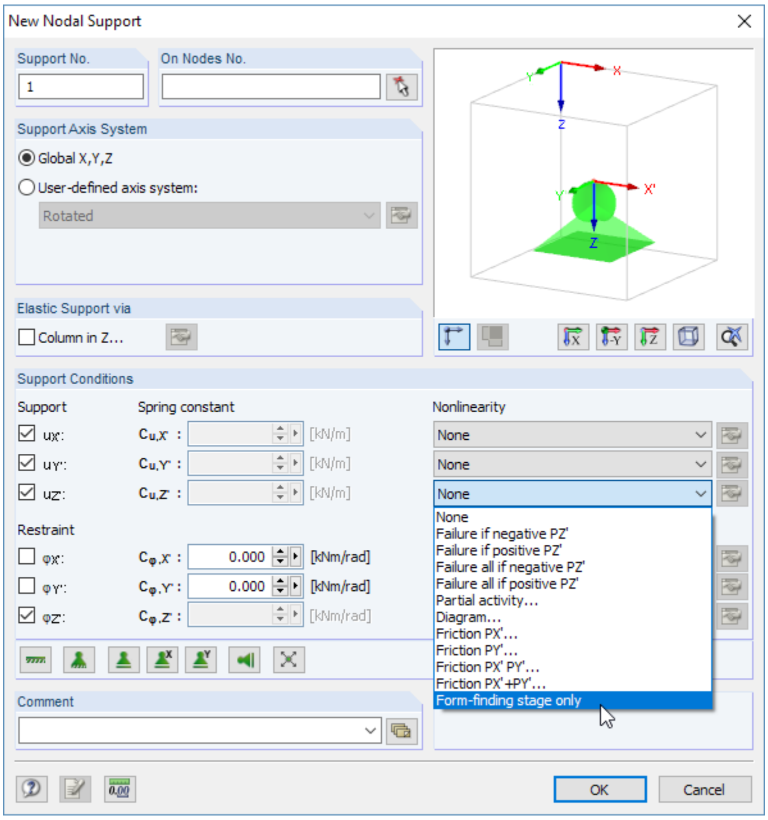


Figure 2.10: Temporary nodal support for form finding stage

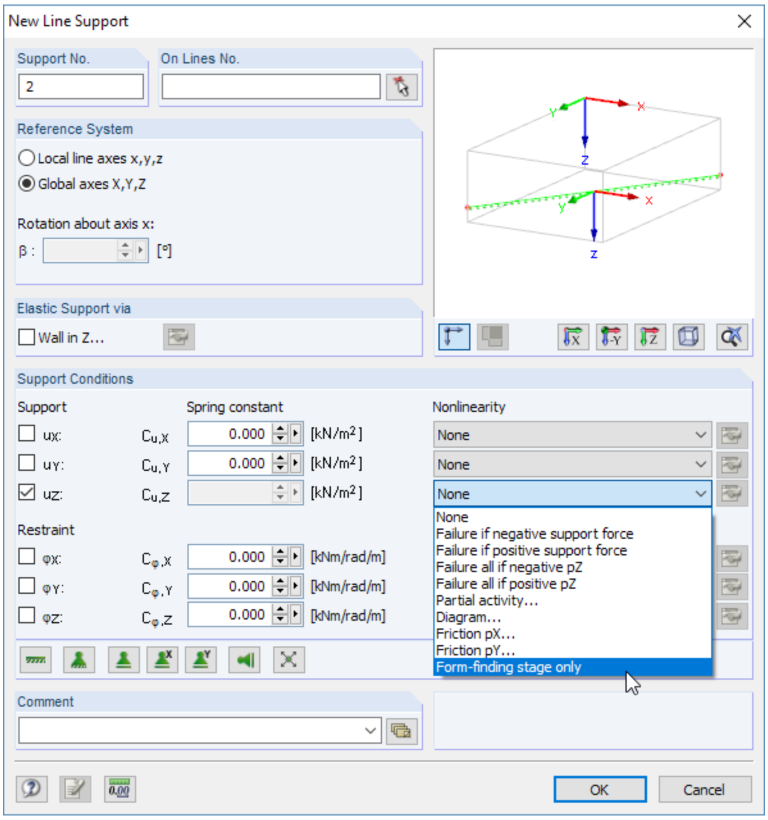


Figure 2.11: Temporary line support for form finding stage

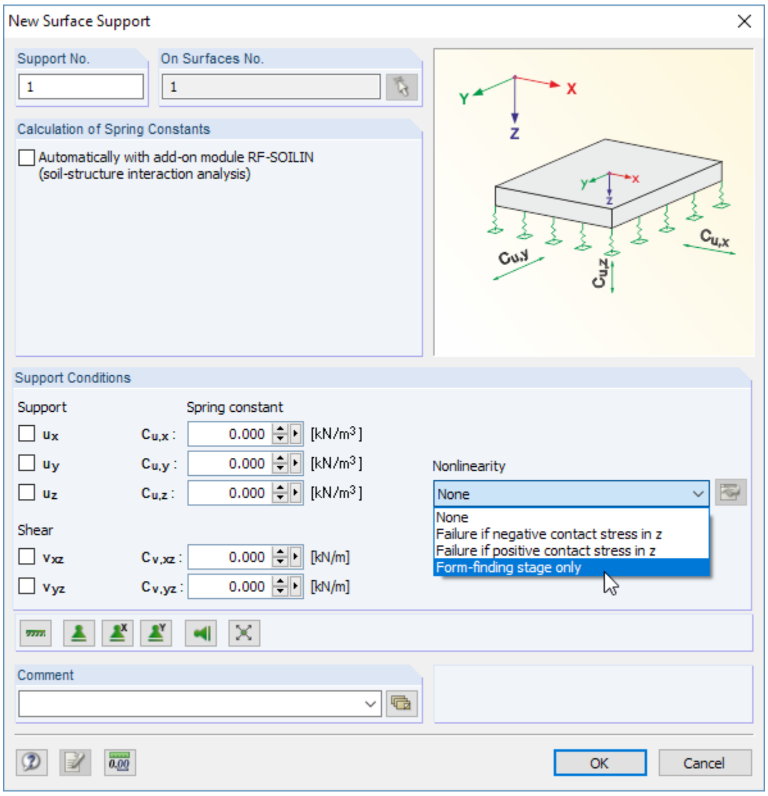


Figure 2.12: Temporary surface support for form finding stage

Chapter 3.2 gives you an example how to apply temporary form finding supports to an arch-supported membrane.

2.2.5 Form Finding Load Case

Loads that are essential for the form finding process, such as self-weight or gas pressure, are to be managed in one specific load case. As *Action Category*, the **Form-Finding** type has to be allocated. Create only <u>one</u> load case featuring this category and define all relevant loads there.

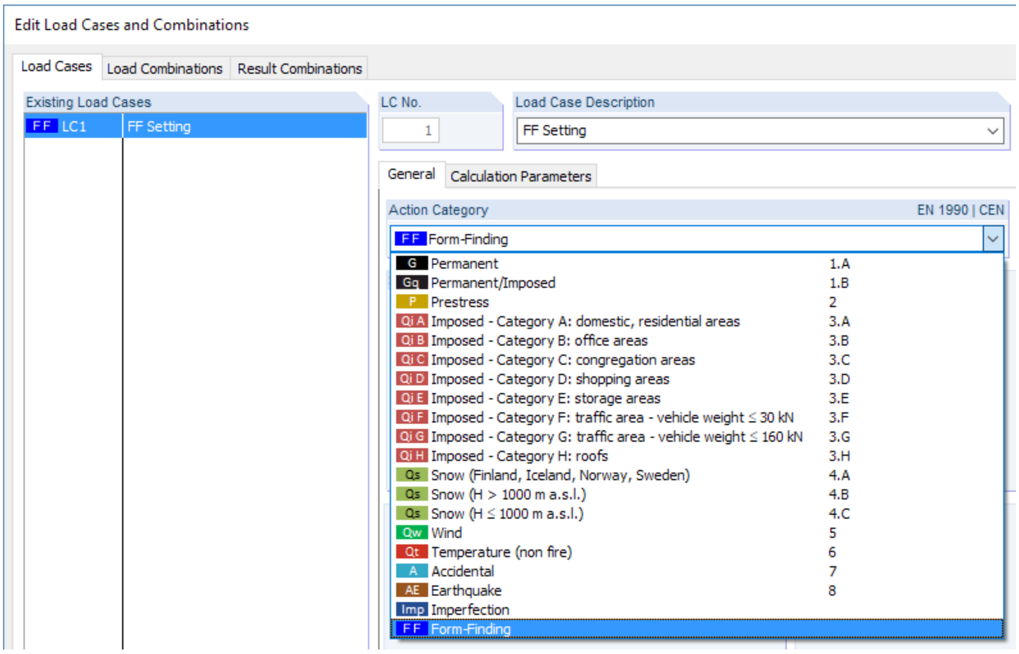


Figure 2.13: Allocating *Form-Finding* action category

The *Form-Finding* action category implies that the specific loads are considered for the form finding process only, not for the subsequent analysis of the final state of the membrane or cable model.

Loads that are relevant for form finding are managed as “ordinary” loads: you can double-click a load in the work window to open its *Edit* dialog box and adjust the parameters, for example.

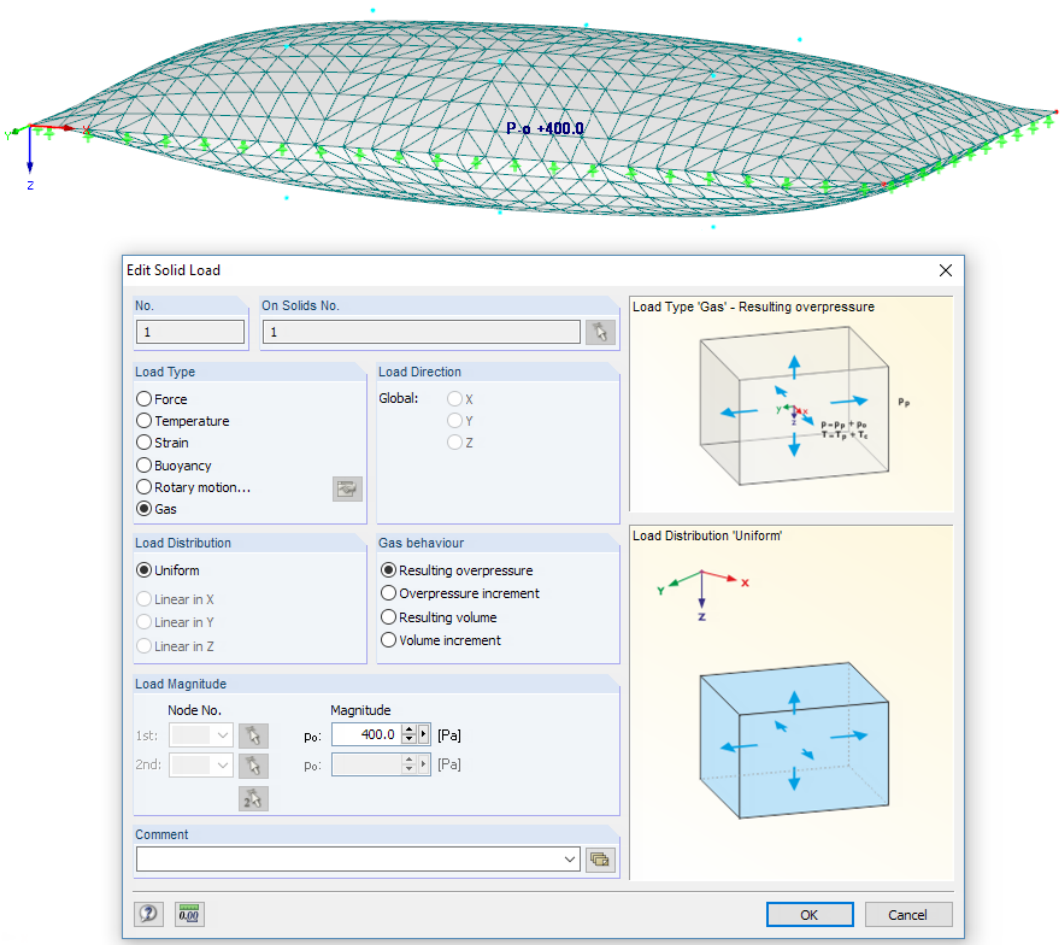


Figure 2.14: Modifiying load in *Edit* dialog box

2.3 Start Calculation

When activating the RF-FORM-FINDING add-on module (see Figure 2.1), a new *RF-FORM-FINDING* load case is created. You can start the calculation of this load case as usual in the RFEM toolbar or the drop-down menu.

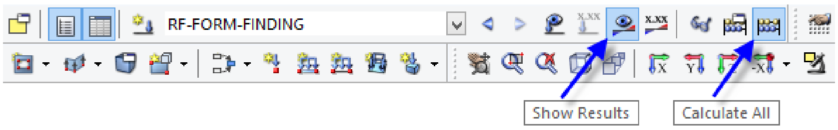


Figure 2.15: Options to start form finding analysis in toolbar

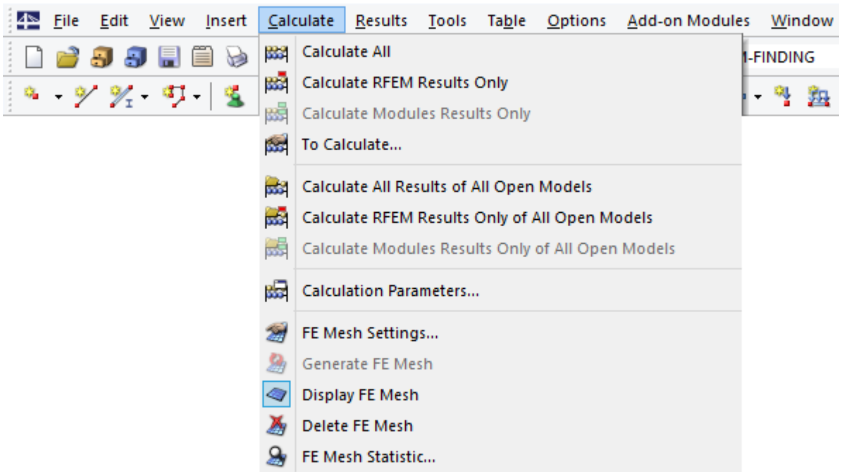


Figure 2.16: Options to start form finding analysis in menu

When the calculation is started, the program initiates the form finding process. It is performed on the model, taking into account the specified values and the *Form-Finding* category load case, if applicable.

As soon as the prestressed shape of the membrane or cable structure has been generated, the load cases (except for the *Form-Finding* type load case) are applied to the new shape of the model. Thus, the form finding process represents the first phase of the calculation where the prestressed shape is created. The loads defined in all load cases and load combinations will be applied to it – but for the *Form-Finding* load case, as mentioned above – when the *Calculate All* option is selected.

When the RF-FORM-FINDING add-on module has been activated, it is not possible to calculate "real" load cases without any previous form finding analysis. When you start the calculation of a load case without the compulsory form finding, the RF-FORM-FINDING case is calculated automatically first. The analysis of the load case will then be based on the resulting prestressed shape.

Do not calculate the *Form-Finding* type load case when the shape has already been found. Its only function is to find the form of the structure. If you calculated it once more manually, its loads would be taken into account again on the distorted model.

2.4 Results Display

The results of the form finding process are displayed in a similar way as load case results. You can control the results display in the *Results* navigator (Figure 2.17). The results are also displayed in Table *4.0 Results - Summary* (Figure 2.18).

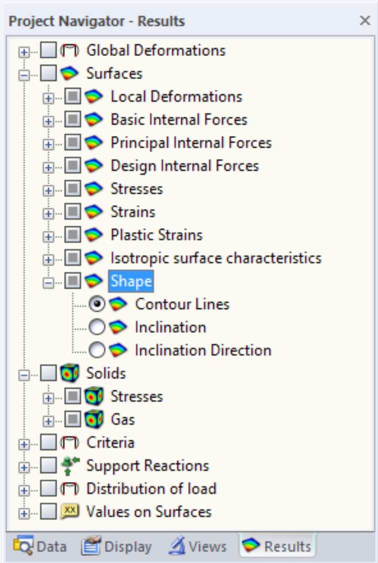
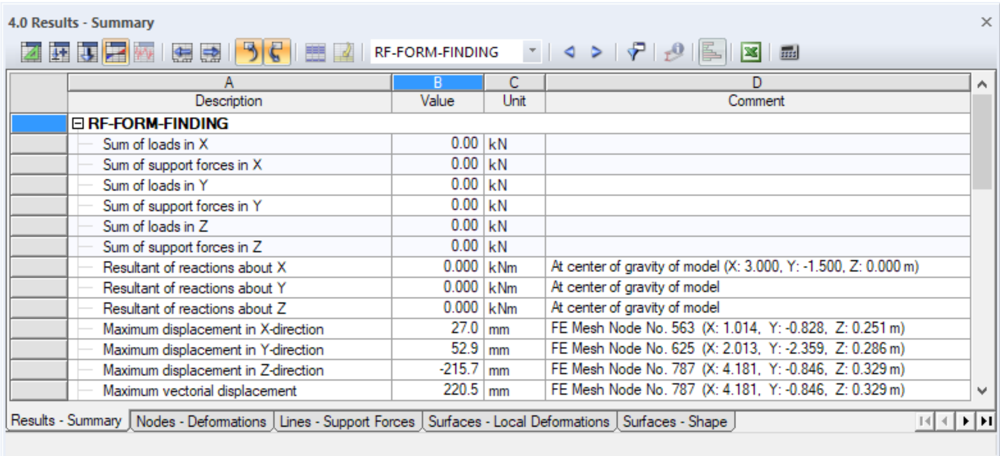


Figure 2.17: *Results* navigator



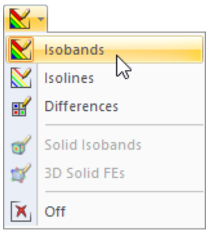
4.0 Results - Summary

A Description	B Value	C Unit	D Comment
RF-FORM-FINDING			
Sum of loads in X	0.00	kN	
Sum of support forces in X	0.00	kN	
Sum of loads in Y	0.00	kN	
Sum of support forces in Y	0.00	kN	
Sum of loads in Z	0.00	kN	
Sum of support forces in Z	0.00	kN	
Resultant of reactions about X	0.000	kNm	At center of gravity of model (X: 3.000, Y: -1.500, Z: 0.000 m)
Resultant of reactions about Y	0.000	kNm	At center of gravity of model
Resultant of reactions about Z	0.000	kNm	At center of gravity of model
Maximum displacement in X-direction	27.0	mm	FE Mesh Node No. 563 (X: 1.014, Y: -0.828, Z: 0.251 m)
Maximum displacement in Y-direction	52.9	mm	FE Mesh Node No. 625 (X: 2.013, Y: -2.359, Z: 0.286 m)
Maximum displacement in Z-direction	-215.7	mm	FE Mesh Node No. 787 (X: 4.181, Y: -0.846, Z: 0.329 m)
Maximum vectorial displacement	220.5	mm	FE Mesh Node No. 787 (X: 4.181, Y: -0.846, Z: 0.329 m)

Results - Summary | Nodes - Deformations | Lines - Support Forces | Surfaces - Local Deformations | Surfaces - Shape

Figure 2.18: Table *4.0 Results - Summary*

After the form finding analysis, the *Results* navigator features the *Shape* entry with three subentries (see Figure 2.17). The *Contour Lines* option shows the outlines of the model. They represent the Z-coordinates of every mesh point as isobands, for example. Alternatively, you can display the maximum *Inclination* on each location of the membrane. The angles can as well be displayed as *Inclination Direction*.



When you select the *Isobands* option for the surface results, the contour lines of the generated model are displayed in a colored view. The *Color Scale* tab of the panel illustrates the legend.

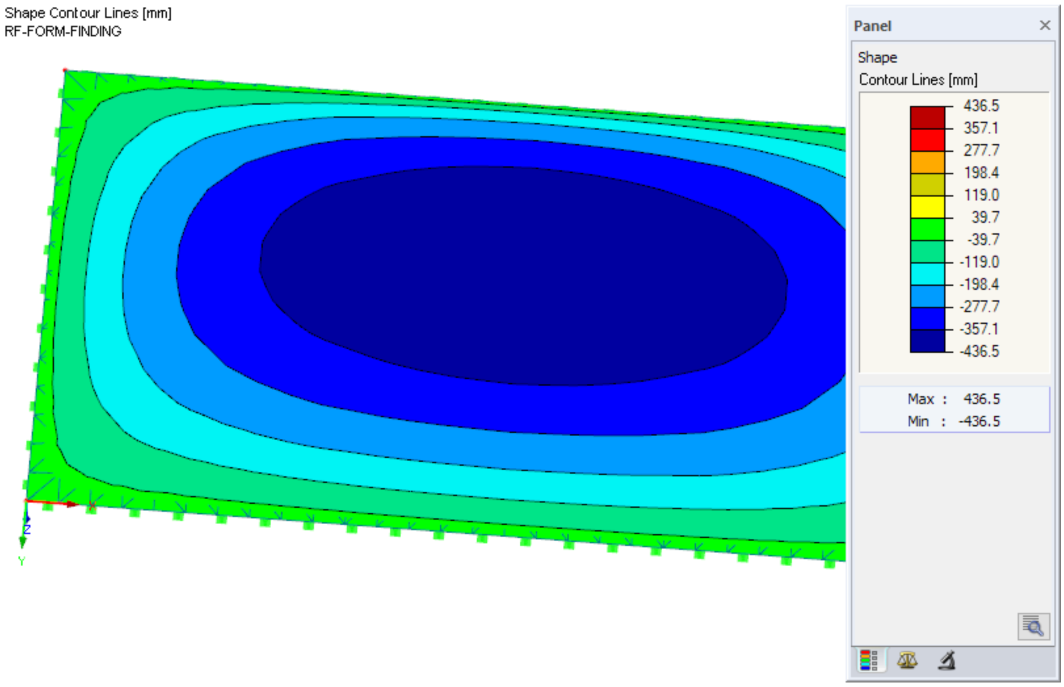


Figure 2.19: Panel illustrating contour lines of membrane model

The button in the panel enables you to display user-defined *level curves* of the generated shape. Enter the *interval* which is to be applied for the black isolines.

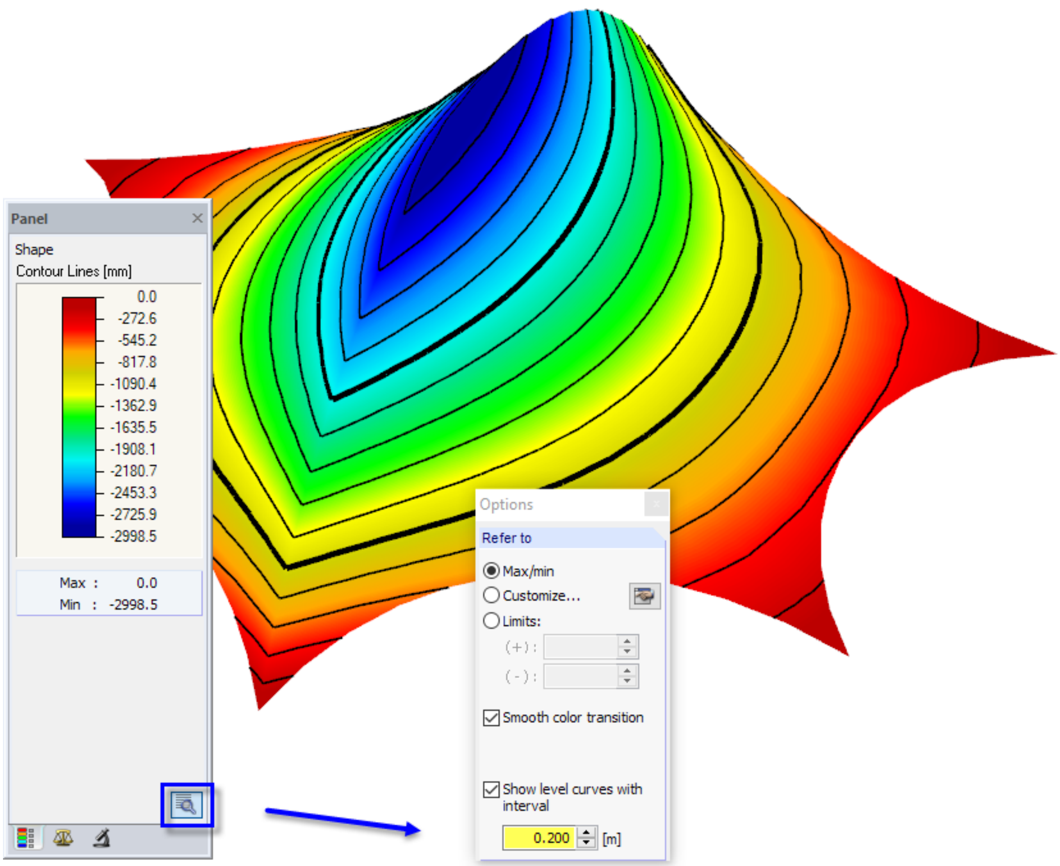


Figure 2.20: *Options* panel for level curves

The form finding analysis entails a distorted shape of the inital FE mesh: the new shape of the mesh replaces the original one. You can activate the new mesh in the *Display* navigator by selecting

FE Mesh (Form-Finding) → **On Members**,

FE Mesh (Form-Finding) → **On Surfaces** and/or

FE Mesh (Form-Finding) → **In Solids**.

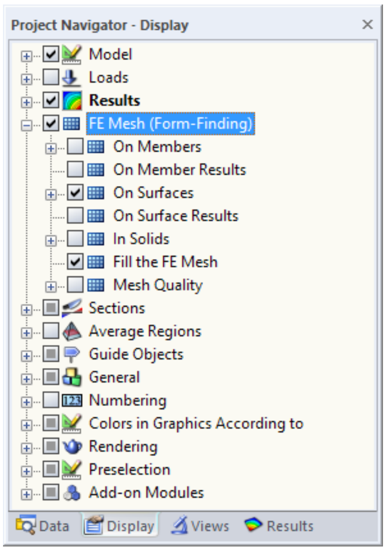


Figure 2.21: Displaying new mesh shape in *Display* navigator

The new mesh – inclusive of its implicated loading prestress – is applied to all subsequent calculations of load cases and combinations.

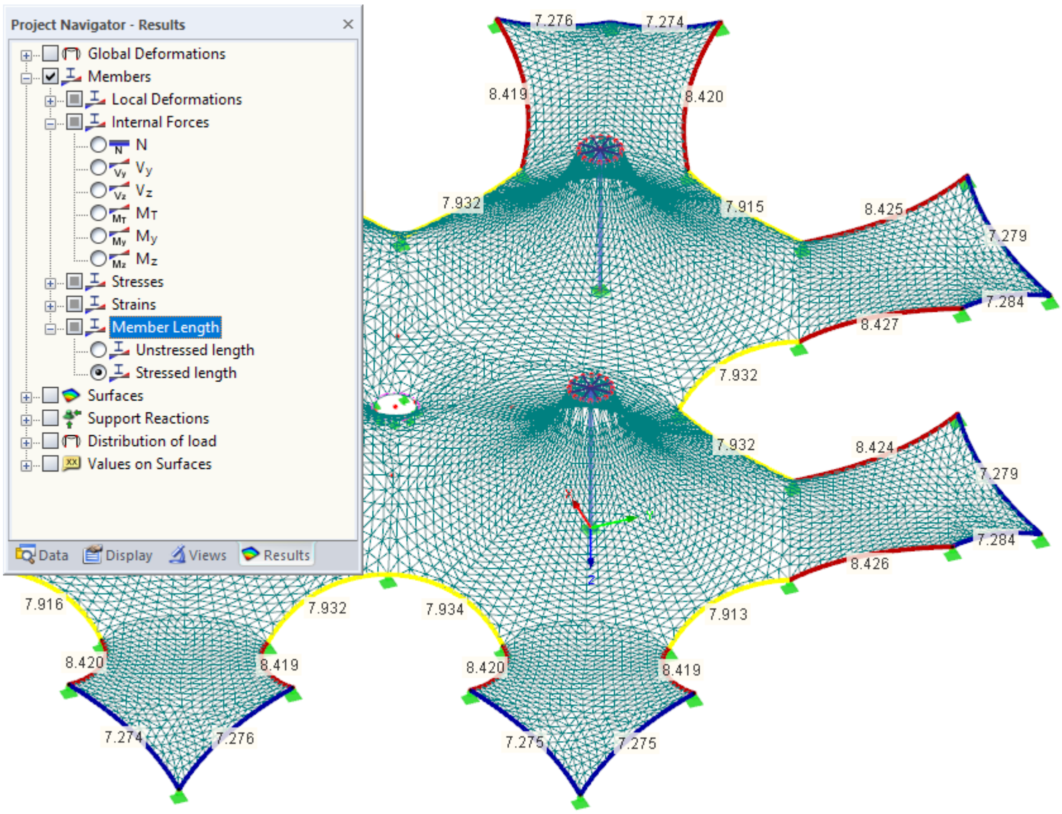
Due to the form finding process, the point of the FE mesh are moved to new locations. The initially modeled surfaces or members remain in their initial positions, however.

To hide or show the original model, use the *Model* selection in the *Display* navigator.

The *Generate NURBS surfaces and lines* option, which is available in the *Calculation Parameters* dialog box (see Figure 2.2, page 7), enables you to transform the shifted mesh into NURBS objects.

Member lengths

For cable or beam members that have been activated for the form finding analysis, the *Unstressed Length* or *Stressed Length* can be displayed in the work window and in Table *4.4.2 Members - Shape*.



4.4.2 Members - Shape

RF-FORM-FINDING

Member No.	A Unstressed Length [m]	B Stressed Length [m]	C Cross-Section
58	7.151	7.284	1 - Round 20
59	7.151	7.279	1 - Round 20
60	8.221	8.419	1 - Round 20
61	8.221	8.420	1 - Round 20
62	8.221	8.424	1 - Round 20
63	8.221	8.426	1 - Round 20
64	7.151	7.284	1 - Round 20
65	7.151	7.279	1 - Round 20
66	7.151	7.274	1 - Round 20
67	7.151	7.276	1 - Round 20
68	8.221	8.425	1 - Round 20
69	8.221	8.427	1 - Round 20
70	8.221	8.419	1 - Round 20

Members - Local Deformations | Members - Shape | Members - Global Deformations | Members - Internal Forces | Members - Strains

Figure 2.22: Displaying unstressed and stressed lengths of cables

As seen in Figure 2.22, the graphics can be controlled by the *Member Length* options of the *Results* navigator. When you activate the *Unstressed length*, you can retrieve the lengths which are required for the fabrication of each cable.

3 Examples

This chapter contains some examples that show you how to work with the RF-FORM-FINDING add-on module. It presents characteristic types of membrane structures, such as point-supported membranes, arch-supported membranes, conical membranes, and pneumatic membranes.

When working with RF-FORM-FINDING, the first step is always to create a model which defines the individual surfaces, cables, supports, and so on. This model represents the "initial approximation" of the final shape. During the form finding analysis, the model is then distorted into its new position. This process represents a function of the specific input parameters for form finding.

The dialog boxes described in Chapter 2 are used to define the loading prestress for the surfaces of the *Membrane* or *Membrane - Orthotropic* types and to set the warp and weft directions. For the cable members, the geometrical conditions or forces are defined.

3.1 Point-Supported Membrane

This example presents an ordinary point-supported membrane. In the first step, a model consisting of cables, a membrane, and supports is created in RFEM (see Figure 3.1).

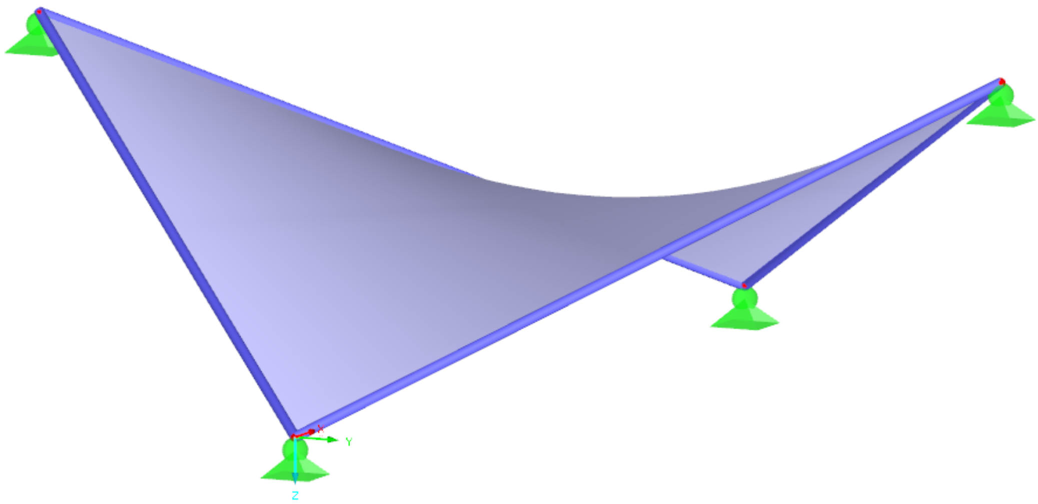
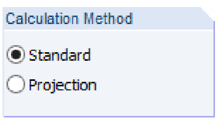


Figure 3.1: Model of point-supported membrane



The calculation method can be defined in the *Edit Surface Stiffness* dialog box (see Figure 3.2). In general, the *Standard* (tension) method is advantageous for membranes that are supported by points or arches, or for pneumatically stabilized membranes, while the *Projection* method is appropriate for conical membranes. For this reason, the default **Standard** method is adequate.

We define values of prestress for the *Membrane* surface and for the *Cable* members. The membrane requires an isotropic prestress of 1.0 kN per meter of width (see Figure 3.2). For an isotropically prestressed shape, the warp and weft directions are not significant. The cables require an average prestress of 10 kN (see Figure 3.3).

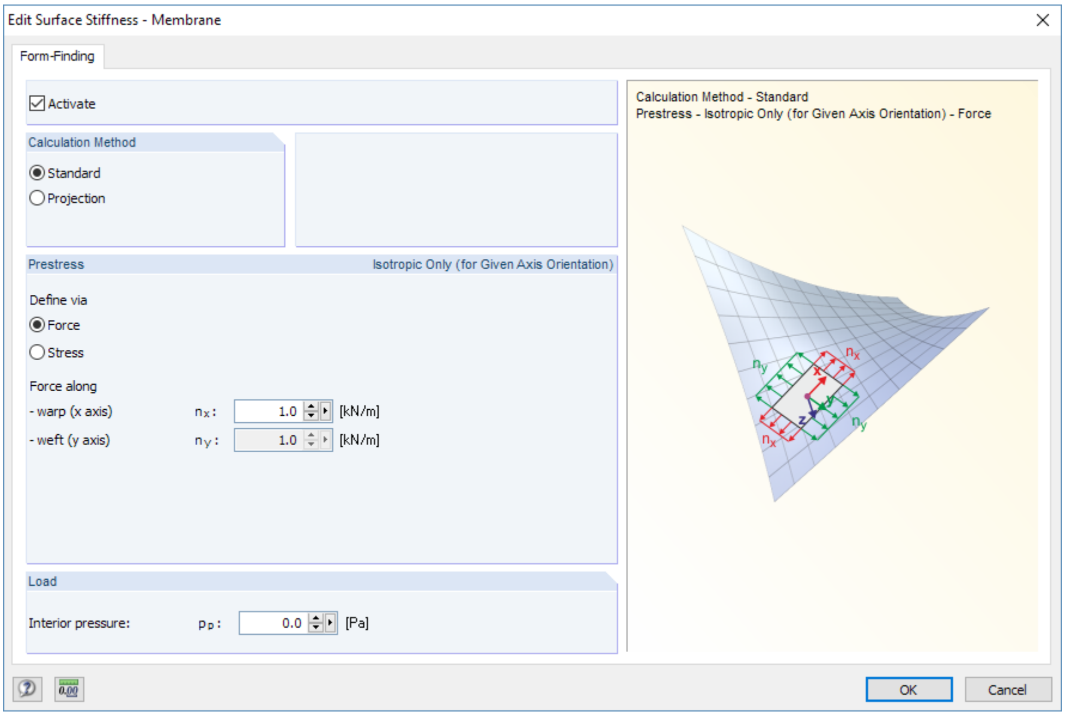


Figure 3.2: Prestress defined for membrane

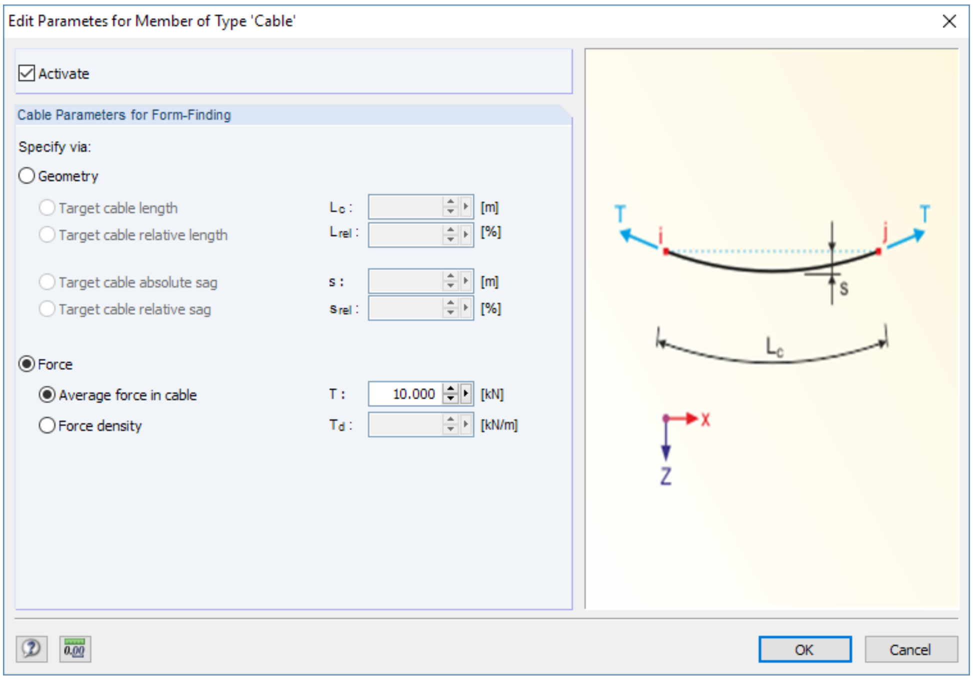


Figure 3.3: Prestress defined for cables

RF-FORM-FINDING

As soon as the model has been created and the input parameters relevant for form finding have been defined, the calculation of the RF-FORM-FINDING case can be started.

As a result of the form finding analysis, a new shape of the structure is created.

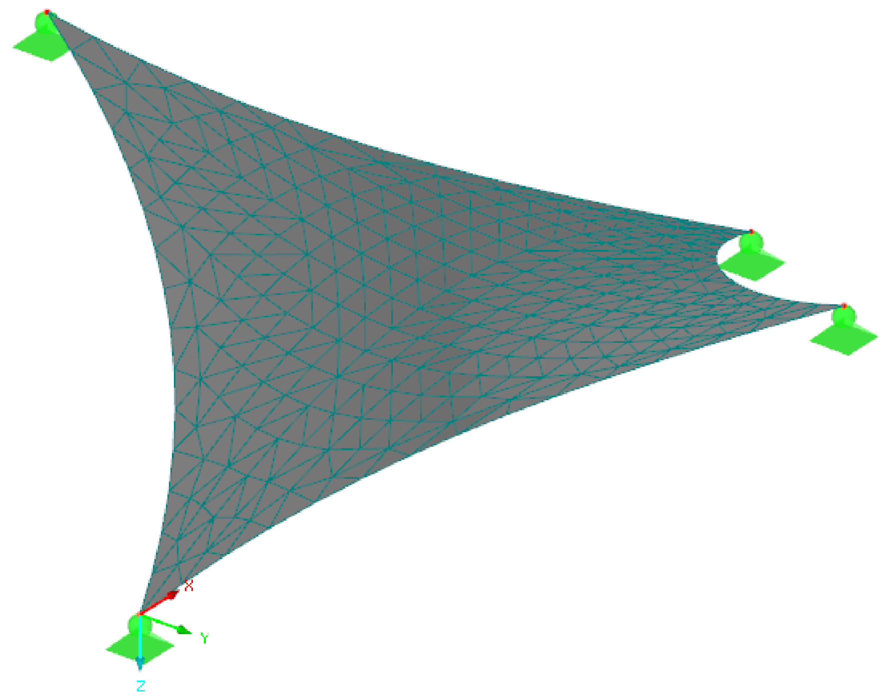


Figure 3.4: New shape of structure

Figure 3.5 shows the of internal forces of the membrane and of the cables. Additionnally, it is possible to display the support reactions.

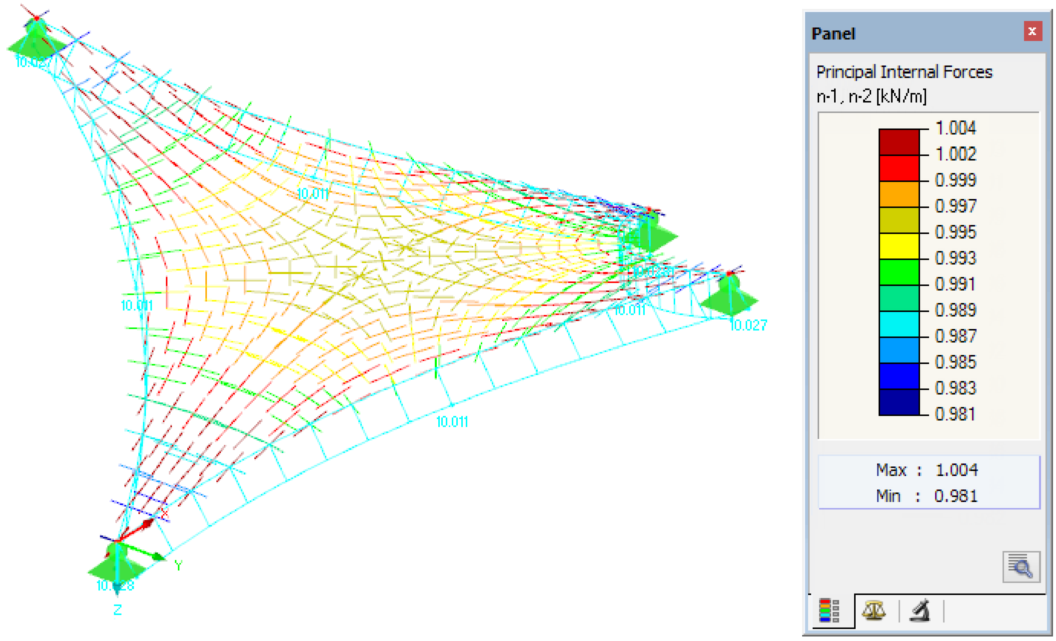


Figure 3.5: Vectorial display of principal internal forces in membrane and axial forces in cables

Figure 3.5 shows that the resulting prestresses already correspond fairly well with the initial values. As the isotropic prestress is a spatial equilibrium prestress, the approximation can yet be improved by modifying the default settings in the *Form-Finding* tab of the *Calculation Parameters* dialog box (see Figure 2.2). Figure 3.6 shows the resulting internal forces after having increased the *Number of iterations for loading prestress*.

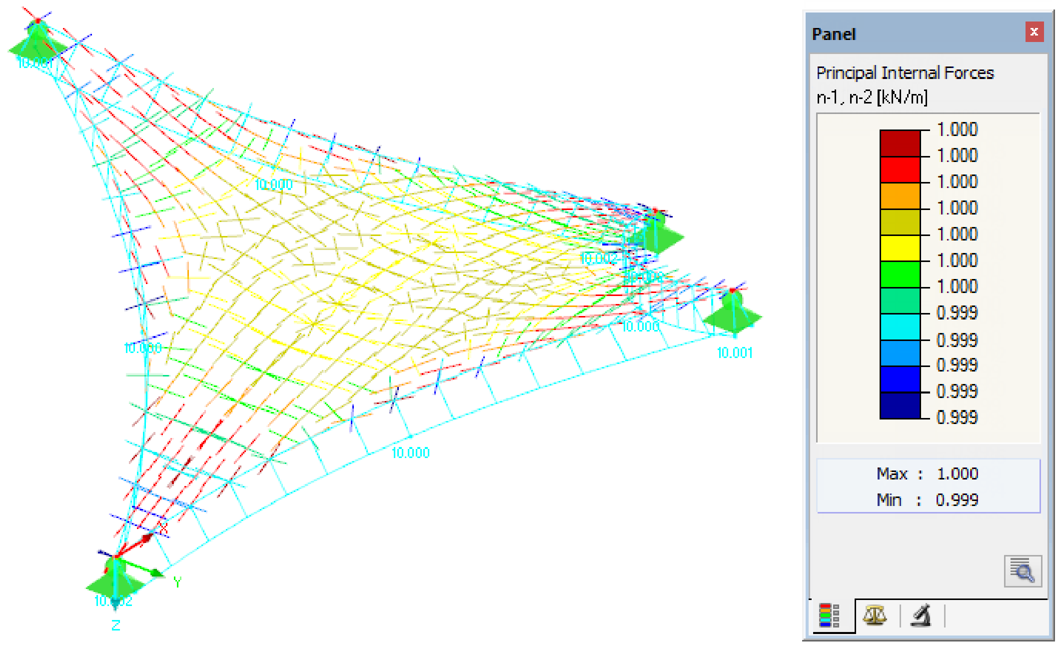


Figure 3.6: Resulting prestress due to increased number of iterations

The prestressed membrane structure can subsequently be subjected to a LC/CO analysis. We can create load cases, define loads, and start the calculation. The load cases will be calculated on the basis of the generated prestressed shape. Thus, the form finding can be considered as the first phase of the analysis, the calculation of all other load cases as its second phase.

If the structural analysis proves the isotropic prestress to be inappropriate for the structure, an orthotropic prestress can be defined instead. For it, the direction of the axis system has to be adjusted accordingly in the *Axes* tab of the *Edit Surface* dialog box first.

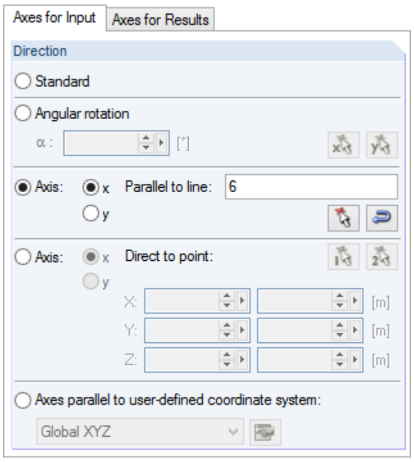


Figure 3.7: Dialog box *Edit Surface*, tab *Axes* (detail)

After that, the input values can be adjusted in the dialog boxes of membranes and cables. In our example, we set the prestress to 2 kN/m in the *warp* and to 1 kN/m in the *weft* directions. For the orthotropic prestress, the warp and weft directions are controlled by the axis system of the surface. The border cables are to be prestressed by average forces of 15 kN now.

In the *Edit Surface Stiffness* dialog box (see Figure 3.2), the text box of the weft direction is available now. If the same value as for the warp direction is set (isotropic prestress), a spatial force equilibrium system is defined. By increasing the precision and number of iterations for the loading prestress, we will obtain more exact results. The size of the FE mesh elements affects the precision to a certain extent as well: in general, the shape in space corresponding with the equilibrium prestress approximates better with a refined mesh.

We define a constant orthotropic prestress by setting the prestress to 2 kN/m in the warp and to 1 kN/m in the weft directions. It cannot be expected that an equilibrium prestress in space is reached by this. Therefore, the selected prestress is applied to a limited number of iterations only – until the membrane structure is stabilized. Thus, the program is in most cases able to find a shape that approximates well to the specified values (see Figure 3.8).

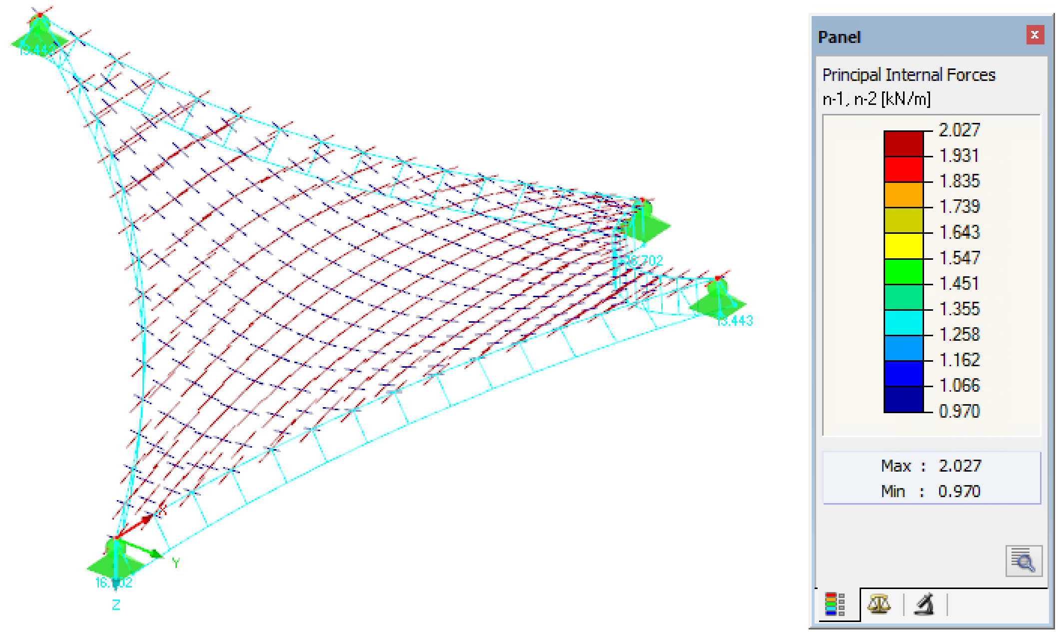


Figure 3.8: Orthotropic prestressed membrane

Regarding the *Number of iterations for loading prestress* of a model, the following is recommended:

- If there is an <u>isotropic</u> prestress, an increased number of iterations will create a shape that better and better approximates to the force equilibrium system in space.
- For a constant <u>orthotropic</u> prestress (i.e. the spatial force system is not in equilibrium), the effect of the prestress must be stopped at the right moment. It will be then when the deformation represented in the *Maximum Displacement* diagram stops rising distinctly (see Figure 3.9).

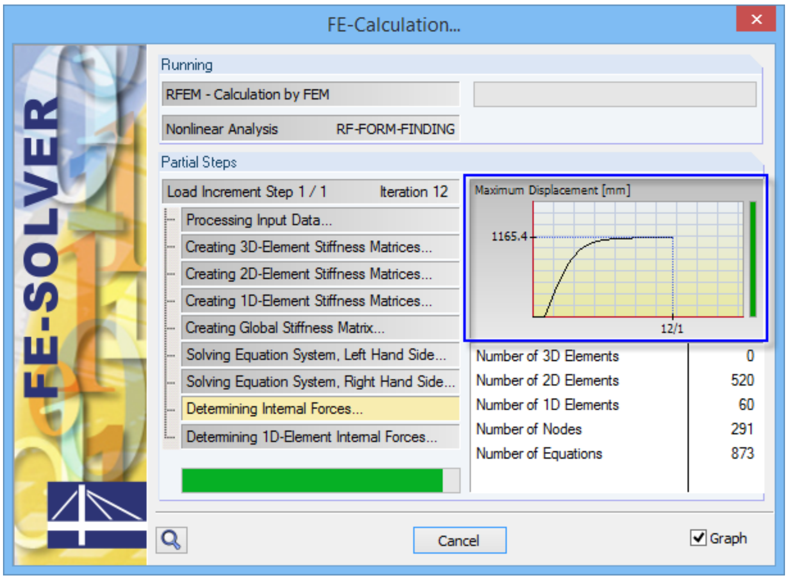


Figure 3.9: Graphical display of deformations during form finding process

Deformations perpendicular to the plane of the membrane (usually in the steep part of the curve) are far more frequent than those within the plane of the membrane. As soon as the increase of the deformation falls off, the program has usually found the shape that corresponds approximately to the required orthotropic prestress. After the last application of the constant orthotropic prestress, the structure is stabilized. Thus, an **equilibrium prestress** is reached which is very close to the specified values.

The equilibrium shape is completely independent of the stiffness of the used materials. The shape of a flexible structure clearly results from the defined boundary conditions and the equilibrium prestress, or it is defined by the boundary conditions and the equilibrium system of loading prestress and loads (overpressure, loads of the *Form-Finding* category load case). The used material only affects the resulting shape when its self-weight is considered for the form finding. This load has only little influence in general, however.

Therefore, it is recommended not to activate the self-weight at the beginning of the form finding process. This way, you can find the prestressed shape that meets your expectations first. Then select the appropriate material, activate the consideration of the self-weight in the *Form-Finding* load case, and start the calculation once again.

Similarly, the final shape is independent of the initial approximation of the shape. In other words, it does not matter how "precisely" the initial structure is modeled. It is only important to adequately arrange the supports and define the loading prestresses. Under these conditions, the final shapes of different initial models will be the same (see Figure 3.10 and Figure 3.11).

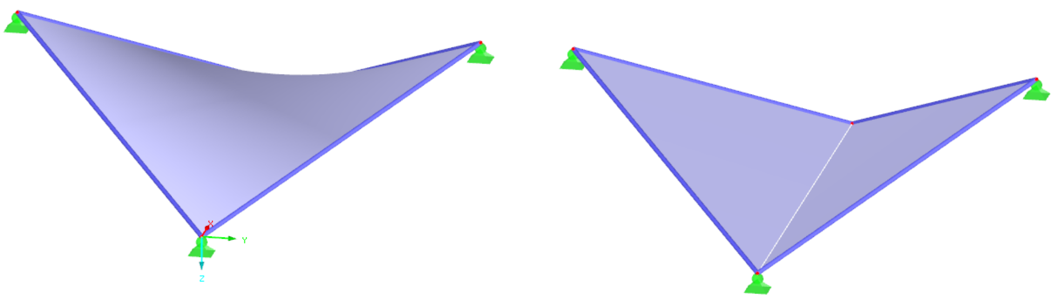


Figure 3.10: Models featuring two different initial shapes

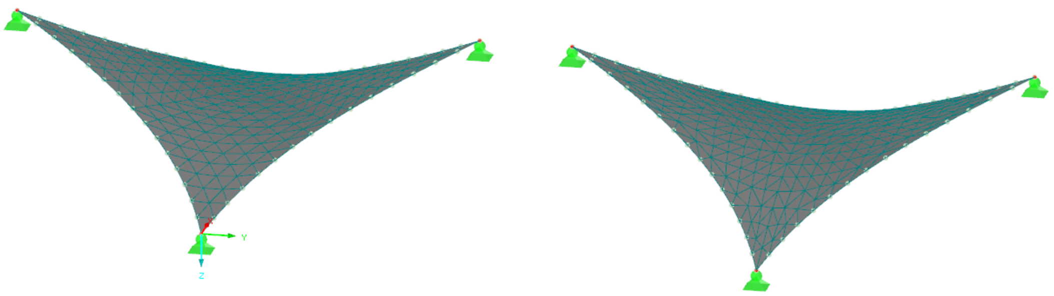


Figure 3.11: Identical shapes after form finding process

Decisive for the final shape are not the absolute values of prestresses, but their ratios: Irrespective of whether you define an isotropic prestress of 1 kN/m for the membrane and an average prestress of 10 kN for the cables, or a prestress of 50 kN/m for the membrane and an average prestress of 500 kN for the cables, the result will be an identical shape for both versions. This does not apply, however, if semi-rigid beams are set as boundary conditions. Those will be deformed differently when varied prestresses are applied to the membrane (see example in Chapter 3.2).

3.2 Arch-Supported Membrane

This example presents an arch-supported membrane. The structure consists of three membrane panels and four steel arches. The membranes are laterally connected to the beams and anchored at their bottom edges. The steel arches are fixed in the foundation (see Figure 3.12).

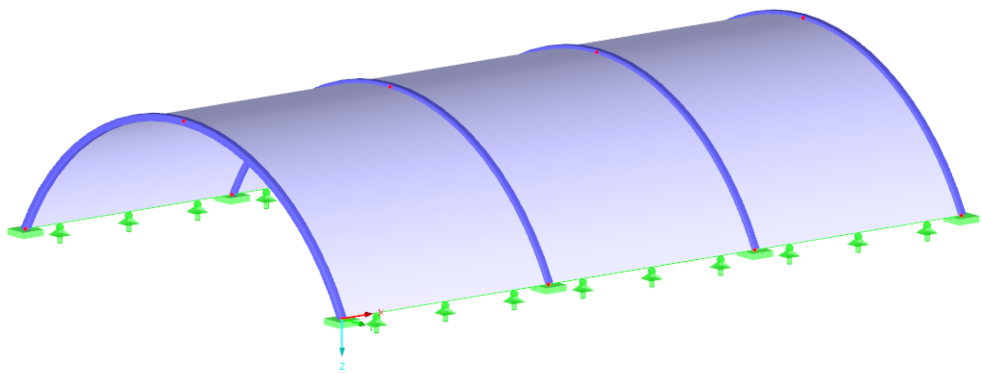
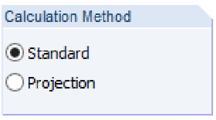


Figure 3.12: Model of arch-supported membrane

In the first step, a model is defined in RFEM representing an initial approximation to the final shape. For example purposes, there is a different loading prestress for each membrane panel. The first panel is defined with an isotropic prestress of 1 kN/m, the second and the third panel with an orthotropic prestress. The loading prestress in the second panel is 1 kN/m in the warp direction and 2 kN/m in the weft direction while the third panel has a loading prestress of 2 kN/m in the warp direction and 1 kN/m in the weft direction.



In the *Edit Surface Stiffness* dialog box, the calculation method for form finding can be defined for each of the three surfaces (see Figure 3.2). In general, the **Standard** (tension) method is advantageous for arch-supported membranes. In the *Calculation Parameters* dialog box (see Figure 2.2 on page 7), the number of iterations for the loading prestress can be set.

As there is an obvious correlation between the loading prestress and shape, different membrane shapes result due to varied prestress values for the panels (see Figure 3.13 and Figure 3.14).

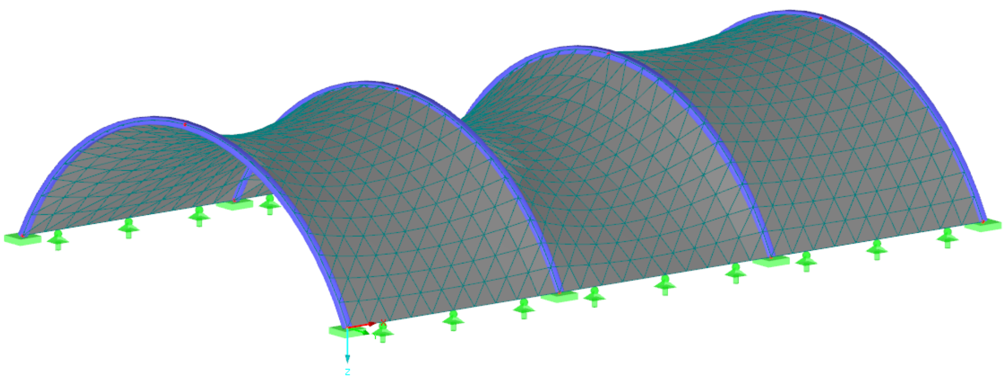


Figure 3.13: Final shape of membrane structure - isometric view

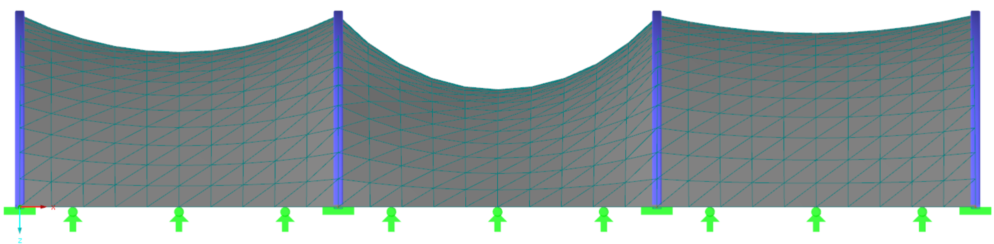


Figure 3.14: Final shape of membrane structure - view in -Y

Figure 3.15 shows the internal forces on the generated shape of the membrane structure. Again, there is a good approximation of the resulting prestress to the required prestress.

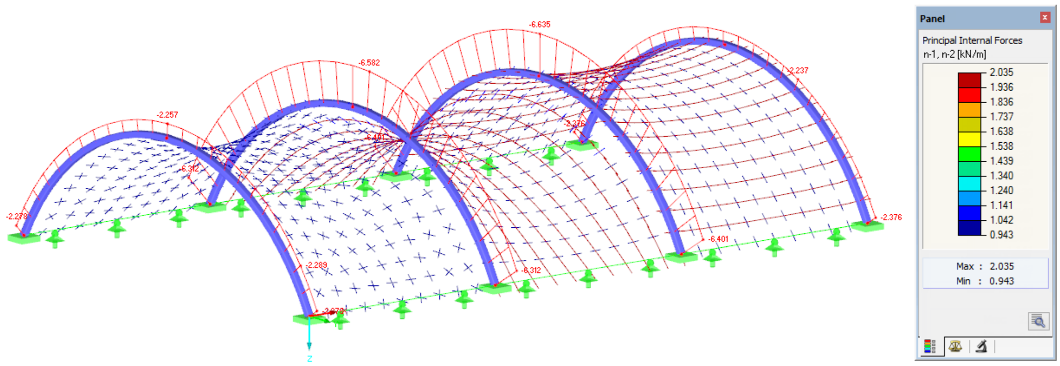


Figure 3.15: Vectorial display of principal internal forces in membrane and axial forces in beams

The steel arches are subjected to the form finding process, too. They represent flexible boundary conditions for the membrane surfaces. The defined loading prestress is applied to the membranes, which the beams must resist. Thus, the result of the form finding is not only the prestressed shape of the membranes, but also the distorted shape and the internal forces of semi-rigid parts of the structure. Due to the loading from the membranes, the steel arches deform according to their stiffnesses.

If you wish to exclude the steel arches from the form finding process, it is possible to support them by temporary form finding supports (see blue supports in Figure 3.16).

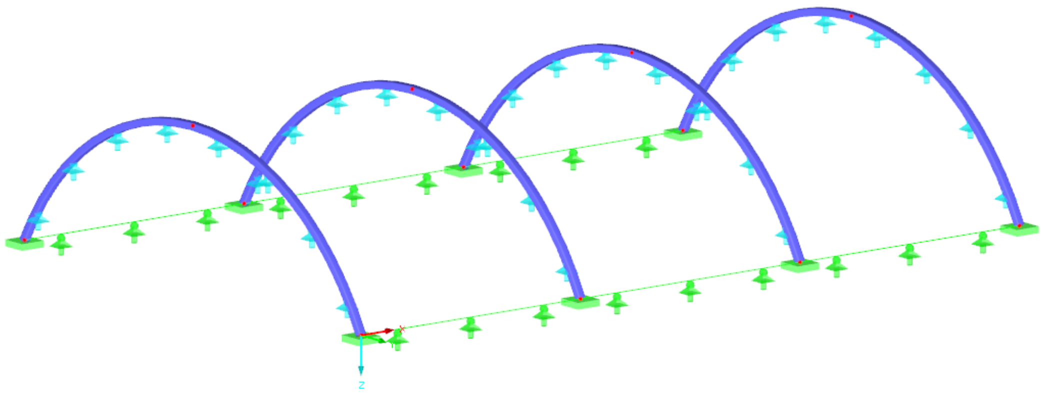


Figure 3.16: Arches with temporary supports for form finding

Those supports have the nonlinear property of *Form-finding stage only* (see Figure 3.17). They are effective for the form finding process only and will not be considered for other load cases – similarly to constructional supports. All loads which the temporary supports have to resist for the form finding will then be absorbed by the steel arches.

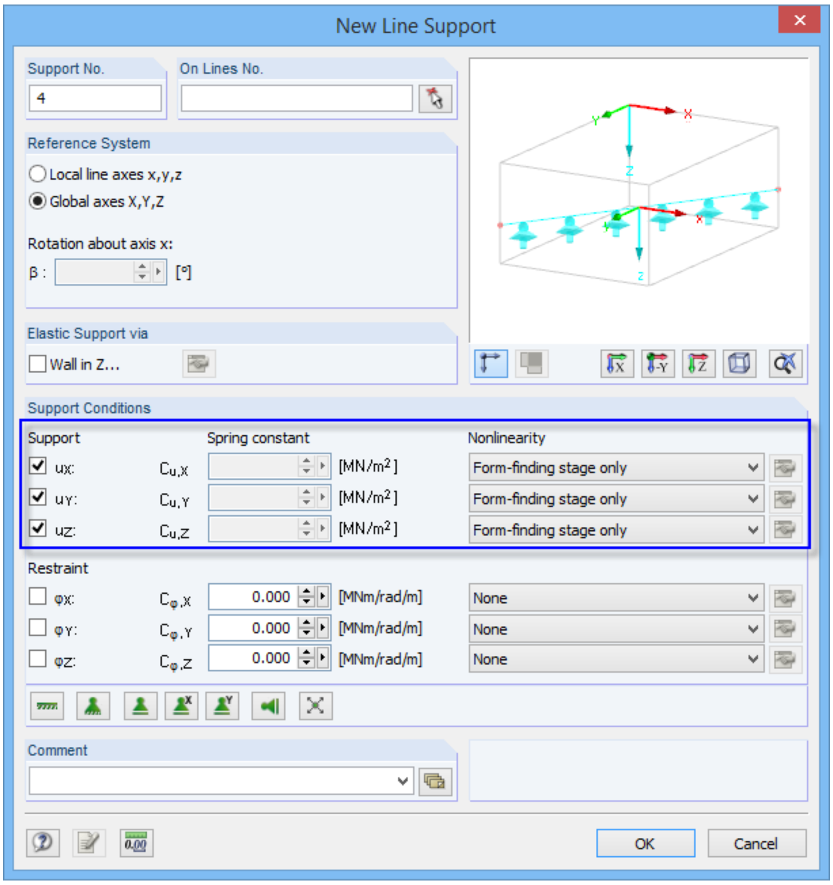


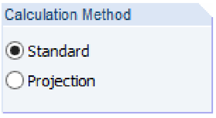
Figure 3.17: Temporary line support for *Form-finding stage only*

It is up to you whether semi-rigid parts of the structure are to be considered for the form finding process or ignored by means of temporary supports. You can make your decision depending on the procedure of assembly.

- If there are no constructional supports for the restrained arches during the assembly, they will deform when the membrane is tightened. This corresponds to the form finding process when no temporary supports are used. The loading prestress which is available after tightening the structure corresponds to the prestress resulting from the form finding process.
- If the beams are supported by construction supports while tightening the membrane, it can be simulated by temporary support for *Form-finding stage only*. When the membrane has been tightened, support reactions occur in the constructional supports. As soon as the supports are removed, the steel arches must resist those forces and will be deformed. Hence, the internal forces in the membrane will be modified as well. This corresponds to the analysis of the structure supplied with temporary supports when starting an "ordinary" load case which has no loads: as the temporary supports are deactivated in this step of form finding, the beams are subjected to all loads and will deform. The deformation of the beams also affects the loading prestress in the membranes. This constitutional change in RFEM due to the eliminated temporary supports corresponds to the process of removing the constructional supports.

3.3 Pneumatically Supported Membrane

This example presents a pneumatically stabilized membrane. The shape of the membrane is determined by the boundary conditions, a loading prestress and internal pressure. For illustration, two membrane structures with different prestress conditions are modeled in the global plane XY.



For this type of membrane structures, the **Standard** (tension) method is advisable.

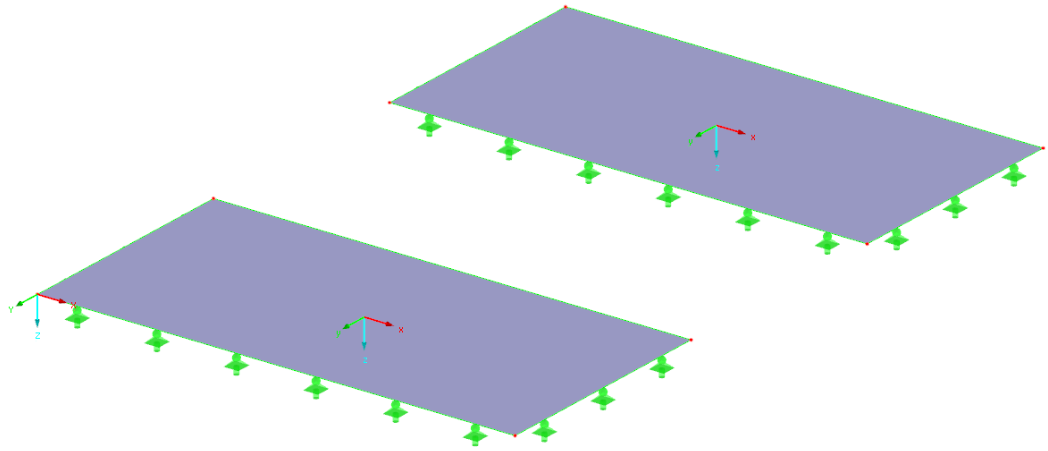


Figure 3.18: Initial geometry of pneumatic membranes

The models will be deformed into an equilibrium position according to the defined values of loading prestress and internal pressure. The model to the left has an isotropic prestress of 1 kN/m, the model to the right orthotropic prestresses of 2 kN/m in the warp direction and of 1 kN/m in the weft direction (see Figure 3.19).

To apply the internal pressure, two alternatives are available: it can be defined either as *Interior Pressure* (static load) or as a *Solid Load* (gas).

3.3.1 Internal Pressure as Static Load

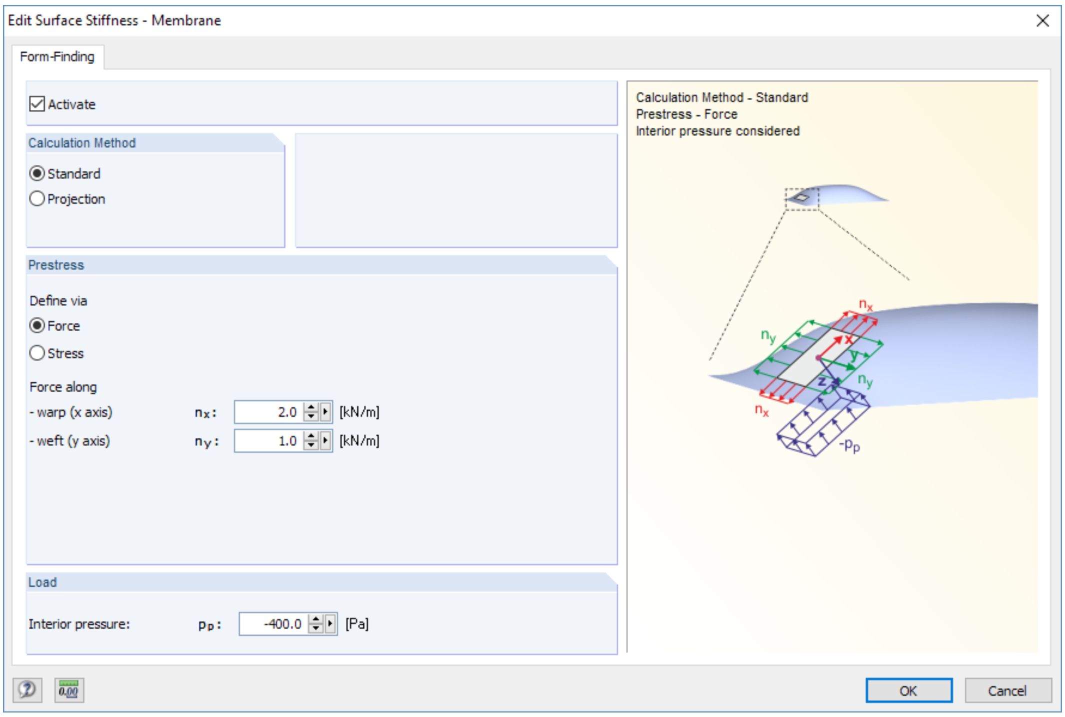


Figure 3.19: Definition of orthotropic prestress and internal pressure

The overpressure of —400 Pa is identical for both models (see Figure 3.19). Positive internal pressure acts in the direction of the local z-axis of the surface, negative interior pressure against the local z-axis of the surface. In our example, the pneumatic models are to be deformed upwards, i.e. against the local z-axis of the surface, which means that the load is entered with a negative sign.

When the calculation has been started, the models converge against the equilibrium shapes.

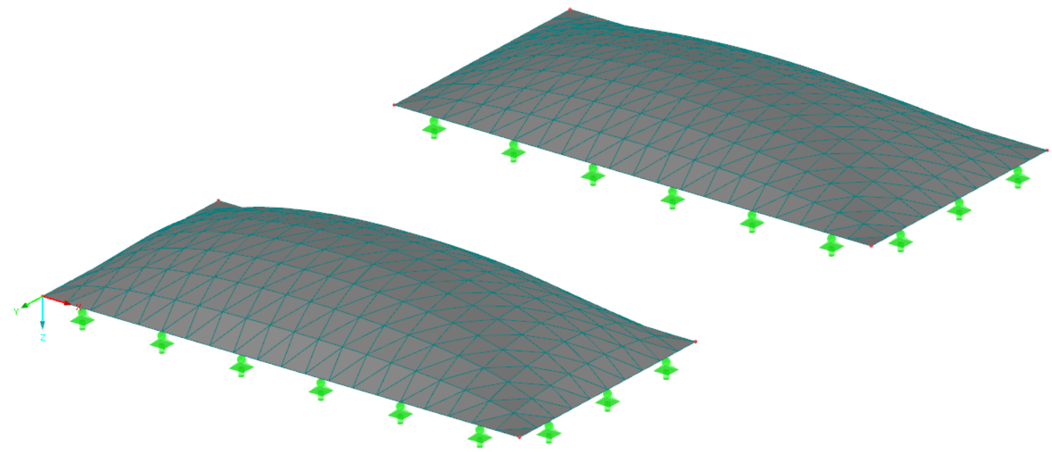


Figure 3.20: Final shapes of membrane structures

Figure 3.21 shows a good approximation of the resulting prestresses to the required prestresses of the membrane structures. Note the difference between isotropic and orthotropic features of the two surfaces.

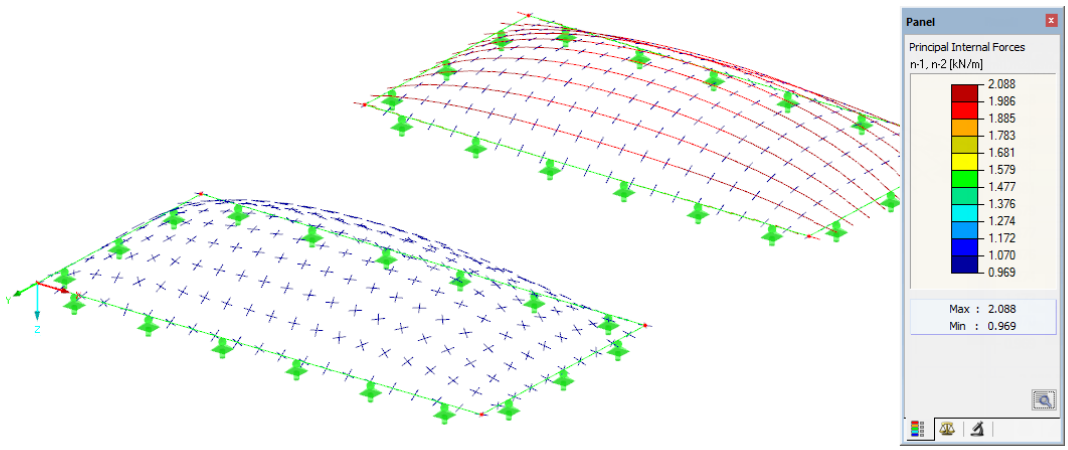


Figure 3.21: Vectorial display of prestresses in membranes

3.3.2 Internal Pressure as Gas Solid

To apply a gas solid between surfaces, the initial models have to modified. The plane initial surfaces each have to be changed into two adequate surfaces which allow for the definition of solids.

In our example, we define *B-Spline* geometrical shapes enclosing the gas.

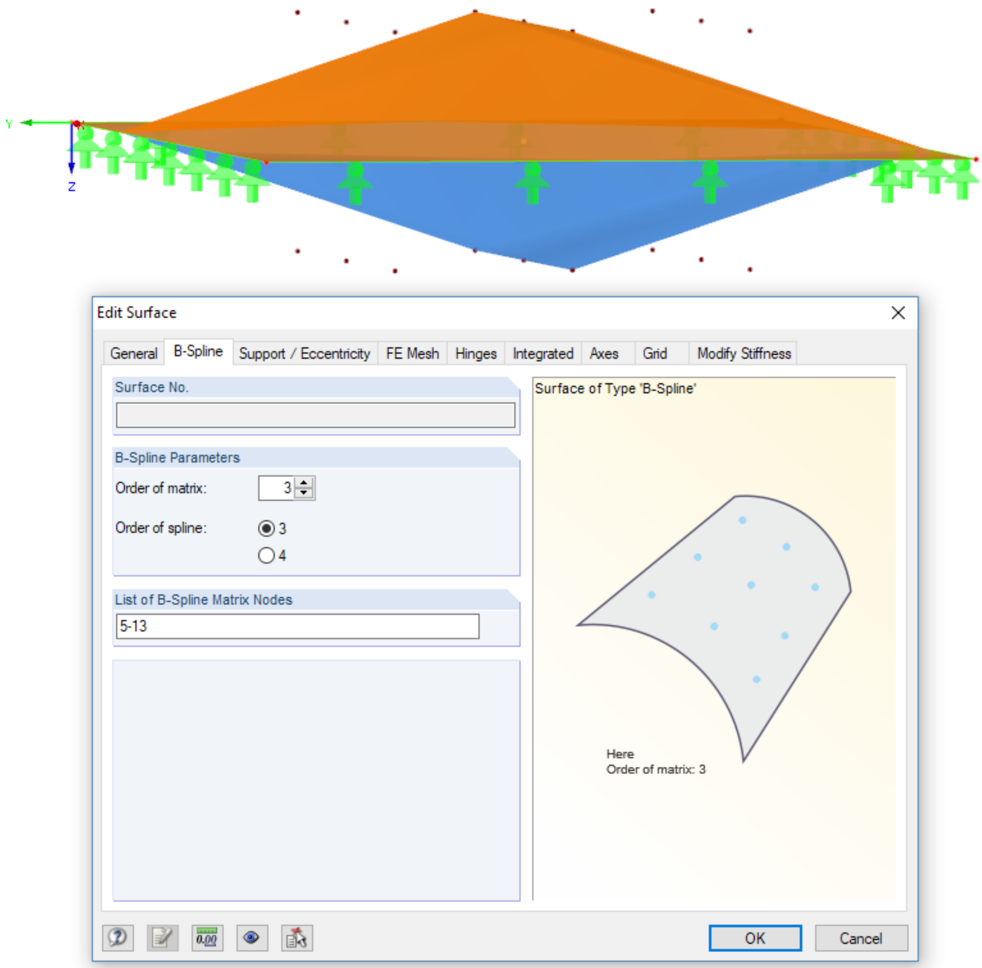
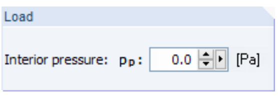


Figure 3.22: Definition of *B-Spline* Surface



In the *Edit Surface Stiffness* dialog box (see Figure 3.19), we set the *Interior pressure*, p_p, to zero for every surface.

We define two solids of the *Gas* type between the membrane surfaces (see Figure 3.23). As material, we can select *Dry Air* from the library.

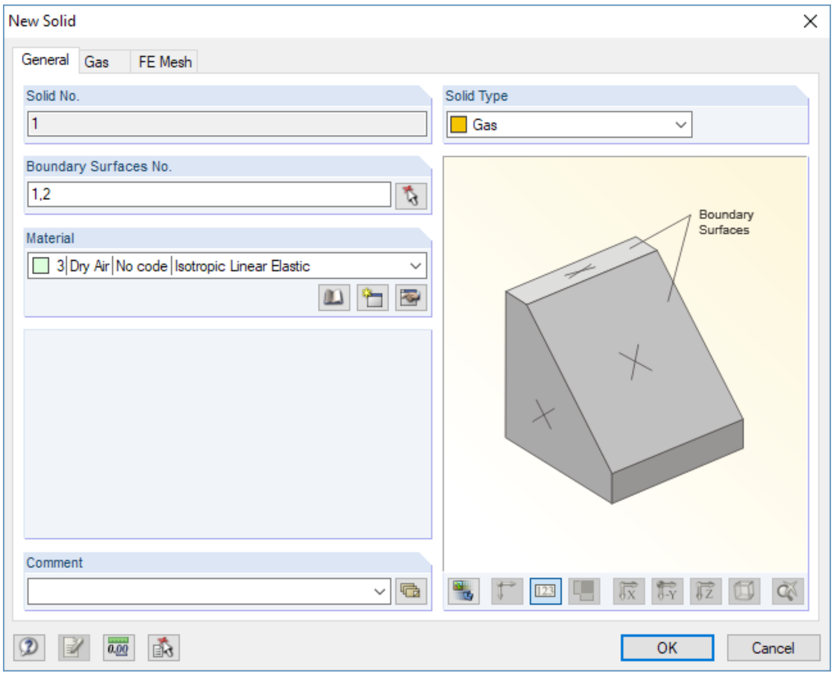
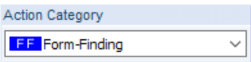


Figure 3.23: Dialog box *New Solid*



Next we create a new load case whose *Action Category* is to be set to **Form-Finding** (see Figure 2.13). We deactivate the self-weight.

For the two solids, we define a new solid load each. We select the *Gas* load type and enter its magnitude of 400 Pa.

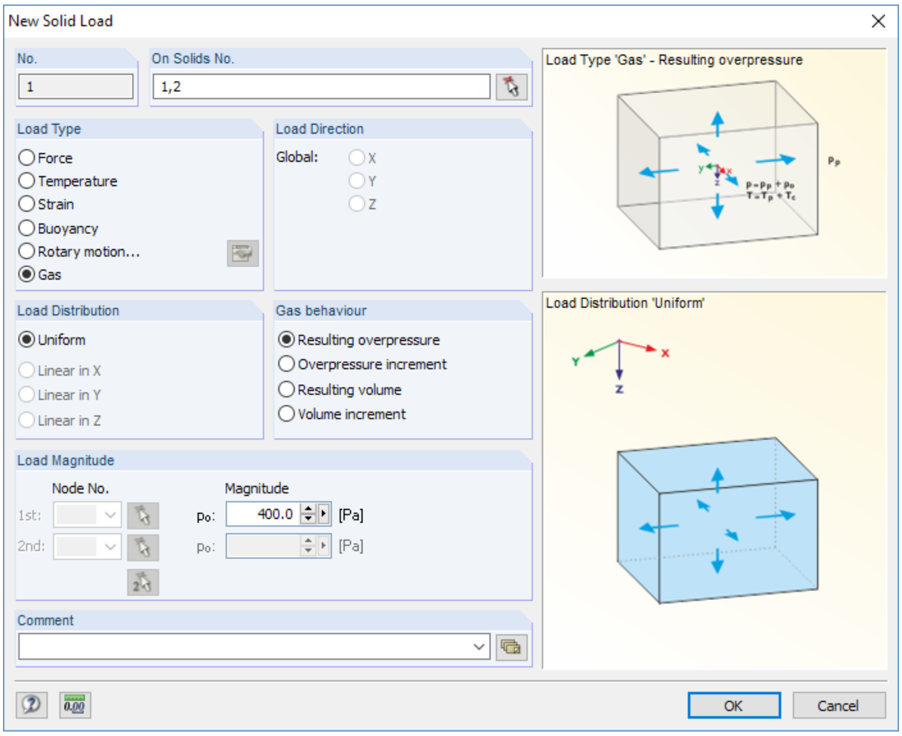


Figure 3.24: Dialog box *New Solid Load*

After the form finding analysis, the shapes of both models have been found. As can be seen in Figure 3.25, the deformation of the membrane structure with the higher (orthotropic) prestress turns out to be slightly less.

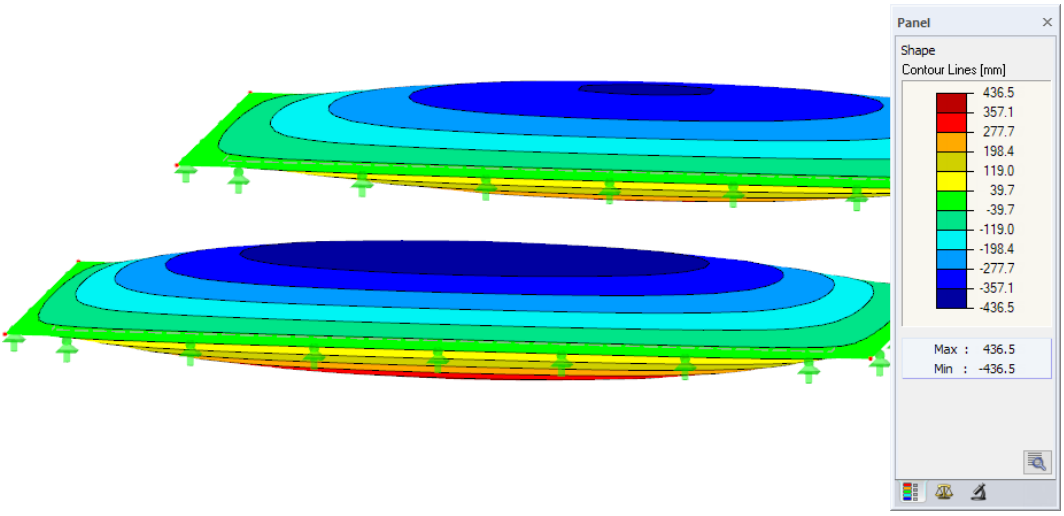


Figure 3.25: Final shapes of membrane structures

The prestresses in the membrane are fairly close to the ones of the model which has been analyzed in the previous chapter (see Figure 3.21).

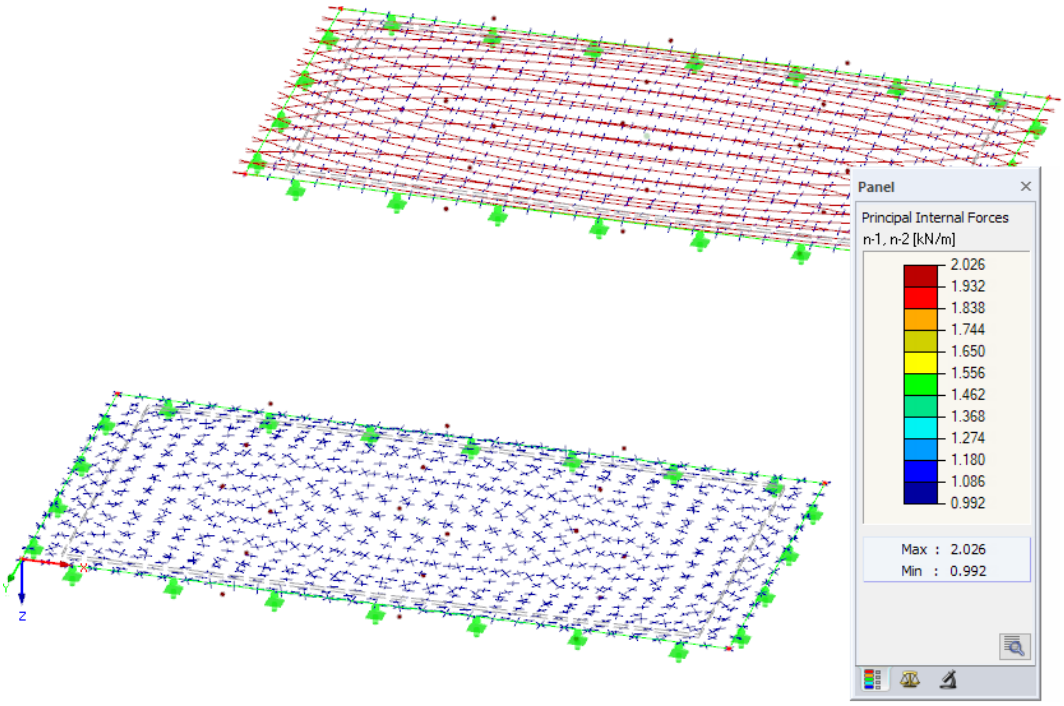


Figure 3.26: Vectorial display of prestresses in membranes

3.4 Conical Membrane

The last example presents a conical membrane structure. The inital model, i.e. the first approximation to the shape of the structure, consists of several membrane surfaces and cables, a peak ring, and a central pylon.

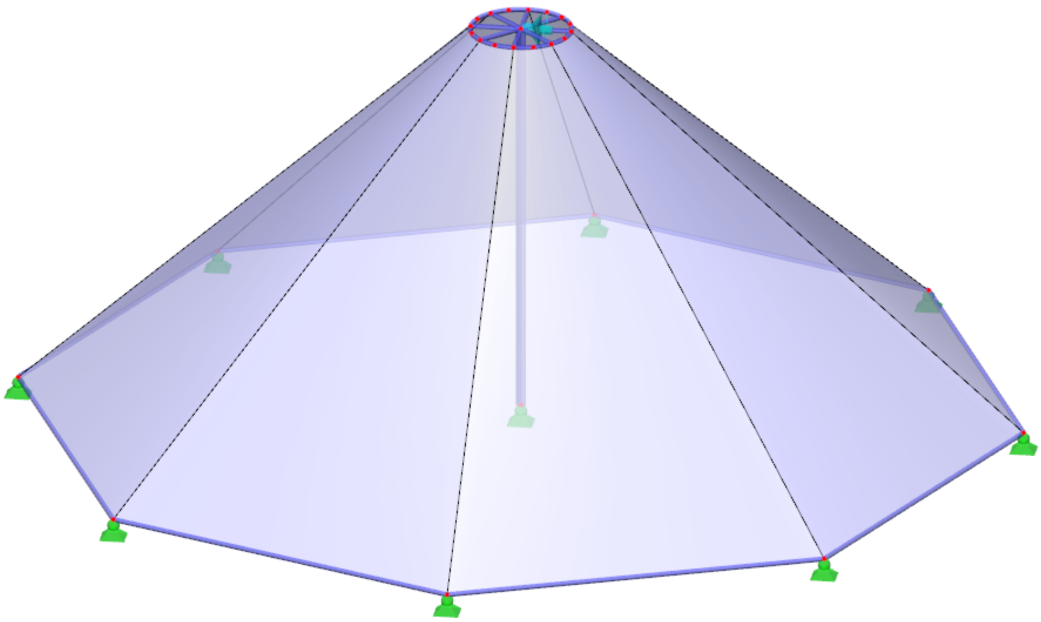
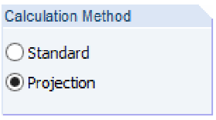


Figure 3.27: Initial geometry of conical membrane structure



For conical membranes, the **Projection** method is recommended for the form finding process.

As described in Chapter 1.4.1 on page 4, the projection method is based on a loading prestress predefined in the projection plane which is then adjusted according to the inclination of the membrane. If the inclination of the membrane increases, the prestress in the directions of the fall lines increases as well, while the prestress in the directions of the contour lines decreases. If an isotropic prestress is defined in the projection, the result will be a general prestress of the membrane structure in space.

The central pylon bearing the membrane structure is simply supported in its footing. This type of support allows for the pylon to tilt during the form finding process as well as when tightening the structure. Please note that it is necessary to secure the pylon for both the form finding analysis and the construction phase, e.g. by a *Form-finding stage only* type (provisional support) at its top (see Figure 3.27). It would also be possible to use some other temporary support, e.g. a rotational restraint at the footing. In any case, the semi-rigid part of the structure must be stable during the form finding and construction phases.

When the initial model has been created, the required prestresses are to be defined. An isotropic prestress (in projection) of 1 kN/m is assigned to the membranes. The cables are prestressed by an average force of 10 kN each.

After the form finding analysis, the equilibrium shape shown in Figure 3.28 is displayed.

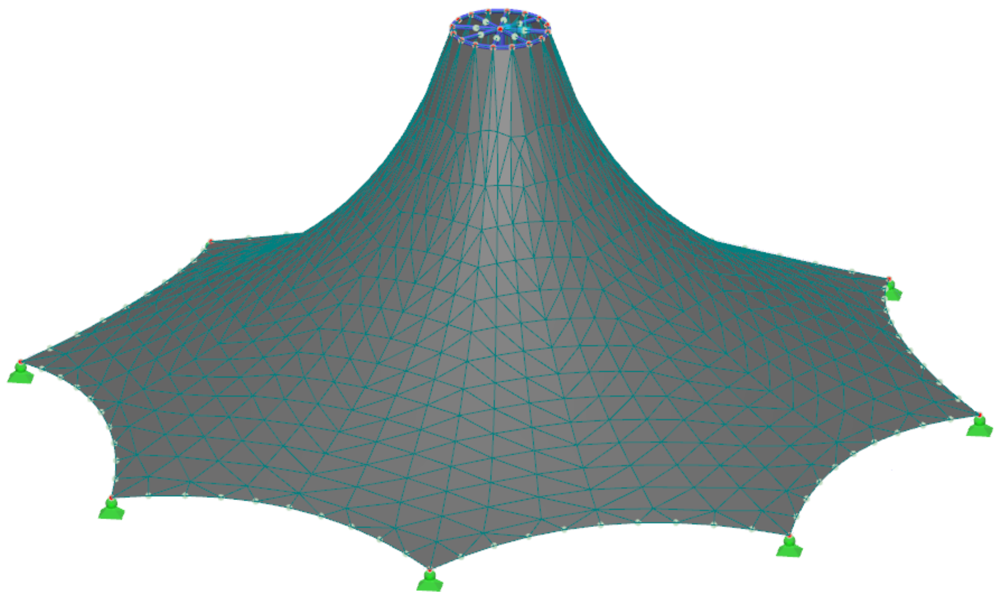


Figure 3.28: Final shape of membrane structure

The resulting prestresses are shown in Figure 3.29.

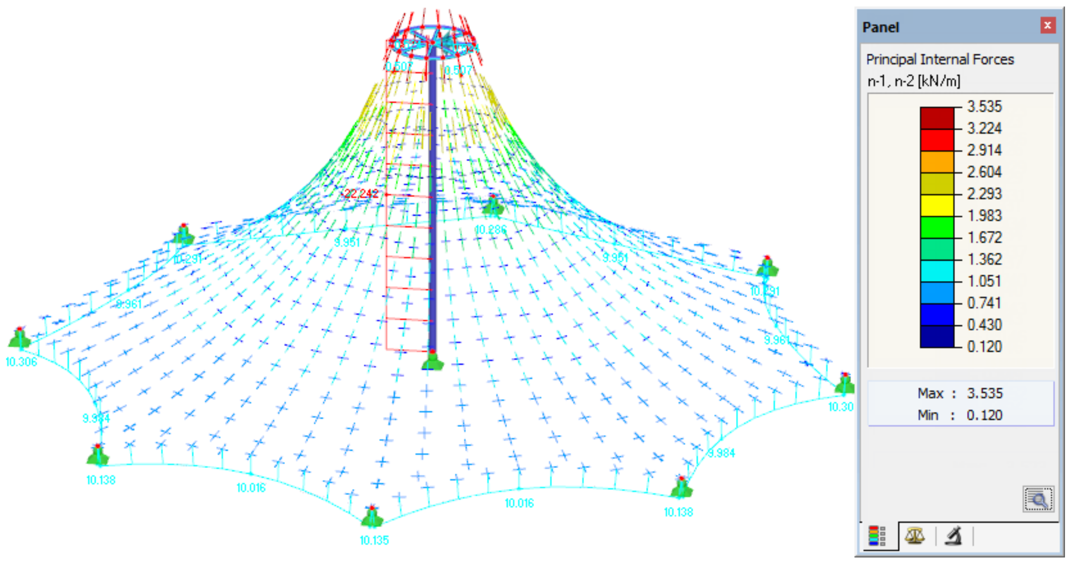


Figure 3.29: Vectorial display of prestresses in membranes

As already mentioned, an isotropic prestress defined in the projection leads to a general orthotropic prestress in the spatial position of the membrane. The loading prestress depends on the inclination in the respective location. In bottom parts of the membrane where the inclination is small, the loading prestress approximates to the isotropic prestress (predefined in the projection), while the loading prestress on top of the cone is changed significantly due to the strong inclination.

The loading prestress can be adjusted in the *Edit Surface* dialog box, if necessary: you can define a radial arrangement of the warp and weft directions (see Figure 3.30) and specify the prestress values accordingly (see Figure 3.32).

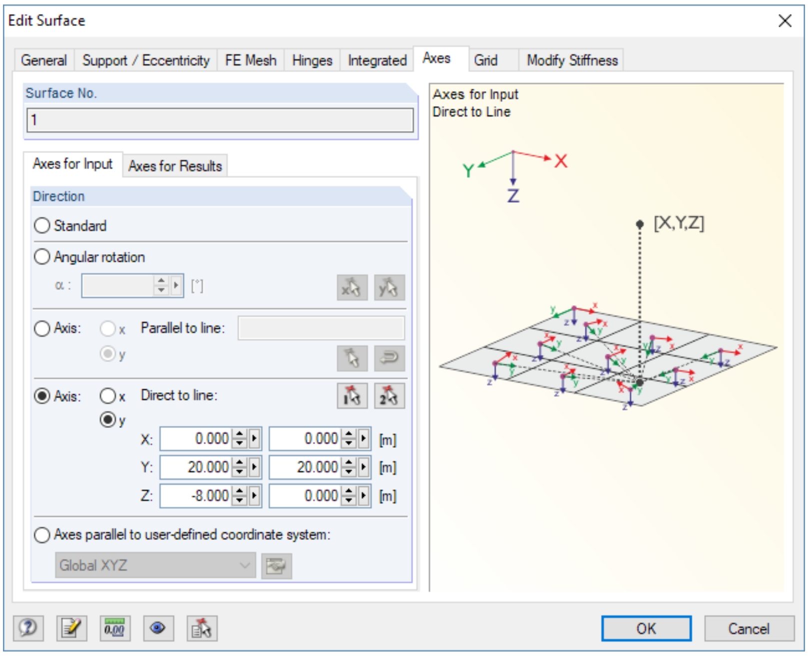


Figure 3.30: Radial arrangement of axes in *Edit Surface* dialog box, tab *Axes*

For radial arrangements, an orthotropic equilibrium prestress existing in the plane cannot have two constant values (in contrast to an orthogonally aligned prestress in the plane). Due to the radial arrangement, the radial forces are affected by the tangential forces: they increase the radial forces in the centric direction because of the curvature.

If you select an orthotropic prestress with radial arrangement, the values of the prestress in the warp and weft directions refer to the point in the projection which is located at a specific distance, *r*, from the center (see Figure 3.31). This radius is determined automatically halfway between the center and most distant point. It is relative to the reference plane and perpendicular to the axis of alignment. In all other distances, the radial prestress is calculated in such a way that it is at equilibrium in the plane. Thus, it is possible to project the force equilibrium system in the spatial position of the membrane.

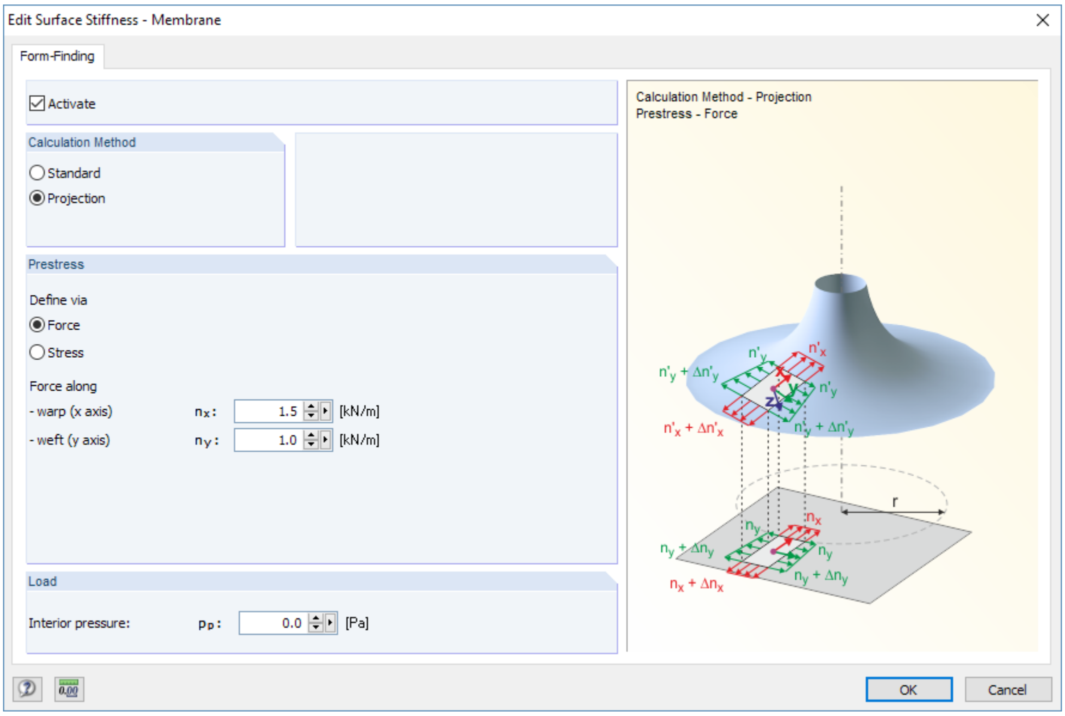


Figure 3.31: Radial prestress in *Edit Surface Stiffness* dialog box

The radial field of the prestress in the projection can be adjusted by means of the forces specified in warp and weft directions as well as by the distance of the defined prestress from the center. The prestress values available in the center, which are small at first sight, are rising concordantly with an increasing inclination of the membrane. This can be illustrated very well in the example of the conical membrane: the modification of loading prestress from isotropic to othotropic (see Figure 3.31) leads to the final shape of the membrane structure shown in Figure 3.32.

With different settings for prestress, different shapes will be generated, too.

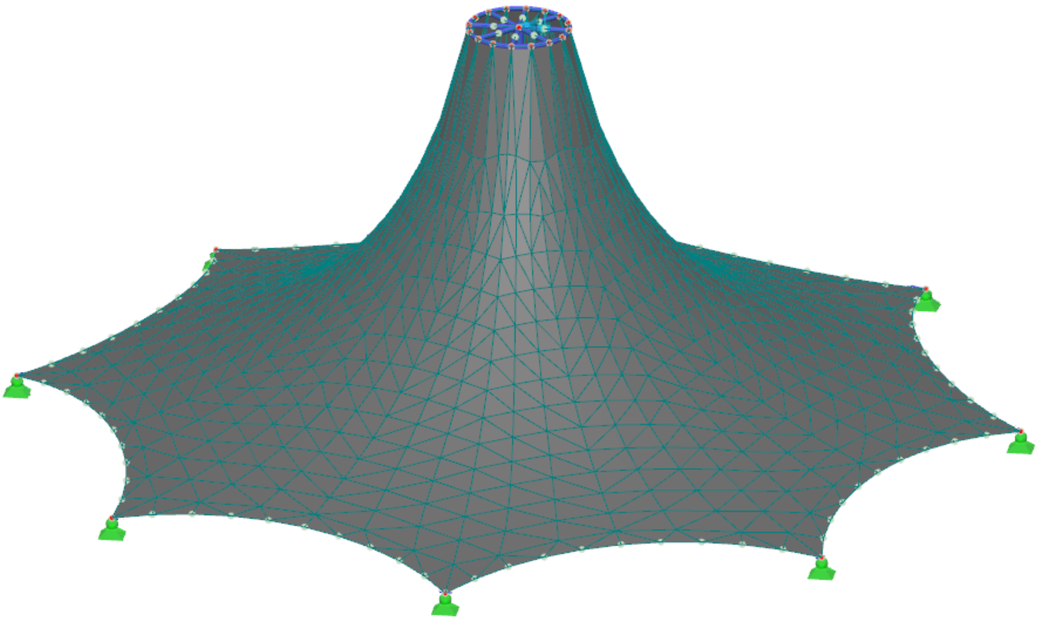


Figure 3.32: Final shape of membrane structure

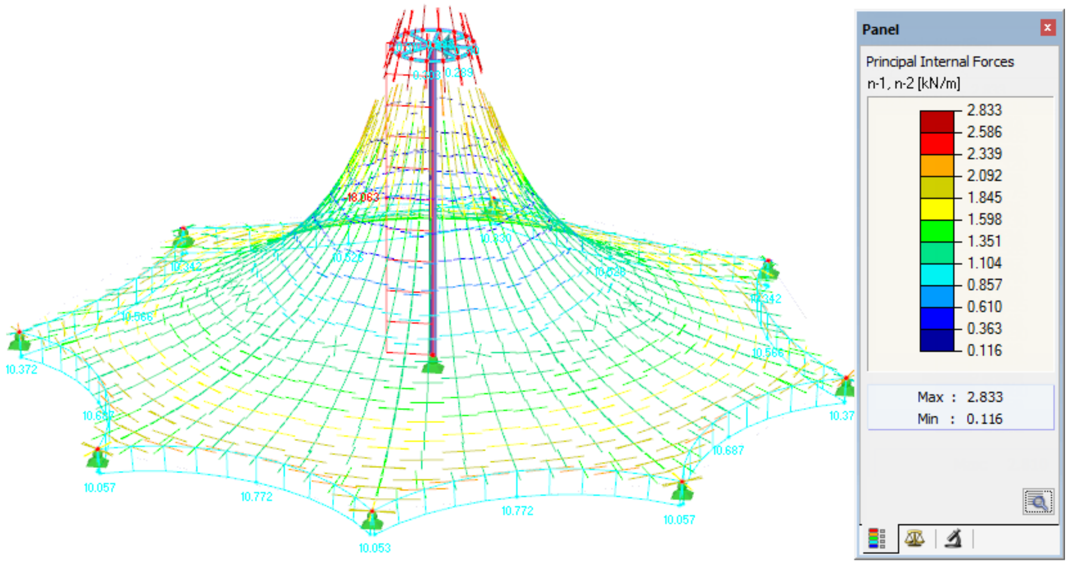


Figure 3.33: Resulting prestress

For further demonstration of the projection method, it would be possible to move the ring points of the conical membrane into a plane and to generate different prestresses in the projection. Then the position of the ring could be moved upwards to compare the changes of the prestress as a function of the inclination. For uniform specifications (isotropy in projection, for example), the resulting prestress in space would vary significantly.

The projection method is suited for high conical membranes as it prevents the membrane gorge from narrowing – an effect of the standard (tension) method. Figure 3.34 shows one of the results which may be created when the *Tension* calculation method is applied.

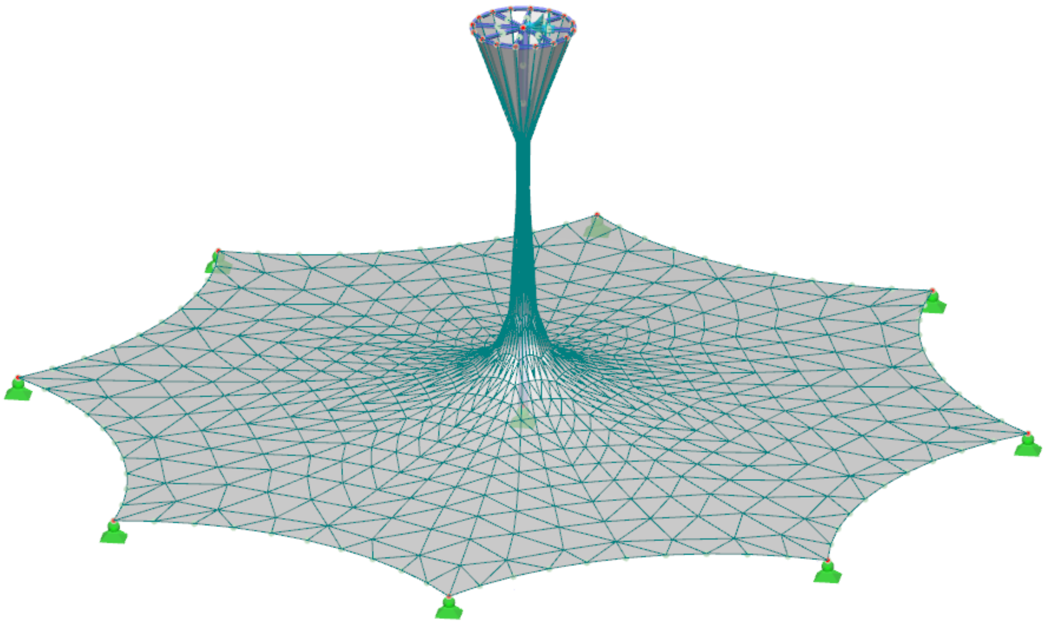


Figure 3.34: Result of standard (tension) method

The final shape is also affected by the number of iterations that have been set for the application of loading prestress.

4 General Notes

4.1 Check of Settings

If a membrane structure consists of several surfaces (see Chapter 3.4), it is assumed that the same values for prestress as well as identical warp and weft directions are defined for all surfaces. Otherwise RFEM will display a corresponding warning.

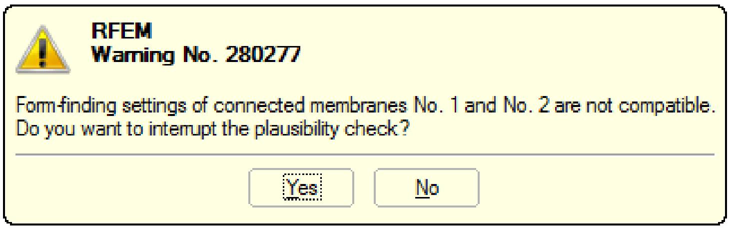


Figure 4.1: Warning

If there are independent membranes or if the individual surfaces are separated by supports, cables or beams, however, they are considered as independent surfaces. The compatibility of input data will not be checked in those cases.

Figure 4.2 shows the layout of an arch-supported membrane. The central panel is divided by a cable. Members are indicated in Roman, surfaces in Arabic numerals. RFEM wil check the compatibility for surfaces 1 and 2 and for surfaces 5 and 6.

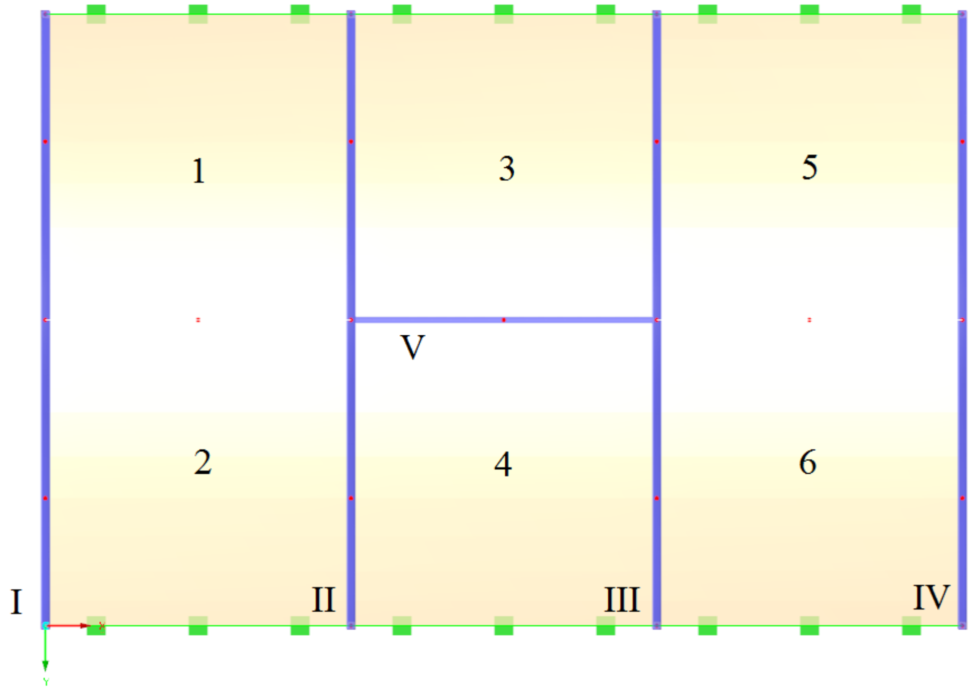


Figure 4.2: Surfaces of arch-supported membrane

4.2 Triangle Elements

For modeling membrane structures, it is recommended to generate the FE mesh with triangle elements. In the case of rectangle elements, warping effects occur. Therefore, membrane surfaces are automatically meshed with triangle elements.

The settings are managed in the *FE Mesh* dialog box that you can access by selecting **Calculate** → **FE Mesh Settings** on the RFEM menu bar.

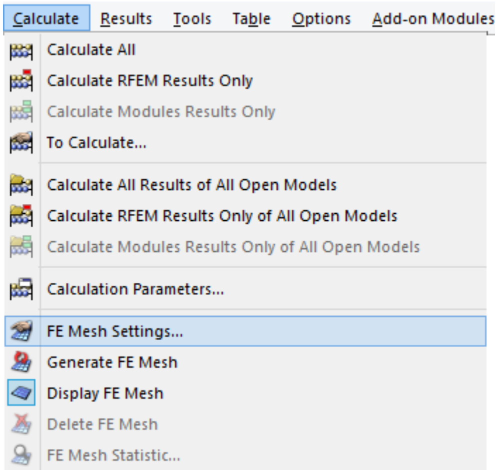


Figure 4.3: Opening *FE Mesh Settings*

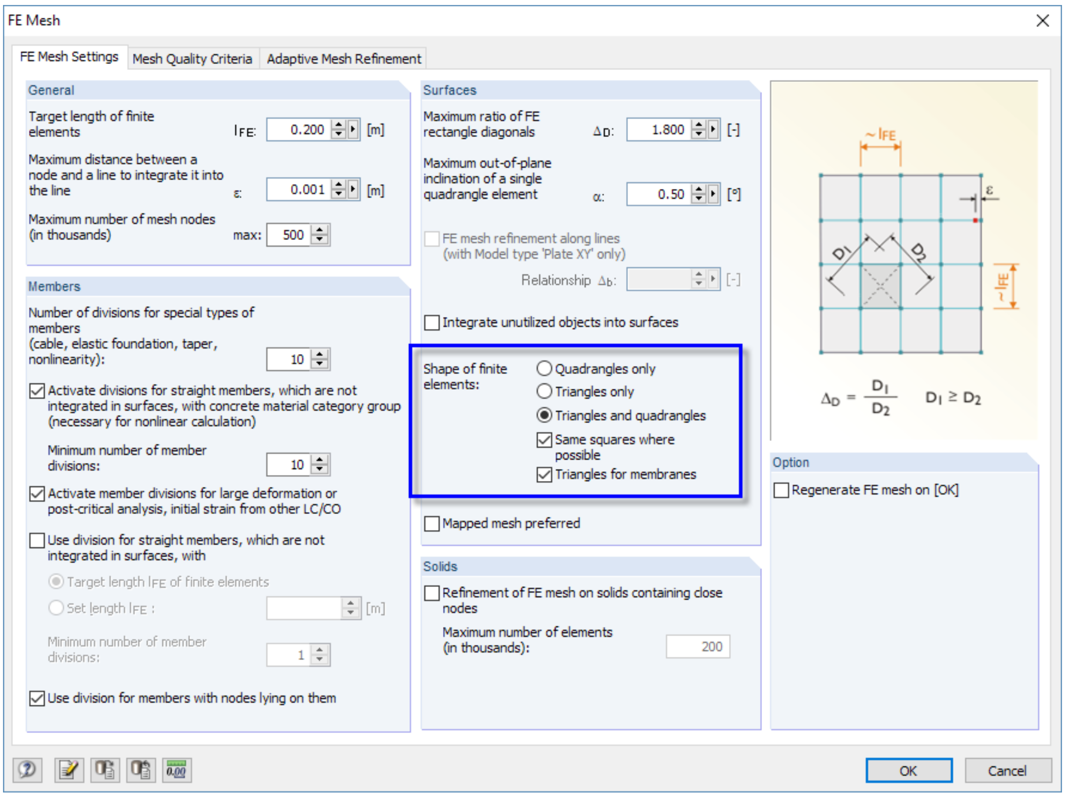


Figure 4.4: Dialog box *FE Mesh*, tab *FE Mesh Settings*

4.3 Cables

There are two options available for defining the *Force* in each cable:

- The *Average force in cable* is to be set for cables that are connected with the membranes of the structure.
- The *Force density* is to be applied to pure cable webs.

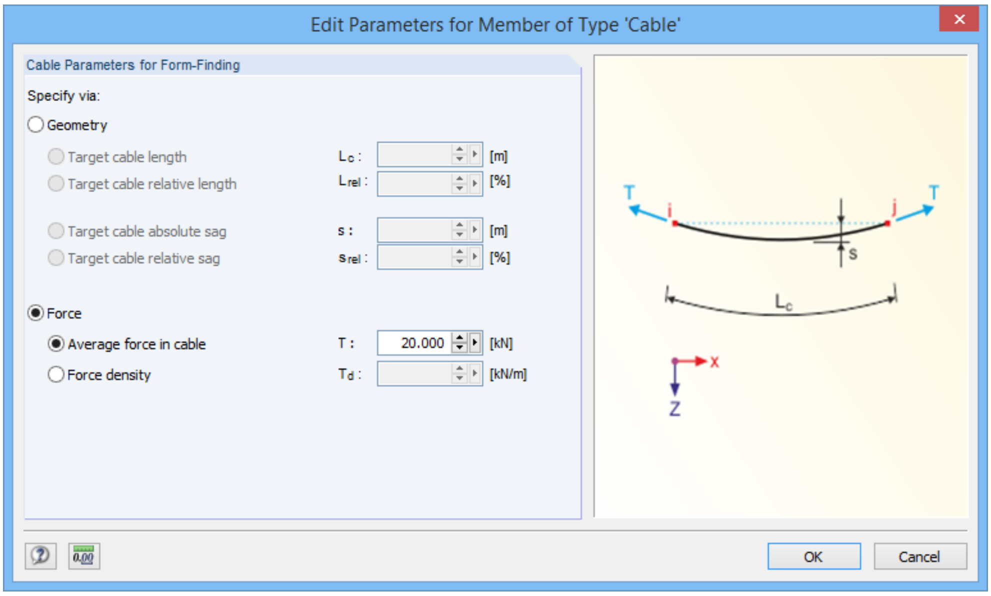


Figure 4.5: Defining force in cable

If an *Average force in cable* is set and the defined value proves to be too low during the calculation, the cable will be stiffened when exceeding the length $\ell_c = 0.4\,\pi\,\ell$. Thus, no strain will occur anymore in the cable (ℓ being the distance between the points *i* and *j*, see Figure 4.5).

The values specified as required prestress or geometry of cables are considered as the resulting values in space. The values entered for cables are not adjusted according to the inclination (such as for membranes) when applying the projection method because they do not represent the explicit, that is the physically real data.

5 RF-CUTTING-PATTERN

This chapter describes the dialog boxes and functions of the RF-CUTTING-PATTERN add-on module.

5.1 Open Add-on Module

To work with RF-CUTTING-PATTERN, it is necessary to activate the add-on module first: Select the **RF-CUTTING-PATTERN** check box which you can access in the *Options* tab of the *New Model - General Data* or *Edit Model - General Data* dialog boxes.

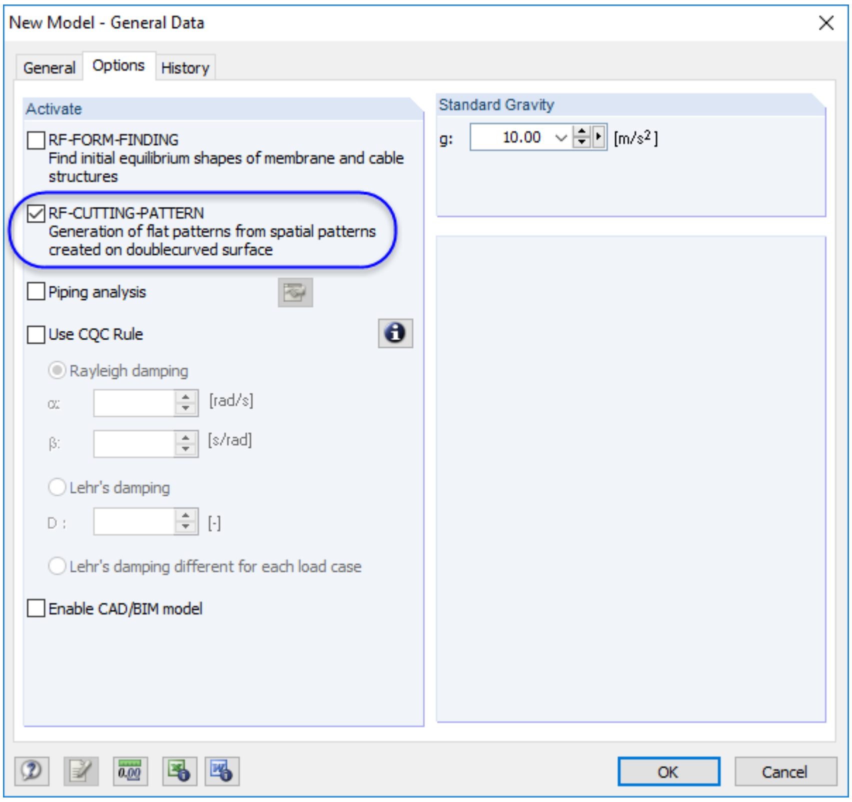
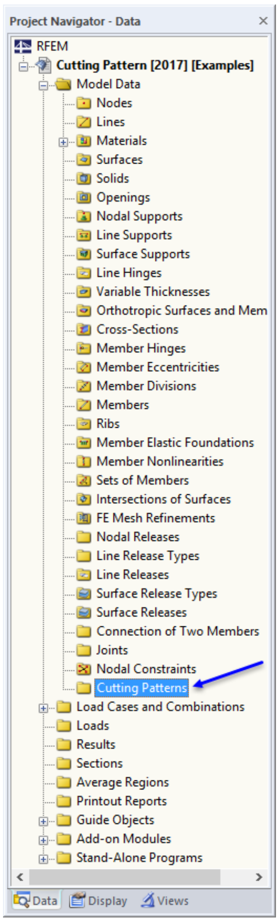


Figure 5.1: Activating the RF-CUTTING-PATTERN in *New Model - General Data* dialog box



RF-CUTTING-PATTERN can be used for self-defined spatial surfaces as well as for structures that have been created with the RF-FORMFINDING add-on module.

When the add-on module has been activated, the additional entry *Cutting Patterns* is available in the *Data* navigator.

5.2 Cutting Lines

The cutting pattern must be defined via boundary lines. If the arrangement of the membrane surfaces in the basic position is too large for cutting, it is possible to divide every surface into partial strips by means of cutting lines. For this, RF-CUTTING-PATTERN offers two special line functions. You can access them on the *Lines* shortcut menu or on the menu

Insert → **Model Data** → **Lines** → **Cut via Two Lines** or **Cut via Section**.

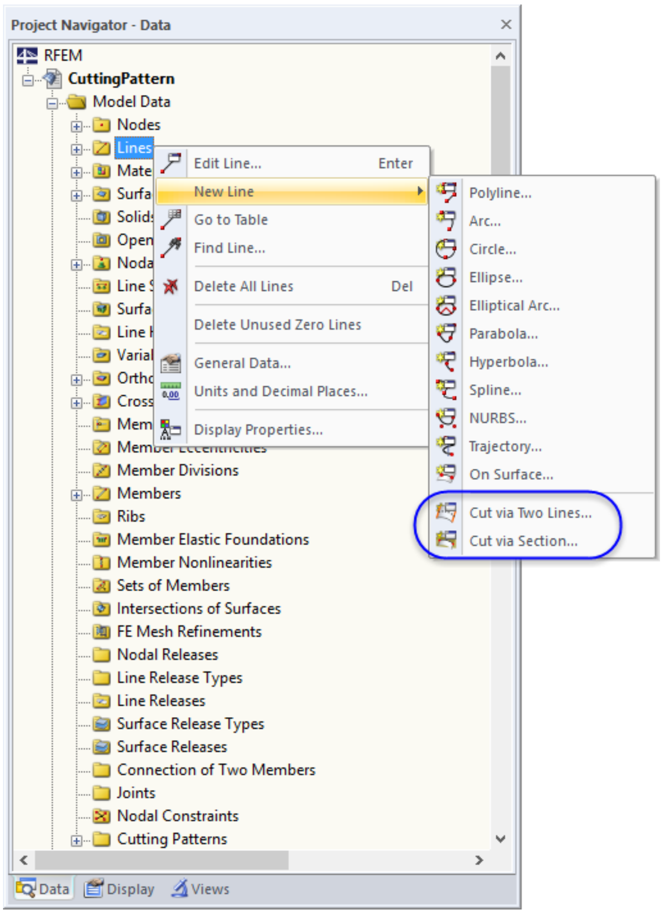
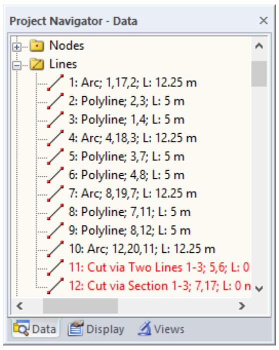


Figure 5.2: Line types for RF-CUTTING-PATTERN on *Lines* shortcut menu



Those two line types do not affect the geometry of the model. Thus, they can be defined independently of the model.

The cutting lines are only shown in the graphic when the FE mesh has been generated. If the mesh is missing, the corresponding navigator entries are marked in red.

In the dialog box of each line type, at first enter the surface on which the line lies (see Figure 5.3).

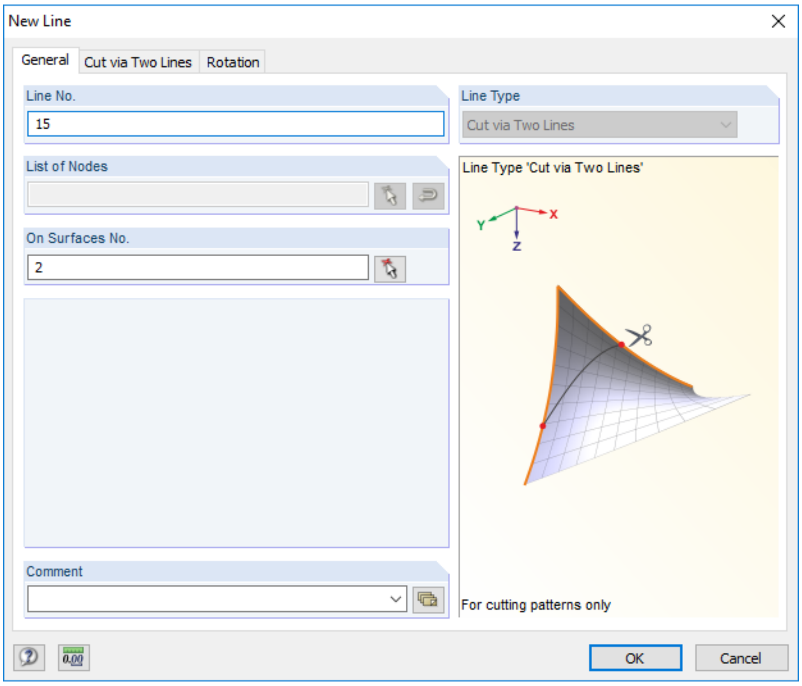


Figure 5.3: Dialog box *New Line*, tab *General*

The surface can be determined graphically with the button. As the lines can also be set on several surfaces, multiple entries are possible.

Cut via two lines

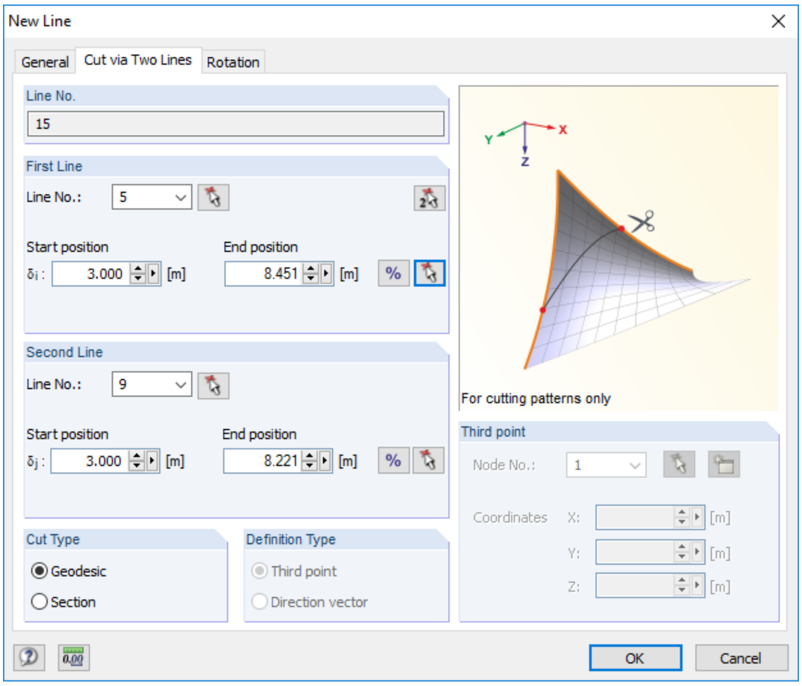


Figure 5.4: Dialog box *New Line*, tab *Cut via Two Lines*

In the second tab of this dialog box, specify the lines and the points on the lines that represent the definition nodes of the cutting line. The lines can be selected graphically with the button.

With the % button, you can switch between the input of relative and absolute distances.

Use the button to determine the relative distances graphically in the work window.

In the *Cut Type* section of the dialog box, you define how the cutting line is to be created:

- *Geodesic*: This is the shortest line between the definition points on the surface area.
- *Section*: A plane is set through the surface. It can be defined by a third point or a direction vector.

Cut via section

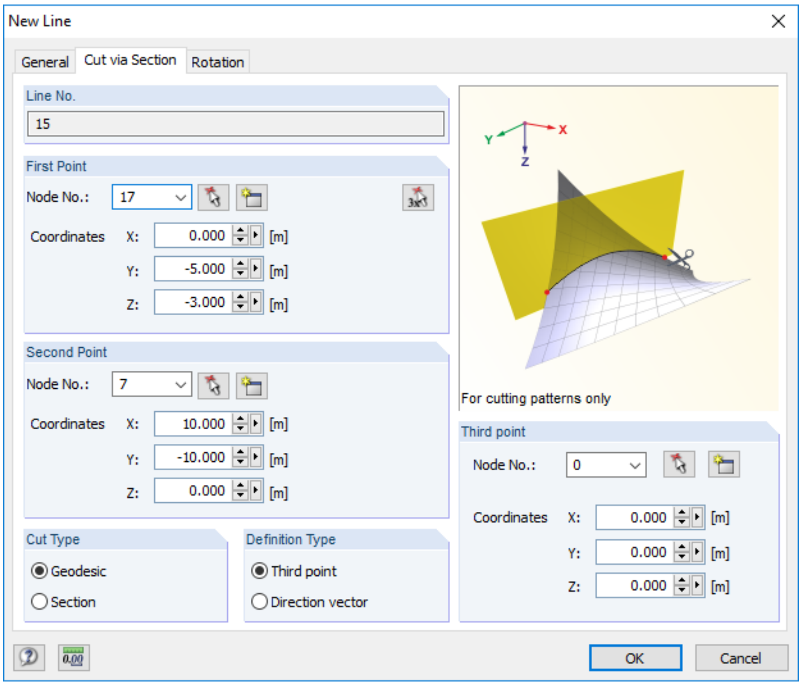


Figure 5.5: Dialog box *New Line*, tab *Cut via Section*

In the second tab, specify the points that define the cutting plane. The nodes can be selected graphically with the button. You can also create a new node with the button.

In the *Cut Type* section, you define how the cutting line will be created:

- *Geodesic*: This is the shortest line between the cutting points on the surface area.
- *Section*: The cutting plane is defined by means of a third point or a direction vector.

Finally, select the *Definition Type* and specify the parameters of the node or vector, respectively. The latter represents a unit vector relative to the line between the *First Point* and *Second Point*.

The cutting lines are displayed after the mesh has been generated.

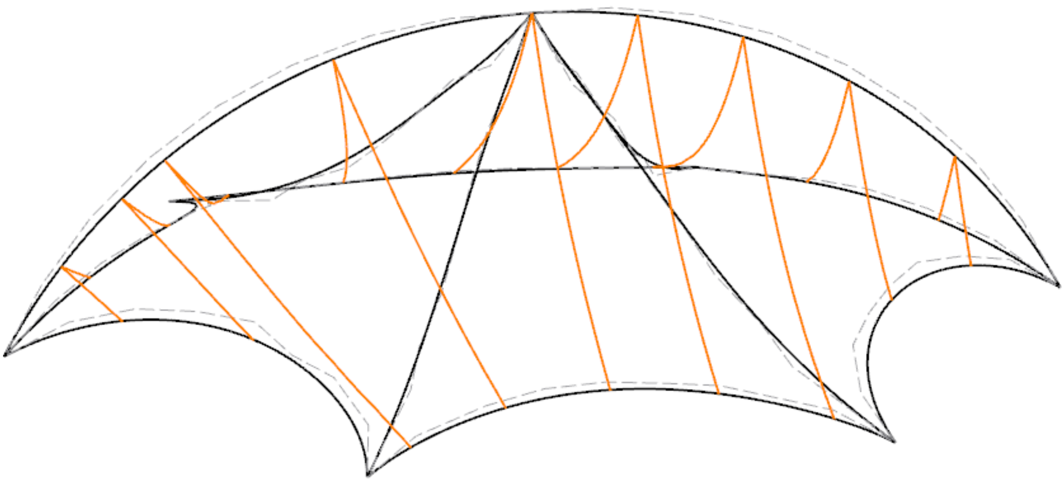


Figure 5.6: Cutting lines (orange)

5.3 Cutting Patterns

To create cutting layouts, the membrane surfaces must be divided into so-called *cutting patterns*. These objects can be created by using the membranes' boundary lines as well as user-defined cutting lines (see Chapter 5.2).

A new cutting pattern can be created on the menu

Insert → **Model Data** → **Cutting Patterns** → **Dialog Box** or **Table**

or on the *Cutting Patterns* shortcut menu in the *Data* navigator.

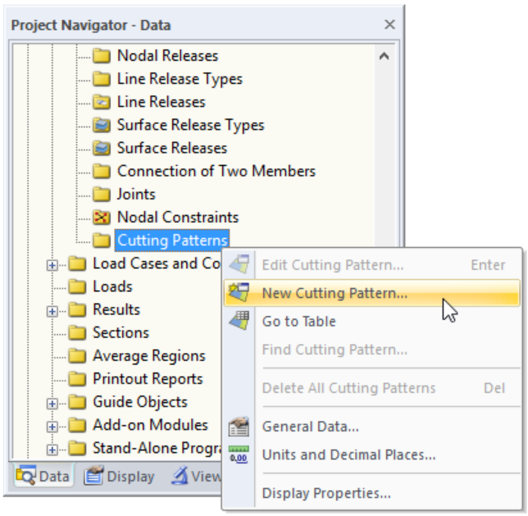


Figure 5.7: Creating new cutting pattern on shortcut menu

Alternatively, you can use a generator which is accessible on the menu

Tools → **Generate Cutting Patterns from Cells**.

If you choose the latter, the *Create Surfaces from Cells* dialog box opens where you can select the relevant surfaces and disable specific lines, if necessary.

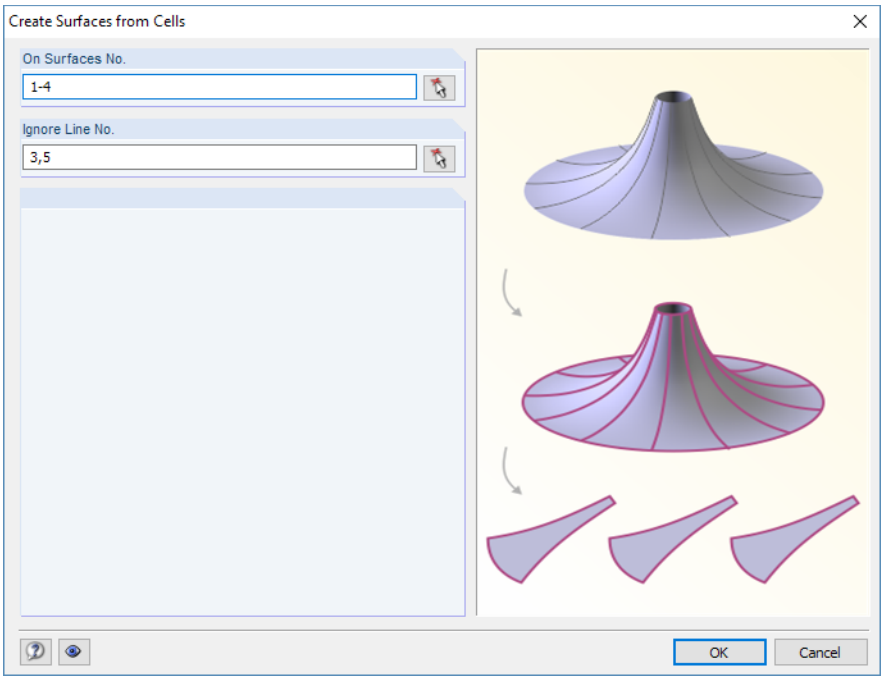


Figure 5.8: Dialog box *Create Surfaces from Cells*

When the generator is not used, the *New Cutting Pattern* dialog box opens (see Figure 5.9).

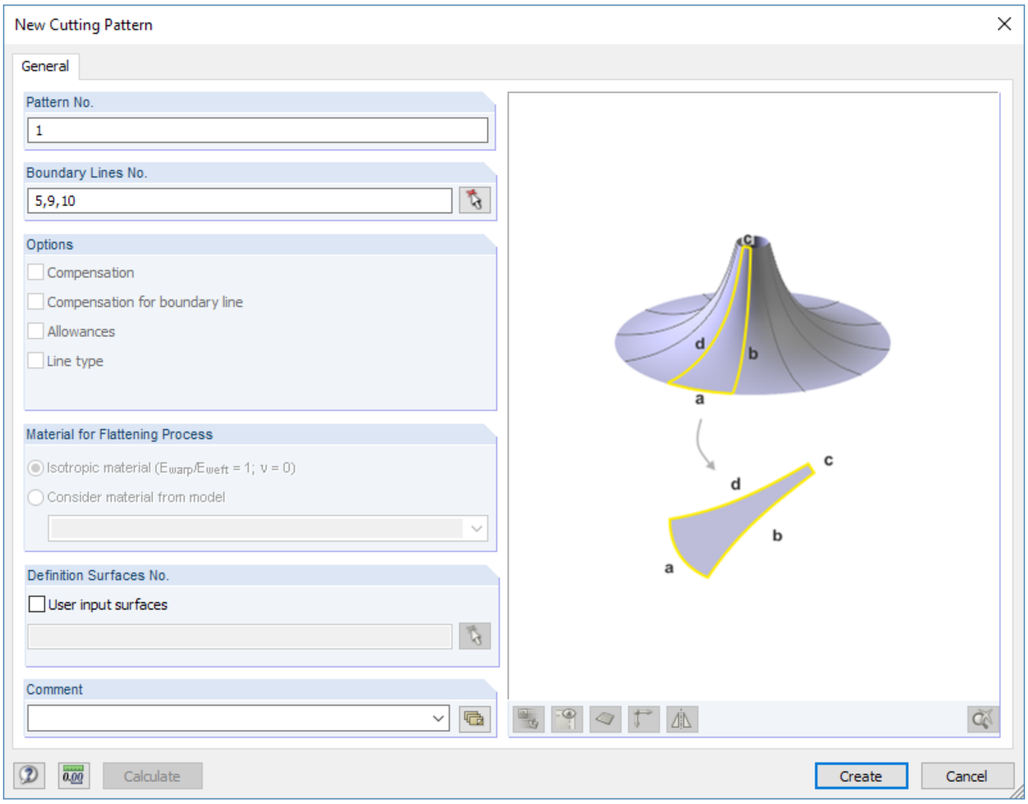


Figure 5.9: Dialog box *New Cutting Pattern*

The *Boundary Lines* of the cutting pattern are represented by "real" lines or cutting lines. They can be defined graphically with the button. RFEM recognizes intersection points automatically.

Create

When the boundary lines are defined, click the [Create] button to generate the cutting pattern. The *Edit Cutting Pattern* dialog box appears.

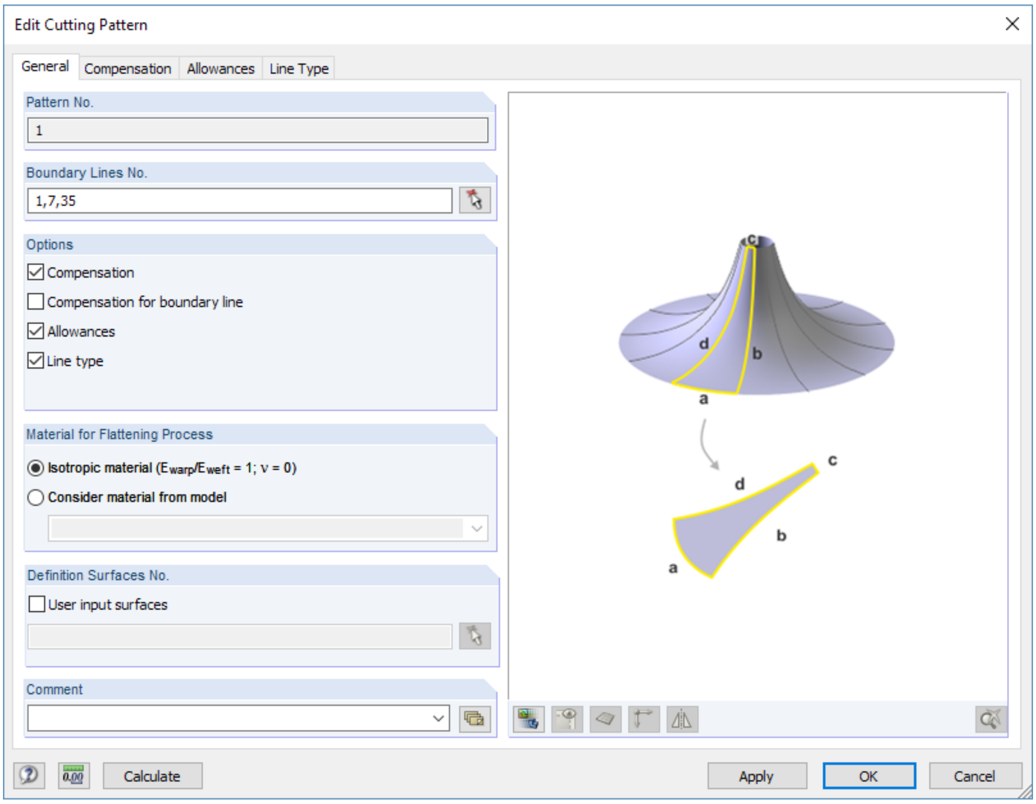


Figure 5.10: Dialog box *Edit Cutting Pattern*, tab *General*

This dialog box allows for specifications concerning compensation and edge tolerances. The other dialog tabs are controlled by the parameters activated in the *Options* section.

General notes

The following principles must be observed when creating a cutting pattern:

- The cutting pattern must not be defined exclusively by cutting lines. There must be at least one system-relevant boundary line.
- RF-CUTTING-PATTERN checks if adjacent cutting patterns are available. The calculation makes sure that common lines have the same lengths.
- RF-CUTTING-PATTERN classifies the lines as boundary or welding lines. In the *Line Type* tab, you can change any welding line to a boundary line (see page 52).

Every cutting pattern is symbolized by a big dot in the work window. If a cutting pattern is marked in red in the *Data* navigator, it is not correctly defined.

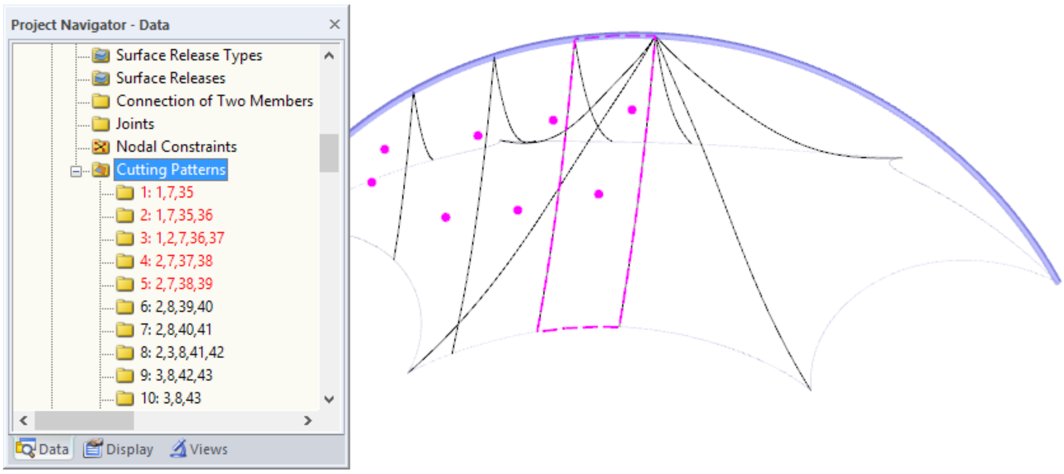


Figure 5.11: *Data* navigator and symbols of cutting patterns

Apply

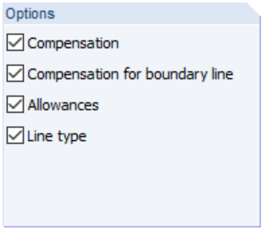
The [Apply] button in the *Cutting Pattern* dialog box creates a preliminary shape of the cutting layout (see Chapter 5.4.2, page 54).

With the [Show Figure or Rendering] button, you can display the view of the cutting pattern in the graphic area (see Figure 5.14, page 51) instead of the symbols. Then the other buttons become accessible as well. They have the following functions:

Button	Description	Function
	Overlap	Shows or hides dashed lines of overlap → Figure 5.15
	Mesh	Shows or hides FE mesh
	Axes	Shows or hides axes of surface
	Mirror	Mirrors shape of cutting pattern about XZ-plane

Table 5.1: Buttons for rendering option

Compensation



This tab is available when the *Compensation* option has been activated in the *General* tab.

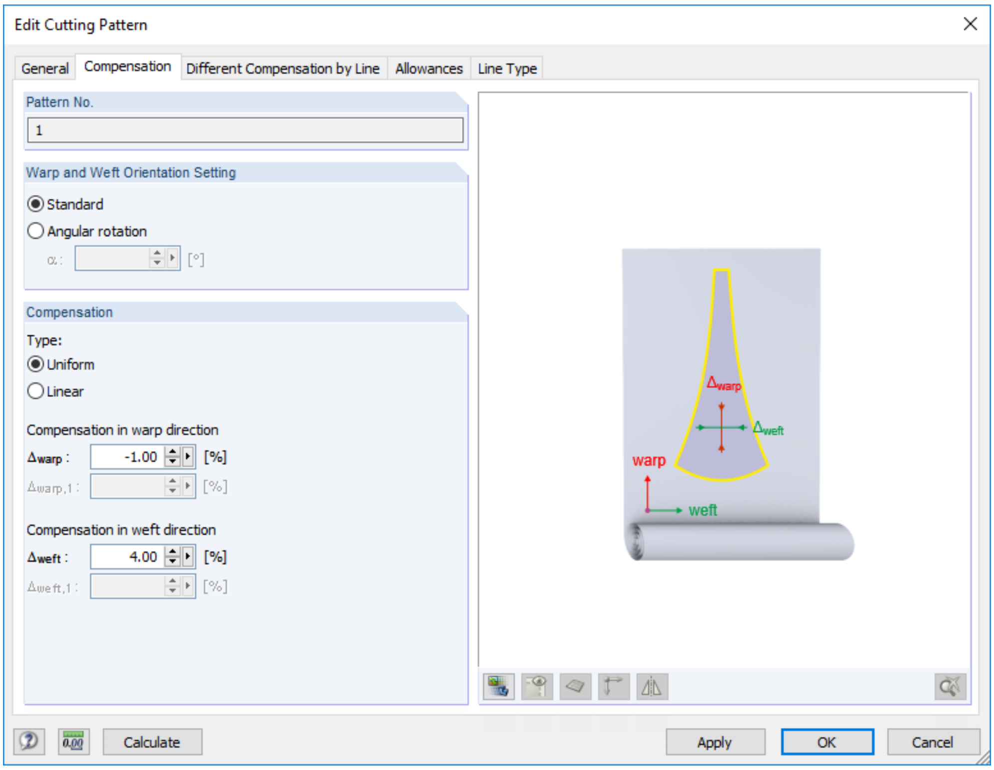


Figure 5.12: Dialog box *Edit Cutting Pattern*, tab *Compensation*

Generally, the dimensions of a cutting pattern are reduced by a small percentage so that the membrane will reach the intended final shape when the prestress is applied. This way, creep effects of the membrane can be compensated.

Warp and Weft Orientation Setting

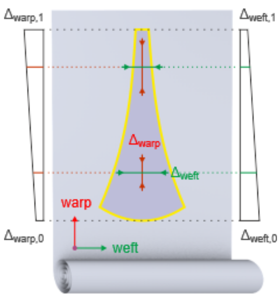
The compensation can be specified separately for the warp and weft directions. As *Standard*, warps in the direction of the surface axis *x* and wefts in the direction of its axis *y* are assumed. If the structure of the textile is aligned differently for the cutting, the orientation can be rotated about the angle α.

The surface axes are shown in the dialog graphic when you switch to the model view by clicking the [Figure or Rendering] button (see Figure 5.14).

Compensation

A positive value of the compensation reduces the cutting layouts, a negative one extends them accordingly in the warp or weft directions.

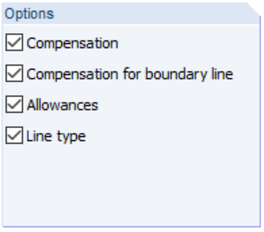
The compensation relates to the non-stressed length of the membrane panel ("original state"). Then, after prestressing, the final shape e.g. determined by RF-FORM-FINDING is reached with consideration of the compensation. The compensation cannot be directly determined by the defined prestress.



You can define a *Uniform* or *Linear* compensation. The latter allows for specifying linearly variable strain components for the warp and weft directions. This may be necessary for conical models. The strains Δ_0 and Δ_1 refer to the edges of the cutting pattern according to the symbols.

When data for compensation is entered, it is necessary to [Calculate] the cutting pattern again (see Chapter 5.4.2, page 54, paragraph "Final calculation").

Different Compensation by Line



This tab is available when the *Compensation for boundary line* option has been activated in the *General* tab.

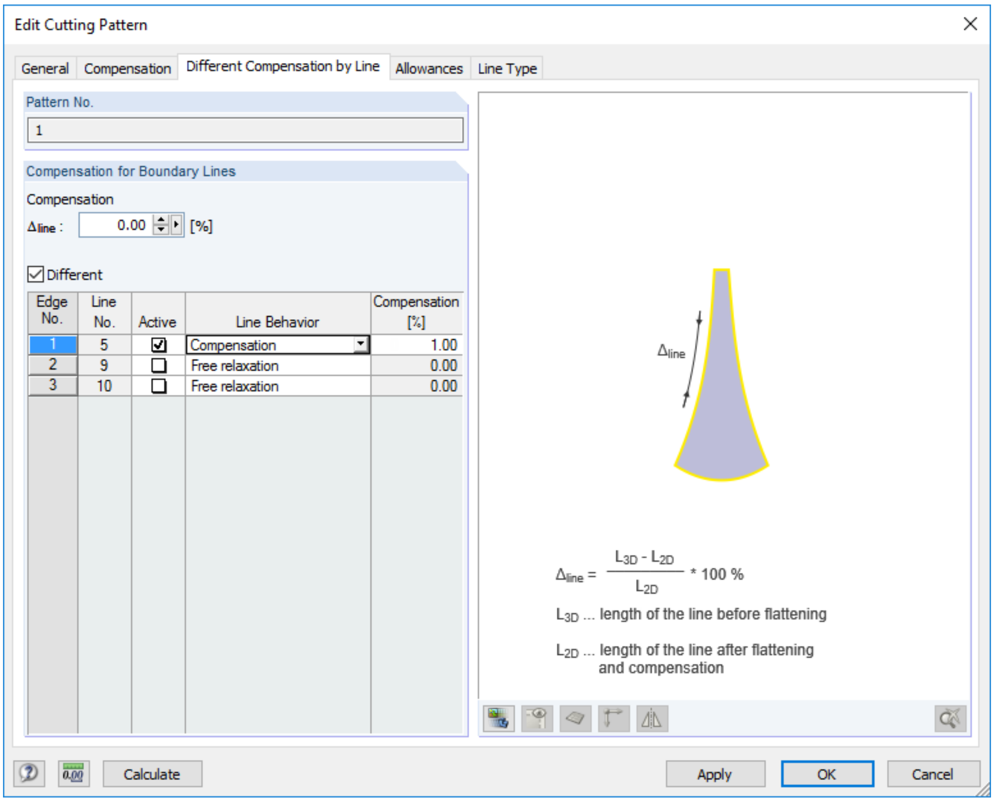
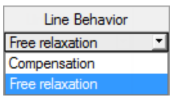


Figure 5.13: Dialog box *Edit Cutting Pattern*, tab *Different Compensation by Line*

Different values of compensation are required when an equality of lines is to be reached, e.g. at rigid supports. Imagine a boundary fixation of a panel to a steel tube: as the length of the steel tube is not compensated, the length of the membrane will remain constant there.

In the Δ_{line} box, you can enter any global value of the compensation. When the *Different* check box has been selected, however, you can set specific lines *Active* or deactivate them in the table below and assign each *Compensation* individually.



The *Line Behavior* can be described by a compensation or a free relaxation (possibility of displacement). Select the appropriate entry from the list. The two options have the following effects for the 'ironing process' when the element nodes of the boundary lines are moved to the final locations to comply with the equilibrium of minimum of energy:

- For a *Compensation* of 0 %, the length of the boundary line is kept equal to the original length. If a value of e.g. 5 % is defined, however, the "ironing" forces the length of the line to be shorter than the original one: the final length for fabrication multiplied by 1.05 will give its length in the FE model after tightening. Therefore, the reference value of 100 % represents the original model length in RFEM, not the stressed length.
- If a *Free relaxation* is set, the above-mentioned procedure ignores the boundary conditions at the edges. The "ironing" will be effectuated without any resistance from the boundary line. Its final length will not feature any specific relation to the original one.

In the rendering mode, the line of the current row is marked in the selection color (see Figure 5.14).

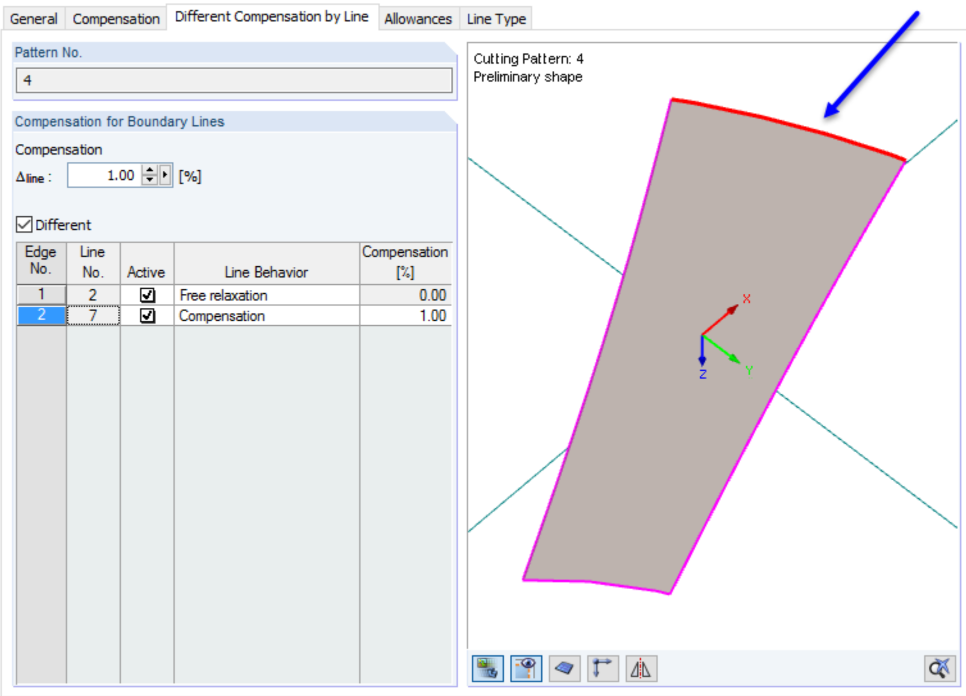
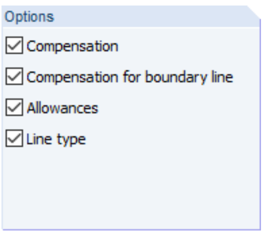


Figure 5.14: Rendering mode with selection of active table row

Allowances



This tab is available when the *Allowances* option has been activated in the *General* tab.

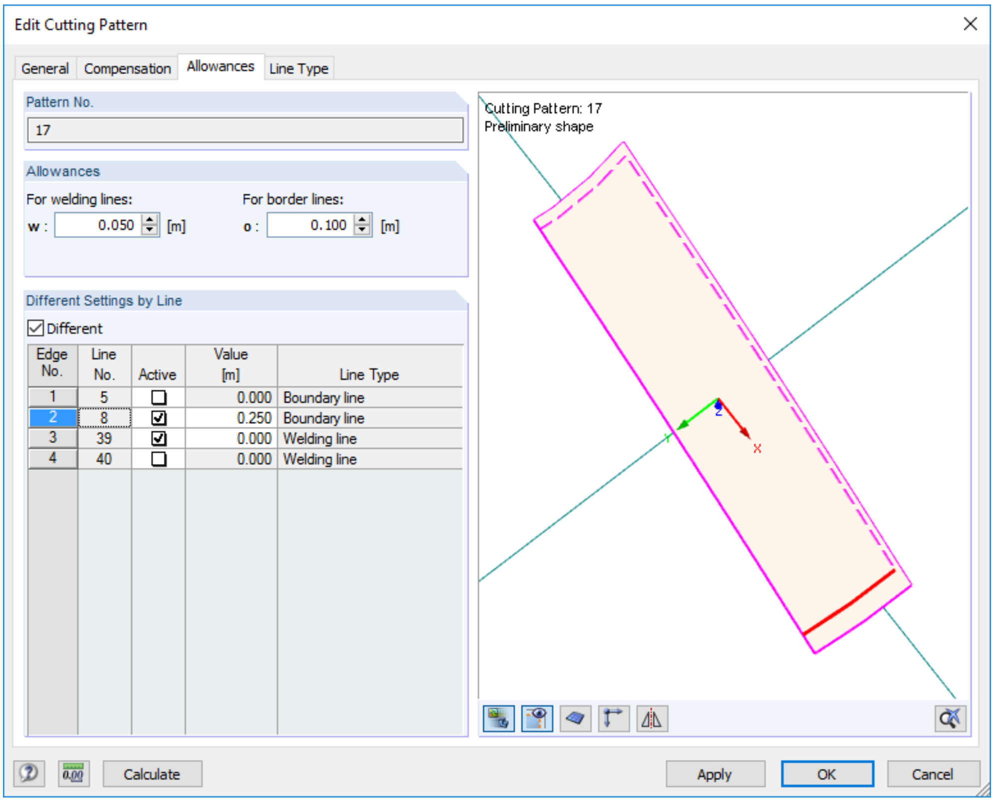


Figure 5.15: Dialog box *Edit Cutting Pattern*, tab *Allowances*

Allowances are needed to attach the membrane to boundary lines, or for overlaps to weld the panels.

It is possible to define allowances globally *For welding lines* and *For border lines*. The line types are managed in the *Line Type* tab where they can be adjusted, if required (see description below).

When the *Different* check box has been selected, it is possible to set specific lines *Active* or deactivate them in the table below and assign the *Value* of each allowance individually.

The parameters can be checked in the dialog graphic when you switch to the model view by clicking the [Figure or Rendering] button (see Figure 5.15). The line of the current table row is marked in the selection color.

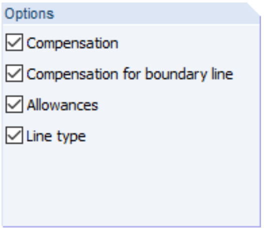
With the [Pattern Overlap] button, you can display and hide the allowances. This way, you can check the cutting pattern with and without tolerances.

You can control the graphic by using the mouse functions familiar from RFEM in order to zoom, shift or rotate the view.

The rendered view is only available when the cutting pattern has been calculated. If you adapt the values of allowances, however, the cutting pattern does not require any recalculation.

In the RFEM work window, the allowances are shown as dashed lines (see Figure 5.6, page 45).

Line Type



This tab is available when the *Line type* option has been activated in the *General* tab.

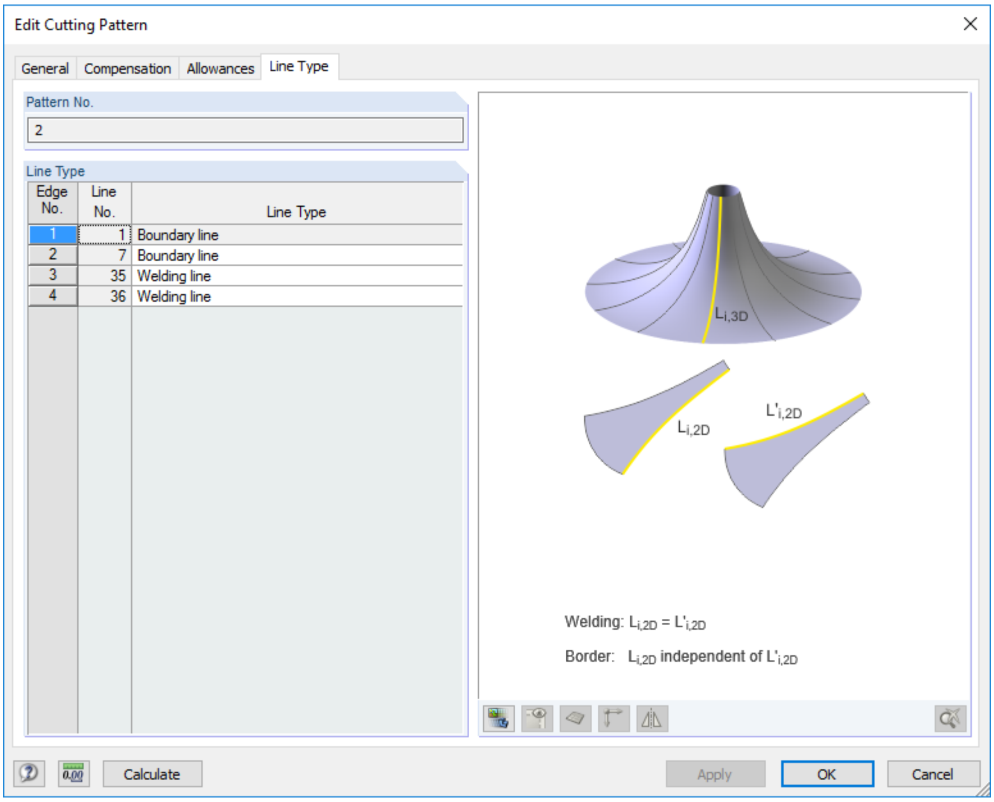


Figure 5.16: Dialog box *Edit Cutting Pattern*, tab *Line Type*

RF-CUTTING-PATTERN defines a line type for every definition line of the cutting pattern. The line is classified either as *Boundary line* or *Welding line*. In general, welding lines are created for adjacent cutting layouts.

Welding lines affect the overall cutting: RF-CUTTING-PATTERN tries to reach a line equality for both adjacent cutting edges, as the common line deforms synchronously. For this, the program uses the average value of the lines from both cutting patterns.

This dialog tab offers you the possibility to change a welding line to the line type *Boundary line*.

On the contrary, a boundary line cannot be transformed into a *Welding line* type. Welding lines are automatically recognized for adjacent cutting patterns that have one common line.

5.4 Calculation

5.4.1 Calculation Parameters

In the *Calculation Parameters* dialog box, the *Cutting Patterns* tab is available when the add-on module has been enabled. It controls the calculation settings specific for RF- CUTTING-PATTERN. To open the dialog box, select on the RFEM menu

Calculate → Calculation Parameters

or click the corresponding button in the toolbar.

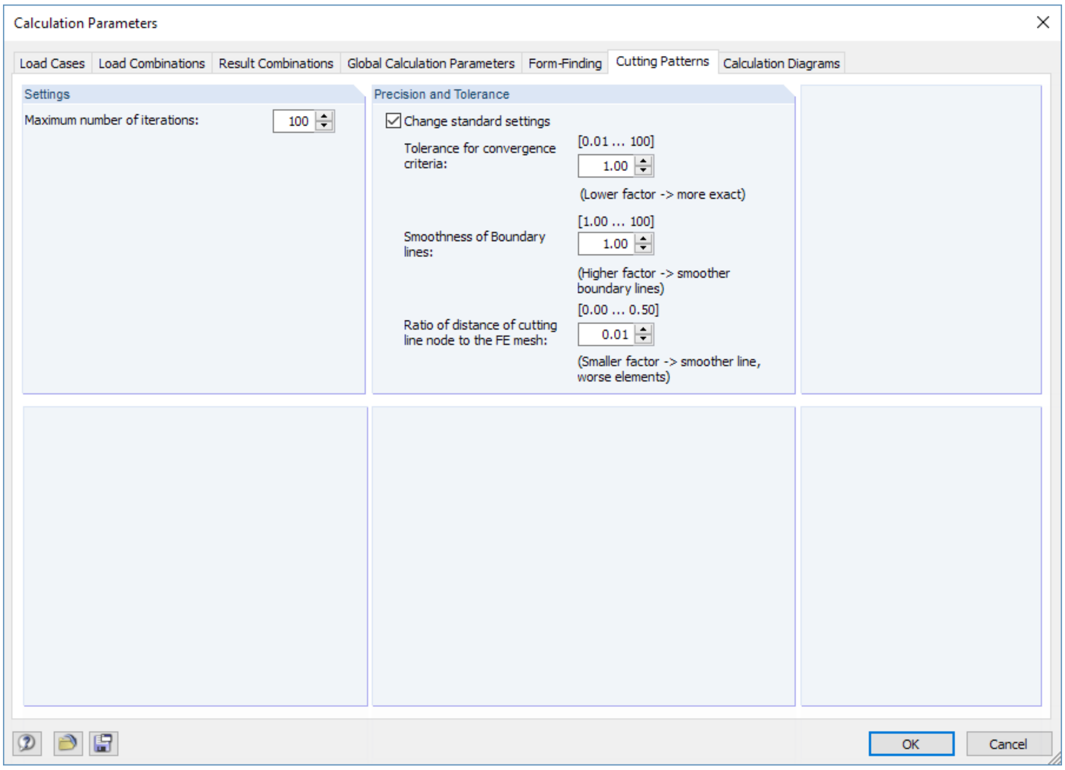


Figure 5.17: Dialog box *Calculation Parameters*, tab *Cutting Patterns*

Settings

As the cutting patterns are calculated independently of any load case or combination, the specifications of the global calculation parameters cannot be applied. In this section, you can define the *Maximum number of iterations* for RF-CUTTING-PATTERN to perform for the flattening procedure.

Precision and Tolerance

It is only rarely necessary to adjust the preset tolerance and smoothing parameters. You can nevertheless access the text boxes when the *Change standard settings* option has been selected.

The convergence behavior of the flattening can be affected by the *Tolerance for convergence criteria*. The factor 1.0 is set by default. The minimum factor is 0.01, the maximum value is 100.0. The greater the value is, the more insensitive the break-off limit will be.

The *Smoothness of Boundary lines* affects the iterative determination of the boundary line shape. Again, the factor 1.0 is preset. The greater the factor is, the more accurately the boundary lines will be smoothed. The calculation time increases accordingly.

The *Ratio of distance of cutting line node to the FE mesh* controls the alignment of cutting lines to adjacent FE mesh points. The smaller the ratio is, the smoother the line will be created – to the disadvantage of a distorted meshing. The ratio of 0.01 is preset.

5.4.2 Preliminary and Final Calculation

General information

During the calculation, RF-CUTTING-PATTERN tries to transform the curved surface components into planar cutting layouts by using an iterative nonlinear method. You may imagine this process as an “ironing procedure” with the aim to smooth out the curved surfaces.

The calculation uses the mesh geometry of the planar, folded, uni- or bidirectionallly curved surface components of the cutting patterns and flattens them by means of the theory of minimum energy. Metaphorically speaking, the program tries to press the meshed geometry of the curved surfaces into a plane, using a press with frictionless press areas. If a state is found where the stresses from flattening are in equilibrium, a minimum of energy and an optimum of accuracy for the cutting layout has been reached. To describe it in one more picture: when an orange half is flattened, the final result will be tension in the external and compression in the internal zones. Those forces will be in equilibrium.

Preliminary calculation

The calculation is organized in two parts. For the preview of the cutting pattern, a preliminary flattening calculation will be carried out as soon as you click [Create] (when creating a new cutting pattern) or [Apply] (when modifying any parameter).

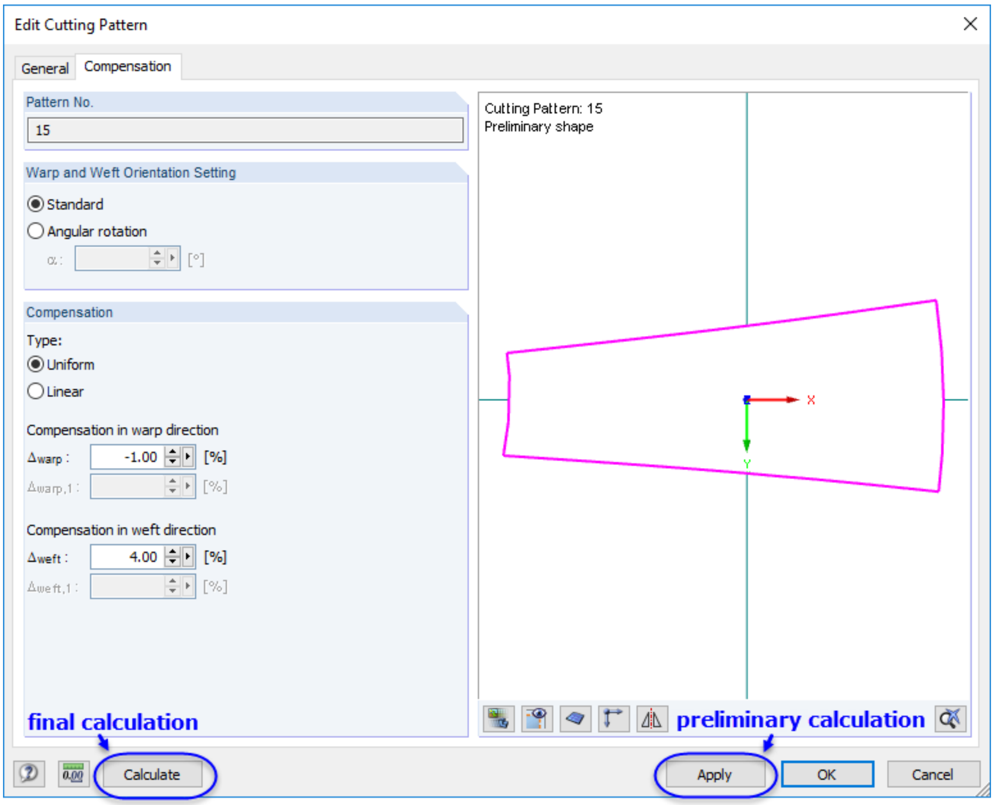
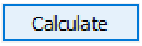


Figure 5.18: [Calculate] and [Apply] buttons in *Edit Cutting Pattern* dialog box

The “preliminary calculation” (*flattening process*) determines the cutting pattern in a high-speed process so that a preview of the flattened pattern can be shown in the dialog graphic.

With the [Apply] function, only the current cutting pattern is analyzed.

Final calculation



The “final calculation” determines the cutting layouts by applying an isotropic material behavior (orthotropic material behavior is in preparation). By clicking the [Calculate] button (see Figure 5.18), a comprehesive flattening calculation is applied to the curved surface components.

The flattening process is carried out by means of the theory of minimum energy. Prestresses and stresses due to flattening are applied in each main direction by means of a compensation. Allowances and reductions for welds and boundary connections are considered separately for every cutting pattern.

The “ironing process” takes an average coordinate system orientation from the curved geometry and applies those systems with the same rotation to the flattened component.

If adjacent cutting patterns have been defined, their parameters are accordingly considered to maintain identical lengths.

With the [Calculate] function, all cutting patterns are globally analyzed.

You can observe the iterative calculation process in the *FE-Calculation* window.

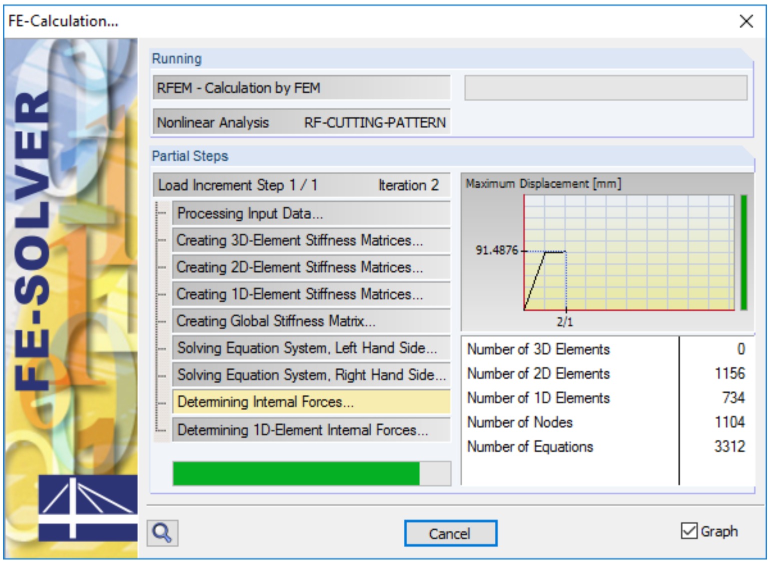


Figure 5.19: Calculation process for RF-CUTTING-PATTERN

As a result of the final calculation, you see the coordinates of the cutting patterns shown in the *Point Coordinates* dialog tab and in Table *4.45.1 Cutting Patterns - Point Coordinates* (see Chapter 5.5).

Cutting patterns which are displayed after the final or preliminary calculation are marked by different colors. In the *Display Properties* dialog box, you can see and adjust the corresponding color settings. To access the dialog box, select on the RFEM menu

Options → **Display Properties** → **Edit**

or right-click a cuttern pattern and select the *Display properties* option in the shortcut menu.

In the *Display Properties* dialog box, you can then modify the colors via

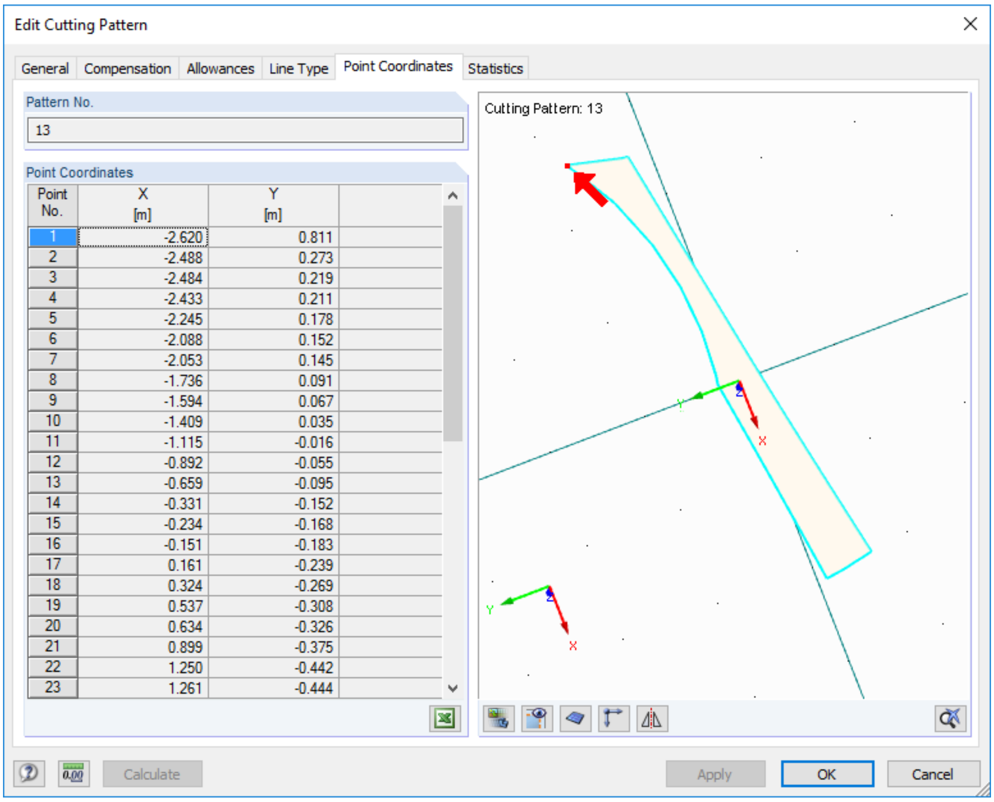
Colors → **Model Data** → **Cutting Patterns**.

5.5 Results and Export

Calculate

When the cutting pattern has been calculated, the new *Point Coordinates* tab is available in the *Edit Cutting Pattern* dialog box.

Dialog box *Edit Cutting Pattern*



Point No.	X [m]	Y [m]
1	-2.620	0.811
2	-2.488	0.273
3	-2.484	0.219
4	-2.433	0.211
5	-2.245	0.178
6	-2.088	0.152
7	-2.053	0.145
8	-1.736	0.091
9	-1.594	0.067
10	-1.409	0.035
11	-1.115	-0.016
12	-0.892	-0.055
13	-0.659	-0.095
14	-0.331	-0.152
15	-0.234	-0.168
16	-0.151	-0.183
17	0.161	-0.239
18	0.324	-0.269
19	0.537	-0.308
20	0.634	-0.326
21	0.899	-0.375
22	1.250	-0.442
23	1.261	-0.444

Figure 5.20: Dialog box *Edit Cutting Pattern*, tab *Point Coordinates*

In this tab, the shape of the cutting pattern is documented in a table of coordinates. It contains the new flattened coordinates of the cutting layout for each FE mesh node. The *Point Coordinates* are related to the centroid of the selected cutting pattern.

In the graphic area, the cutting pattern with its coordinate system in the centroid is displayed. The point which has been selected in the table is marked by an arrow. To hide or show the overlapping areas, use the button. To hide or show the FE mesh of the pattern, use the button.

The areas of the cuttings are displayed in the last *Statistics* dialog tab.

Cutting Pattern Statistics

Cutting Pattern Area			
2D area	A_{2D}	4.492	m^2
2D area with allowances	$A_{2D,a}$	5.129	m^2
3D area without compensation	A_{3D}	4.629	m^2
Additional properties			
Length (in X direction)	L	5.044	m
Length (in X direction) with allowances	L_a	5.257	m
Width (in Y direction)	W	1.419	m
Width (in Y direction) with allowances	W_a	1.533	m

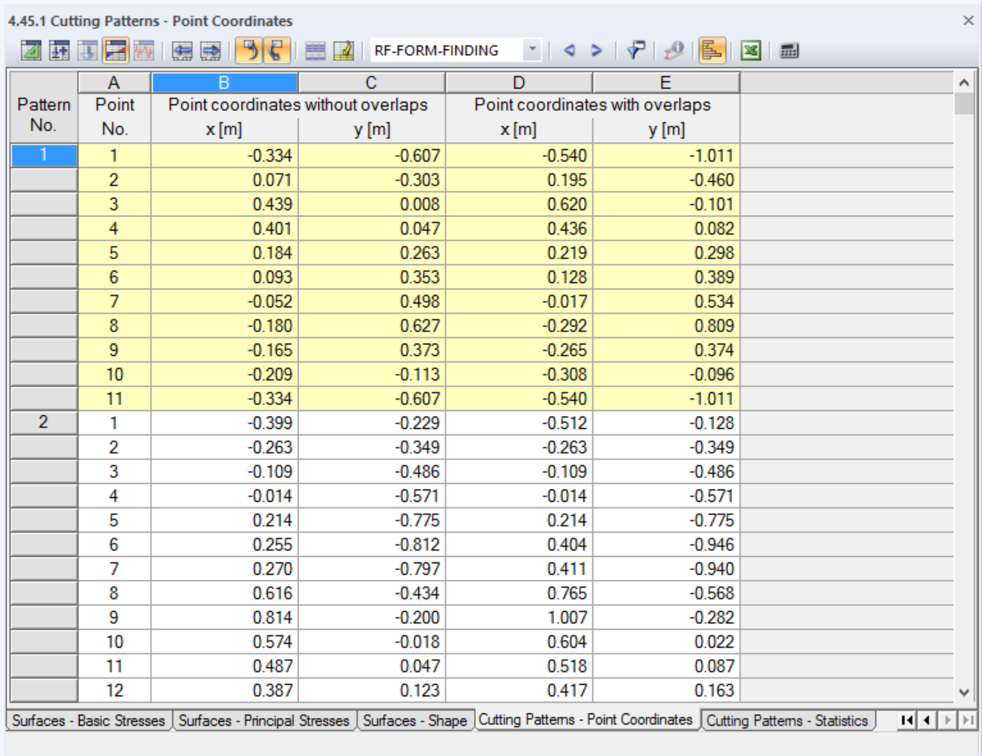
Figure 5.21: Dialog box *Edit Cutting Pattern*, tab *Statistics* (detail)

The *2D area* represents the surface area of the “ironed” cutting pattern. The *3D area* (the surface area of the curved surface) is displayed as well for checking purposes. If the values are only slightly different, a good quality of the flattening process has been obtained.

The dimensions of the cutting pattern are shown as *Additional properties*. You can see at a glance whether the production of the panels is possible from the roll material.

RFEM Tables *4.45 Cutting Patterns*

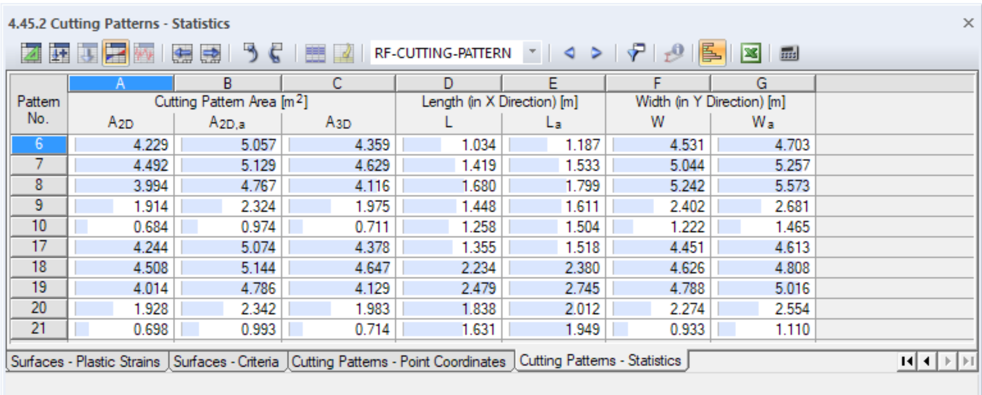
The RF-CUTTON-PATTERN results are also listed in Table *4.45.1 Cutting Patterns - Point Coordinates* and Table *4.45.2 Cutting Patterns - Statistics* of RFEM.



Pattern No.	A	B	C	D	E
	Point No.	Point coordinates without overlaps x [m]	y [m]	Point coordinates with overlaps x [m]	y [m]
1	1	-0.334	-0.607	-0.540	-1.011
	2	0.071	-0.303	0.195	-0.460
	3	0.439	0.008	0.620	-0.101
	4	0.401	0.047	0.436	0.082
	5	0.184	0.263	0.219	0.298
	6	0.093	0.353	0.128	0.389
	7	-0.052	0.498	-0.017	0.534
	8	-0.180	0.627	-0.292	0.809
	9	-0.165	0.373	-0.265	0.374
	10	-0.209	-0.113	-0.308	-0.096
	11	-0.334	-0.607	-0.540	-1.011
2	1	-0.399	-0.229	-0.512	-0.128
	2	-0.263	-0.349	-0.263	-0.349
	3	-0.109	-0.486	-0.109	-0.486
	4	-0.014	-0.571	-0.014	-0.571
	5	0.214	-0.775	0.214	-0.775
	6	0.255	-0.812	0.404	-0.946
	7	0.270	-0.797	0.411	-0.940
	8	0.616	-0.434	0.765	-0.568
	9	0.814	-0.200	1.007	-0.282
	10	0.574	-0.018	0.604	0.022
	11	0.487	0.047	0.518	0.087
	12	0.387	0.123	0.417	0.163

Figure 5.22: RFEM table *4.45.1 Cutting Patterns - Point Coordinates*

The output is sorted by cutting patterns. For each FE mesh point you see the *Point coordinates without overlaps* and the *Point coordinates with overlaps*.



Pattern No.	A	B	C	D	E	F	G
	Cutting Pattern Area [m²] A_{2D}	$A_{2D,a}$	A_{3D}	Length (in X Direction) [m] L	L_a	Width (in Y Direction) [m] W	W_a
6	4.229	5.057	4.359	1.034	1.187	4.531	4.703
7	4.492	5.129	4.629	1.419	1.533	5.044	5.257
8	3.994	4.767	4.116	1.680	1.799	5.242	5.573
9	1.914	2.324	1.975	1.448	1.611	2.402	2.681
10	0.684	0.974	0.711	1.258	1.504	1.222	1.465
17	4.244	5.074	4.378	1.355	1.518	4.451	4.613
18	4.508	5.144	4.647	2.234	2.380	4.626	4.808
19	4.014	4.786	4.129	2.479	2.745	4.788	5.016
20	1.928	2.342	1.983	1.838	2.012	2.274	2.554
21	0.698	0.993	0.714	1.631	1.949	0.933	1.110

Figure 5.23: RFEM table *4.45.2 Cutting Patterns - Statistics*

The second results table includes the 2D areas of the “ironed” cutting pattern, the 3D areas of the curved surfaces as well as the *Length* and *Width* of each pattern.

Printout report

The cutting patterns can be documented in the general printout report of RFEM. In the *Printout Report Selection* dialog box, the cutting patterns are managed in the *LC/CO Results* tab.

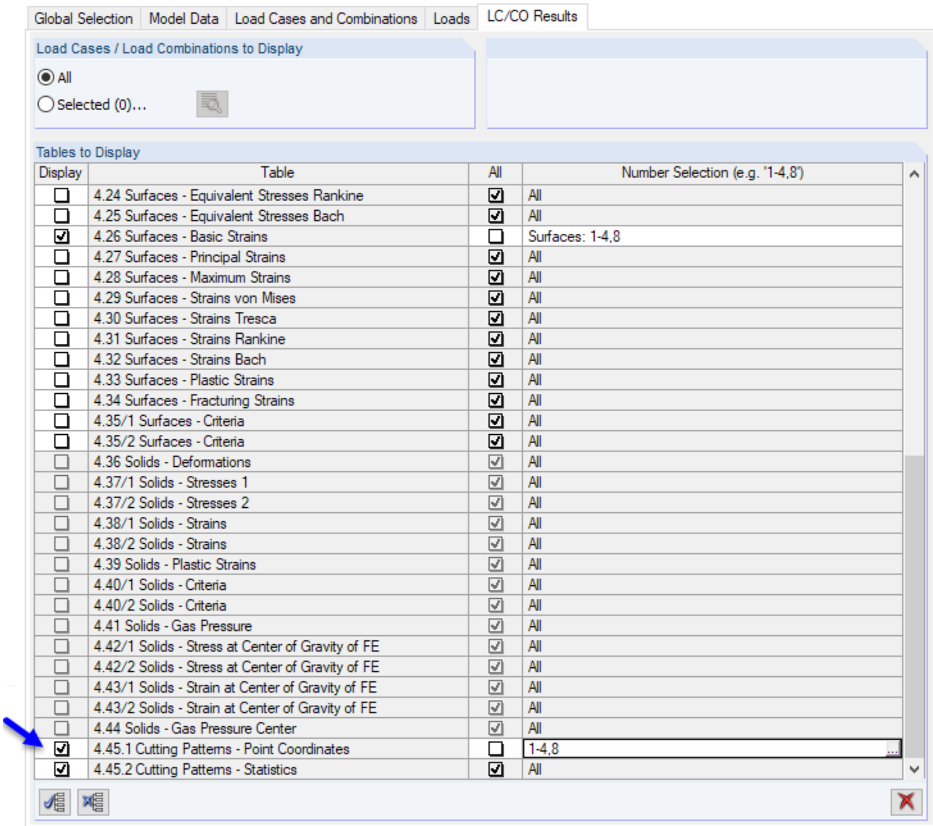
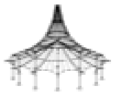


Figure 5.24: Selection of cutting patterns in *Printout Report Selection* dialog box

If you do not want *All* cutting patterns to appear in the printout report, clear the corresponding check box. Then, enter the numbers of the relevant objects, or select them graphically by clicking the button.

The point coordinates and graphics of the selected cutting patterns are incorporated in the printout report, as seen in Figure 5.25.



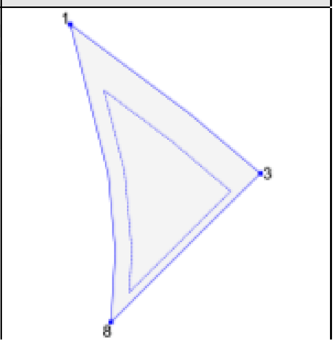
Engineering Office Bavaria Construction
Joseph street, 12345 Rainbow Valley

Page: 11/19
Sheet: 1
MODEL

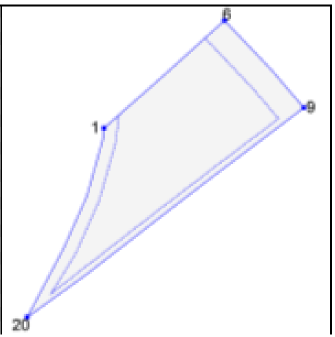
Project: | Model: CuttingPattern | Date: 25.04.2017

4.45.1 CUTTING PATTERNS - POINT COORDINATES

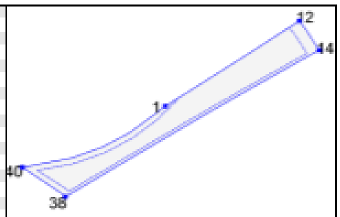
Pattern No.	Point No.	Coordinates without overlaps x [m]	Coordinates without overlaps y [m]	Coordinates with overlaps x [m]	Coordinates with overlaps y [m]
1	1*	-0.334	-0.607	-0.540	-1.011
	2	0.071	-0.303	0.195	-0.460
	3*	0.439	0.008	0.620	-0.101
	4	0.401	0.047	0.436	0.082
	5	0.184	0.263	0.219	0.298
	6	0.093	0.353	0.128	0.389
	7	-0.052	0.498	-0.017	0.534
	8*	-0.180	0.627	-0.292	0.809
	9	-0.165	0.373	-0.265	0.374
	10	-0.209	-0.113	-0.308	-0.096
	11	-0.334	-0.607	-0.540	-1.011



Pattern No.	Point No.	Coordinates without overlaps x [m]	Coordinates without overlaps y [m]	Coordinates with overlaps x [m]	Coordinates with overlaps y [m]
2	1*	-0.399	-0.229	-0.512	-0.128
	2	-0.263	-0.349	-0.263	-0.349
	3	-0.109	-0.486	-0.109	-0.486
	4	-0.014	-0.571	-0.014	-0.571
	5	0.214	-0.775	0.214	-0.775
	6*	0.255	-0.812	0.404	-0.946
	7	0.270	-0.797	0.411	-0.940
	8	0.616	-0.434	0.765	-0.568
	9*	0.814	-0.200	1.007	-0.282
	10	0.574	-0.018	0.604	0.022
	11	0.487	0.047	0.518	0.087
	12	0.387	0.123	0.417	0.163
	13	0.195	0.269	0.225	0.309
	14	-0.056	0.461	-0.026	0.501
	15	-0.084	0.482	-0.053	0.522
	16	-0.107	0.501	-0.077	0.540
	17	-0.366	0.701	-0.336	0.740
	18	-0.528	0.827	-0.498	0.866
	19	-0.663	0.932	-0.632	0.971
	20*	-0.919	1.130	-1.097	1.306
	21	-0.730	0.827	-0.818	0.780
	22	-0.549	0.421	-0.644	0.387
	23	-0.426	-0.016	-0.524	-0.036
	24	-0.399	-0.229	-0.512	-0.128



Pattern No.	Point No.	Coordinates without overlaps x [m]	Coordinates without overlaps y [m]	Coordinates with overlaps x [m]	Coordinates with overlaps y [m]
3	1*	-0.189	-0.144	-0.401	-0.012
	2	0.086	-0.316	0.086	-0.316
	3	0.230	-0.406	0.230	-0.406
	4	0.404	-0.516	0.404	-0.516
	5	0.681	-0.691	0.681	-0.691
	6	0.706	-0.707	0.706	-0.707
	7	0.735	-0.726	0.735	-0.726
	8	1.002	-0.897	1.002	-0.897
	9	1.204	-1.027	1.204	-1.027
	10	1.310	-1.095	1.310	-1.095
	11	1.401	-1.154	1.401	-1.154
	12*	1.654	-1.318	1.822	-1.427
	13	1.767	-1.163	1.932	-1.275
	14*	1.944	-0.884	2.145	-0.939
	15	1.743	-0.771	1.768	-0.728
	16	1.563	-0.670	1.587	-0.627
	17	1.239	-0.487	1.263	-0.444
	18	1.229	-0.482	1.254	-0.439
	19	1.222	-0.478	1.247	-0.435
	20	0.913	-0.303	0.938	-0.259
	21	0.689	-0.175	0.713	-0.131
	22	0.598	-0.123	0.623	-0.080
	23	0.417	-0.019	0.442	0.024
	24	0.273	0.064	0.298	0.107
	25	0.006	0.219	0.031	0.262
	26	-0.069	0.263	-0.044	0.306
	27	-0.155	0.314	-0.130	0.357
	28	-0.435	0.479	-0.410	0.522
	29	-0.635	0.597	-0.609	0.640
	30	-0.828	0.711	-0.803	0.754
	31	-1.083	0.861	-1.058	0.904
	32	-1.242	0.955	-1.217	0.998
	33	-1.364	1.027	-1.339	1.070
	34	-1.641	1.191	-1.615	1.234
	35	-1.673	1.210	-1.648	1.253
	36	-1.792	1.281	-1.767	1.324
	37	-1.965	1.377	-1.930	1.420
	38*	-2.006	1.407	-2.058	1.496
	39	-2.067	1.361	-2.118	1.440
	40*	-2.526	1.057	-2.771	1.007
	41	-1.928	0.948	-1.954	0.851
	42	-1.402	0.762	-1.442	0.670
	43	-0.950	0.522	-1.004	0.437
	44	-0.567	0.240	-0.632	0.163



RFEM 5.16.01 - General 3D structures solved using FEM | www.dlubal.com

Figure 5.25: Point coordinates and graphics of cutting patterns in printout report

DXF export

Exporting cutting patterns

The flattened geometry of the cutting pattern can be exported in a DXF file by selecting

File → Export

on the RFEM menu or by clicking the corresponding button in the toolbar.

The *Export* dialog box familiar from RFEM appears.

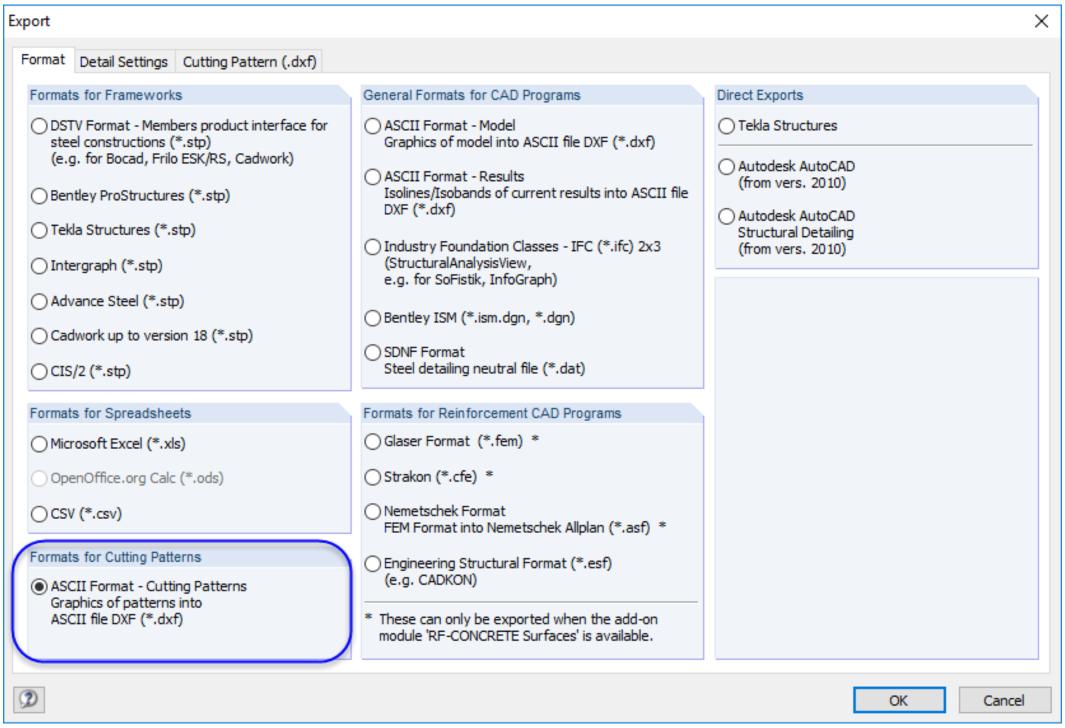


Figure 5.26: Dialog box *Export*

To export cutting patterns, select the *ASCII Format* in the *Formats for Cutting Patterns* dialog section.

In the *Cutting Pattern* tab, you can check and adjust the *Number* of patterns per row and the *Distance* between the patterns, if necessary.

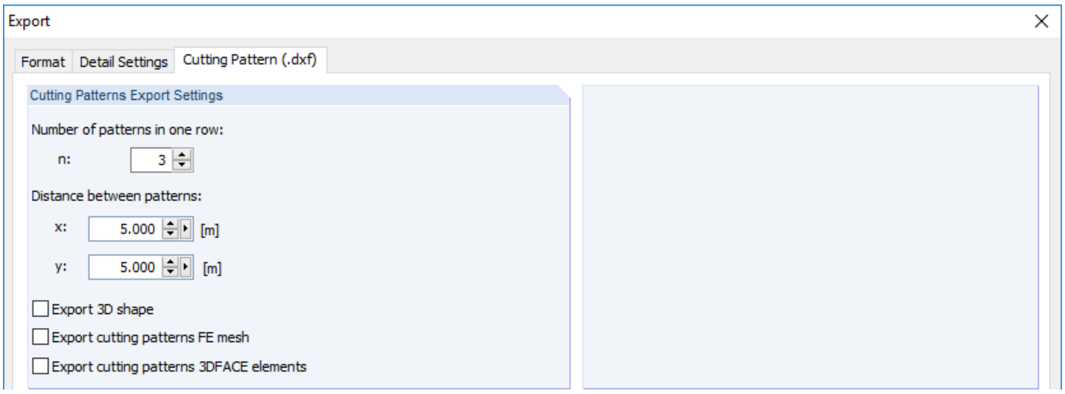
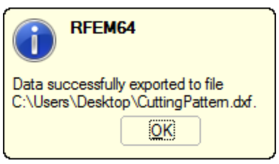


Figure 5.27: Dialog box *Export*, tab *Cutting Pattern (.dxf)*



Click [OK] to start the export. Then, in the Windows *Save As* dialog box, enter a file name and the location where store the DXF file.

A corresponding message appears when the export has been successful.

Now you can open the DXF file in a CAD program where it can be further edited.

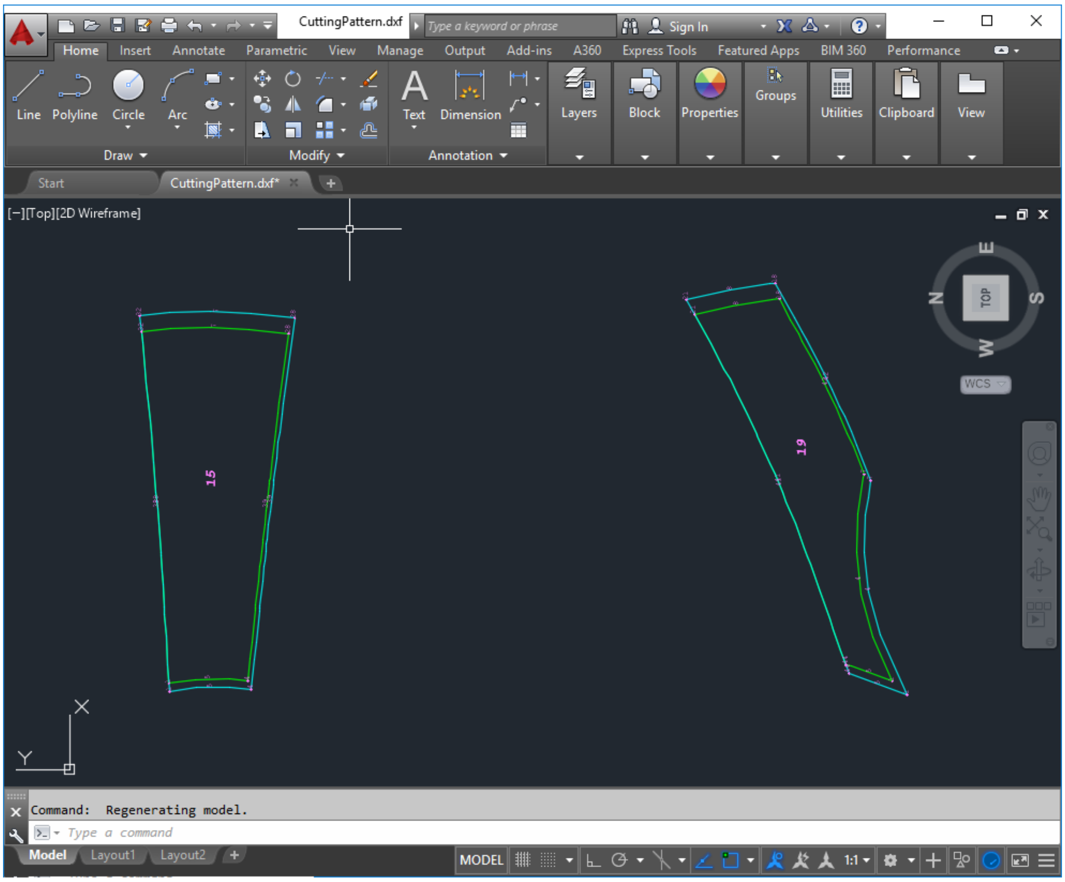


Figure 5.28: Cutting patterns in AutoCAD

Cutting lines as well as allowance lines are organized in different layers.

Exporting shape of model

If you want to export the geometry of the deformed mesh, use the *ASCII Formal - Model* option which is available in the *General Formats for CAD Programs* section of the *Export dialog box* (see Figure 5.28). You will then be able to use the shape of the model in a CAD program.

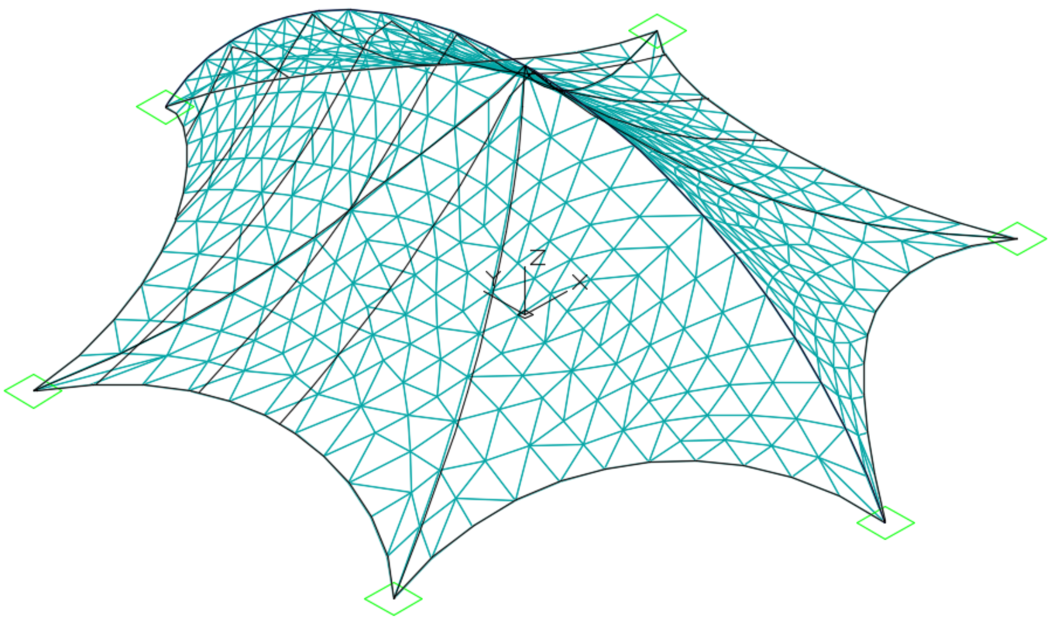


Figure 5.29: FE mesh of deformed model in AutoCAD

Index