Strut-and-Tie Model Concrete BSM+

Table of contents

Basic documentation - overview

In addition to the individual program manuals, you will find basic explanations on how to operate the programs on our homepage www.frilo.eu in the download area (Manuals).

Tip: Go back - e.g. after a link to another chapter/document - in the PDF with the key combination "ALT" + "arrow key left"

FAQ - Frequently asked questions

We have answered frequently asked questions about our programs on our homepage in the
‣ Service ‣ Support ‣ FAQ area.

Application options

The BSM+ program generates strut-and-tie models for the design of reinforced concrete components. Right-angled structures with any number of re-entrant corners and recesses can be calculated.

In the Assistant, the user selects a basic system suitable for his application, which he can then further edit and adapt with the editing tools.

In the first version of the program, the focus is on generating rigidity-optimized strut-and-tie models for a given component geometry and load. In the future, it is conceivable that proof of the load capacity of the compression and tension struts and the nodes will be carried out according to DIN EN 1992-1-1 + NA, Section 6.5.

The generated strut-and-tie models can be influenced by changing the calculation parameters ("Fineness of the mesh" and/or the "Mesh density of the bars").

Calculation bases

The calculation is based on a topology optimization algorithm based on the so-called Ground Structure Method, see for example [1]. An initial basic structure is referred to as a ground structure, which consists of truss rods connected to one another in nodes. Each bar is assigned a design variable xi ∈ [0,1], which can be interpreted as a "normalized cross-sectional area". If xi = 1, the bar is "activated" and if xi = 0, it is "deactivated".

Note: In the program, the term „Ground Structure" has been replaced by the term "Mesh density of the bars".

The optimization problem consists of an objective function and a restriction function. The objective function is to maximize the elastic stiffness of the truss structure. Since the solution without a restriction function is trivial (maximum stiffness through maximum number of members = Ground Structure), the number of bars that can be used from the ground structure is limited [2].

The static calculation is based on the finite element method. Starting from the initial bar structure, an optimization algorithm, based on the optimality criteria method [3], iteratively reduces the number of "active" bars over the distribution of design variables to the number of bars allowed by the constraint, such that the elastic stiffness is maximized. The following applies: the more bars that can be used, the stiffer the system becomes, but the more complex the resulting strut-and-tie model becomes.

After the search algorithm has converged, the generated truss is first smoothed using a filter method in such a way that numerical effects are eliminated (e.g. bars with a very small normalized cross-sectional area xi ≠ 0 that hang freely in space) and the result as a strut-and-tie model for the design of the component can be used.

Two quality criteria are checked for this:

▪ Crossing compression struts: if compression fields cross in a reinforced concrete component, a compression node is created at the crossing point. The method used is not able to inherently consider this effect. Crossing compression struts in the strut-and-tie model are always "one behind the other", so that no new node is created. This means that results can be calculated that are statically consistent but cannot occur in reality. Such results are therefore marked as "invalid" for the design of the component. The result is then not presented to the user.

▪ According to the work on the strut-and-tie model theory by Schlaich/Schäfer [4], acute angles <30° between compression and tension struts are unrealistic or associated with major incompatibility of the deformations. Therefore, the strut-and-tie model found is analyzed accordingly in the program. If there are acute angles <30°, the result is displayed, but accompanied by a corresponding warning message to draw the user's attention to the potentially problematic use.

Literature

[1]   G. Rozvany, M. P. Bendsoe, U. Kirsch. Layout Optimization of Structures. Applied Mechanics Reviews, 48(2):41-119, 1995. ISSN 0003-6900. doi: 10.1115/1.3101884

[2]   A. Asadpoure, J. K. Guest, and L. Valdevit. Incorporating fabrication cost into topology otimization of discrete structures and lattices. Structural and Multidisciplinary Optimization, 51:385-396, 2015.

[3]   M. P. Bendsoe and O. Sigmund. Topology optimization: Theory, methods, and applications. Springer, Berlin, 2004.

[4]   J. Schlaich and K. Schäfer. Konstruieren im Stahlbetonbau: Beton-Kalender 2001. Ernst & Sohn, Berlin, 311-492, 2001.

Input

The Wizard - Fast entry of a basic system

The Wizard opens by default when creating a new item. If necessary, this can also be switched off by deactivating the check mark in the lower window area. In the Assistant, predefined basic systems can be selected using symbols.

Input options directly in the graphic

It is planned to adopt prefabricated systems and edit them accordingly using the "Add geometry", "Openings", "Loads" and "Supports" functions.

Additional geometry, openings, supports and loads can be added via the right-click context menu or the ribbon bar.
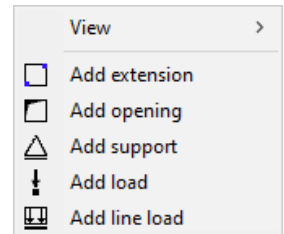
Interactive texts

As in all PLUS Programs, the texts displayed in the graphic at the top left are interactive and can be clicked on. In this way, dialogs are called up directly in the graphic, which can otherwise only be reached via the left menu. See also Basic operating instructions.
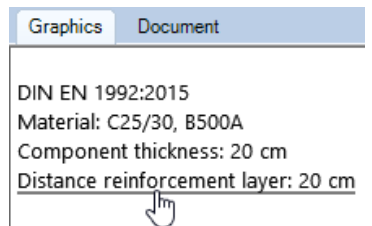
Interactive component edges

By clicking on a component edge, an additional dimension chain appears. The length of the edge can be changed using this dimension chain.
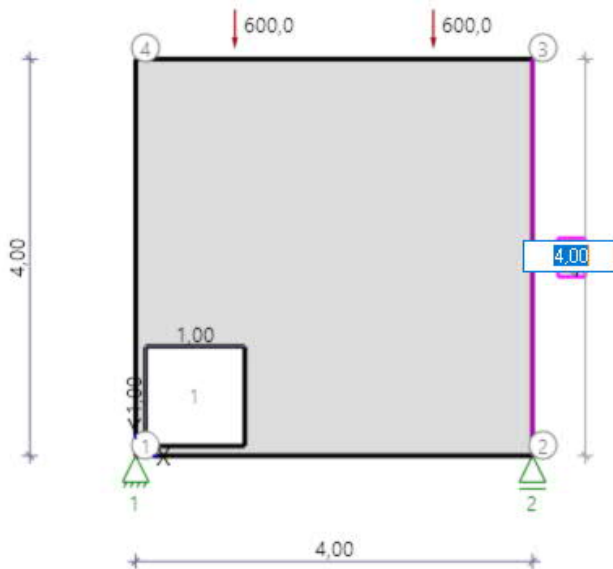
For graphic input in the PLUS Programs, see also Basic operating instructions.
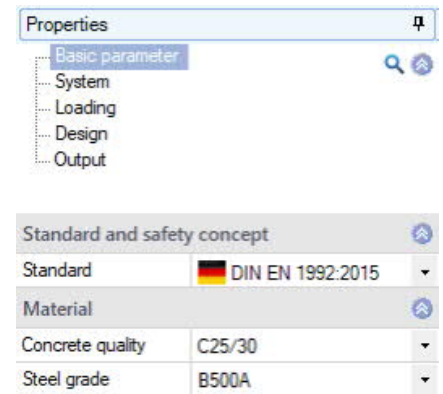
Basic parameters

Information on the standard and the material can be entered under the basic parameters.

Standard and safety concept

Standard: definition of the design standard with national annex.

Material

Depending on the selected standard, the corresponding material parameters - concrete and steel quality - are listed for selection.

System

You can enter geometries, openings, supports and loads directly in the graphic using the right-click context menu, using the functions in the ribbon bar or alternatively using a table (tabs below the graphic).
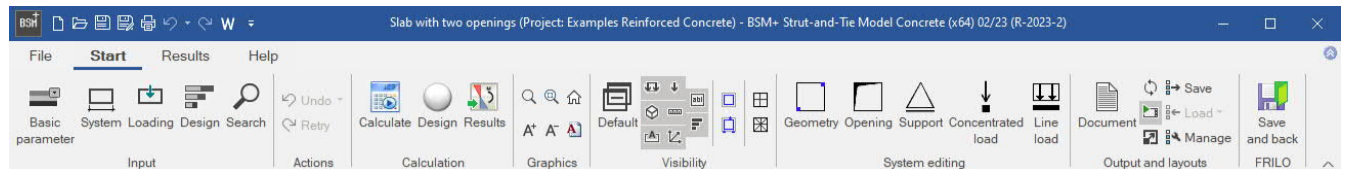
Fig. Ribbon bar

In addition, the plate thickness and the concrete cover can be adjusted under "System".

Remarks

Remarks can be entered via a remarks editor, which can then be seen in the output at the relevant point.
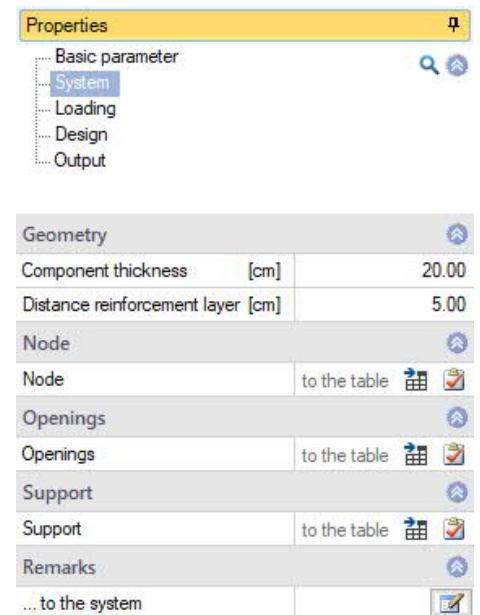
Nodes

The table of nodes is for informational purposes only. No further nodes can be added via the table.

	X	Y	
	[m]	[m]	
1	0.00	0.00	
2	8.00	0.00	
3	8.00	4.00	
4	0.00	4.00	

Add/extend geometry

The geometry can be extended by graphical input using the "Add extension" function.
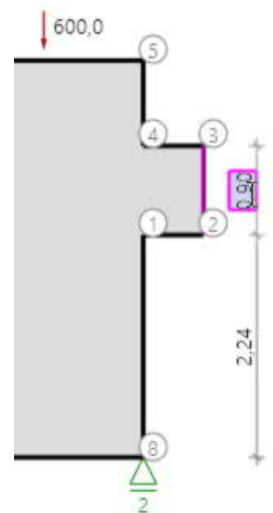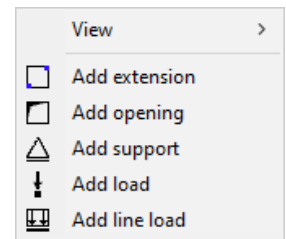
Only rectangular contours can be added.

Graphic input

Call via context menu ▸ Add extension:

Define the first point of the rectangle with a mouse click, then pull out the rectangle and define the second point diagonally opposite with another mouse click.

Note: *The additional rectangle contour must intersect the existing contour so that the two contours are merged.*

By clicking on a component edge, an interactive dimension chain appears. The dimension of the component edge can also be adjusted via this dimension chain.

Openings

For <u>tabular</u> input, click on the "Openings" tab below the graphic. You can add or delete input lines using the buttons on the right of the table.

Only rectangular openings can be added.

	X	Y	Width	Height
	[m]	[m]	[m]	[m]
1	1.00	1.50	2.00	1.00
2	5.00	1.50	2.00	1.00

Graphical input of an opening

Openings can also be added via graphical input (context menu).

The graphical input of an opening rectangle works in principle as described under ▸ <u>Nodes</u> ▸ Graphical input.

If the opening element is selected with the left mouse button, the <u>dimensions of the opening element</u> and additional dimension chains appear. The size of the opening can be edited by clicking on the dimension. The edge distance/position of the opening element can be precisely defined using the additional dimension chains.
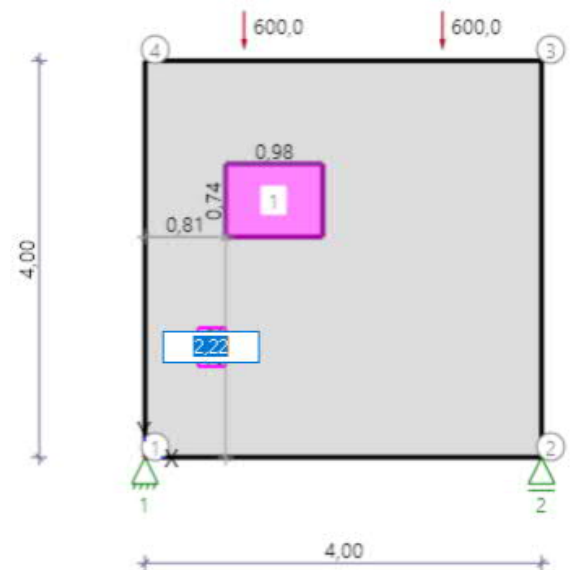
Supports

For the <u>tabular</u> input, click on the "Supports" tab below the graphic.
You can add or delete input lines using the buttons on the right of the table.

Concentrated loads can be added in vertical and horizontal directions. For the calculation, the load must be in contact with the outer contour.

	X	Y	Cx	Cy
	[m]	[m]		
1	0.00	0.00	starr	starr
2	8.00	0.00	frei	starr

Graphical input of a support

Supports can also be added via graphical input (context menu).

Select "Add support" from the context menu and click on the location for the support. The numbered support symbol is displayed.

You can also edit the properties of the support (position, free, rigid) via the context menu (right-click on the support symbol).
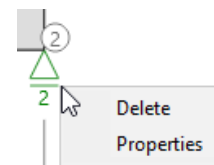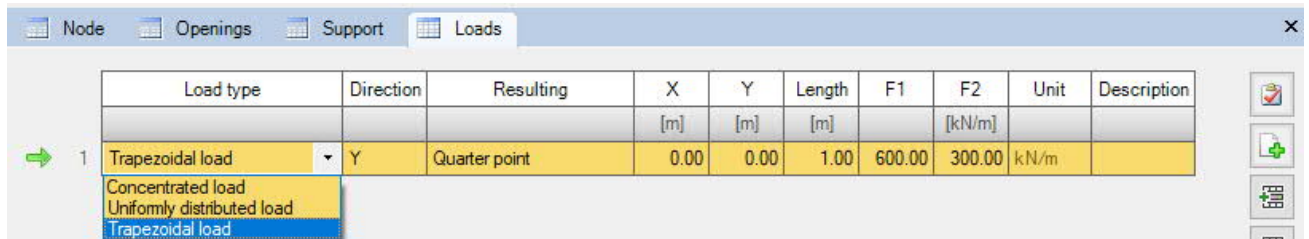
Load

Loads can be entered directly in the graphic using the right-click context menu, using the function in the ribbon bar or alternatively using a table (tabs below the graphic).

Only concentrated loads can be entered as design loads in vertical and horizontal directions. Distributed loads must - as known from the strut-and-tie model theory - be summarized beforehand in equivalent concentrated loads.

For <u>tabular input</u>, click on the "Loads" tab below the graphic. You can add or delete input lines using the buttons on the right of the table.

Load type	Direction	Resulting	X	Y	Length	F1	F2	Unit	Description
			[m]	[m]	[m]		[kN/m]		
1 Trapezoidal load ▾	Y	Quarter point	0.00	0.00	1.00	600.00	300.00	kN/m	
Concentrated load									
Uniformly distributed load									
Trapezoidal load									

Graphical input of loads

Loads can also be added via graphical input (context menu).

Select "Add load" from the context menu and click on the position for the load. The load value is displayed and can be edited.

You can also <u>edit</u> the properties of the load (x/y coordinates, direction, load value F) via the context menu (right mouse click on the load).

Design

Under the menu item "Design" you can choose between the "Automatic determination of the parameters" and a manual parameter input (remove checkbox).
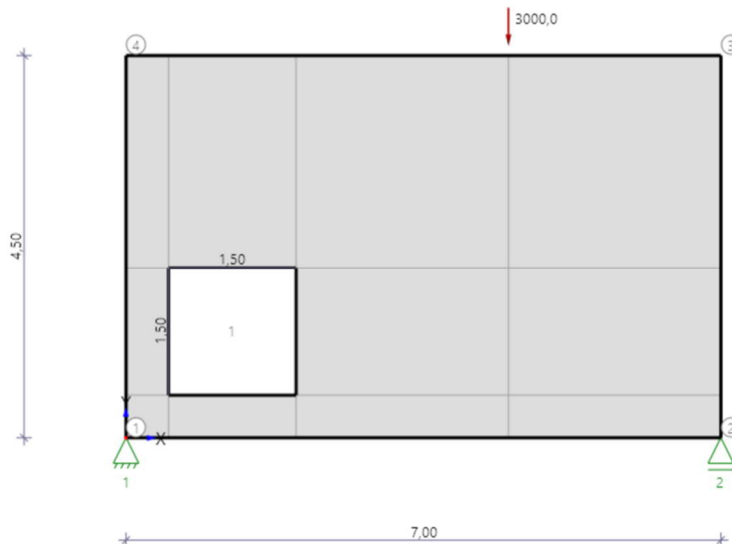
## Definitions of terms

Maximum edge length of the mesh elements:

The fineness of the mesh in which bars can be created can be influenced via the "*Maximum edge length of the mesh elements*". The mesh is comparable to an FE mesh. Supports, geometric indentations, openings and loads are constraint points for the mesh of the system. In contrast to a finite element calculation, however, when searching for practicable strut-and-tie models, it is advisable to use coarser meshes in order to obtain strut-and-tie models with as little complexity as possible for the design of the component, so that the computational effort is kept within limits.

In general, the following applies: the finer the mesh, the finer the strut-and-tie model generated, and the stiffer and less cracked the load transfer in the component.

The coarsest mesh for this system is shown in the following figure. The system is only subdivided by the constrained geometry.

Properties			⊞
Basic parameter			
System			🔍 ⊗
Loading			
Design			
Output			

Calculation parameters			⊗
Automatic determination of the parameters			☐
Fineness of the mesh	Manually	Default	Fine
Max. edge length of the mesh elements	[m]		8.00
Determine mesh density automatically			☐
Mesh density of the bars			3
Remarks			⊗
... to the results			📝

Note: For visual control, the underlying mesh can be displayed in the ribbon bar in the "Visibility" group.
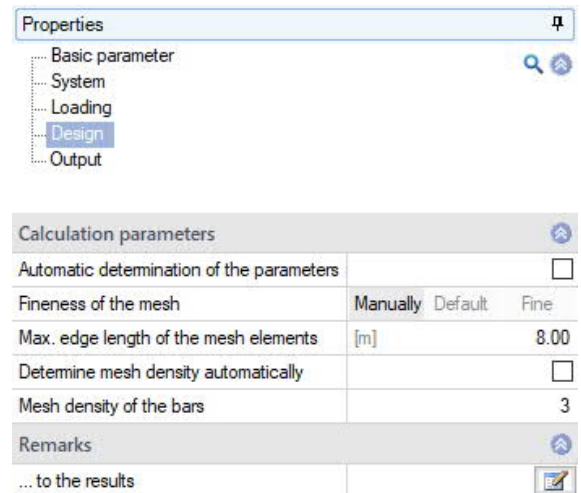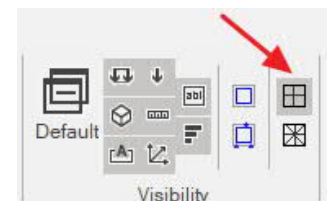
Mesh density of the bars:

The mesh density of the bars indicates how many bars are available for the algorithm to guide the load through the component. This means that all of these bars can be used to determine the strut-and-tie model.

The following applies: the greater the meshing density of the bars, the more bars are available for the load transfer, and the more complex the strut-and-tie model generated tends to be.
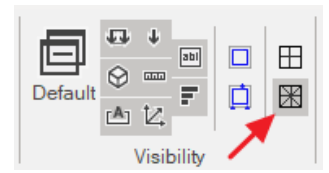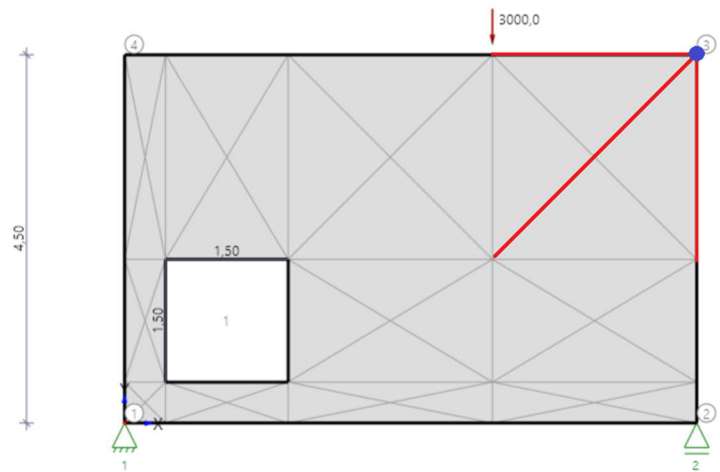
Note: For visual control, the mesh density of the bars can be displayed in the ribbon bar in the "Visibility" group.

If the value 1 is assumed for the "Mesh density of the bars", then a bar is generated from each node of the mesh to the neighboring node (Fig. right).

Legend:
Blue: regarded node

Red: bar to neighboring node

If the value 2 is assumed for the "Mesh density of the bars", the neighboring nodes of the neighboring node are also connected by bars (Fig. on the right).

Legend:
Blue: regarded node

Red: additional bars to the second neighboring node
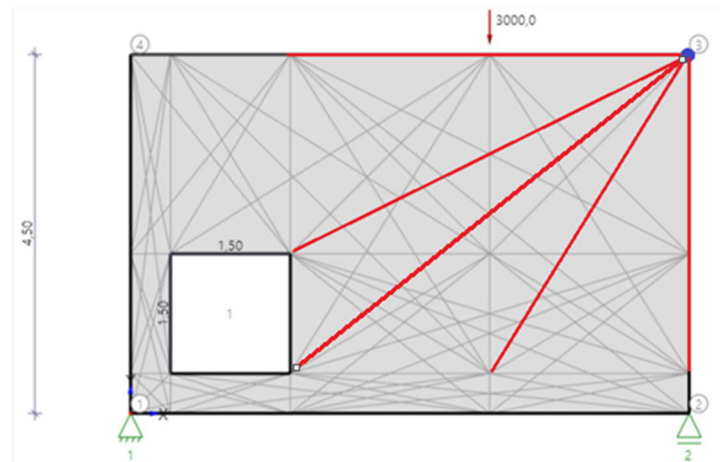
A value of 3 means that the neighboring nodes of the second order of the neighboring node are also connected by bars.

The figure on the right shows a mesh density of 3.

Legend:
Blue: regarded node

Red: additional bars to the third neighboring node
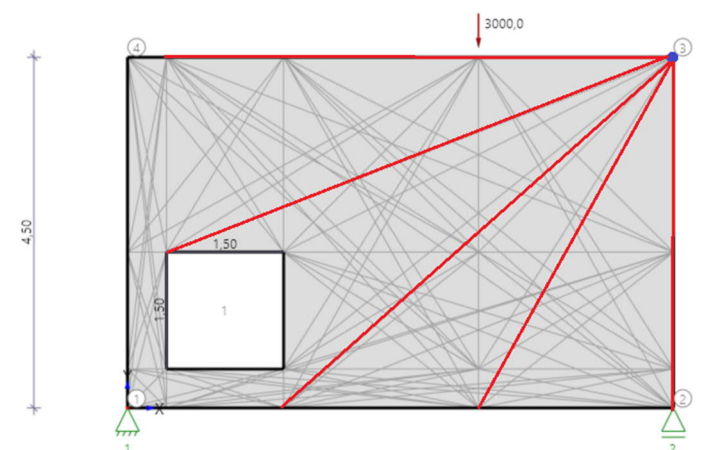
Automatic determination of the parameters

By default (<u>option activated</u>), the calculation parameters are automatically determined by the program based on the geometry, supports and load position. The "*maximum edge length of the mesh elements*" is always the same as the maximum contour length in the system. Supports, geometric indentations, openings and loads represent additional constraint points and subdivide the system into a finer mesh.

In addition, the "*Mesh density of the bars*" is determined automatically. The program automatically searches for possible strut-and-tie models for a mesh density spectrum (1-5).

Note: *This approach is the default, as a coarse mesh usually produces the most practical results. The settings generate simple, stiffness-optimized strut-and-tie models that are well suited for designing the component.*

If the program does not find a strut-and-tie model via the "*automatic determination of parameters*" or the quality of the generated strut-and-tie model is insufficient, the checkmark for this can be deactivated. In this case, further calculation parameters become visible. These can then be set freely.
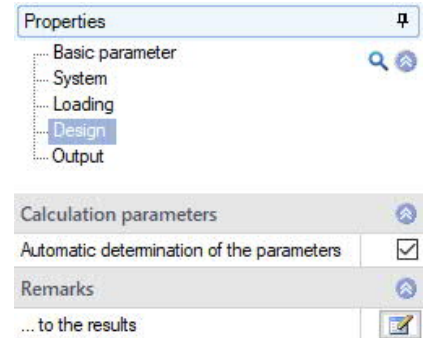
Manual/automatic determination of the parameters

If the option is <u>deactivated</u>, you can choose between "Manual", "Default" and "Fine" for the "*Fineness of the mesh*":

Manually: Maximum edge length freely selectable.

Default: Automatic "minimum mesh density", analogous to the automatic determination of the parameters.

Fine: Generates a mesh twice as fine as "Standard".

In addition, the option for "*Determine mesh density automatically*" can be deactivated. This means that the mesh density of the bars can be set freely, for which a strut-and-tie model is finally determined.
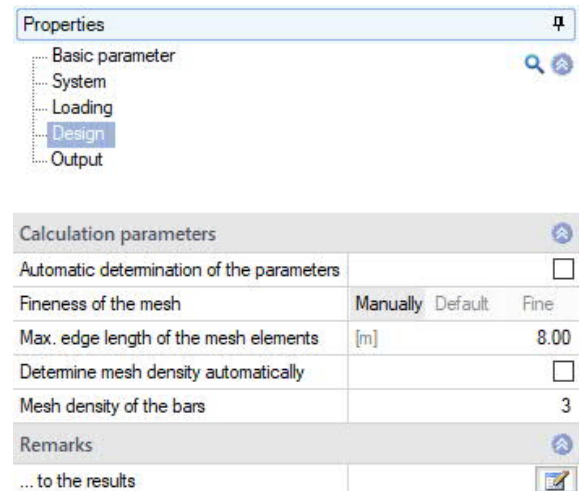
Output

Output of system data, results and graphics.

Use the "Calculate" button to start the calculation.

Call up the output document via the "Document" tab. You can view and print the issue in PDF format here.

See the Output and Printing document.

The scope of the output can be specified individually using the options offered.