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"

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 ϵ [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Note: In the program, the term „Ground Structure" has been replaced by the term "Mesh density of the bars".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 Assistant/Wizard - Fast entry of a basic system

The Assistant (former 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.
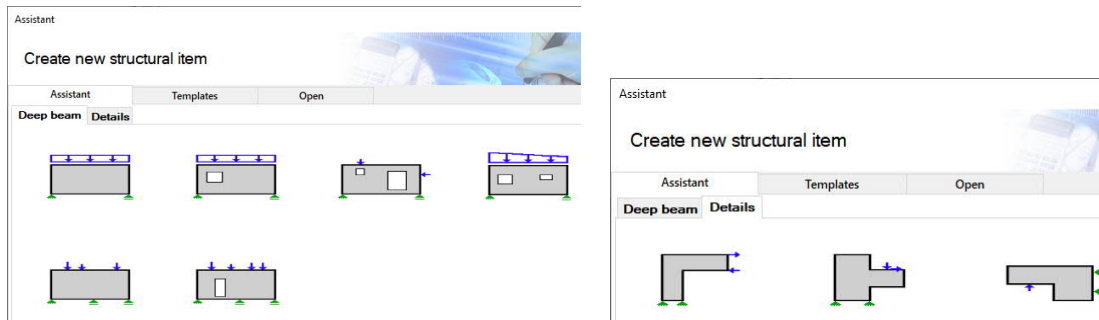
Input options directly in the graphic

After selecting a basic system in the assistant, adjustments can be made using the "Add extension/geometry", "Openings", "Loads" and "Supports" functions. These editing functions can be accessed via the right-click context menu (fig. right) or the ribbon bar.

See System. For graphical input in the PLUS programs, see also Basic operating instructions.
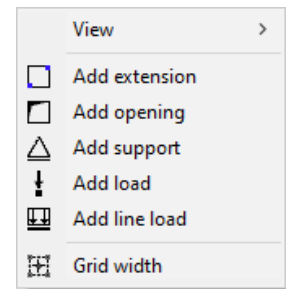
Interactive texts

As in all PLUS Programs, the texts displayed in the graphic at the top left (& right "Grid width) 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.
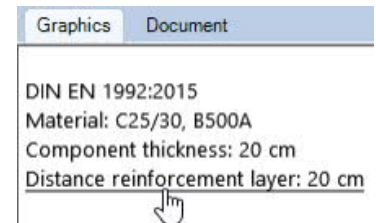
Interactive component edges

By clicking on a component edge, an additional dimension chain appears. The length of the edge can be changed using this dimension chain.

Guidance through the program

The information line above the graphic (in the signal colors red/grey/green) supports you with an overview of the processing phase you are in and also allows you to switch directly to a previous processing phase by clicking on it.

Modeling ▶ Selection ▶ Truss processing ▶ Results
Edit your truss or continue (click on "Check").

Gray = still pending processing, red = current processing phase, green = already processed

See Start Calculation
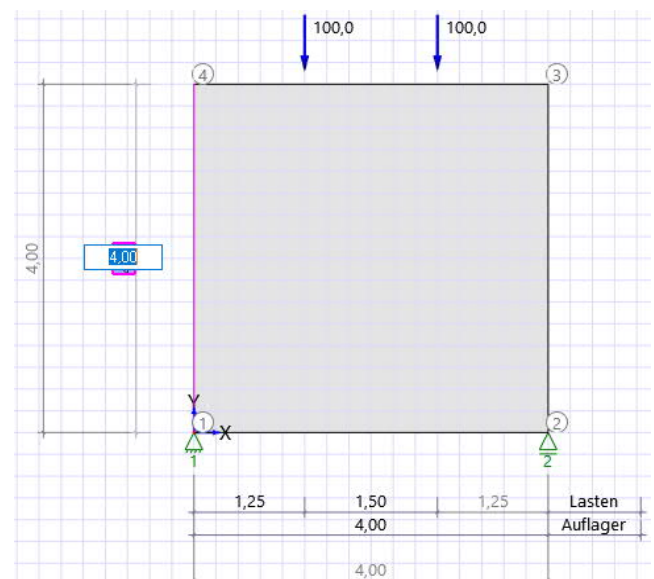
Basic parameters

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.

Visibility

The loads, the coordinate axes, the design space contour, the mesh and the mesh density of the members and the grid can be made visible.

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).
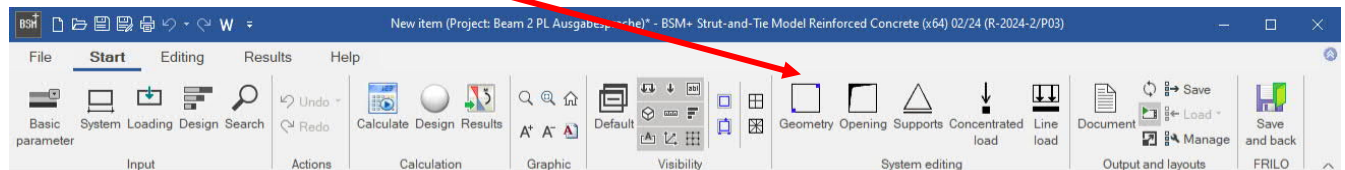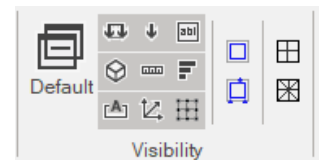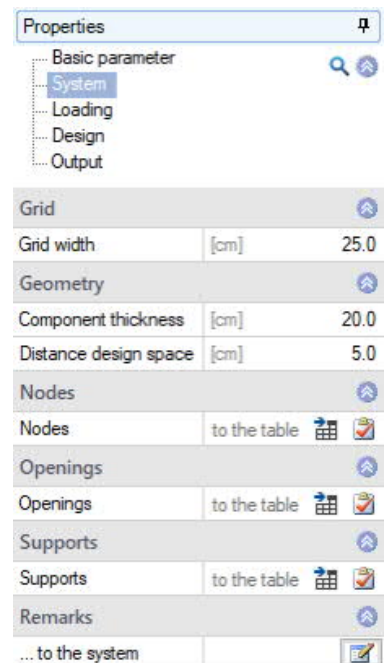
Fig. Ribbon bar

In addition, the grid width, component thickness distance of the design space contour to the component edge can be adjusted under "System".
The design space describes the area in the component that is available for the framework modeling.

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.

Grid and grid snap

A grid is available in the "Graphics" tab, which can support you in the graphic creation of the component. The grid can be hidden under "Visibility".

The grid width can be defined either using the interactive text at the top right or in the left-hand menu tree under "System". If the grid is visible, the grid width can also be adjusted using the context menu (right mouse button).

The grid points can be snapped.
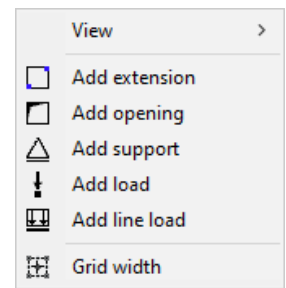
Graphic input

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.
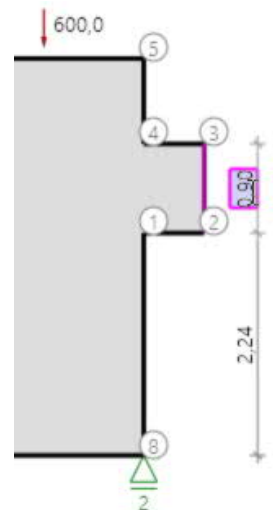
Openings

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.
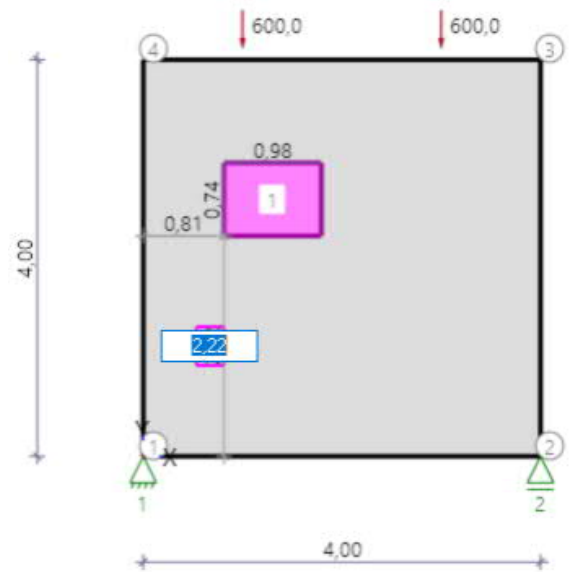
Supports

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).
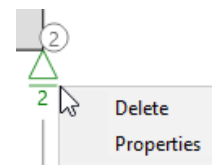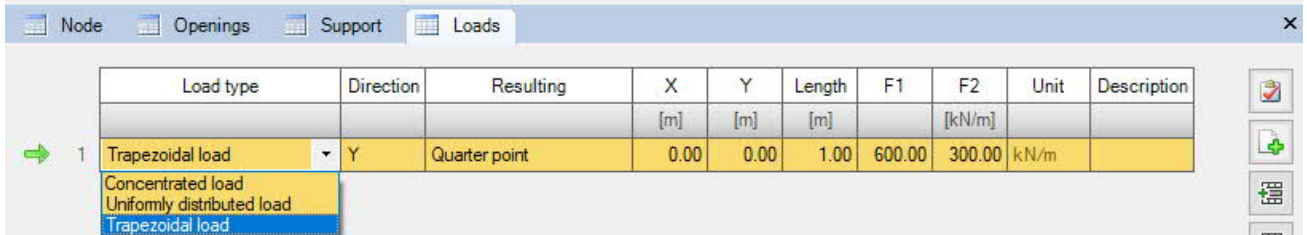
Load

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 tabular input, 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									

Note: For the calculation, the load must be in contact with the outer contour.

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 edit the properties of the load (x/y coordinates, direction, load value F) via the context menu (right mouse click on the load).

Design / Calculation

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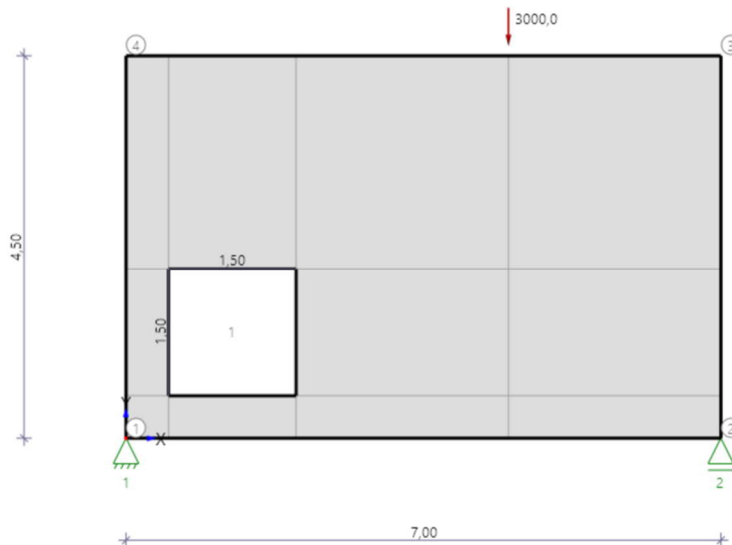
Definitions of terms

### 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.
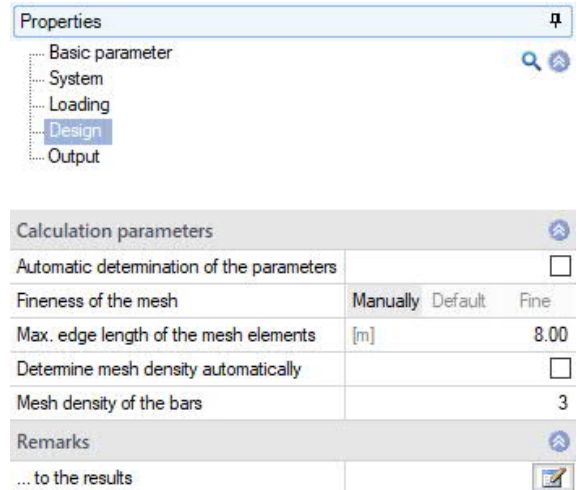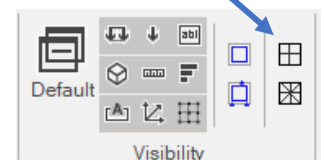
Mesh density of the bars:

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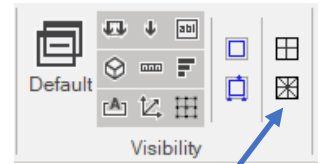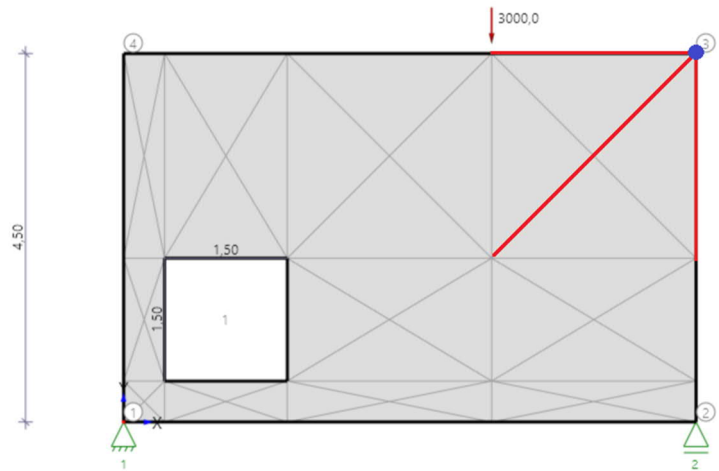
Note: *For visual control, the mesh density of the bars can be displayed in the ribbon bar in the "Visibility" group.*

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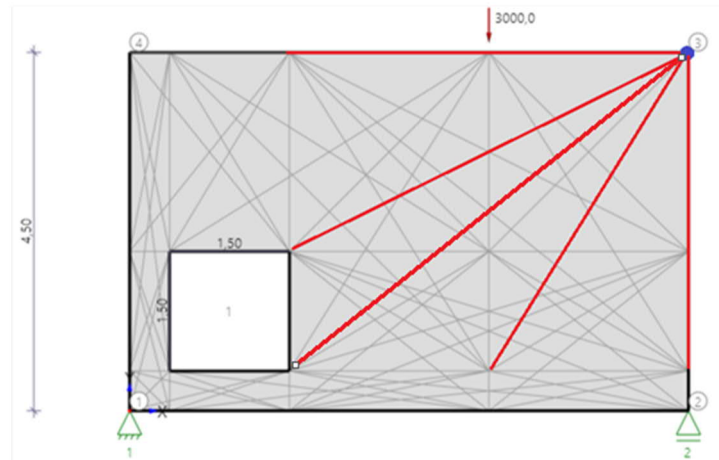
A value of 3 means that the neighboring nodes of the second order of the neighboring node are also connected by bars.

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

Legend:
Blue: regarded node

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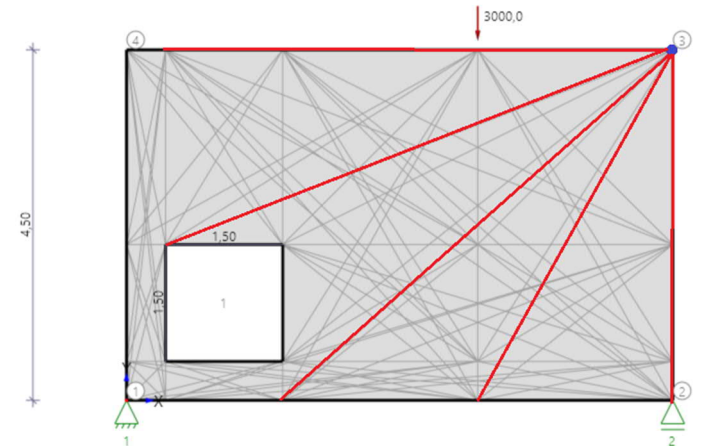
Automatic determination of the parameters

By default (option activated), 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.
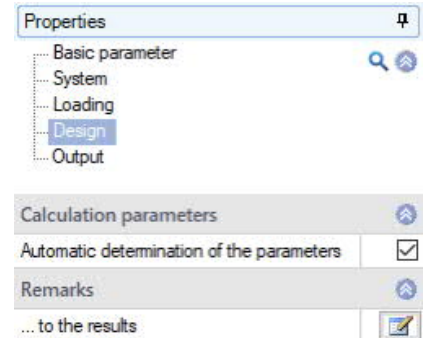
Manual/automatic determination of the parameters

If the option is deactivated, 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.
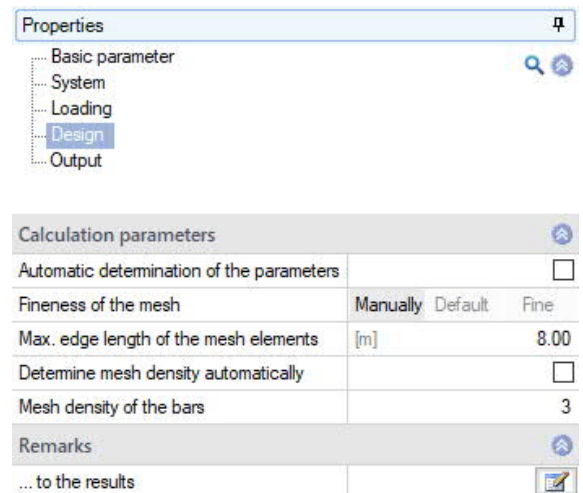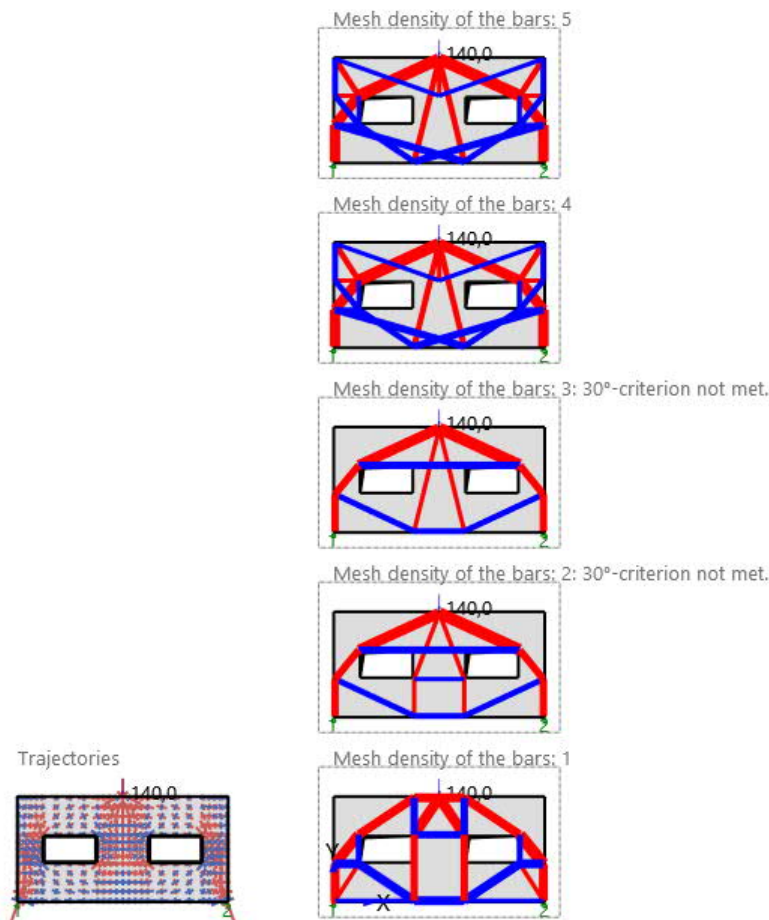
Start calculation

The strut and tie models are generated via the "Calculate" button and displayed under the Results tab - see illustration below. You can now select a model and use the Editing menu item to edit this model further.
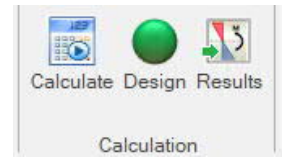
Modeling ▶ **Selection** ▶ Truss processing ▶ Results
Select a truss.

The information line above the graphic (in the signal colors red/grey/green) supports you with an overview of the processing phase you are in and also allows you to switch directly to a previous processing phase by clicking on it.
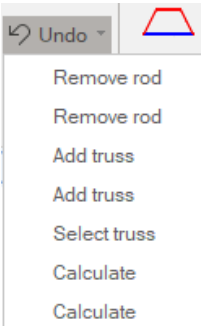
Gray = still pending processing, red = current processing phase, green = already processed

Editing

After a calculation, you can use this menu item to edit the generated/selected strut- and tie model. You can use the icons to deactivate active members, add or remove members, move nodes and start a generation of the modified framework.
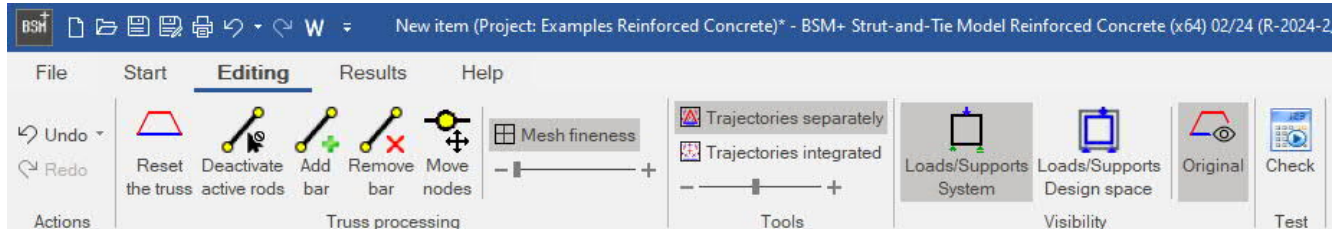

If necessary, you can reset the framework or use the undo function to jump to an individual editing step (click on the list arrow to the right of "Undo").

Reset the truss	Resets the framework to the status prior to editing. You can then start editing again.
Deactivate active rods	Deactivates all used members
Add bar	First click on the icon. Then click on one of the displayed nodes for the start of the member and also for the end of the member. You can add further members continuously.
Remove bar	First click on the icon. Then click on the members to be deleted one after the other.
Move nodes	First click on the icon. The cursor is displayed with a move symbol and you can grab a node with the mouse and move it to another grid point.
Mesh fineness	Move the slider to "+" to refine the grid. Check the grid displayed in the graphic. To reset the grid, you can click directly on "Mesh fineness".
Trajectories	By default, the trajectory is displayed in a separate image. Select "Trajectories integrated" if the trajectory and the framework to be processed are to be displayed in one image. The force lines can be displayed shorter or longer using the slider. The trajectories are displayed for information purposes and serve as an aid when creating a user-defined framework model.
Visibility	In this area you will find switches (on/off) for displaying the loads/supports.
Original	Additional display of the original framework from the first calculation.
Check	After editing, click on the "Check" icon. A second determination of a possible framework is started, taking into account the selection of members to be used. In order to be able to calculate user-defined truss systems (including kinematic ones), all truss nodes are supported by very soft springs in the X and Y directions. Furthermore, a spring is also applied in the unsupported direction of a single-value support. No spring is inserted for fixed supports. All springs are given the same compliance in each direction. This is calculated from the sum of all bar stiffnesses per bar length in relation to the total number of bars. In order to keep the spring stiffness as low as possible (no force absorbtion, only stabilization of the framework), the resulting value is divided by 1,000,000. The springs show that the framework is supported by the surrounding concrete. This means that it is also possible to calculate bar structures that would be kinematic without a spring attachment.

Results

The results of the framework generation are displayed here in the graphics window.
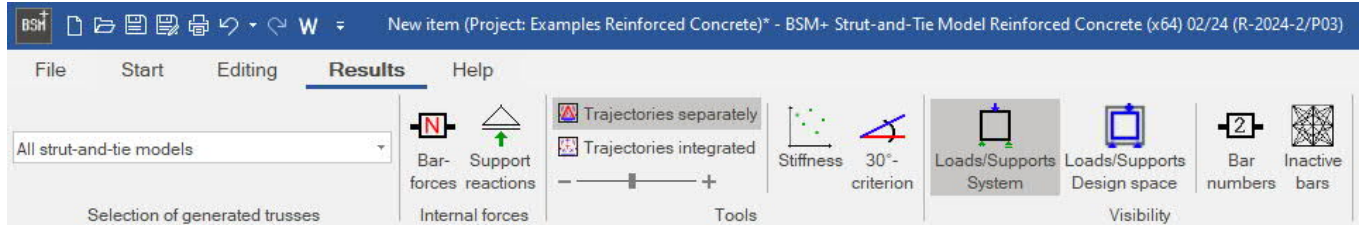
Overall comparison	All generated framework models are displayed by default (all framework models). You can also expand a list here and select an individual result for display.
Internal forces	Display (on/off) of the member forces and support reactions.
Trajectories	See Editing
Stiffness	Show/hide diagram of the scaled stiffness of the framework.
30°-criterion	Show/hide angle between two members smaller than 30°.
Visibility	Show/hide loads, supports, member numbers and inactive members.

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.