Mechanically Jointed Beams HNV+

Contents

Basic Documentation – Overview

In addition to the individual program manuals, you will find basic explanations on the operation of the programs on our homepage www.frilo.com in the Campus-download-section.

Application options

The program designs mechanically connected wooden beams.
Different timber cross-sections can be stacked and connected by means of fasteners. This increases the load-bearing capacity. Different timber materials can be used. The fasteners can have different spacing in order to be able to use them as economically as possible.

Fasteners

- Dowel pins
- Fit bolts
- Connectors (special dowels)
- Nails

Structural system

- Single-span girder
- Multi-span girder
- Cantilevers

Vertical load

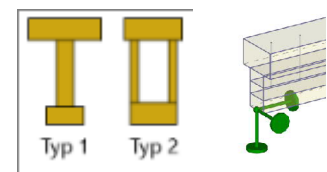- Uniformly distributed linear load
- Concentrated load
- Trapezoidal load

Cross-sections

Type 1:

- Rectangular cross-sections

- Any number of stacked cross-sections

Type 2:

- Cross-section with upper and lower chord and webs

- Different materials

Standards

- DIN EN 1995:2013

- ÖNORM EN 1995:2019

- PN EN 1995:2010

- BS EN 1995:2019

- EN 1995:2014

Design

The design is performed using the method of shear analogy. Two virtual beams are generated which are coupled in their deformation but have different stiffnesses. The stresses of the two beams are then used to perform the verifications.

In the ultimate limit state, both the initial and the final state are calculated. This provides for compliance with the standard, which requires that when the distribution of internal forces is influenced by the stiffness distribution, the deformation coefficient affects the moduli of elasticity, shear, and displacement.

Data entry

General notes concerning the data-entry fields

This program allows the calculation in accordance with various standards and National Annexes. Some of these standards differ considerably in regard to the load application, the combination rules, the determination of the decisive internal forces and the verification process.

Therefore, the data-entry fields and options can differ from those described below depending on the selected standard.

Basic parameters

You can select the standard and the service class.

Average kmod for wind — If the option is selected, the modification coefficient kmod for wind is used as the mean value for the classes of the load duration short and very short.

$\psi 2$ for crane loads — determines the combination coefficient $\psi 2$ for crane loads (ratio of permanent share to total crane load).

Cite in Wind Zone 3 or 4 — Select this option if the building is located in wind zone 3 or 4. In this case the action 'snow' is not considered as an accompanying action to the leading action 'wind'.

Equal γG for permanent loads — if this option is checked, all permanent loads or load cases are applied together with the same partial safety factor (γG,sup or γG,inf), otherwise permanent loads are combined independently with their lower and upper partial safety factors.

In order to make the input easier, the material and geometry can also be set to "same" for all components.

Structural system

Recommendation: Enter the structural system directly via the context-sensitive GUI!

Lateral restraint (Side fixure)

Continuously supported	the beam is continuously supported over the entire length. The buckling length is 0.

Position of the restraints	You can define the position of the lateral restraints
	    at the supports
	    in the centre of the span
	    in the third or the quarter points
	At the distance x0: in the displayed field, enter the distance of the restraint to the left edge of the beam.
	User-defined: Enter in the table the distance of the individual restraints to the left edge of the beam (click "Edit" button).
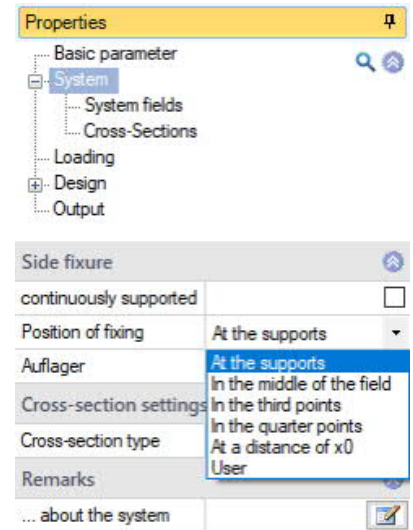
Cross-section type

Type according to the graphic shown. Depending on the type, the corresponding functions are displayed (upper chord, lower chord, web or cross-sections).

input
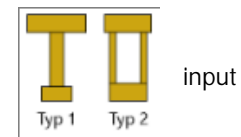
Support (Auflager)

Click on the tab "Support" (Auflager) below the graphic to open the table. Here you can edit the support width, depth and the Kc90 value.

System spans

Enter the length for the individual spans of the beam. The option "cantilever" is available for the outermost spans.

You can also add or remove spans and cantilevers directly in the graphic using the +/- icons.

Cross-sections

Enter the timber species, material standard, strength class (here you can also define a user-defined material via the F5 key), specific weight as well as the cross-sectional dimensions width/height of the individual beam layers. Materials and cross-sections of individual layers may differ.

The input of the individual cross sections can also be done via the tab below the graphic.

For type 2: input of top, bottom chord and web.

Loads

Select first whether self-weight should be included in the calculation or not.

Via the 'Loads' tab (below the graphical representation), the load table is displayed.
See also Data entry via tables (Basic Operating Instructions).

Member	select where the load should apply: over the entire system length or over a span
Load type	selection of the load type: Uniform linear load, concentrated load or trapezoidal load.
Load value	enter the load values pi/pj and, depending on the selected load type, the distance *a* to the left beam end and the length *l*.
	By clicking on the arrow icon you can access a load value compilation - see the description of the LOAD+ program.
distance a	Distance of the load from the left end of the beam.
Field by field	A load over several fields is considered as <u>one</u> load and is not applied alternatively field by field. If this option is selected, the load is applied field by field.
Action	select an action from a list.
Simultaneous group	the loads of a simultaneous group always apply simultaneously.
Alternative group	only one of the loads of an alternative group applies at a time.
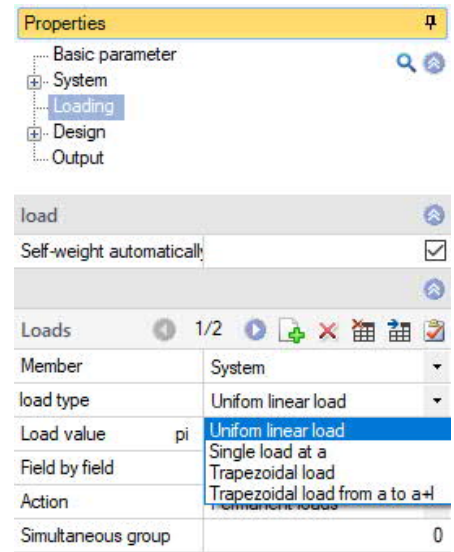
	Member	load type	pi	pj	a	l	Field by field	Action	Sim	Alt	
					[m]	[m]	☑ ☐				
1	System ▾	Unifom linear load	1.00 🔽	—	—	—	☐	Permanent loads	0	0	
2	System	Unifom linear load	0.50 🔽	—	—	—	☐	Cat. A: domestic, residental areas	0	0	

Design

Serviceability

w,inst	limit value of the elastic deflection
w,net,fin	limit value of the summarized elastic deflection and creep deformation
w,fin	limit value of the final deformation

Calculation options

Consider shear from components   check this option if the shear deformations of the components are to be considered.
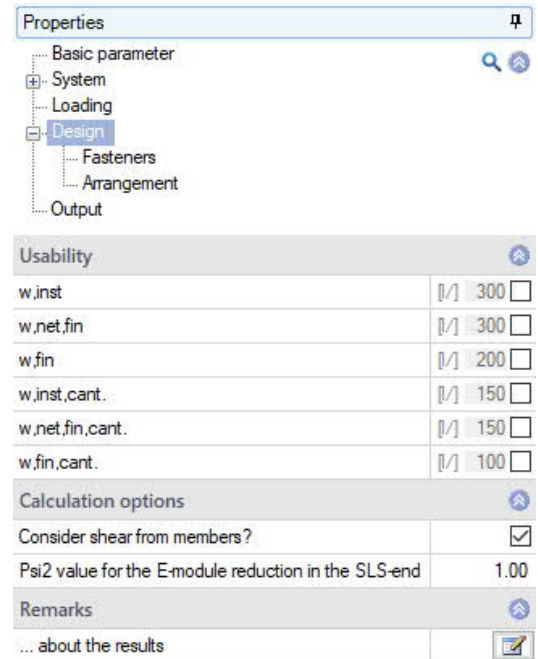
Psi2 value   the Psi2 value that is included in the reduction of the E-modulus of the design of the ULS in the final state. This value applies to all design combinations in this design situation.

Remarks

Call up the FRILO Remarks Editor.

You can enter texts, images or even tables relating to the system, which optionally appear in the output.

See also Remarks Editor.

Fasteners

Select the type of fastener. Dowel pins, fit bolts, connectors (special dowels) and nails are available for selection.

Depending on the selected fastener, further specifications are required for the definition of the fastener properties.

By default, the selection of fasteners from preferred values is enabled. A list of default values is available for selection. The associated property values are displayed. You can edit them after ticking the corresponding checkbox - user-defined entry of all values is also possible.
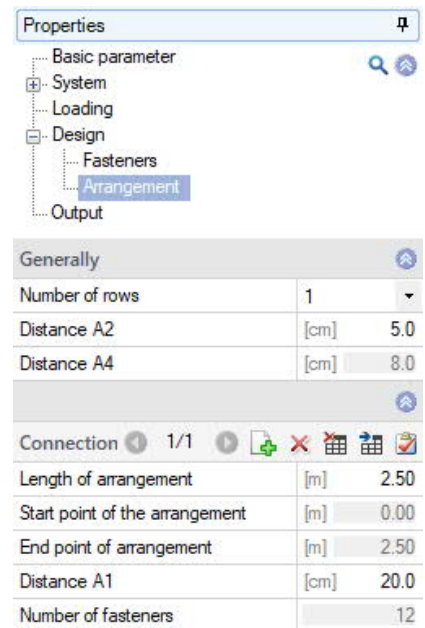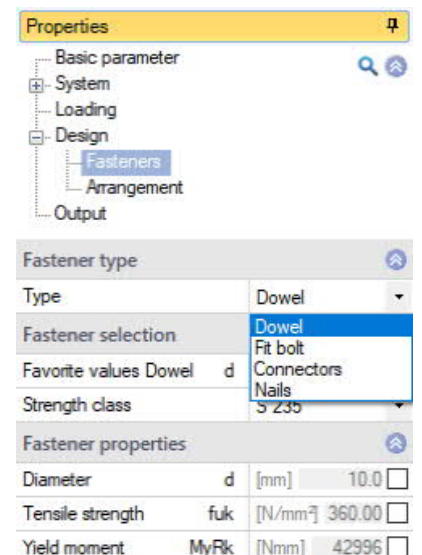
Arrangement

The arrangement of the fasteners can differ in the different sections.

Via the tab "Arrangement fasteners" (below the graphical representation), you can display the data-entry table.
See also Data entry via tables (Basic Operating Instructions).

Enter the length and spacing for the positioning of each fastener. Start and end point of the arrangement as well as the number of fasteners are displayed.

Properties		⊓
⋯ Basic parameter		🔍 ⌃
⊞ System		
⋯ Loading		
⊟ Design		
⋯ Fasteners		
⋯ Arrangement		
⋯ Output		

Generally		⌃
Number of rows		1 ▾
Distance A2	[cm]	5.0
Distance A4	[cm]	8.0
		⌃
Connection ◀ 1/1 ▶ 📑 ✕ 📇 📇 📝		
Length of arrangement	[m]	2.50
Start point of the arrangement	[m]	0.00
End point of arrangement	[m]	2.50
Distance A1	[cm]	20.0
Number of fasteners		12

Load transfer

Via "Connected programs" in the upper menu bar, the support forces can be transferred to the column programs

- Reinforced concrete column B5+
- Wooden column HO1+
- Steel column STS+
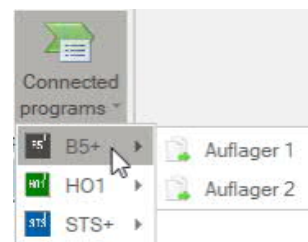
Output

Before starting the output, click on the calculation icon if the option "automatic calculation after each input" is switched off ("Auto off" icon).

After the calculation, the utilization is displayed in the lower right section of the graphics window and provides a good overview of the economic efficiency of the structural system entered.
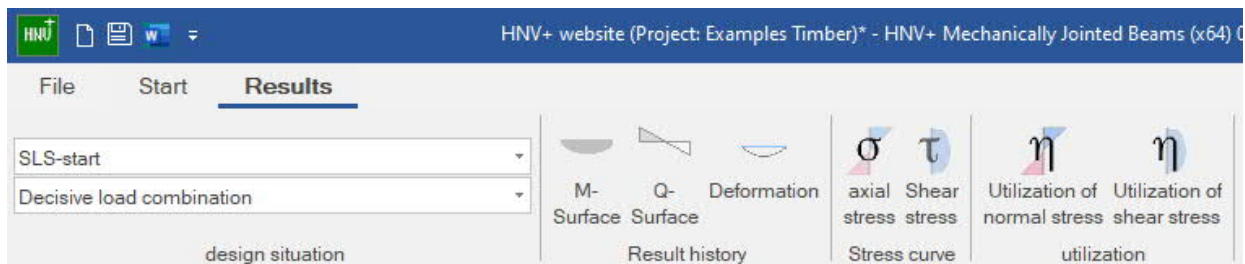
Output scope

By checking the desired options, you can determine the scope of data to be put out.
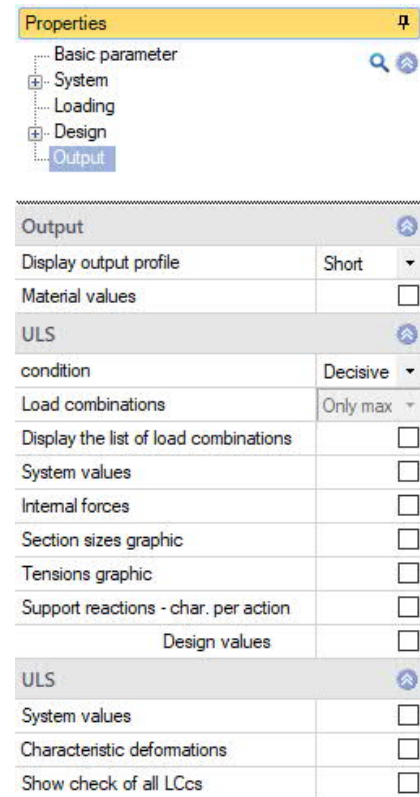
Result options

Via the "Results" tab, you can display the different result graphs.

Output as a PDF document

Via the Document tab, you can display the document in PDF and print it.

See also Output and printing.pdf