

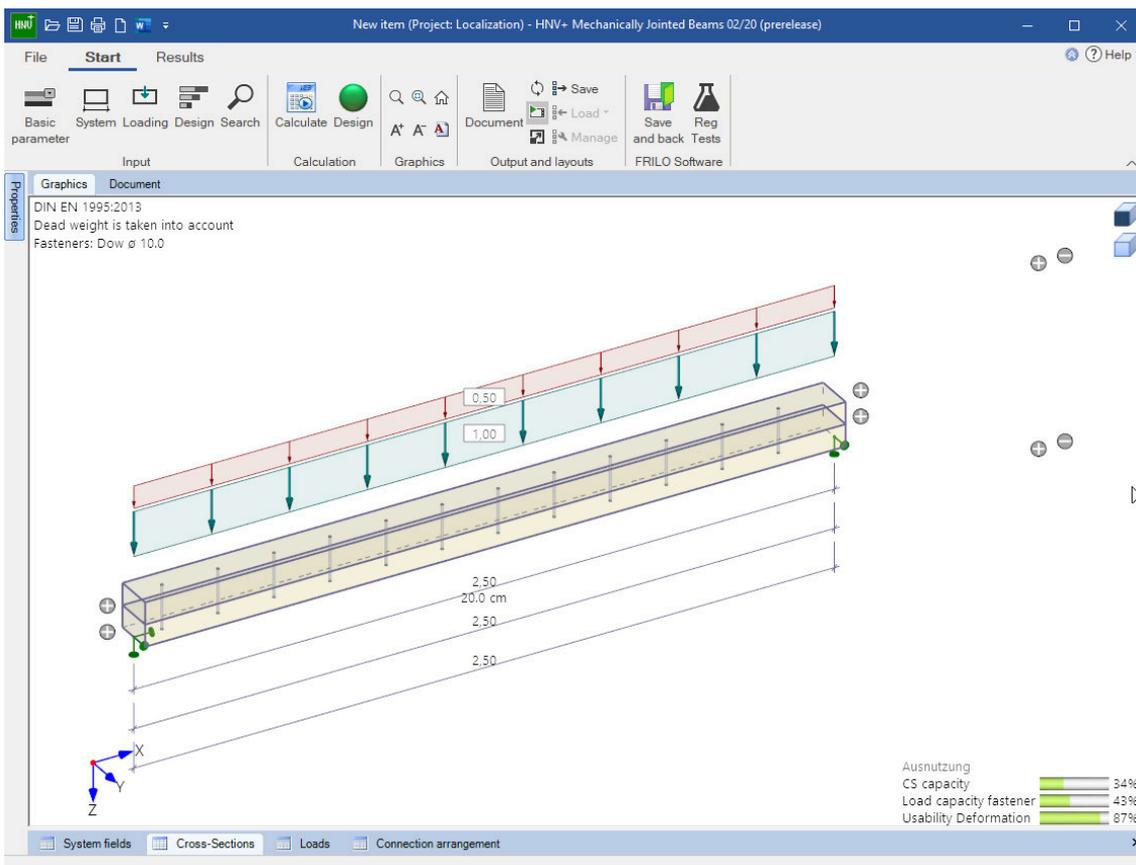
# Mechanically Jointed Beams HNV+

FRILO Software GmbH

[www.friilo.eu](http://www.friilo.eu)

[info@friilo.eu](mailto:info@friilo.eu)

As of 04/03/2021



# Mechanically Jointed Beams HNV+

## Contents

<b>Application options</b>	<b>3</b>
<b>Data entry</b>	<b>4</b>
<b>Basic parameters</b>	<b>4</b>
<b>Structural system</b>	<b>5</b>
System spans	5
Cross-sections	5
<b>Loads</b>	<b>6</b>
<b>Design</b>	<b>7</b>
Fasteners	7
Arrangement	8
<b>Output</b>	<b>9</b>

## Basic documentation - overview

In addition to the individual program manuals, you can find basic explanations on how to operate the programs on our homepage [www.frilo.eu](http://www.frilo.eu) ( ▶ Service ▶ Articles Information ▶ Basic Operating Instructions).

## 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

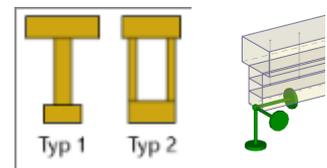
- Single-span girder
- Multi-span girder
- Cantilevers

### Vertical load

- Uniformly distributed linear load
- Concentrated load
- Concentrated moment
- Trapezoidal load

### Cross-sections

- Type 1: Rectangular cross-sections with any number of stacked cross-sections
- Type 2: Cross-section with upper and lower chord and webs
- Different materials



### 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.

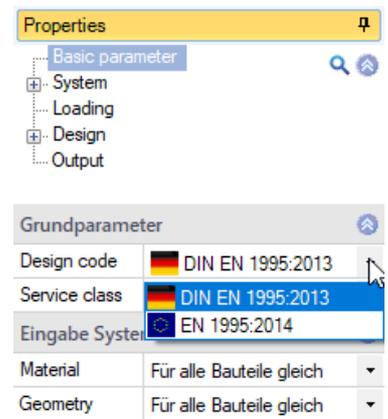
Therefore, the data-entry fields and options can differ from those described below depending on the selected standard.

## Basic parameters

You can select the standard and the materials. Moreover, you can define the strength class and the service class as well as the specific weight in this section.

Strength class:

You can adjust the strengths and stiffnesses. To do this, click in the data-entry field and press the F5 key. In the pop-up menu "User-defined material" you can enter/edit/save/load new materials.



### Comments

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](#).

## Structural system

Recommendation: Enter the structural system directly via the [context-sensitive GUI!](#)

### Lateral restraint

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  $x_0$ : 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).

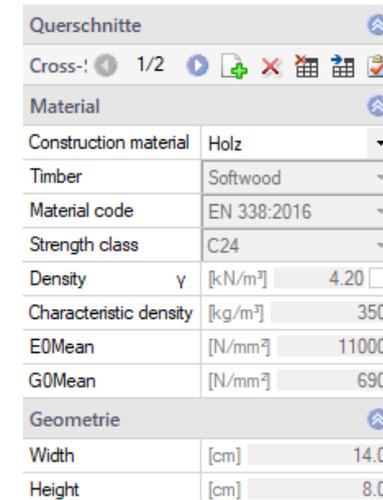
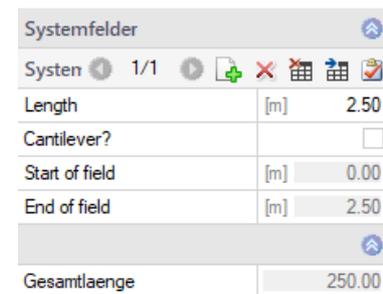
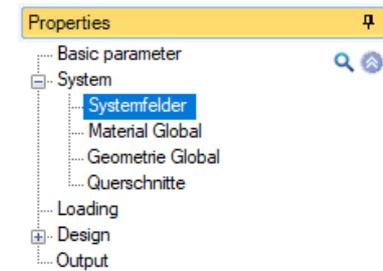
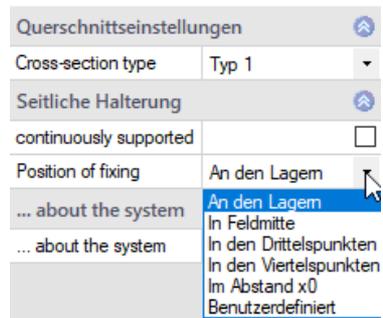
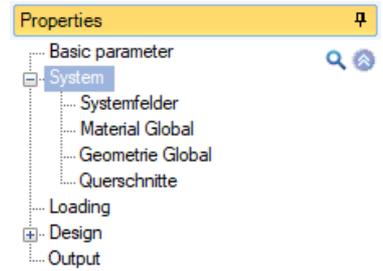
### System spans

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, 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.



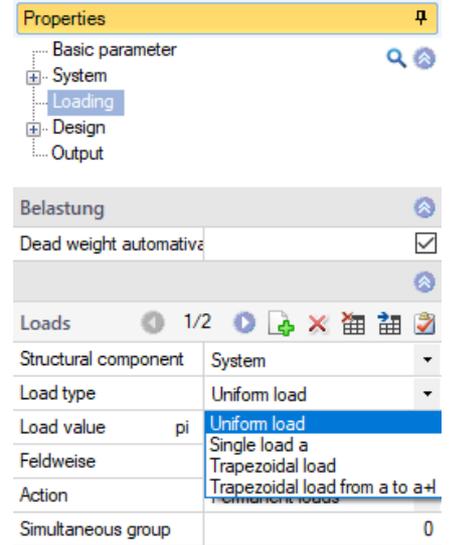
## Loads

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).

- Component select where the load should apply: over the entire system length or over a span
- Load type selection of the load type: UDL, concentrated load, concentrated moment, or trapezoidal load.
- Load value enter the load values  $p_i/p_j$  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.
- Action select an action from a list.
- Concurrency group the loads of a concurrency group always apply simultaneously.
- Alternative group only one of the loads of an alternative group applies at a time.



The screenshot shows the 'Properties' window with the 'Loading' tab selected. Below it, the 'Belastung' (Loading) section has 'Dead weight automatic' checked. The 'Loads' section shows a table with a dropdown menu open for 'Load type'. The dropdown options are: Uniform load, Single load a, Trapezoidal load, Trapezoidal load from a to a+l, and Permanent loads. The 'Action' column shows 'Permanent loads' and the 'Simultaneous group' is set to 0.

	Structural compo...	Load type	$p_i$	$p_j$	$a$ [m]	$l$ [m]	Feldwei...	Action	Sim.	Old
1	System	Uniform load	1.00	--	--	--	<input checked="" type="checkbox"/>	Permanent loads	0	0
2	System	Uniform load	0.50	--	--	--	<input type="checkbox"/>	Cat. A: domestic, residential areas	0	0

## Design

### Calculation options

Consider shear from components  check this option if the shear deformations of the components are to be considered.

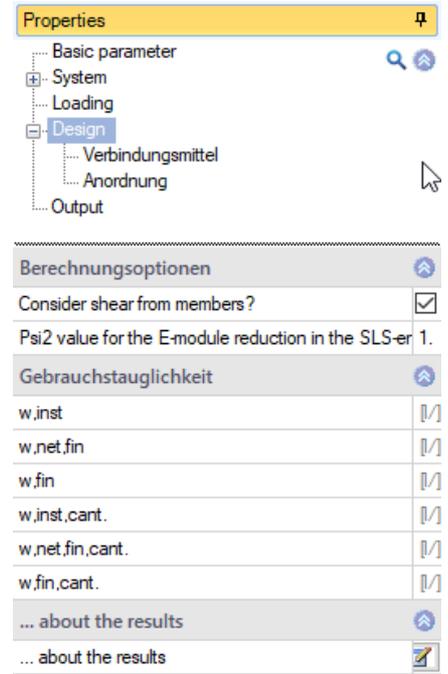
Psi2 value  the Psi2 value that is included in the reduction of the E-modulus of the design of the ULS in the final state. This value applies to all design combinations in this design situation.

### 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

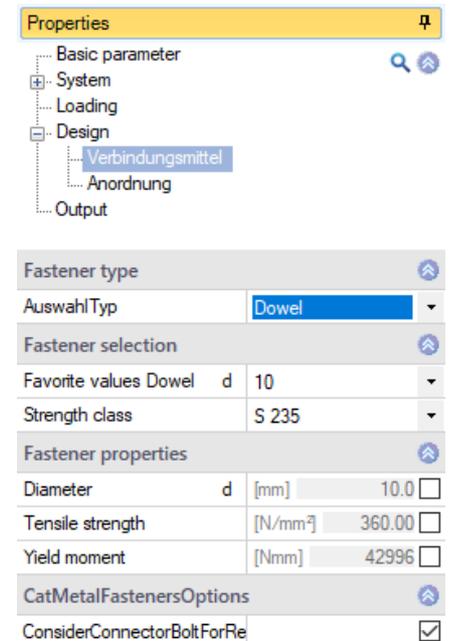


## Fasteners

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.



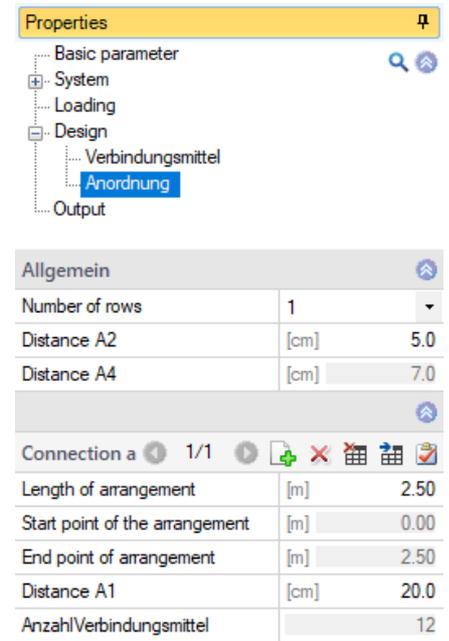
## Arrangement

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.



The screenshot shows the 'Properties' panel with the following structure:

- Properties
  - Basic parameter
  - System
  - Loading
  - Design
    - Verbindungsmittel
    - Anordnung** (highlighted)
    - Output

The 'Allgemein' (General) tab is active, showing the following table:

Allgemein		
Number of rows		1
Distance A2	[cm]	5.0
Distance A4	[cm]	7.0
Connection a 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
Anzahl/Verbindungsmittel		12

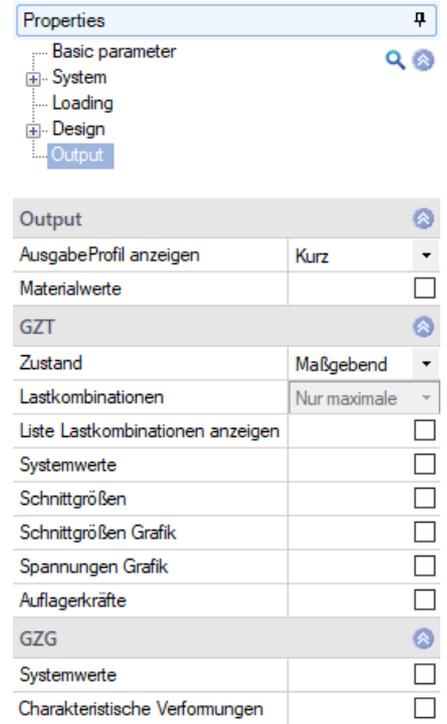
## Output

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

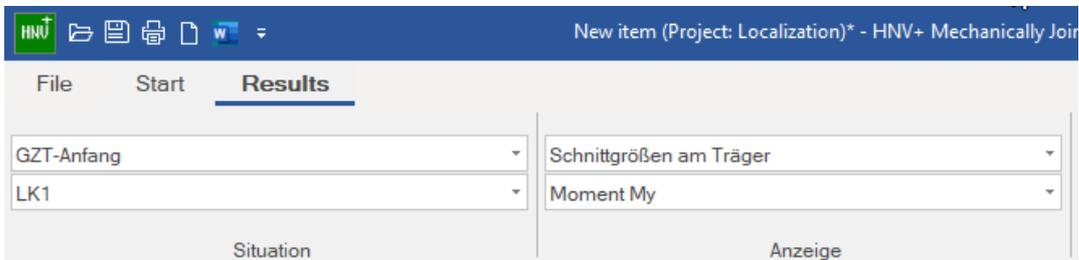
After the calculation, the utilization is displayed in the lower right section of the graphics window and provides a good overview of the economic efficiency of the structural system entered.

### 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](#)