

Engineering Manual No. 4

Update 09/2022

# Formwork for concrete bridge

### Project description

Program: Truss2D

File(s): FineTrial.trs

Formwork.dxf

The target of this manual is a step-by-step input guide for a formwork truss. The example was taken from a real bridge project carried out in Germany.

## Regelquerschnitt M 1:50





# Starting new project

Start Truss2D



Create new project

	TRUSS4 - Tru	uss 2D [Untit
<u>F</u> ile	Edi <u>t</u> Data	a <u>e</u> ntry An
5	<u>N</u> ew	Ctrl+N
ß	<u>O</u> pen	Ctrl+0
۲	<u>S</u> ave	Ctrl+S
	Save <u>a</u> s	

Use FineTrial.trs attached to this engineering manual as template

New	×
General Templates	
Empty structure as a Truss Generator template Generator	
Keep project description	VOK X Cancel



#### Add FineSupplierTrial catalogue to your catalogues and confirm by clicking "OK"



If the window does not appear, it means the catalogue was found, then just check if the timber and nail plates supplier is chosen correctly in "Truss properties". If not click on the appropriate supplier and change it.

Truss properties	[Modify]
Standard :	EN 1995-1-1 (EC5); national annex : EN 1995-1-1
Parameters :	truss count <u>1</u> ; total number of plies is 1;
:	number of plies <u>1</u> ; loading width of truss is <u>1000.0</u>
Mounting :	Type : truss; mounting mode : below outline
Symmetry :	symmetry check is switched on (general); truss symmetry was not recognized
Thickness :	Truss thickness is <u>50</u> mm
Material :	truss material <u>S10 (C24) - coniferous</u>
Suppliers	timber (catalogue) FineSupplierTrial (max, length 6000 mm); nail plates (catalogue) FineSupplierTrial (types; BV15, BV20);



# Truss settings

Open Truss settings

TRUSS	4 - Truss 2D [Untitled.trs*	]			
<u>File</u> Edi <u>t</u>	Data <u>e</u> ntry Anal <u>y</u> sis	T <u>o</u> ols <u>D</u> isplay	<u>H</u> elp		
¥ D	A		4 <b>~</b> - /	🖈 - 👸 👸 -	

### Set loading width to 600 mm

Truss settings		×
O Standard	Truss count:	1
Truss properties	Number of plies:	1
O Material	Total number of plies:	1
-O Thickness -O Mounting	Loading width of truss:	600.0 [mm]
-O Symmetry	Girder	
O Suppliers	The girder is loaded symmetrically	

### Set truss thickness to 60 mm

Truss settings		×
O Standard	Truss thickness (timber width):	
Truss properties	Truss thickness:	60 [mm]
O Parameters		
Material     O     Thickness     Mounting		
O Symmetry		

### Change mounting to "General"

Truss settings			×
O Standard	Type:	truss	Ψ
<ul> <li>Truss properties</li> </ul>			
-O Parameters	Mounting mode	general geometry	•
-O Material	Mounting mode:		Ψ
O Thickness Mounting	Automatically d	etermine truss geometry	
O Symmetry	Geometry:	[not recognized]	¥
Cuppliarr			



#### Choose left upper detail and click the button "Modify"



### Choose the "Intersection" detail





### Project input

Import the drawing of the bridge's cross-section by clicking on



Use Formwork.dxf file attached to this EM. Import window pops up and should be accepted with "OK". This drawing will be placed as a construction layer in the background of the main workspace window and provides snap points when creating the truss. Note that lines of the construction layer cannot interact with Truss 2D's drawing functions, such as trim/extend, move, etc.

lmport of construction layer		-		×
List of layers	+ 🔍 🖂			
Pormwork Bridge				
✓ Draw axes	Units of imported structure : mm 💌 Range of coordinates : x : (-25.000; 9825.000), y : (-	79.961; 1	220.000) [	mm]
	🗸 OF	(	X Can	cel

#### Adjust your view to maximum







Turn off the bridge layer



And leave only the "Formwork" layer turned on

						^			
			Layer na	ime					
	✓ Fe	ormwork				-			
	B	ridge							
						1			
	:								
						-			
	L		_						
				OK	X Cancel				
<b>- b a b c c c c c c c c c c</b>									
ne formwork layer									
					/				
					/				
	\ \								
	L								
Change the propertie	s of the	e construct	ion layer						
Change the propertie	s of the	e construct	ion layer			1			
Change the propertie	s of the	construct	ion layer Constructi	on lines	•				
hange the propertie	s of the	e construct	ion layer Constructi	on lines	▼ <mark>溜</mark> ※ 中 な				
Change the propertie	s of the	e construct	ion layer Constructi	on lines ZBZ	▼ 【 ※ 円 び				×
Change the propertie	s of the	e construct	ion layer Constructi	on lines こ Bて	▼ 【¥ ?? 円 ☆		_		×
Change the propertie	s of the	e construct	ion layer Constructi	on lines ζ Βζ	▼ 【¥ 父 凸 文		_		×
Change the propertie	s of the	e construct	ion layer Constructi	on lines	▼ 副 ② 円 之 Colour	Thickn		Linetvi	×
Change the propertie	s of the	e construct	ion layer Constructi	on lines ∠ B ed Printable S ()	▼ 課 ☆ 円 な Colour	Thickn 0.20mm 0.10mm		Linetvr	X De
Change the propertie	s of the	e construct	ion layer Constructi	on lines 군 B군 ed Printable 爱	マー語 名 Colour	Thickn 0.20mm 0.10mm		Linetvi	× - •
Change the propertie	s of the	e construct	ion layer Constructi	on lines ∠ B ed Printable ☆	マ 評 な Colour	Thickn 0.20mm 0.10mm	ess ▼ - ▼ -	Linetvr	
Change the propertie	s of the	e construct	ion layer Constructi	on lines 군 B군 ed Printable 爱	マージョン そ一当 な Colour	Thickn 0.20mm 0.10mm	ess – T –	Linetvi	×
Change the propertie	s of the	e construct	ion layer Constructi	on lines ∠ B ed Printable ☆	マ 学 一 Colour マ マ マ マ マ マ マ マ マ マ 、 、 、 、 、 、 、 、 、 、 、 、 、	Thickn 0.20mm 0.10mm	ess ▼ - ▼ -	Linetv	× 
Change the propertie	s of the	e construct	ion layer Constructi	ed Printable	マ で Colour マ マ マ	Thickn 0.20mm 0.10mm	ess – T –	Linetvi	× 
Change the propertie	s of the	e construct	ion layer Constructi	ed Printable	▼ ∰ X	Thickn 0.20mm 0.10mm	ess v –	Linetv	× 



Now draw construction lines that will help us add joints and members. Select the line tool



Draw the first line starting on the top right point of the formwork layer and with a length of 2000 mm.

1

Copy the line by using "Equidistant"



Enter the required length (See the picture below, e.g., 125, 975, 625, 1600...) and click on the line and towards the left direction. Now repeat with the newly created line. Click on it, place your mouse cursor to the left and click.



Select "Add member" from the data entry window on the left





Click on the left end of the lower sloped line, then on the right end. This creates the bottom chord of the truss:



Boxed numbering of the joints and members will appear. There are always two numbers above each other for joints and members. An upper number is a serial number, the lower bold number indicates the type of joint and member, respectively. The joint numbers are placed close to the joints, and the location of the member numbers may vary.

The two blue arrows show the lifting points for the "truss" and the blue circle with the cross indicates the centroid of gravity of the truss. If these shouldn't appear on your screen, tick the following checkboxes. Make sure that "General" is active in the Data entry window.





Use Extend/Trim from the CAD-tools menu to extend the bottom chord to the required length, which will be defined by the two vertical drawing lines.



First click on the left vertical construction line to select to where the bottom chord shall be extended, then click on the bottom chord. Repeat for the other side, first click on the function Extend/Trim again, then on the right vertical construction line, and finally on the bottom chord.







Zoom into the area where you want to place the joint by using the mouse wheel, click on the intersection point of the structural line and the upper edge of the bottom chord and repeat for the other points. Your joints should look like this:



These joints will be turned into supports for the truss in a later step. It is easier to add them before the entire truss geometry has been created, as the drawing is clearer now. Joints can either be absolute or relative. Absolute joints have a fixed position in the coordinates system, relative joints can be moved in a simple way along members between two absolute joints. They also will move automatically if the coordinates of an absolute joint are changed.

#### If any of the newly added joints turned out to be absolute, right-click on it and convert it to a relative.

If you miss a point that you want to click on, you can either cancel the function while it hasn't been

finalized or click on "Undo" or use <CTRL+Z>.

#### Note:

2

Webs must be connected through a relative joint to a chord (main member), otherwise no plate will be positioned automatically on such a joint. Change such unwanted absolute joints to relative ones.



![](_page_10_Figure_10.jpeg)

![](_page_11_Picture_0.jpeg)

Your table of joints should look like this:

Þ	Number 🔺	Input style	Coordin	ates
			Y [mm]	Z [mm]
	1	abs.	-25,0	-252,2
-	2	rel. to 1; 125,2mm from prim.j., in axis 1	100,0	-244,7
1	<mark>≯ 3</mark>	rel. to 1; 1728,1mm from prim.j., in axis 1	1700,0	-148,0
۶ [	4	rel. to 1; 3331,1mm from prim.j., in axis 1	3300,0	-51,3
	5	rel. to 1; 4929,2mm from prim.j., in axis 1	4895,2	45,1
	6	rel. to 1; 5334,7mm from prim.j., in axis 1	5300,0	69,6
	7	rel. to 1; -3335,9mm from end j., in axis 1	6495,2	141,8
	8	rel. to 1; -1733,0mm from end j., in axis 1	8095,2	238,5
	9	rel. to 1; -130,1mm from end j., in axis 1	9695,2	335,2
	10	abs.	9825,0	343,1

Make the intermediate joints special supports with vertical spring properties of 2,0 MN/m.

Hit the <ESC>-key to end any other functions. Select joints 2 to 5 and 7 to 9 within the table using Windows functionality (i.e. hold down the <Shift> key, klick on joint No.2, then on joint No. 5. Repeat the same procedure for joints No.7 and No.9). These relative joints should be selected and displayed in green colour.

![](_page_11_Figure_5.jpeg)

Right mouse click on any of the selected joints, select "Support for selected joints" from the context menu

![](_page_11_Figure_7.jpeg)

![](_page_12_Picture_0.jpeg)

Change the support width to 160mm

Suppore		
Free	Displacement along axis	
Dis <mark>p</mark> l. Y	Y: fixed Z: 2.0	2.000
Hinge	Rotation round axis:	Ē
Fixed	Support rotation:	-
Displ. Z	X: 0.000 °	
Special	out of plane buckling pre	nt vented
Wall plate -		
upport type:		
upport type: imber wall pl	ate	2
upport type: imber wall pl Properties	ate	
upport type: imber wall pl Properties - Support widt	ate	160 [mm]
upport type: imber wall pl Properties – Support widt Centre distan	ate h: ice from joint:	160 [mm] 0.0 [mm]
upport type: imber wall pl Properties – Support widt Centre distan Additional m	ate h: ice from joint: ember position	160 [mm] 0.0 [mm] 0.00 [–]
upport type: imber wall pl Properties – Support widt Centre distan Additional m	ate h: ice from joint: ember position structural layout	160 [mm] 0.0 [mm] 0.00 [-]

Select "Special..." in the dialog and specify the spring properties as shown.

Fixe	d displaceme	nt along axis:	
<u>Y</u> :	free	• к	[MN/m]
<u>Z</u> :	spring	• к	2.000 [MN/m]
Fixe	d rotation abo	out axis:	
<u>X</u> :	free	• К	[MNm/i
	Free	Fixed	Spring
Sup Sup	port rotation	about axis:	
12		0.00	0 <i>fx</i> [°]

![](_page_13_Picture_0.jpeg)

Edit joint no 1 and make it a horizontal support. Right mouse click on that joint, select "Edit joint" from the context menu, switch to the "Support" tab and check Y: fixed.

lopology	Code	Support	Wall plate	Bottom detail	Joint edit	 	
Free	2	Displace	ment along	axis			
Displ.	Y	Y: 🔽 Z:	fixed fixed				
Hing	e	Rotation	round axis				Image: Second
Fixe	d	X: Support	rotation:				
Displ.	Z	X: 0.00	00 °				
Specia	al	Linear st	ability - dis	placement kling prevented			

Hit <ESC> to deselect all objects.

Joint table with coordinates and support indicators

Num	nber 🔺	Input style	Coordin	ates	Support			
			Y [mm]	Z [mm]	Py	PZ	Ox	
>	1	abs.	-25.0	-252.2	1			
	2	rel. to 1; 125.2mm from prim.j., in axis 1	100.0	-244.7		~		
	3	rel. to 1; 1728.1mm from prim.j., in axis 1	1700.0	-148.0		1		
	4	rel. to 1; 3331.1mm from prim.j., in axis 1	3300.0	-51.3		~		
	5	rel. to 1; 4929.2mm from prim.j., in axis 1	4895.2	45.1		1		
	6	rel. to 1; 5334.7mm from prim.j., in axis 1	5300.0	69.6				
	7	rel. to 1; -3335.9mm from end j., in axis 1	6495.2	141.8		1		
	8	rel. to 1; -1733.0mm from end j., in axis 1	8095.2	238.5		1		
	9	rel. to 1; -130.1mm from end j., in axis 1	9695.2	335.2		1		
-	10	abs.	9825.0	343.1				

![](_page_14_Picture_0.jpeg)

### Adding the other members

We will add some more members (chords) which will describe the contour of the concrete cross-section and some diagonal and vertical webs to hold these chords and minimize their deflections.

Zoom in the left part of the truss and add members that describe the contour of the crosssection. Use the lines that represent the framework, it will look like this

![](_page_14_Figure_4.jpeg)

Continue by adding points that will connect top and bottom chords. All of these points have to be relative, in case they will turn absolute convert them to relative position.

![](_page_14_Figure_6.jpeg)

![](_page_15_Picture_0.jpeg)

Before adding more members keep only snapping to objects which will ensure that you will click precisely on the required joint

![](_page_15_Figure_2.jpeg)

Create members between joints 2-15, 17-5, 16-3, 3-17

![](_page_15_Figure_4.jpeg)

![](_page_16_Picture_0.jpeg)

Your final structure should look like this

![](_page_16_Figure_2.jpeg)

![](_page_17_Picture_0.jpeg)

The entering of the truss geometry is completed.

In order to have the same numbering for joints and members, we do the following: Right mouse click in the free workspace and call the function "Remove idle rel. joints" from the context menu. Then right mouse click again and select "Renumber structure".

![](_page_17_Figure_3.jpeg)

If the renumbering didn't give you the same results, adjust the numbering manually with the "Arrow" buttons in order to get the same result as the tables below. It is not necessary to have the same numbering, however in order to smoothly follow the tutorial it is strongly suggested to do so.

Jo	ints	Memb	pers		
- -	Nur	mber 🔺			
~	1		abs.		
<u>^</u>		2	rel. to 4; 65.		
Ŷ		3	rel. to 4; 16		
÷		4	rel. to 4; 28		
-		5			

Joints - Numbers and coordinates

Number 4	Input style	Coordin	ates	Support			
		Y [mm]	Z [mm]	Py	Pz	OX	
1	abs.	-25.0	-252.2	1			
2	rel. to 4; 125.2mm from prim.j., in axis 1	100.0	-244.7		~		
3	rel. to 4; 1728.1mm from prim.j., in axis 1	1700.0	-148.0		1		
4	rel. to 4; 2855.1mm from prim.j., in axis 1	2824.9	-80.0				
5	rel. to 4; 3331.1mm from prim.j., in axis 1	3300.0	-51.3		1		
6	rel. to 4; 4929.2mm from prim.j., in axis 1	4895.2	45.1		1		
7	rel. to 4; 5334.7mm from prim.j., in axis 1	5300.0	69.6				
8	rel. to 4; -3335.9mm from end j., in axis 1	6495.2	141.8		1		
9	rel. to 4; 7014.2mm from prim.j., in axis 1	6976.4	170.9				
10	abs.	7167.0	582.3				
11	rel. to 6; 1575.2mm from prim.j., in axis 1	8725.0	814.5				
12	rel. to 6; 2561.0mm from prim.j., in axis 1	9700.0	959.9				
13	abs.	9825.0	978.5				
14	rel. to 4; -1733.0mm from end j., in axis 1	8095.2	238.5		1		
15	rel. to 4; -130.1mm from end j., in axis 1	9695.2	335.2		1		
16	abs.	9825.0	343.1				
17	abs.	-25.0	540.1				
18	abs.	2633.8	311.3				
19	rel. to 2; 125.5mm from prim.j., in axis 1	100.0	529.3				
20	rel. to 2; 1104.1mm from prim.j., in axis 1	1075.0	445.4				

![](_page_18_Picture_0.jpeg)

# Members - primary and secondary joints, member lengths

Joints	Members							
+ N	umber 🔺	-	Je	pint		Length	Rotation	Cross-section
4		first	primary	secondary	end	[mm]	[°]	
¥ .	1	2	2	19	19	774.0	90.000	60 x 80
^	2	17	17	18	18	2668.6	-4.918	60 x 80
Ŷ	3	18	18	4	4	435.5	-63.970	60 x 80
<b>†</b> >	4	1	1	16	16	9868.0	3.459	60 x 80
	5	9	9	10	10	453.4	65.142	60 x 80
	6	10	10	13	13	2687.4	8.478	60 x 80
	7	15	15	12	12	624.7	89.557	60 x 80
	8	20	20	3	3	861.8	-43.516	60 x 80
	9	3	3	18	18	961.6	23.503	60 x 80
	10	10	10	14	14	914.8	-17.233	60 x 80
	11	14	14	11	11	853.5	42.445	60 x 80

![](_page_19_Picture_0.jpeg)

### Creating load cases, load case combinations, and loads

Generally, for creating loads, load cases, and load case combinations, the load's dialog can

be used by clicking on the green flash "Generate" in the Data entry window Generate This does not work for irregular shapes such as this formwork truss.

In the first step, we will create three load cases: self-weight, concrete load, and working load. Then we generate combinations for ULS (ultimate limit state) and SLS (serviceability limit state). Finally, we apply loads to different members.

Select "Load" from the Data entry window. Hit in the table window with the Load cases" tab being active.

Lo	ad cases	Combinations ULS	Combinations SLS
5	Numbe		
÷			
$\ll^{-1}$	8		
×			

The first load case is self-weight.

	Transfer to the local division of						
Load type:	load not	t specified					
Name:	G1	self weight-p	ermanent		[tem	plate]	•
Code:	self weig	ght	▼ Type:	permanent			×
Load factor -	unfavourable	e effect of load :			γ <sub>f,Sup</sub> =	1.35	[-]
Load factor -	favourable e	ffect of load :			y <sub>f,inf</sub> =	0.90	[-]
Category:	[default	input]					•
Factor of perr	nanent load	reduction in alte	ernative combination	on :	ξ =	0.85	[-]
Factor of com	bination val	ue:			ψ <sub>0</sub> =		[-]
Factor of freq	uent value :				ψ1 =		[-]
Factor of qua	si-permanen	t value :			ψ2 =		[-]
Use for th	e main varia	ble load in autor	matic generating o	f combinations.			
<u>01</u>						_	
lumber	1				🕂 Ado	d 🔰	Cance

Click on 4dd, a

, and continue defining two more load cases.

![](_page_20_Picture_0.jpeg)

Type "Concrete" as the name of this load case Q2 and select short-term variable and Category E: storage areas.

LUau Case	Loading width			
Load type:	load not spe	cified		3
Name:	Q2 C	oncrete	[temp	late] 🔹
Code:	force	▼ Type: shor	t-term variable	
Load facto	r - unfavourable eff	ect of load :	γ <sub>f,Sup</sub> =	1.50 [-]
Load facto	r - favourable effect	of load :	γ <sub>t,int</sub> =	[-]
Category:	Category E:	storage areas		3 <b>-</b>
Factor of p	ermanent load redu	ction in alternative combination :	ξ =	[-]
Factor of c	ombination value :		ψ <sub>0</sub> =	1.00 [-]
Factor of fi	requent value :		ψ1 =	0.90 [-]
Factor of q	uasi-permanent val	ue:	ψ2 =	0.80 [-]
	ne v sava		ations	

Add a third load case Q3 called "Working load", being the same type and category. Hit "Add" again then "Cancel" to close the dialog window.

Load case	oading width				
Load type:	load not specified	[			1
Name:	Q3 Workin	g load	[temp	late]	37
Code:	force	▼ Type: sho	rt-term variable		1
Load factor ·	unfavourable effect of	load :	γ <sub>f,Sup</sub> =	1.50	[-]
Load factor ·	favourable effect of lo	ad :	γ <sub>f,inf</sub> =		[]
Category:	Category E: storag	ge areas			a.
Factor of per	manent load reduction	in alternative combination :	ξ =		[-]
Factor of cor	mbination value :		ψ₀ =	1.00	[-]
Factor of fre	quent value :		ψ1 =	0.90	[-]
Factor of qu	asi-permanent value :		ψ2 =	0.80	[-]
	he main variable load ir	automatic generating of combir	nations.		

![](_page_21_Picture_0.jpeg)

The table window shows 3 load cases, one permanent and two short-term.

Los	i cases	Combinations ULS Combinations 9.5											
5	umbe-	ben Load cases					Load factor						
		Name Code Type Ottegory				YtSup	Ward	3	40	41	92	loading width	
	1	G1 self weight-permanent	Self weight	Permanent	(default input)	1,35	0,90	0,85					
*	2	Q2 Concrete	Force	Short-term variable	Category E: storage areas	1,50			1,00	0,90	0,80	-	
×	3	Q3 Working load	Force	Short-term variable	Category E: storage areas	1,50			1,00	0,90	0,80	1	
a													
Ň													
13													

Click on the tab "Combinations ULS" next to the tab "Load cases", then on the green flash

to create all load combinations for the ultimate limit state (strength). Confirm and close the dialog with "OK".

Properties of combinatio	×		
— Ultimate limit state —			
Generate for type o	f combinatior	Basic	•
Use optimized generation	ting of combin	ations	
Generation style	genera	l TRUSS <mark>sty</mark> le	•
		✓ ОК	🗙 Cancel

The program generated five load combinations, one with permanent load only, and the other four combinations of the permanent and short-term load cases.

Lo	ad	cases	<b>Combinations ULS</b>	Combinations SLS
ş	N	umbe	-	
+	>	1*	G1	
41		2*	Q3:G1	
×		3*	Q2:G1	
-		4*	Q2:G1+Q3	
Ŷ		5*	Q3:G1+Q2	

Switch to the tab "Combinations SLS" and click on the green flash again to generate all load combinations for the serviceability limit state (deflections). Remove the check from "Final deflection" as the load is not acting long enough upon the truss to cause any long-term deflection effects.

![](_page_22_Picture_0.jpeg)

- Sanviceshility limit stat		
Serviceability infint star	ic.	
Generate for select	ed types of combinations	
Characteristic		
CHAPTERSUIC		
Final deflection		
Final deflection     Use optimized gener	ating of combinations	
Final deflection     Use optimized gener	ating of combinations	

Five load combinations were generated by the program

Lo	ad cases	Combinations ULS	Combinations SLS
5	Numbe≜		
÷	1*	G1	
4	2*	Q3:G1	
×	3*	Q2:G1	
-	4*	Q2:G1+Q3	
Т П	5*	Q3:G1+Q2	

In the "Active load case window" select [2] Q2 Concrete as the active load case. That is the one for which we will now quantify the load values.

![](_page_22_Picture_5.jpeg)

![](_page_23_Picture_0.jpeg)

## Applying loads to members

After having created load cases and combinations, the "real" loads must be connected with members. In the following table, the vertical loads are given

Member No	Joint Start - End.	Load Z-Dir.	Load Y-Dir.
		[kN/m²]	[kN/m²]
2	17 – 18	-6,0 - (-12,0)	-
3	18 – 4	-12,0 - (-22,5)	-12,0 - (-22,5)
4	4 – 9	-22,5 - (-22,5)	-
5	9 – 10	-22,5 - (-12,0)	+22,5 - (+12,0)
6	10 - 13	-12,0 - (-6,0)	-

![](_page_23_Figure_4.jpeg)

![](_page_23_Figure_5.jpeg)

![](_page_24_Picture_0.jpeg)

Select "Load individually" from the Data entry with the activated load case Q2.

Active load case
[2] Q2 Concrete 🔹
Data entry
General
X Remove
Dimensioning
Topology
Add members
/* Reconnect
O Tools
Load
Generate
Load individually
O Load on selected

Click on 1 in the table window and make the following entries:

	ng wiath					
o member: 2	▼ L = 2668	8.6 mm	[temp	olate]	•	2
ype: conti	nuous force				•	m/Ma 0
ontinuous: load	letermined by dis	tance from beg	in. and	end pt.	-	
rientation: on pr	ojection in direct.	of gl. z-axis			-	
o <mark>cation</mark> unit lengt	h unit [mm]				•	
1: -6.00 [k	N/m²] a1:	0.0 [mr	n] r1:	Edge	•	Load values will be multiplied by loading width.
2: -12.00 [k	N/m²] a2;	0.0 [mr	n] r2:	Edge	•	<ul> <li>f1, f2: intensity of uniformly distributed area load</li> <li>a1: distance of load beginning from the reference ioint in</li> </ul>
						direction of the member avir

On member 2 apply a continuous load determined by distance from beginning and end point with the orientation on projection of global z-axis. Use the shown values for the load. Then hit "Add".

![](_page_25_Picture_0.jpeg)

Select member 3, look up the definition of the positive direction of the member by reviewing the order of the joint numbers at r1 and r2 (should be 18 for r1 and 4 for r2), change the load values to -12,0 for f1 and -22,5 for f2.

			//m2	N/m2
o member:	3 ▼ L = 435.5 mm	[template]	× 200 K	12.50 k
/pe:	continuous force		-	
ontinuous:	load determined by distance f	om begin. and end pt.	- 八	
	on projection in direct, of gl. z-axis			
rientation:	on projection in direct. of gl. z	-axis		2
rientation:	on projection in direct. of gl. z	-axis		a l
rientation: ocation unit	on projection in direct, of gl. z length unit [mm]	-axis		E A Ø
rientation: ocation unit 1: -12	on projection in direct, of gl. z length unit [mm]	-axis 0 [mm] r1: 18	Load values will be multiplied	3 4 S I by loading width.
rientation: ocation unit 1:	on projection in direct. of gl. z           length unit [mm]           .00         [kN/m²]         a1:         0           .50         [kN/m²]         a2:         0	-axis 0 [mm] r1: 18 0 [mm] r2: 4	<ul> <li>Load values will be multiplied</li> <li>f1, f2: intensity of uniform</li> </ul>	I by loading width.

Hit "Add" and continue entry. Look at the table window, it now should contain two lines.

Jo	nt load	Member loads		
÷	Numbe	r 🔺	Member	Load type
4	> 1		3	Continuous force
~	2		2	Continuous force

Simply change the orientation to "on projection in direction of gl. y-axis" and hit "Add". This added the horizontal hydrostatically distributed concrete load to member 3.

o member:	3 ▼ L = 435.5 mm [ter	plate] ▼	-12.00 kM/m2
ype:	continuous force	•	
ontinuous:	load determined by distance from begin. an	end pt. 🔻	
rientation:	on projection in direct, of gl. y-axis	<b>*</b>	۳
			14
o <mark>cation</mark> unit	length unit [mm]		-22.50 kAlrin 2
ocation unit	.00 [kN/m <sup>2</sup> ] a1: 0.0 [mm]		e multiplied by loading width.
2: -22	length unit [mm]           1.00         [kN/m²]         a1:         0.0         [mm]         i           1.50         [kN/m²]         a2:         0.0         [mm]         i	18   Load values will be f1, f2: intensity     4   1	e multiplied by loading width.

![](_page_26_Picture_0.jpeg)

#### Continue with member 4 as shown:

member:	4 ▼ L = 9868.0 m	m [template]	-	
/pe:	continuous force			-22.50 kN/m2
ontinuous:	load determined by distance	from begin. and end p	: <b>т</b>	
rientation:	on projection in direct, of gl. z-axis			
	on projection in anect, or gr	. z-axis	- Ī	°⊢₩∽∞
cation unit 1: -22 2: -22	Iength unit [mm]           .50         [kN/m²]         a1:           .50         [kN/m²]         a2:	0.0 [mm] r1: 4 0.0 [mm] r2: 9	• []	oad values will be multiplied by loading width. 1. f2: intensity of uniformly distributed area load

Add loads to member 5 with the vertical load and the horizontal load. Don't forget to change the sign for the horizontal load (positive when pointing to the right)

	by loading width		04/m2
member:	5 🔹 L = 453.4 mm	[template]	×
pe:	continuous force		-
ontinuous:	load determined by distance from	n begin. and end pt.	
rientation:	on projection in direct. of gl. z-ax	is	-
			5
cation unit	length unit [mm]		•
-22	2.50 [kN/m <sup>2</sup> ] a1: 0.0	[mm] r1: 9	Load values will be multiplied by loading width.
	2.00 [kN/m <sup>2</sup> ] a2: 0.0	[mm] r2: 10	f1, f2: intensity of uniformly distributed area load
- 14			al: distance of load beginng from the reference joint in

![](_page_27_Picture_0.jpeg)

<ul> <li>Multiply I</li> </ul>	by loading width		10
o member:	5 🔹 L = 453.4 mm	[template]	▼ 12.00.kN/m2
/pe:	continuous force		• 📫 / 🛰
ontinuous:	load determined by distance from	) begin. and end pt.	-
rientation:	on projection in direct. of gl. y-axi	is	• / /
o <mark>cation</mark> unit	length unit [mm]		▼ 22.50 kN/m2 >9
		[mm] r1.0	<ul> <li>Load values will be multiplied by loading width.</li> </ul>
1: 22	2.50 [kN/m <sup>2</sup> ] a1: 0.0	fund may	
1: 22 2: 12	k.50         [kN/m²]         a1:         0.0           k.00         [kN/m²]         a2:         0.0	[mm] r2: 10	<ul> <li>f1, f2: intensity of uniformly distributed area load</li> <li>a1: distance of load beginning from the reference joint in</li> </ul>

#### Last member 6 with concrete load

Multiply I	by loading width			
o member:	6 ▼ L = 2687	.4 mm	[template]	× 1000
ype:	continuous force			• 00.2
ontinuous:	load determined by dist	ance from begi	n. and end pt.	
rientation:	ion: on projection in direct. of gl. z-axis			·
	3			10 10 10
o <mark>cation</mark> unit	length unit [mm]			•
1: -12	2.00 [kN/m <sup>2</sup> ] a1:	0.0 [mm	i] r1:	<ul> <li>Load values will be multiplied by loading width.</li> </ul>
2: -6	5.00 [kN/m <sup>2</sup> ] a2:	0.0 [mm	n] r2: Edge	<ul> <li>f1, f2: intensity of uniformly distributed area load</li> <li>a1: distance of load beginng from the reference joint in direction of the member axis</li> </ul>

### "Add" and "Close" the dialog

![](_page_27_Figure_5.jpeg)

![](_page_28_Picture_0.jpeg)

Switch the active load case to Q3 Working load.

In Data entry select "Load on selected".

![](_page_28_Picture_3.jpeg)

In the main workspace, window click on those members that should be loaded with Q3, i.e. members 2 through 6. Selected members and their numbers are highlighted in green colour.

![](_page_28_Figure_5.jpeg)

![](_page_29_Picture_0.jpeg)

In the table window click on the "plus" sign. In the dialog only specify the orientation of the load, i.e. global z-axis and enter a value of  $-2 \text{ kN/m}^2$  for f.

New member lo	oad		*	
Mechanical	oad of member y loading width			
Load is applie	d to selected members	[template]	▼	
Type:	continuous force		-2.00 kN/m2	
Continuous:	uniformly distributed load on th	e whole member		
Orientation:	on projection in direct. of gl. z-axis 🔹 👻			
f:	.00 [kN/m <sup>2</sup> ]	]	Load values will be multiplied by loading width. f: intensity of uniformly distributed area load	
			▲ <u>A</u> dd × <u>C</u> lose	

Hit "Add", then "Close".

![](_page_29_Figure_4.jpeg)

The working load is distributed along the entire bottom chord and needs to be adjusted to act only between joints 4 and 8. Switch back to "Load individually" in Data entry, pick the load in the table window that is related to member 4 (row 5) and either double click on the line or on the edit button *Load*.click

![](_page_30_Picture_0.jpeg)

Change "Continuous" in the dialog and specify joints 4 and 8 as r1 and r2, respectively. Close with "OK".

![](_page_30_Figure_2.jpeg)

The input of loads is done and we can move on to the design.

![](_page_30_Figure_4.jpeg)

![](_page_31_Picture_0.jpeg)

# Design and analysis of the truss

Analyse the truss <F8> or

![](_page_31_Figure_3.jpeg)

Review the results, internal forces, reactions, and deflections.

![](_page_31_Figure_5.jpeg)

Members check:

![](_page_31_Figure_7.jpeg)

#### Joints check:

Overall ve	Jverall verification of joints: NOT SATISHED 44					
Number +	Description		Verification - envelope			
			Max. utilization	Decisive combination		
2	wall plate width: 160 mm	1	[28.8 %]			
3	wall plate width: 160 mm	1	[56.9 %]			
5	wall plate width: 160 mm	1	[89.6 %]			
6	wall plate width: 160 mm	X	[106.6 %]			
8	wall plate width: 160 mm	<b>v</b>	[91.7 %]			
14	wall plate width: 160 mm	1	[57.2 %]			
15	wall plate width: 160 mm	1	[-10.0 %]			
2	nail plate BV15 1014 (105 x 147)	1	[76.9 %]	[4] Q2:G1+Q3		
3	nail plate BV20 2819 (280 x 198)	1	[79.5 %]	[4] Q2:G1+Q3		
4	nail plate BV15 0712 (70 x 126)	1	[67.5 %]	[4] Q2:G1+Q3		
7	nail plate BV15 1714 (175 x 147)	1	[76.9 %]	[4] Q2:G1+Q3		
9	nail plate BV15 0712 (70 x 126)	1	[70.5 %]	[4] Q2:G1+Q3		
10	nail plate BV15 2812 (280 x 126)	~	[85.4 %]	[4] Q2:G1+Q3		
11	nail plate BV15 2808 (280 x 84)	1	[83.9 %]	[4] Q2:G1+Q3		
12	nail plate BV15 1014 (105 x 147)	1	[83.9 %]	[4] Q2:G1+Q3		
14	nail plate BV20 1633 (160 x 330)	1	[81.9 %]	[4] Q2:G1+Q3		
15	nail plate BV15 1010 (105 x 105)	1	[77.0 %]	[4] Q2:G1+Q3		
18	nail plate BV15 2812 (280 x 126)	1	[86.2 %]	[4] Q2:G1+Q3		
19	nail plate BV15 1414 (140 x 147)	1	[72.4 %]	[4] Q2:G1+Q3		
20	nail plate BV15 1021 (105 x 210)	1	[98.8 %]	[4] Q2:G1+Q3		

In the table, we can see that Joint 6 does not satisfy. This is due to the high compression force in the joint, which means that we have to increase the width of the support.

![](_page_32_Picture_0.jpeg)

### Double click on the joint in the table and change the width to 180mm

Edit detail properties Number 6		×
Topology Code Support Wall plate Joint edit		
Support type:		
timber wall plate	▼ 2.	000
Properties Support width:	[mm]	Ø
Centre distance from joint:     0.0       Additional member position     0.50       Image: Constructural layout     Don't check compression parallel to the grain		
Number: 6		VK X Cancel

#### Analyse the truss again by pressing <F8>

Overall verification of joints: SATISFIED				🛹 Edit	
Number +	Description	Verification - envelope			
			Max. utilization	Decisive combination	
2	wall plate width: 160 mm	1	[28.8 %]		
3	wall plate width: 160 mm	1	[56.9 %]		
5	wall plate width: 160 mm	1	[89.6 %]		
> 6	wall plate width: 180 mm	1	[97.7 %]		
8	wall plate width: 160 mm	1	[91.7 %]		
14	wall plate width: 160 mm	1	[57.2 %]		
15	wall plate width: 160 mm	1	[-10.0 %]		
2	nail plate BV15 1014 (105 x 147)	1	[76.9 %]	[4] Q2:G1+Q3	
3	nail plate BV20 2819 (280 x 198)	1	[79.5 %]	[4] Q2:G1+Q3	
4	nail plate BV15 0712 (70 x 126)	1	[67.5 %]	[4] Q2:G1+Q3	
7	nail plate BV15 1714 (175 x 147)	1	[76.9 %]	[4] Q2:G1+Q3	
9	nail plate BV15 0712 (70 x 126)	1	[70.5 %]	[4] Q2:G1+Q3	
10	nail plate BV15 2812 (280 x 126)	1	[85.4 %]	[4] Q2:G1+Q3	
11	nail plate BV15 2808 (280 x 84)	1	[83.9 %]	[4] Q2:G1+Q3	
12	nail plate BV15 1014 (105 x 147)	1	[83.9 %]	[4] Q2:G1+Q3	
14	nail plate BV20 1633 (160 x 330)	1	[81.9 %]	[4] Q2:G1+Q3	
15	nail plate BV15 1010 (105 x 105)	1	[77.0 %]	[4] Q2:G1+Q3	
18	nail plate BV15 2812 (280 x 126)	1	[86.2 %]	[4] Q2:G1+Q3	
19	nail plate BV15 1414 (140 x 147)	1	[72.4 %]	[4] Q2:G1+Q3	
20	nail plate BV15 1021 (105 x 210)	1	[98.8 %]	[4] Q2:G1+Q3	

All joints and supports are satisfied and we are done with this engineering example.

For more engineering manuals visit https://www.finesoftware.eu/.