

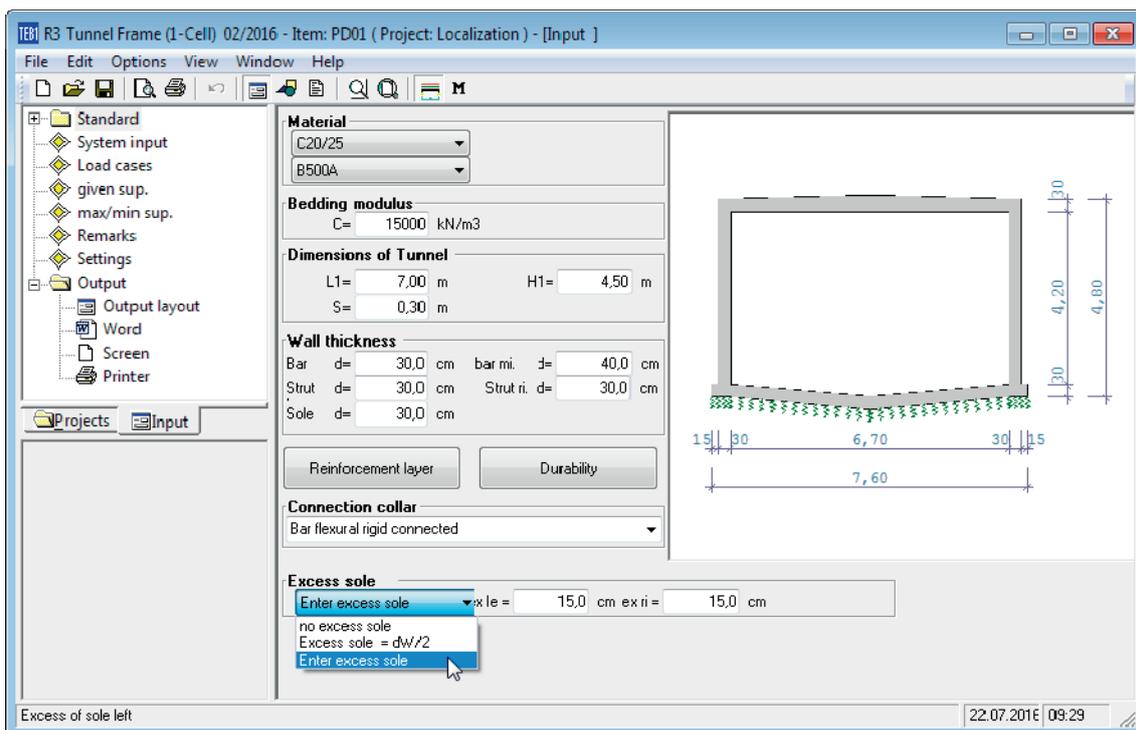
TEB – Tunnel Frame

FRILO Software GmbH

www.friilo.com

info@friilo.eu

As of 29 June 2016



TEB – Tunnel Frame

Contents

Application options	4
Basis of calculation	4
Definition of the structural system	5
Material	5
Subgrade reaction modulus	5
Tunnel dimensions	6
Wall thicknesses	6
Reinforcement layers	7
Connection of horizontal member (single-cell frame)	7
Base projection (single-cell frame)	7
Loads	8
Load cases	8
Definition of the loads	9
Max./min. superposition	10
Pre-set superposition	11
Output	12
Output profile	12

Further information and descriptions are available in the relevant documentations:

FDC – Basic Operating Instructions	General instructions for the manipulation of the user interface
FCC	Frilo.Control.Center - the easy-to-use administration module for projects and items
FDD	Frilo.Document.Designer - document management based on PDF
Frilo.System.Next	Installation, configuration, network, database
FDC – Menu items	General description of the typical menu items of Frilo software applications
FDC – Output and printing	Output and printing

Application options

The TEB application allows the calculation of rectangular single-cell (TEB-1) and twin-cell (TEB-2) frames with a continuously supported base plate.

TEB-1: You can define a camber or a projection for the base.

The horizontal member can have a greater thickness in the middle than at the ends. The connections of the horizontal member can be resistant to bending or pinned. The horizontal member can be dispensed with completely.

The vertical members, the ceiling and the floor slab may have different thicknesses. The minimum width required for the definition of the system and the loading is one metre at least.

Any loads can apply to the horizontal and vertical members.

Available standards

The design can be performed in accordance with the following standards:

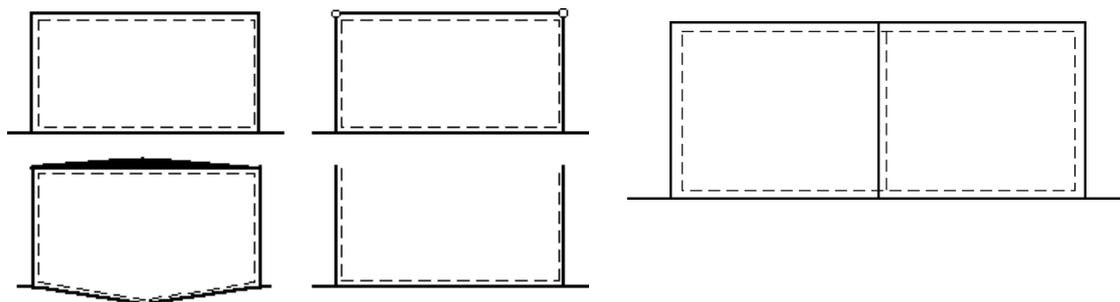
- DIN EN 1992
- BS EN 1992
- ÖNORM EN 1992
- NTC EN 1992
- PN EN 1992
- EN 1992

The standards DIN 1045 / DIN 1045-1 are still optionally available.

Construction materials

Depending on the selected standard, particular types of material may not be available.

Exemplary drawings of the structural system



Basis of calculation

The calculation is based on the displacement method and the subgrade reaction modulus method. The stiffness matrix is generated from the system values. Subsequently, the displacement and torsion of the nodes is assessed and the internal forces and the soil pressure are calculated on this basis. The stiffness of the members is calculated for state I. The structural system is a sway system.

Definition of the structural system

First select the desired standard in the main menu.

Define subsequently the system parameters in the following sections:

- Material
- Subgrade reaction modulus
- Tunnel dimensions
- Wall thicknesses
- Reinforcement layers
- Connection of horizontal member (single-cell frame)
- Base projection (single-cell frame)

Material

Select the concrete quality and the reinforcing steel grade in the drop down lists.

Depending on the selected standard, particular types of material might not be available.

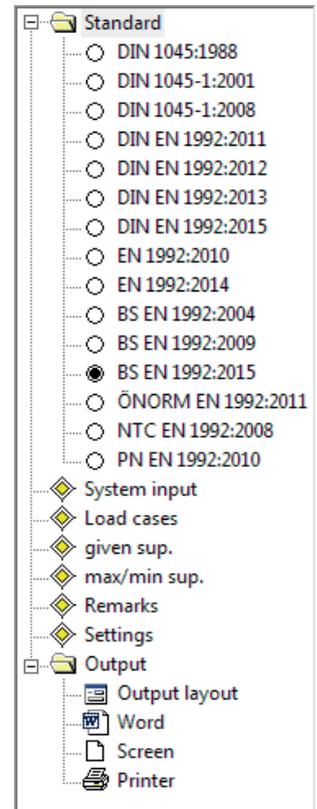
Subgrade reaction modulus

Specify the subgrade reaction modulus.

You can take the subgrade reaction modulus C [kN/m^3] either from general expert literature or from a soil expertise. It is typically in the order of $10,000 \text{ kN}/\text{m}^3$.

Hahn (reference literature) gives the following guiding values for the subgrade reaction modulus C :

Wet clay soil:	20,000	to	$30,000 \text{ kN}/\text{m}^3$
Dry clay soil:	60,000	to	$80,000 \text{ kN}/\text{m}^3$
Fine gravelly sand soil:	80,000	to	$100,000 \text{ kN}/\text{m}^3$
Coarse gravelly sand soil:	150,000	to	$200,000 \text{ kN}/\text{m}^3$



Material

C20/25

B500A

Bedding modulus

C= 15000 kN/m³

Dimensions of Tunnel

chamber 1 L1= 5,00 m

chamber 2 L2= 5,00 m

Height H1= 3,50 m

Reinforcement layer

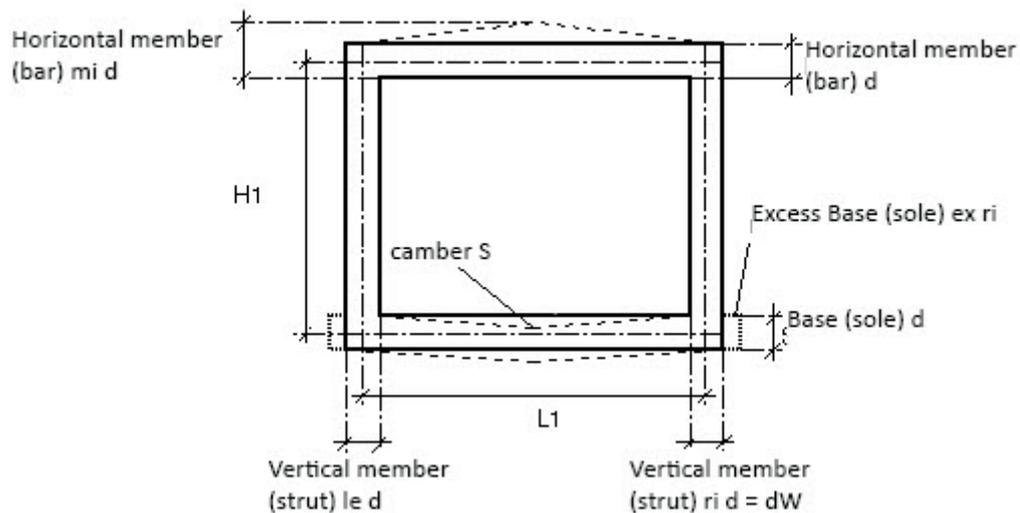
all reinf. layers identical (5 cm)

Predefine reforc. layers Durability

Starts the one-cell tunnel frame.

Tunnel dimensions

- L1** width of the tunnel
- L2** width of the second chamber
- S** camber, is assumed centrally at L/2
- H1** height of the tunnel



Abbreviations: le = left, ri = right, mi = mid

Wall thicknesses

All wall thicknesses are defined in centimetres. The dimensions and designations are shown in the sketches above.

Wall thickness											
Bar le	d=	<input type="text" value="35,0"/>	cm	Bar ri	d=	<input type="text" value="35,0"/>	cm				
Strut le	d=	<input type="text" value="30,0"/>	cm	Strut mi	d=	<input type="text" value="30,0"/>	cm	strut ri	d=	<input type="text" value="30,0"/>	cm
Sole le	d=	<input type="text" value="40,0"/>	cm	Sole ri	d=	<input type="text" value="40,0"/>	cm				

Reinforcement layers

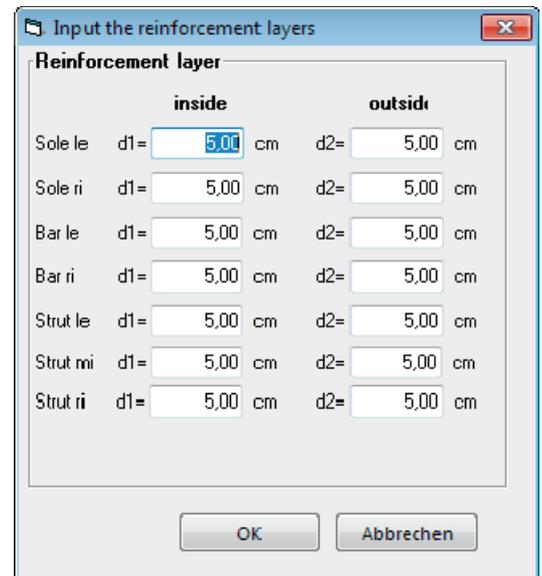
Definition of the interior and exterior reinforcement layers.

d1 interior reinforcement layer

d2 exterior reinforcement layer

Each of the two reinforcement layers must exceed the size of one centimeter.

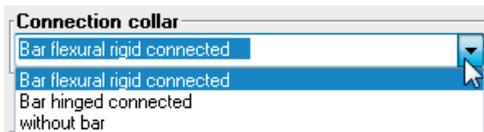
Twin-cell frame: The option "All reinforcement layers have same size (5 cm)" allows you to set all values of d1 and d2 to 5 cm with a mouse click.



	inside	outside
Sole le	d1= 5,00 cm	d2= 5,00 cm
Sole ri	d1= 5,00 cm	d2= 5,00 cm
Bar le	d1= 5,00 cm	d2= 5,00 cm
Bar ri	d1= 5,00 cm	d2= 5,00 cm
Strut le	d1= 5,00 cm	d2= 5,00 cm
Strut mi	d1= 5,00 cm	d2= 5,00 cm
Strut ri	d1= 5,00 cm	d2= 5,00 cm

Connection of horizontal member (single-cell frame)

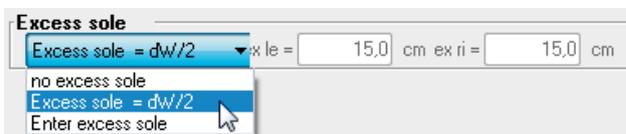
Select the type of connection from the drop-down list.



When you select the option "Without horizontal member", the system is designed as a U-channel structure.

Base projection (single-cell frame)

The option $dW/2$ refers to a projection that is as wide as half the wall thickness. To set a user-defined projection on the left and the right, select "Specify base projection" and enter the corresponding dimensions in the data-entry fields ProLe (base projection on the left), ProRi (base projection on the right).

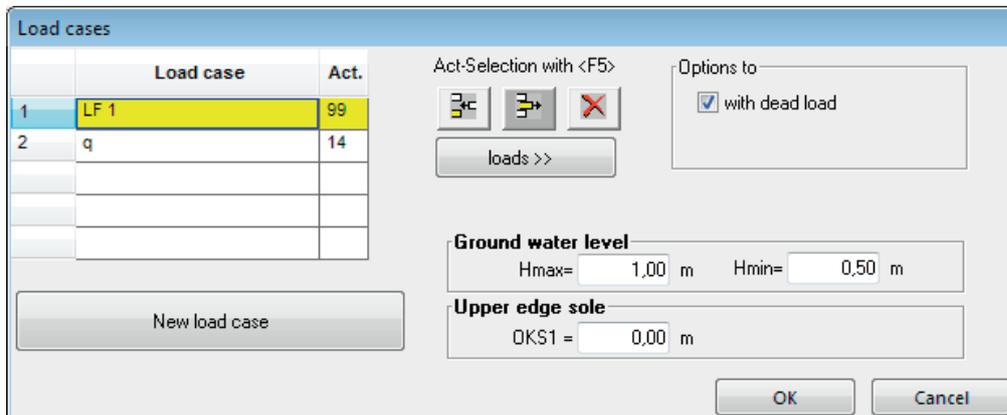


The projection is taken into account in the calculation; currently it is not considered in the design, however. If required, the moment is to be determined from the difference of the corner moments.

Loads

Load cases

Double-click on the “Load cases” option in the main menu to display the load case dialog.



Selecting and renaming load cases

To define a load case, you must select it first. Click into the respective row in the Load case list to select it. The selected load case is highlighted. You can edit the name of the load case and overwrite it, if you wish.

Generating a new load case

Click on “New load case” to add a new load case to the list. To be able to do this you must have entered loads in the previous load case.

Defining loads

Click on the “Loads” button to define loads for the selected and highlighted load case. The button shows the number of loads that you have already entered for your information.

Functions of the F5 key: pressing the F5 key when the mouse cursor is in the load case field accesses the dialog for the load definition. Pressing the F5 with the mouse cursor in the “Act.Grp.” field F5 accesses the dialog for the definition of action groups.

Act. Grp. = action group.

Click into the Act.Grp. row and press the F5 key to select the action group from the displayed list.

Default setting: action group 14 (other variable loads).

Options

With dead load select this option to take the self-weight of the beam into account in the calculation.

Ground water level this query supports the calculation of the safety against buoyancy (single-cell frame).

Hmax maximum ground water level

Hmin minimum ground water level

Base top edge the specification refers to the altitude above MSL (single-cell frame).

Definition of the loads

Click on the “Loads >>” button to access the [Load cases](#) dialog.

Load input

Loads of load case << LC >>

	Span	Type	Gle	Dist	Gri	Length	from item
1	1	0					
2	2	0					
3		1					
4		2					
5	3	3					
6	4	5					

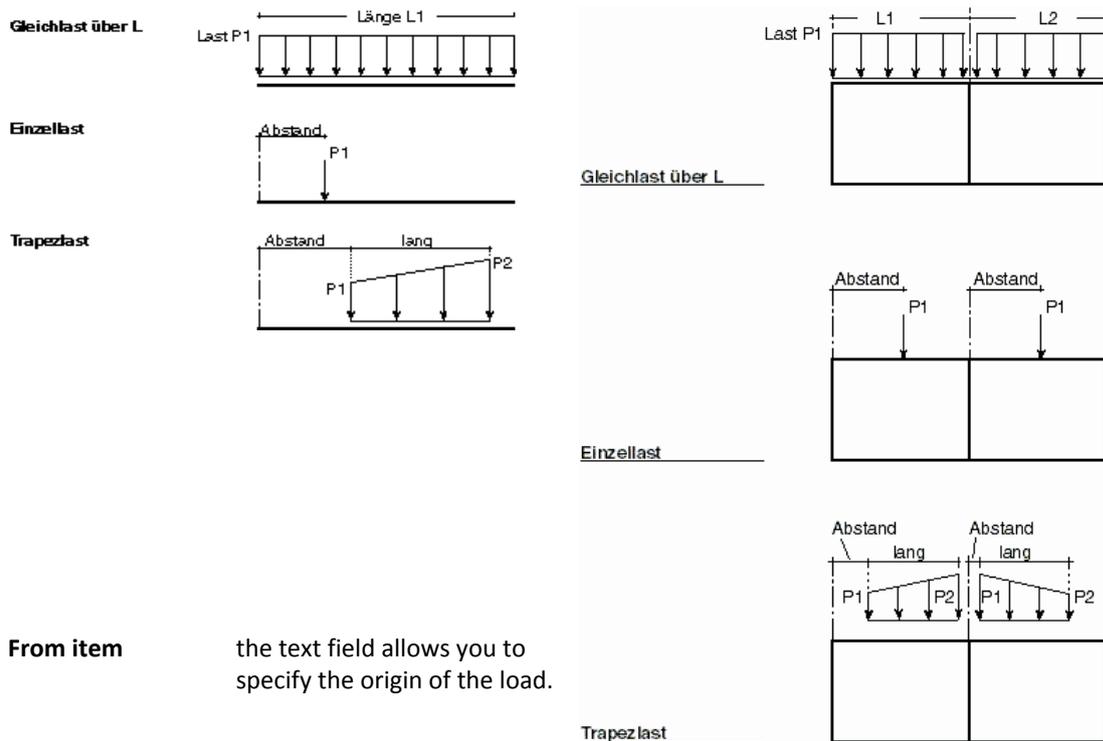
No. of beams
 1 = bar
 2 = strut left
 3 = strut right
 4 = sole

Length of span = 7,00 m

OK Abbrechen

Load types

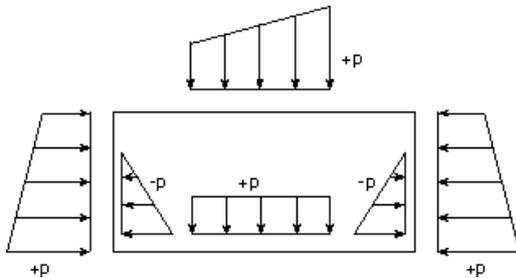
You can select among six different types of loads (“Type” column).



Defining the direction of action of loads

Loads applying to vertical members are positive when they act from the outside to the inside.

Loads applying to horizontal members or the base are positive when they act from the top to the bottom.



Max./min. superposition

Double-click on the "max./min. sup." item in the main menu to activate the function.

Max Min Superposition

Name:

Load case	perman ent	Variable load			Factor	not
		normal	+/-	altern.		
LF 1	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	1,35	<input type="checkbox"/>
q	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	1,50	<input type="checkbox"/>

OK Abbrechen

You can calculate only one single max./min. superposition at a time. When you enter an additional max./min. superposition it will overwrite the previous one.

First enter a name for the superposition. The current max./min. superposition will be put out with this designation. If you have already defined a max./min. superposition its name is shown in the list.

The results of a load case are treated as follows in the superposition:

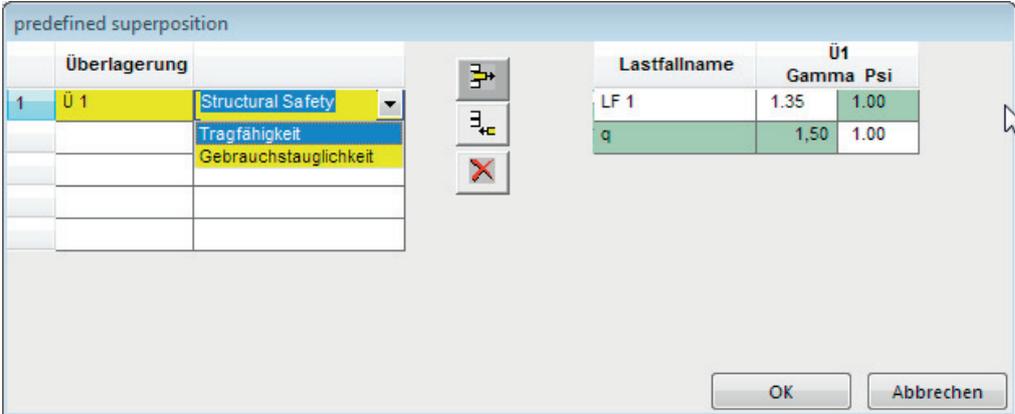
Permanent the results of all load cases that are defined as permanent, are multiplied with the respective factor and are added up.

Normal the load case acts as a normal live load.

The results of all load cases that are defined as normal (live load cases) are multiplied with the respective factor and are added up, if they increase the absolute amounts of the superposition results.

Pre-set superposition

Double-click on the “Pre-set sup.” option in the main menu to access the dialog.



	Überlagerung	
1	Ü 1	Structural Safety
		Tragfähigkeit
		Gebrauchstauglichkeit

Lastfallname	Ü1	
	Gamma	Psi
LF 1	1.35	1.00
q	1.50	1.00

In the left table (superposition) you can specify for each pre-set superposition rule whether a structural safety verification or a serviceability verification is concerned.

In the right table, the safety factor gamma and the combination coefficient Psi are specified for each load case.

Each load case represents an action. For a permanent action, you can optionally select a value for gamma (1.35, 1.0, 0) in the superposition rule. The combination coefficient Psi for permanent loads is set to 1.0 and cannot be adjusted.

For variable actions, the safety coefficient gamma is set invariably to 1.5. You can optionally set the associated combination coefficients to Psi0; Psi1; Psi2; 1.0 and 0. This allows you to describe various pre-set superpositions with different factor combinations.

When you select “Serviceability verification” for a superposition in the left table, all safety factors and combination coefficients of this superposition are set to 1.0 in the right table.

Output

Output of the system data, results and graphical representations on the screen or the printer.

The Output item in the Main menu allows you to start the output on a printer or the screen.

Screen	displays the values in a text window on the screen
Printer	starts the output on the printer
Word	if installed on your computer, the text editor MS Word is launched and the output data are transferred. You can edit the data in Word as required.

Output profile

The output profile allows you to determine the scope of data that should be put out on a printer. This feature avoids unnecessarily detailed printouts.

Note: these settings have no effect on the output on the screen.

System data the data of the structural system are represented in tables. In addition, the modulus of elasticity, the self-weight of the structural system as well as the cross sectional properties A and I of horizontal members, vertical members and the base are put out.

System graph the system graph shows a graphical representation of the structural system and the dimensioning of the individual components.

Load cases

the internal forces N , Q , f , A_s for all members at the break points of the corresponding state line as well as the displacement of the floor slab and the pertinent soil pressure can be put out as results of the individual load case calculations.

In addition to this, the bending design (M , N , h , kh , A_s and A_s'), the shear stresses (Q , τ_0 and τ) and the moment diagram can be put out as graphical representations.

The "Graphic" menu allows you to control the output of the graphics.

Pre-set superposition

Calculation and output of the superposition results (internal forces, floor slab shifts and pertaining soil pressures, design, graph of moments).

Max./min. superposition

The "Graphic" menu allows you to control the output of the relevant graphics.

You can select or deselect all rows for printing.