SSD

SOFiSTiK Structural Desktop

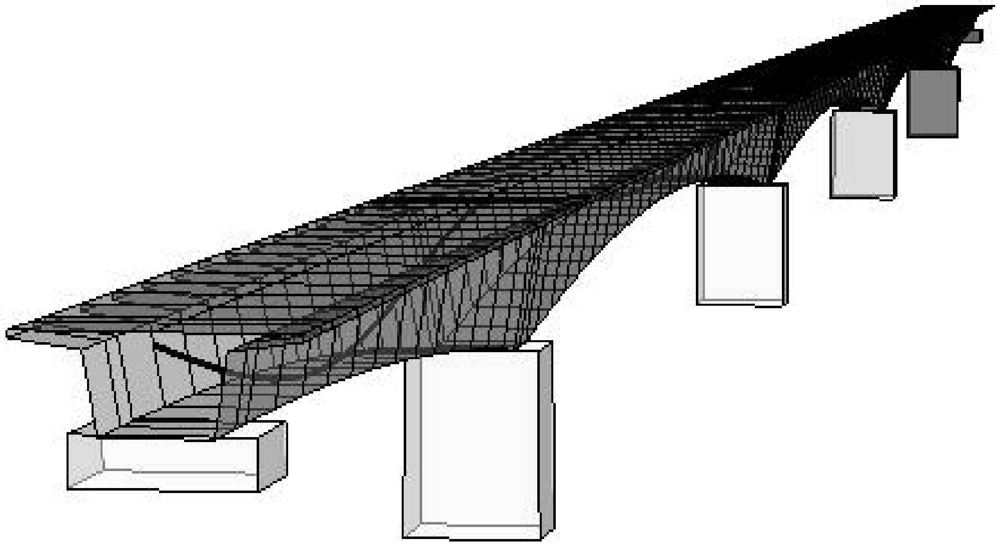
Introduction Bridge Wizard

Program Version 10.89-23

Contents

1 Introduction

1.1 Contents



A detailed introduction of the task „Beam and Slab Bridge“ will be given in the following Tutorial. Using an example out of the German literature [1], the complete workflow from the input up to the analysis and documentation is described in this example. The results can be easily checked with the literature.

1.2 Intention

Working through this entire example will help the user to understand the logic and design of this task. Also the user should be able to work independently on any bridge design using the SSD-Task “Beam and Slab Bridge”.

1.3 Basics

To understand the following Tutorial, the user must know the basic functionality of the SSD. We provide a general SSD-Tutorial **ssd_tutorial_1.pdf** you will find in the SSD menu “HELP > Quick Reference”

Checking input and results according to the latest available codes is in the responsibility of the user and has to be done for each project!

This example is based on the codes listed below

DIN 1045-1 (2.Berichtigung 06/05)

DAfStB Heft 525 (Berichtigung 1:2005-05)

DIN-FB 101 and 102 (2003) and ARS from BMVBW (Federal Ministry of Transport, Building and Urban Affairs)

Additional interpretations of the National Committee NABau and experiences according the usage of the DIN-Fachberichte up to 22nd August 2007

The SSD-Task “Beam and Slab Bridge” in the version 23.10.17 is available for the following codes.

- German DIN Fachbericht 102
- Eurocode EC 2 1991-2
- Spain EHE IAP 2003
- British Standard BS 5400

1.4 Scope of Work

The SSD-Task “Beam and Slab Bridge” is designed to support the bridge engineers in his daily work. Beginning with the system generation including tendon geometry and system, loading according to DIN-FB 101, design of the main structure according to DIN–FB 102 and also definition to construction stages, the task guides the user dialog based through the complete input.

This is a fast and easy possibility to check different alternatives, getting fast results for the primary design and support forces. This can be done with the well known SOFiSTiK accuracy. The automatically generated input file is consistent to the SOFiSTiK environment and can be used with all other SOFiSTiK programs.

2 General Description

2.1 Insert Task

To add the Task “Beam and Slab Bridge” to your project, please use the right mouse click inside the task tree and select the command “Insert Task”. Now the following dialog including all available tasks appears. Select the task “Beam and Slab Bridge” and confirm with OK.

Picture 1: Dialog – Insert New Task

Now the task is available to use in the task tree. You may move the task up and down inside the tree. A double-click on this task will open it for input or modification.

> Using the SSD menu ‘File’ à ‘New Project from Template’ you can select a complete Bridge Example 'dinfb_bruecke' as a template. Please note this template is done for the German market and contains all tasks with German titles.
>
> For a complete Project in English language simply start a new project, delete and add the required tasks.

Picture 2: Standard Template Projects

2.2 Modify Task

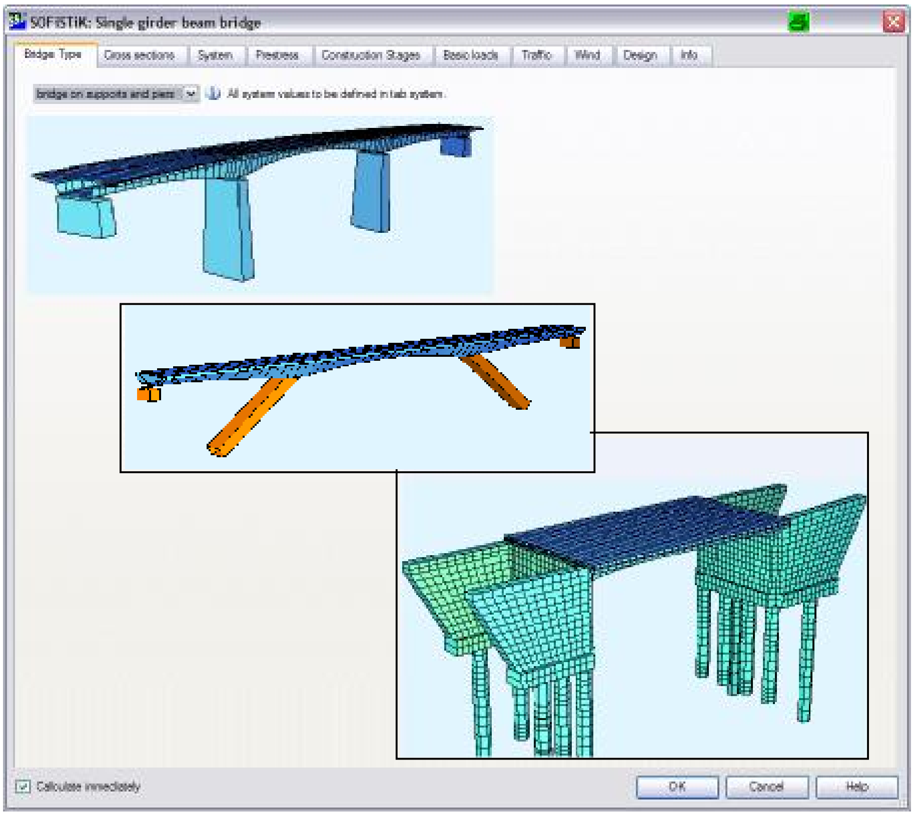
The dialog contains several tabs to define Bridge Type, Cross Sections, System, Prestress, Constructions Stages, Basic loads, Traffic loads, Wind loads and to control analysis and design. Starting a new project or modifying a project just go through every tab and change the input.

> For additional help please read the tooltip's inside the dialogs.

2.2.1 Tab Bridge Type

The task "Beam and Slab Bridge" contains three types of bridges

- Bridge on support and piers
- Inclined strut bridge
- Portal frame bridge (with rigid connection of superstructure to abutment.)



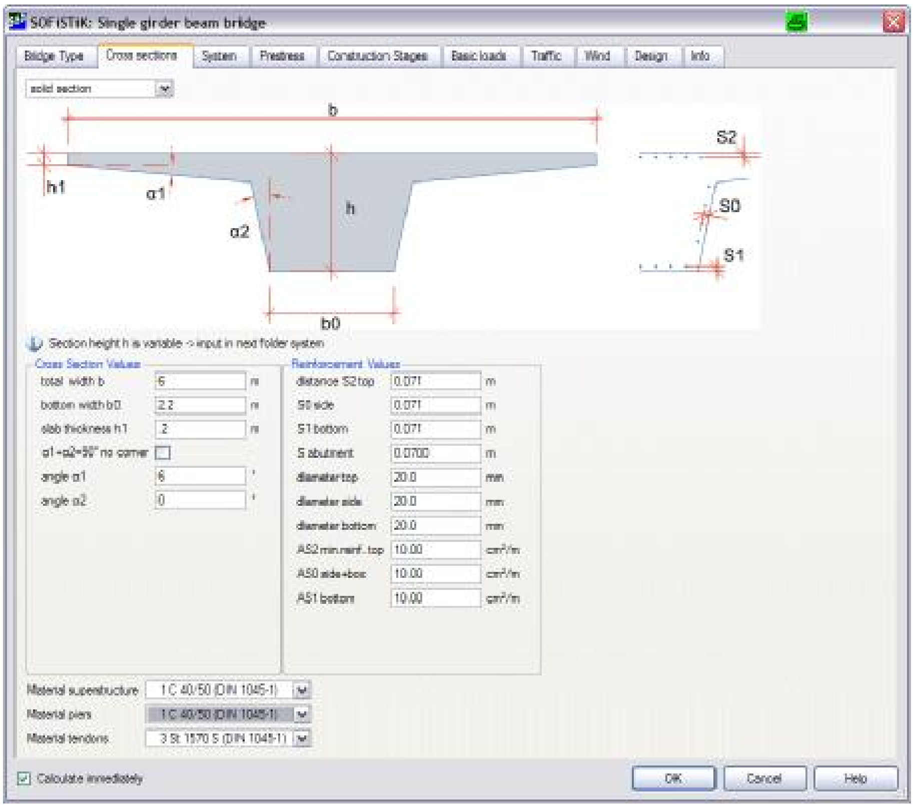
Picture 3: Tab Bridge Type

2.2.2 Tab Cross section

Different types of cross sections are available in this tab. Every type is shown graphically in a basic sketch with all relevant variables. The cross section height is variable according to the alignment and will be defined in the System tab.

The following cross sections are available:

- Solid section
- Box girder declined web
- Box girder vertical web
- Double web T beam
- Three web T Beam
- Slab bridge
- Programmable section

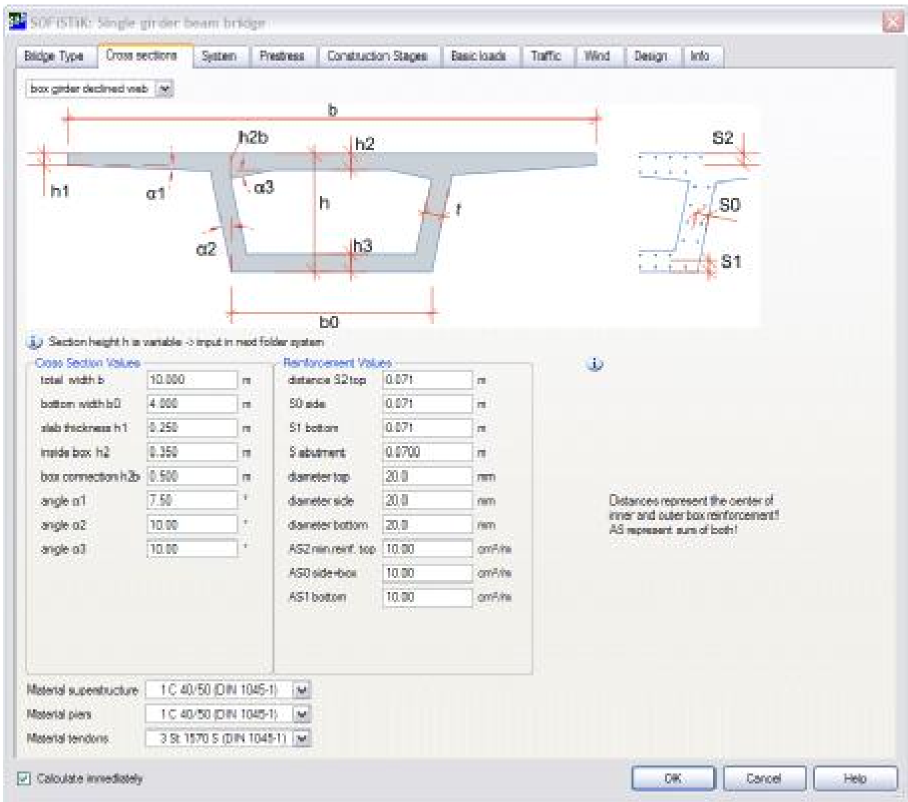


Picture 4: Tab Cross Section – solid section

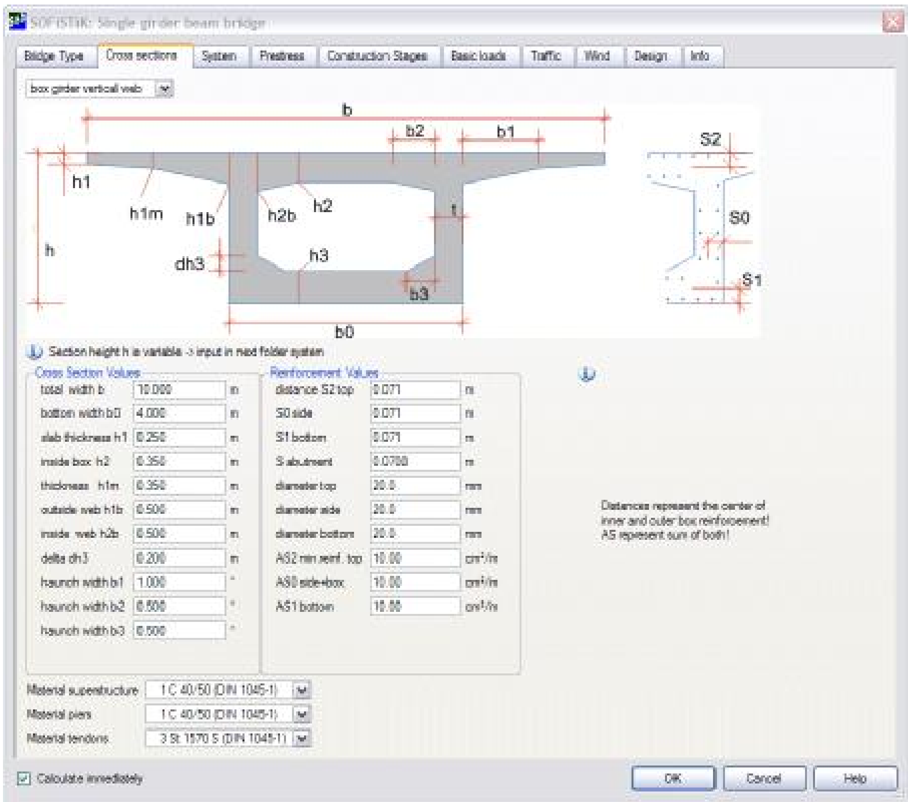
A detailed description you will find in the following example.

> Reinforcement distance is measured to the center line of each reinforcement layer.
>
> Please note the different definition of reinforcement distance. Using box girder cross sections the reinforcement distance is measured up to the center line of inner **and** outer reinforcement!

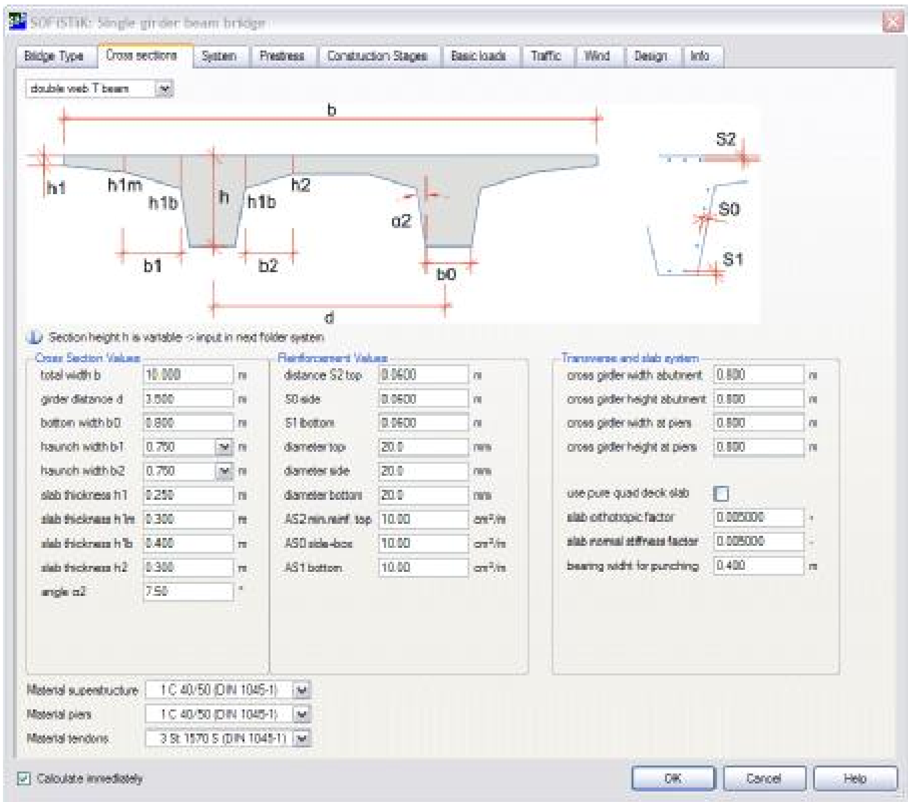
For creating a “programmable section” (see Picture 10) please follow the explanation of the tooltip inside the tab. Advanced knowledge of the SOFiSTiK input language CADINP is strongly recommended for this cross section.



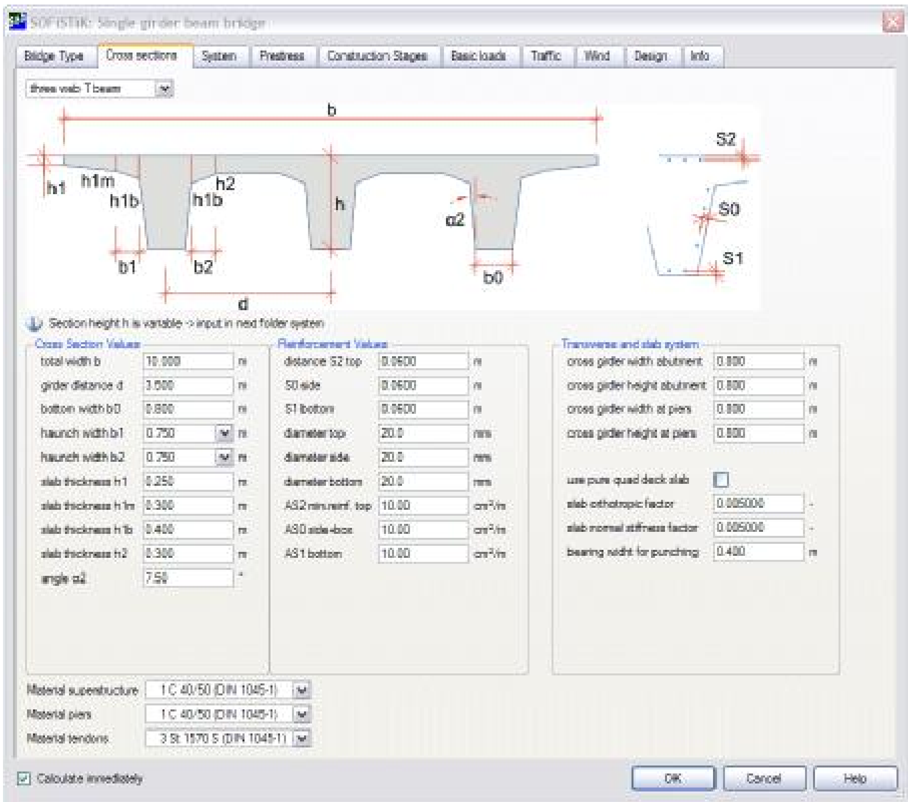
Picture 5: Box girder declined web



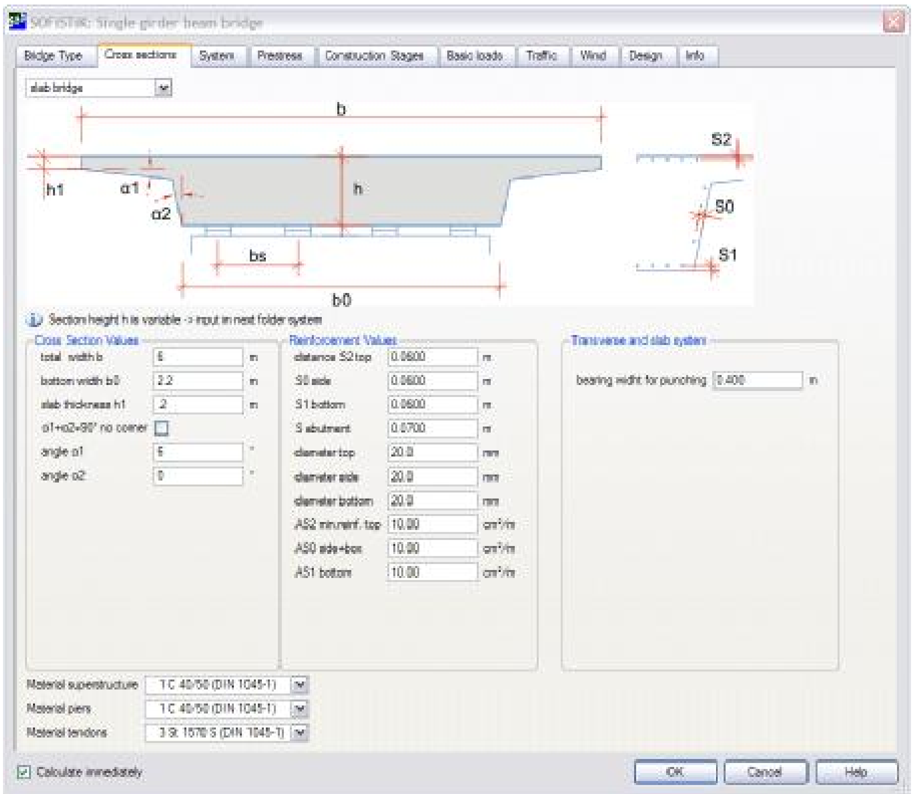
Picture 6: Box girder vertical web



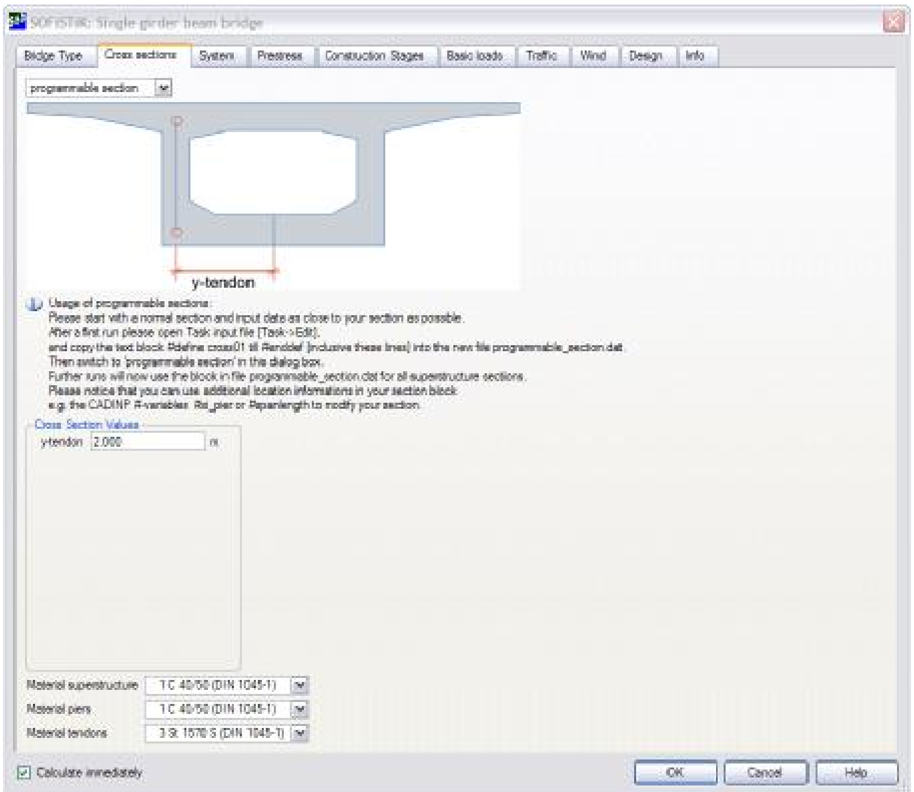
Picture 7: Double web T Beam



Picture 8: Three web T-Beam



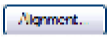
Picture 9: Slab bridge



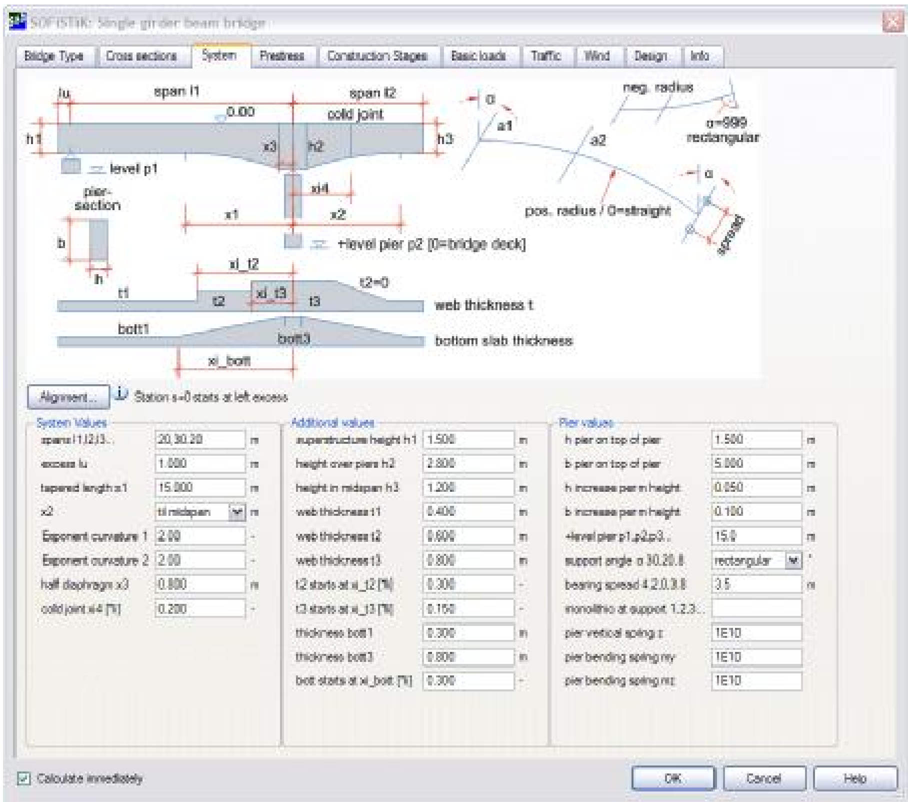
Picture 10: Programmable Section

2.2.3 Tab System, Alignment

This tab deals with all kind of system and alignment information. There are three areas for the input of "System values", "Additional Values" and "Pier Values". The dialog varies according to the cross section. Also the location of cold joints for the span by span erection has to be defined in this tab.

Use the button 

to define the horizontal and vertical alignment.



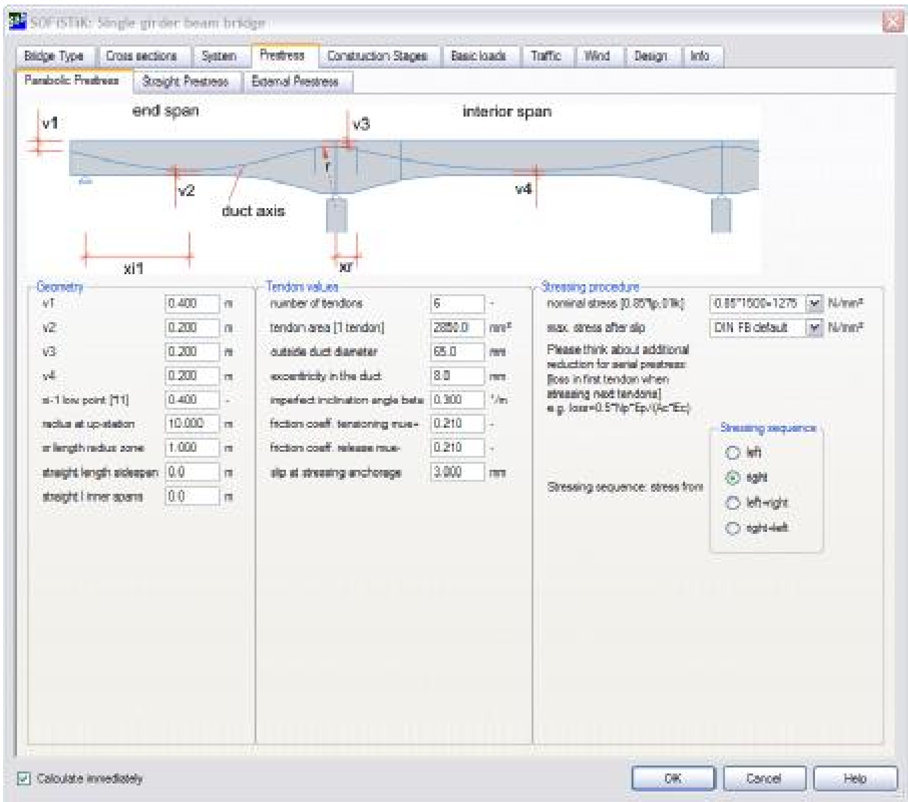
Picture 11: System (box girder with variable web width)

2.2.4 Tab Prestress

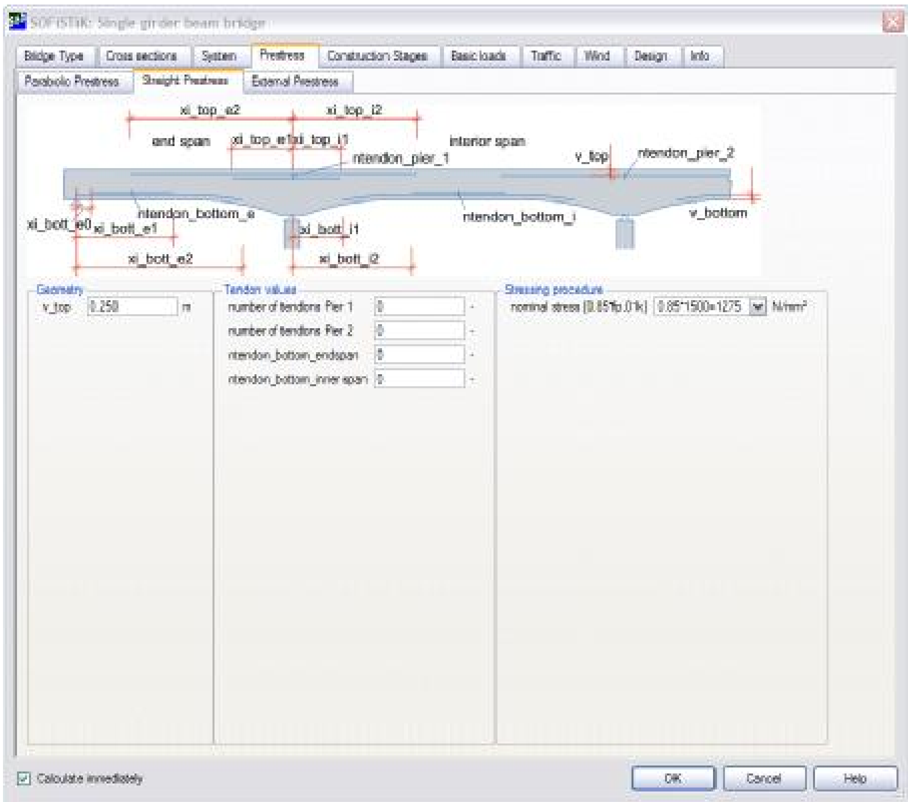
Defining the Prestress is also dialog based. There are three different tendon geometries available:

- Parabolic Prestress
- Straight Prestress
- External Prestress

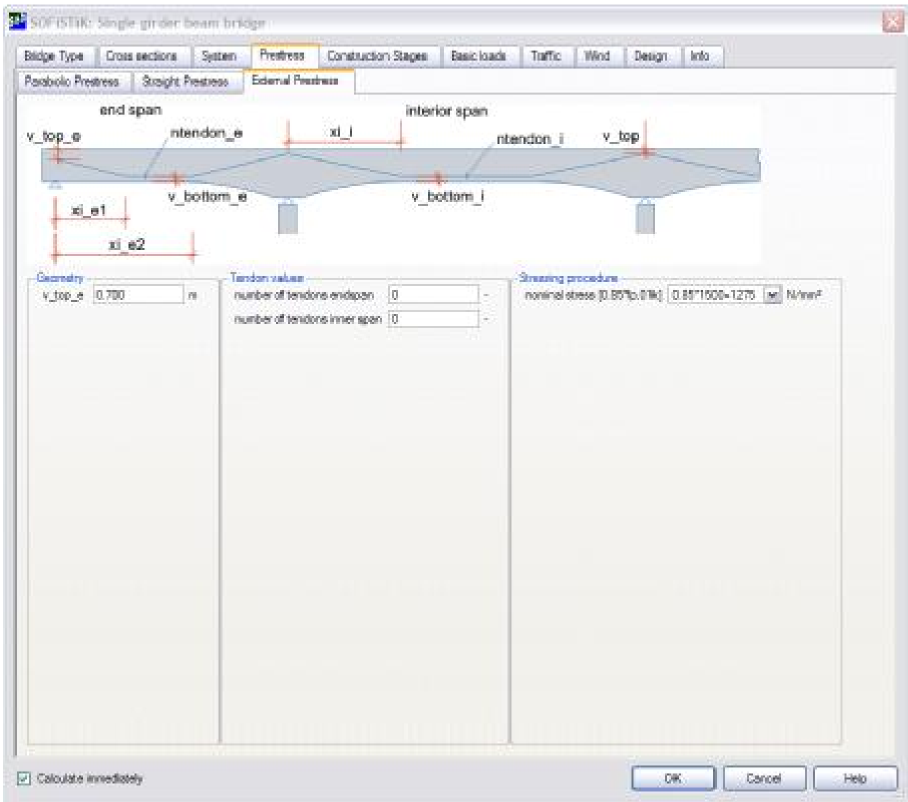
In a basic sketch all input variables are explained.



Picture 12: Prestress- Definition of Parabolic Prestress



Picture 13: Straight Prestress



Picture 14: External Prestress

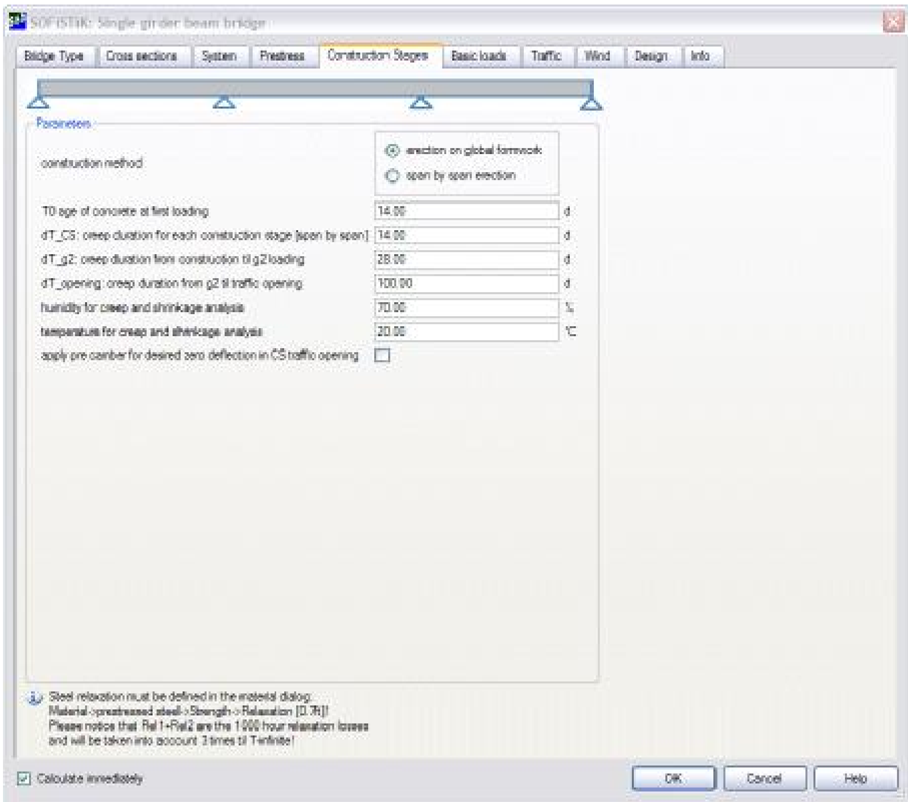
2.2.5 Tab Construction Stages

At them moment there are two different erection types available:

- Erection on global framework
- Span by span erection

All important variables especially for the creep, shrinkage and relaxation analysis are defined here. An automatic precamber analysis [6] for a desired zero deflection in construction stage traffic opening is also available.

> Additional information for automatic creep, shrinkage and relaxation analysis including construction stages and precamber analysis including several examples you will find in the SOFiSTiK manual **csm_1.pdf** .



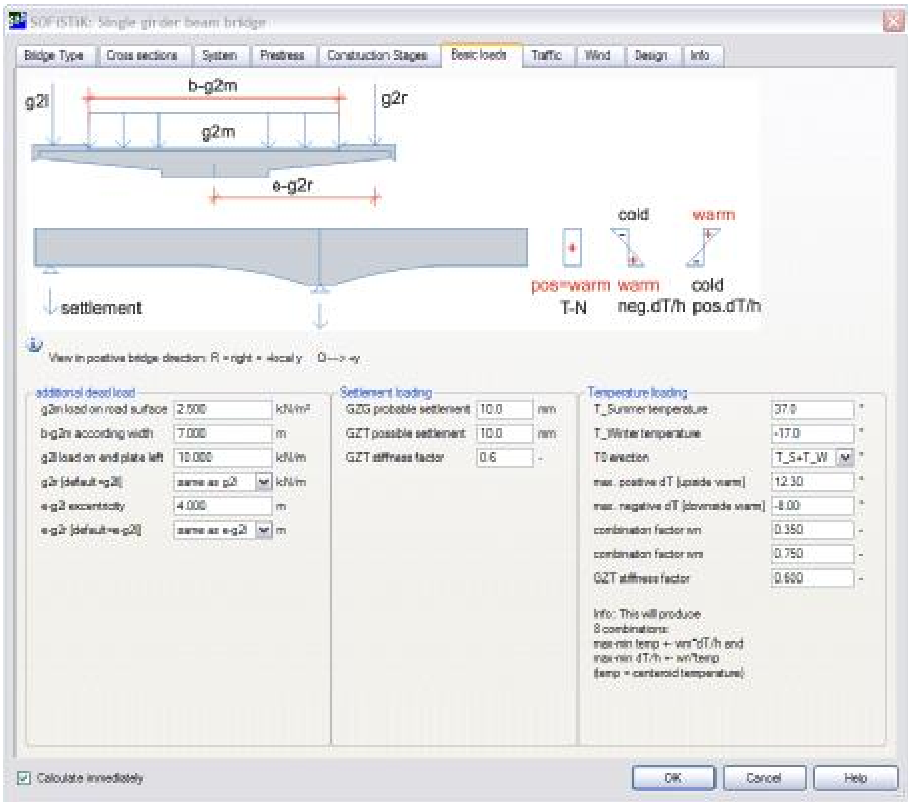
Picture 15: Construction Stages

2.2.6 Tab Basic Loads

Inside this tab all additional dead loads (dead load on road surface including allowable tolerance, load on end plates), settlements and temperatures will be defined.

For the Ultimate limit design (GZT in German language) the stiffness, which will be applied only on the possible settlement loading and on the temperature loading can be reduced by a factor. This factor is according to DIN-FB.

> Additional loads of 0.5 kN/m² according to DIN-FB (allowance for tolerance for the road surface), must be added manually to the basic load on road surface.

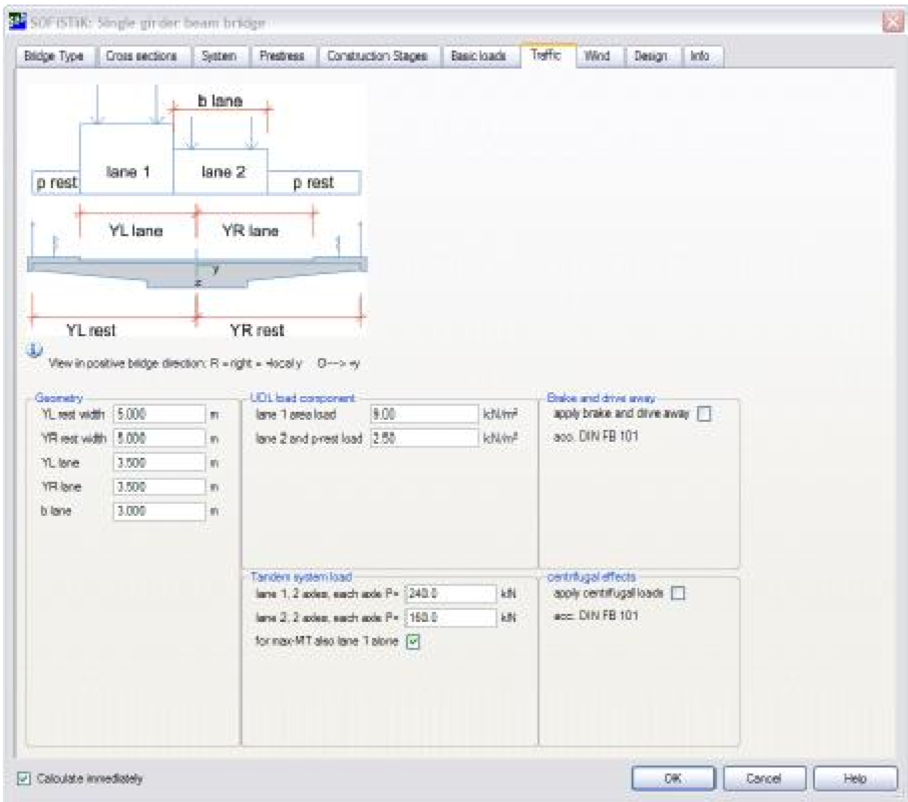


Picture 16: Basic loads

2.2.7 Tab Traffic

With only a few inputs all necessary information for the entire traffic load according to DIN-FB 101 will be automatically generated. The UDL load will be applied according to the different lanes and spans. The tandem system (TS) load can be applied also alone to the structure.

Modifying all values is very easy and comfortable. As an option you can also apply additional loads for brake and drive away and for centrifugal effects.



Picture 17: Traffic loads

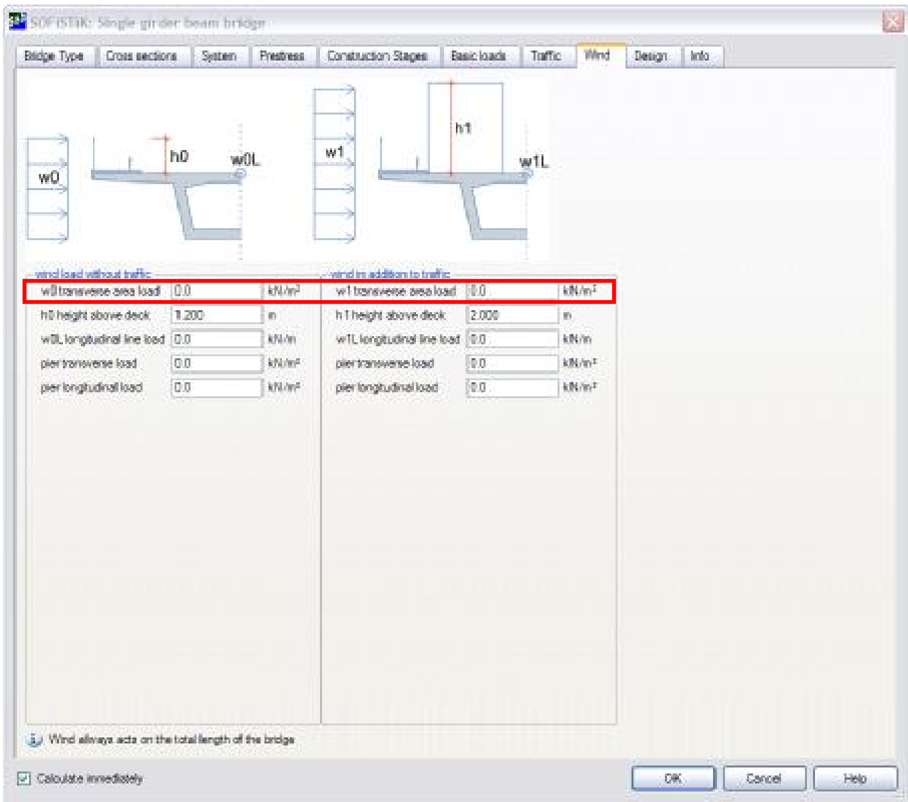
2.2.8 Tab Wind

In this tab you can activate additional wind loads. There are two separate loads available.

– Wind load without traffic

– Wind load with traffic

The default setting is without wind load, because this load depends very much on the bridge geometry



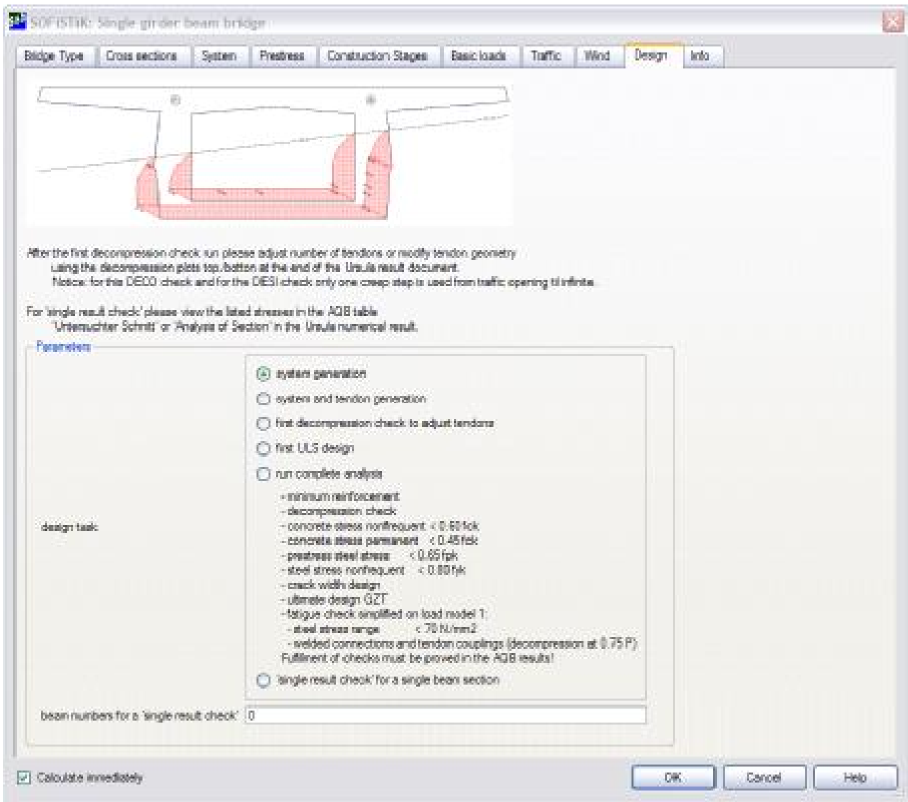
Picture 18: Wind loads

2.2.9 Tab Design

Inside the tab design different design tasks are available. This is very important for the design process, because it allows you to go stepwise through the design process.

- **system generation**: In a first step the program generates all cross sections including the effective width and the structural system. We use beam elements with cubic ansatz function, variable cross sections including tapered sections and eccentric reference axis [5].
- **system and tendon generation**: in the next step the tendon geometry and prestress system will be applied on the structure. We strongly recommend to check the geometry using the ANIMATOR and URSULA output plots, generated by the module GEOS.
- **first decompression check to adjust tendons**: This step helps the user to optimize the tendon geometry. Creep, shrinkage and relaxation results will be analysed ant the results can be checked in the URUSLA output plots. If necessary, changes are easy to apply to the system by modifying the input inside the task.
- **first ULS design**: After the tendon optimisation the next step is the ULS design of the structure. This will help you to check the general bearing capacity.
- **run complete analysis**: This is the most advanced design task. A complete analysis will be done for the entire structure. All relevant design checks are included. The URSULA output contains plots for every design check including necessary reinforcement plots.
- **'single result check' for a single beam section**: This check starts a complete analysis including a full output just for a single beam section. The program automatically select a beam section from midspan and out of the middle of the bridge. You may directly select a beam element using the ANIMATOR and the "select element" function.

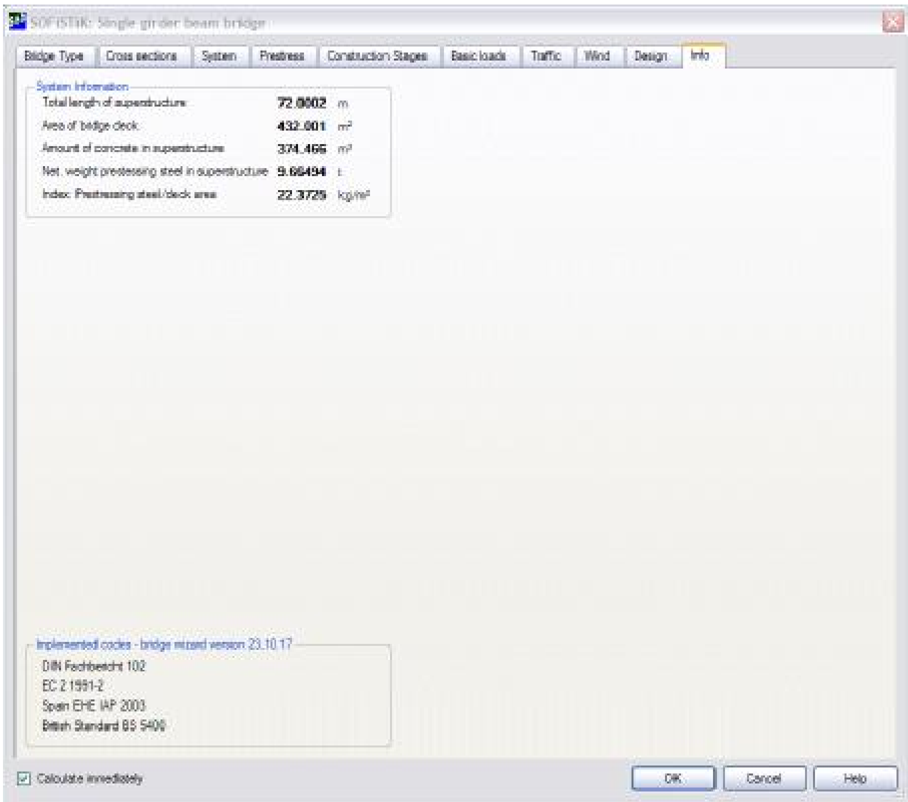
Don't use the 'single result check' before the complete analysis, because the design will be done only for a single beam section.



Picture 19: Design

2.2.10 Tab Info

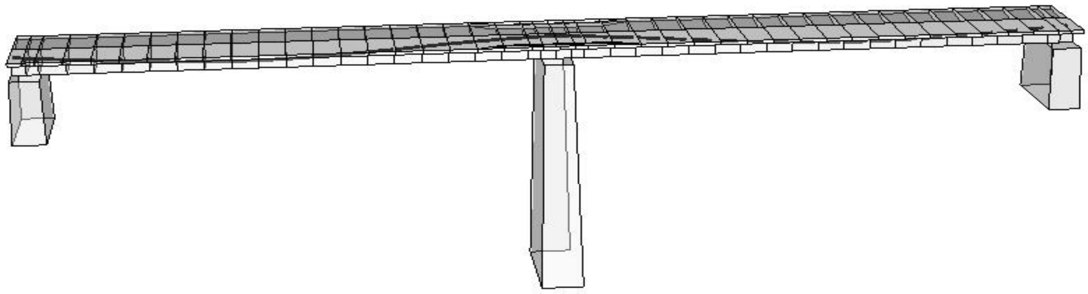
The info tab provides you with some additional information of your structure. This can be very important in early design steps.



Picture 20: Information about bridge structure

3 Example Two Span Bridge

3.1 Problem



We took the following example out of [1]. It is a small two span bridge with 2 x 26.60 m span width and a single T Beam. The tendon geometry is parabolic. All relevant parameters are show in the following table.

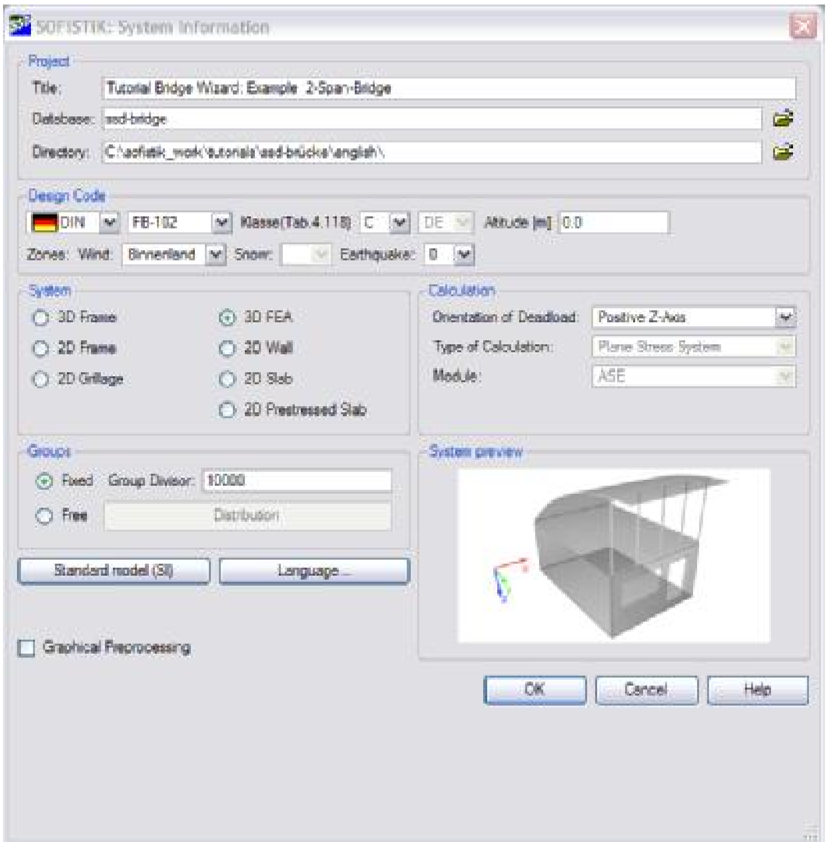
This system seems to be very simple, but it will show the entire functionality of the task. Also it is very easy to modify the input data to analyse alternative systems.

Geometry			
Total length	L	=	54.60 m
Span width	l	=	2 x 26.60 m
Total width including end plates	B	=	6.50 m
Width between guardrail	B	=	6.00 m
Width cross section top	b_{top}	=	6.00 m
Width cross section bottom	b_{bottom}	=	2.20 m
Road width	b_{road}	=	6.00 m
Total cross section height	H	=	1.55 m
Structural height	h	=	1.25 m
Slenderness ratio	λ	=	21.3
Bridge deck	A	=	319.20 m²
Clearance	h	=	4.70 m
Material and Prestress System			
Concrete	C 40/50		
Reinforcing Steel	BSt 500 SB		
Prestressing Steel	St 1570/1770		
Prestressing System	SUSPA - strands		
Building Categorie			
Design longitudinal direction	Categorie C		
Design transverse direction	Categorie D		
Boundary conditions			
Exposition class	XC4, XD1 and XF2		
Design velocity	V_E = 100 km/h		
Load Model			
Tandem axis for global design	Load Model 1		$\alpha_Q = 0.8$ and $\alpha_q = 1.0$
Single axis for local design	Load Model 2		$\beta_Q = 0.8$
Fatigue load model	Load Model 3		

Table 1: Bridge Parameter

3.2 Create New Project

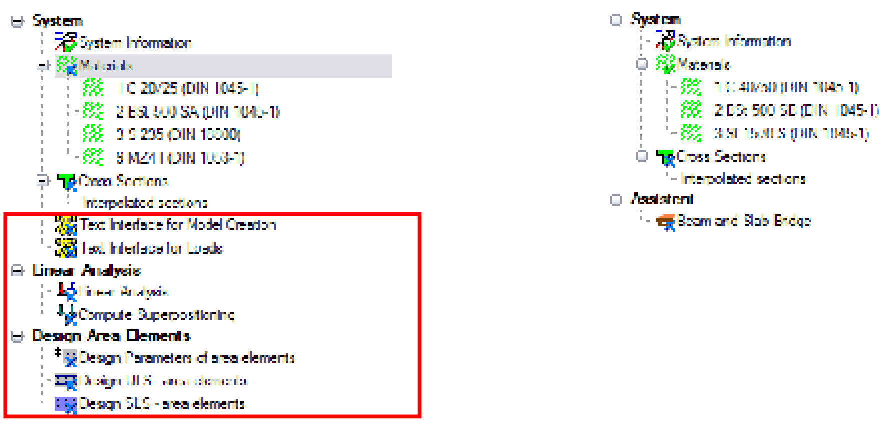
First open the SSD and create a new project.



Picture 21: System information new project

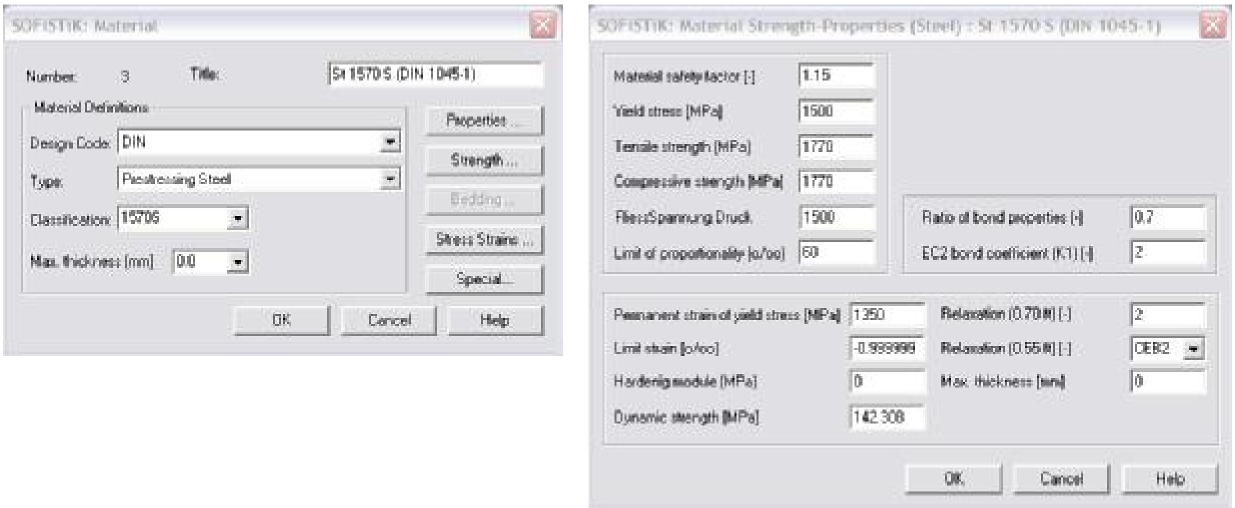
After that you see the task tree as shown on the left hand side. Please change Materials as listed in Table 1. Delete all tasks except for System Information, Materials and Cross sections. You may use the right mouse button „✖ Delete“.

After that insert a new group name it “Assistant” and insert the task “Beam and Slab Bridge” under the new group.



3.3 Define Materials

All relevant materials are defined outside the task “Beam and Slab Bridge”. Therefore the definition has to be done in the task “Materials”. To change the materials use the right mouse button “Edit” and use the Material Dialog for all changes.



Picture 22: Dialog Material

3.4 Input Task "Beam and Slab Bridge"

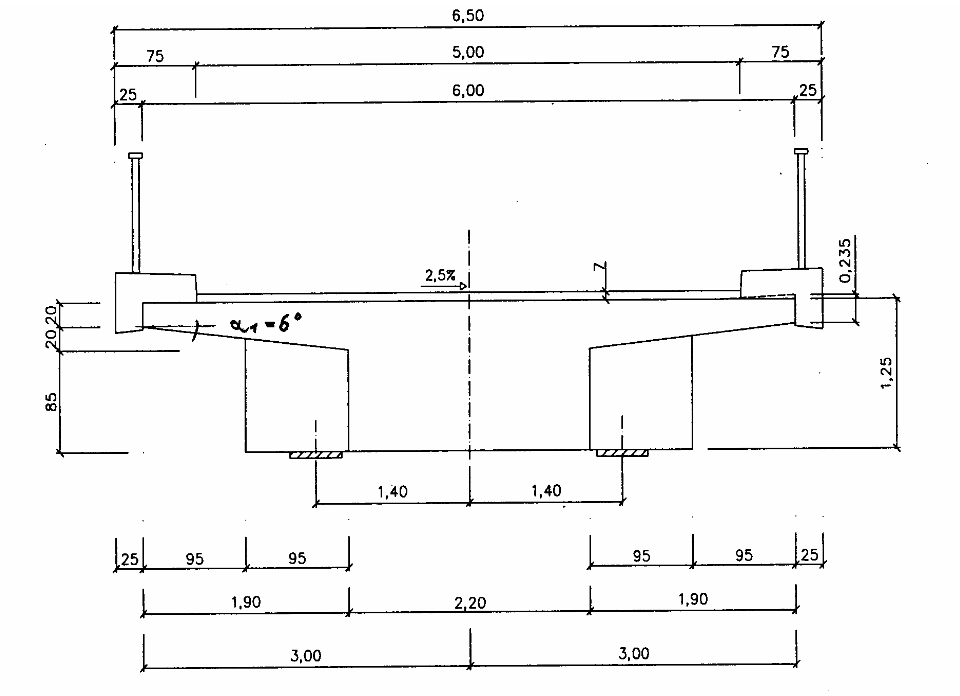
The complete definition except the materials will be done in the task " Beam and Slab Bridge" Pleas open the task with a double click and generate all input step by step working through all tabs from left to right.

3.4.1 Define Bridge Type

In this small example we deal with a "bridge on supports and piers"

3.4.2 Define Cross Section

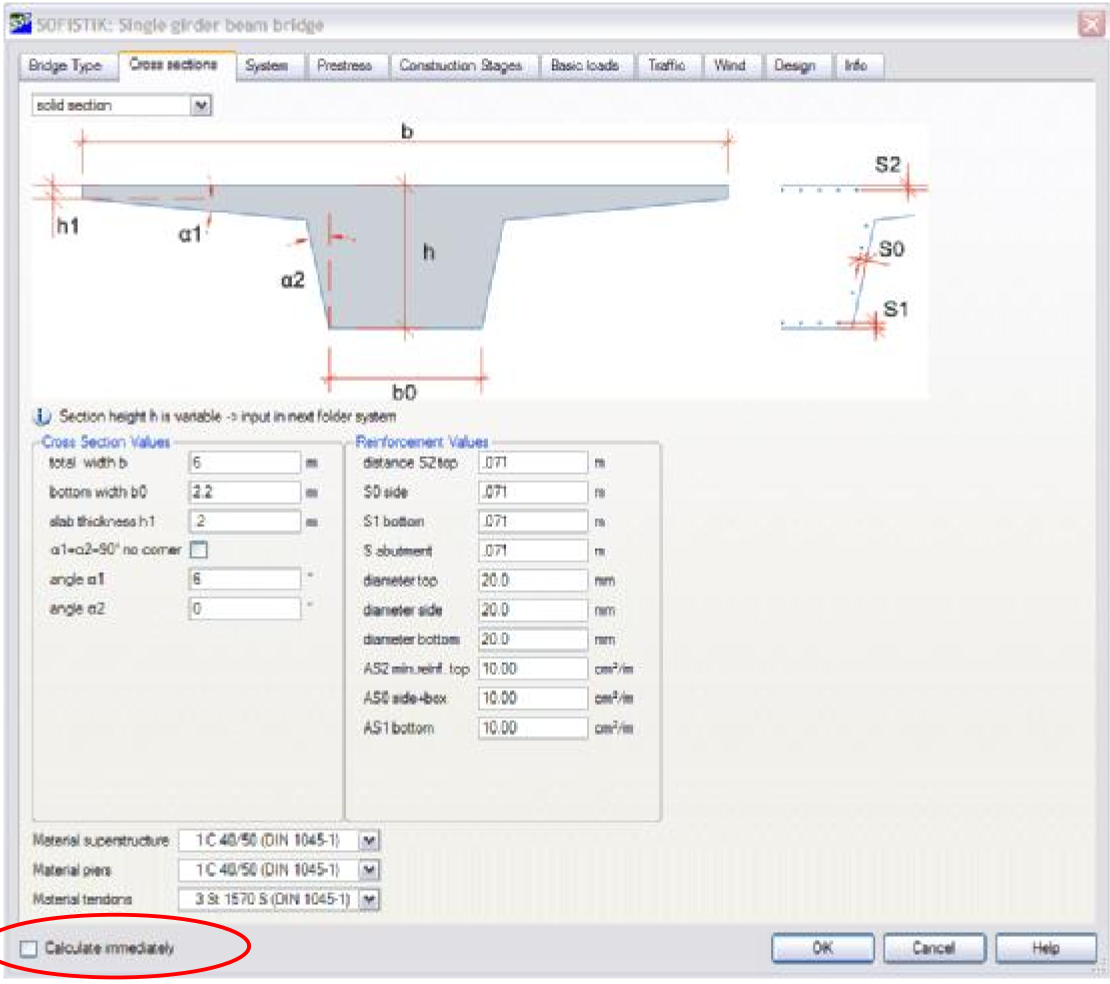
Use the "solid section" to define the T Beam cross section shown below.



Picture 23: Cross section example [1]

Now insert all variables, total width = 6.00 m, bottom width = 2.20 m, slab thickness = 0.20 m angle α_1 = 6° and angle α_2 = 0°, for the entire cross section in the tab cross section. The total height will be defined in the tab "System". Please note that the program does the calculation for the **effective width automatically**. The distance from concrete surface to the center line of the different reinforcement layers are calculated to 7.1 cm, due to a nominal cover of 45 mm according to DIN-FB 102, a presumed stirrup diameter of 16 mm and a diameter of 20 mm for the longitudinal reinforcement. (45 + 16 +20/2 = 71 mm).

To select or change the materials use the combo-box at the bottom of the dialog. In case the required materials are not there or have to be modified please proceed as written in chapter 3.3 . Additional input to minimum reinforcement and bar diameter see picture below.



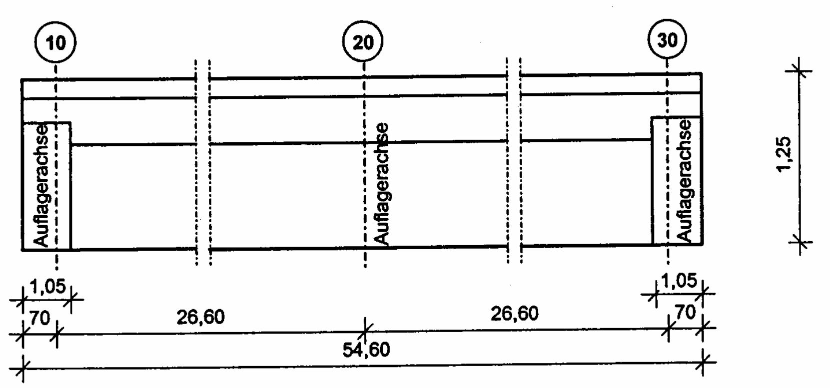
Picture 24: Input Cross section T Beam

In case you stop the input with the CANCEL button all input will be lost. During the input procedure we recommend to deactivate the option "Calculate immediately". Now if you close the dialog with OK all input will be saved without directly starting the calculation.

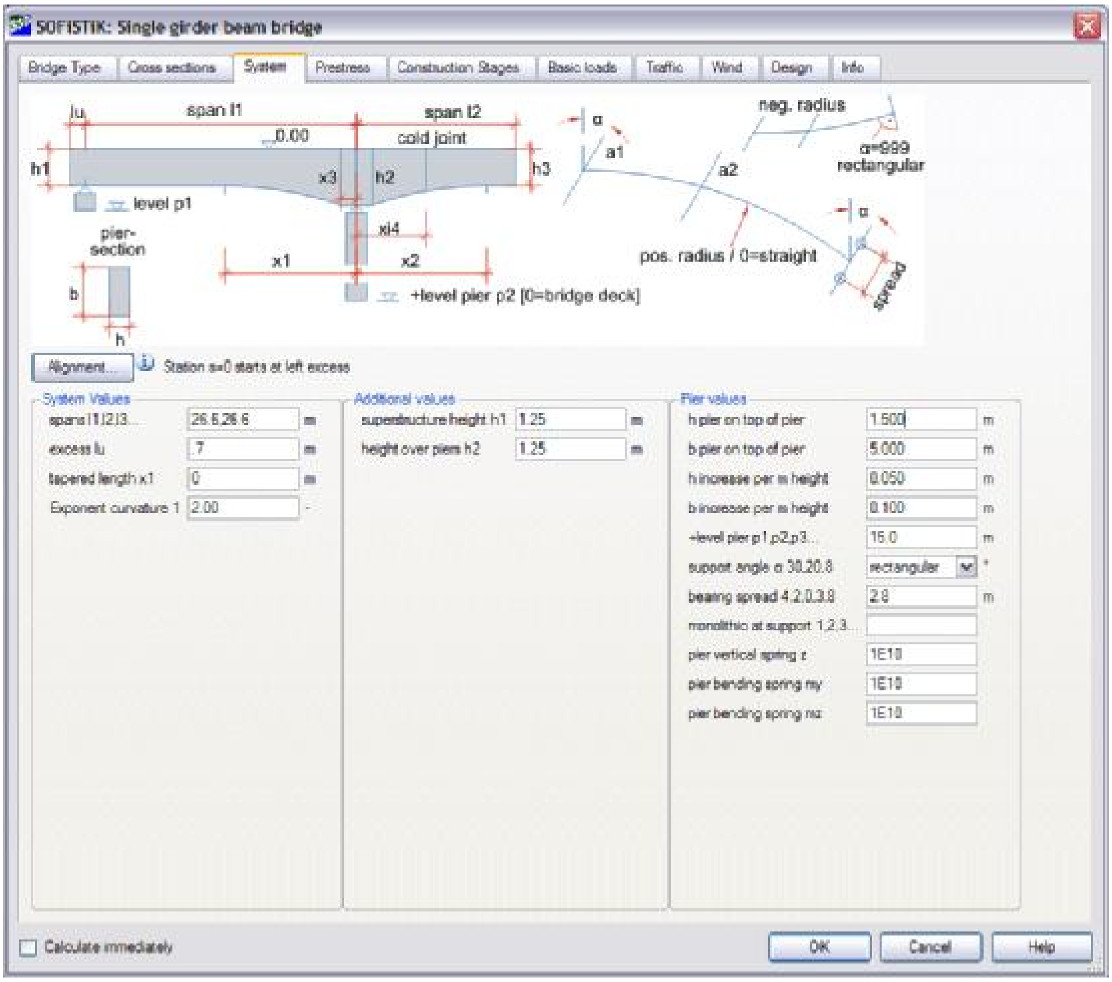
3.4.3 System

After defining the cross section we move to the next tab, to define the bridge system. Inside this tab only a few inputs are necessary. We define the two spans with 26.6 m each. We do have an excess length of 0.70 m on both ends and a tapered length of 0.0 m. As you see, the dialog changes dynamically depending on your input. All other input will be shown in the picture below.

The pier values are not relevant in our example. Simply use the default settings.



Picture 25: Longitudinal sketch of bridge [1]



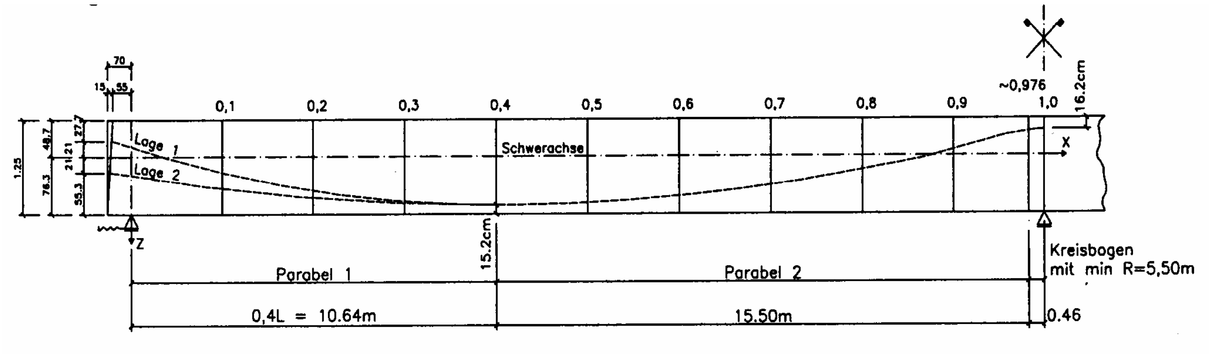
Picture 26: Input System

You may define multiple spans with the same span length for example using the input 20,4*30,20 for a 6 span bridge.

As a default setting we use floating supports on elastomeric springs. A change is possible only in the input data file.

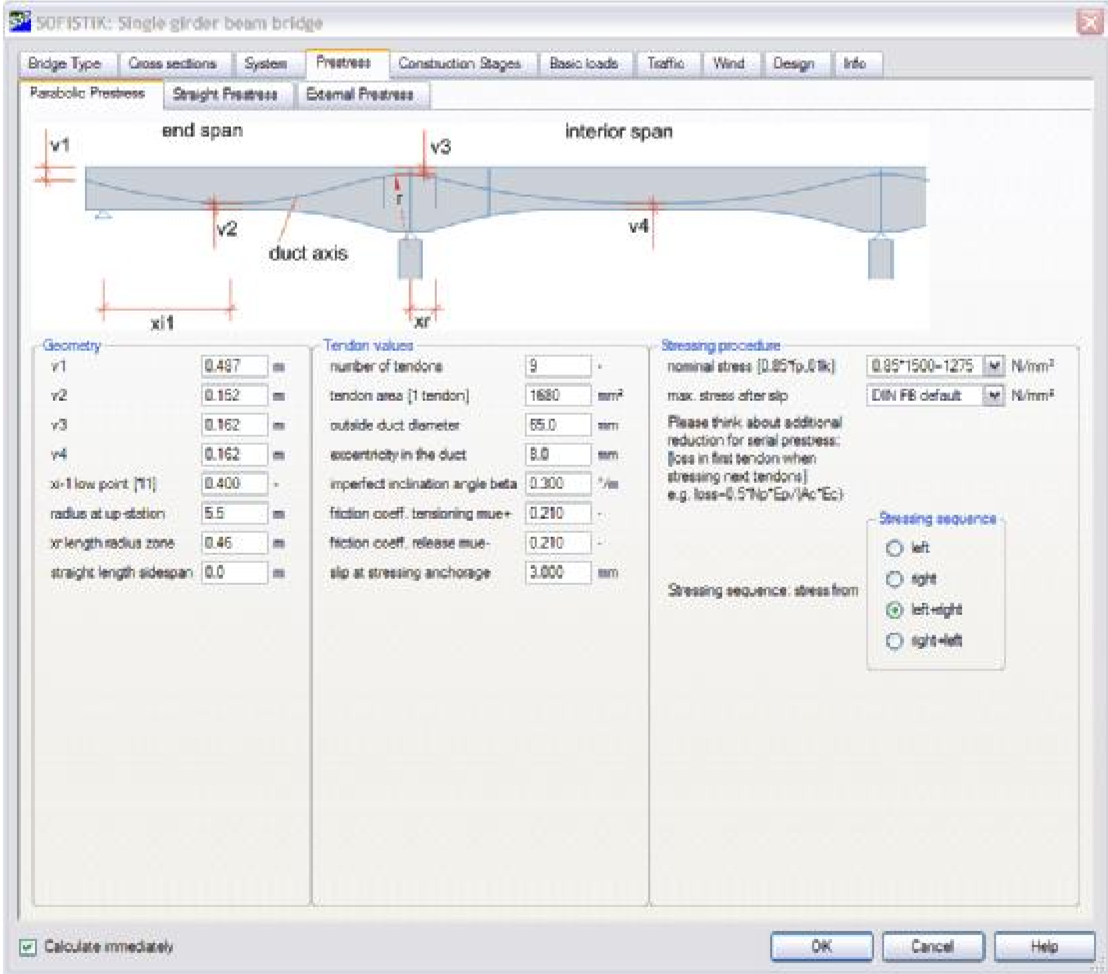
3.4.4 Prestress

In the next input step we deal with Prestress and tendon geometry. This will be done in the tab Prestress. In our example symmetrical parabolic tendon geometry will be used. Over the middle support the tendons have a circular arc shape with a radius of 5.5 m. The edge distance in the anchorage area equals to 0.487 m. In midspan (0.4*L) the bottom edge distance equals to 0.152 m. We will use a SUSPA® post tensioned system with 140 mm² tendons. Every tendon contains 9 strands Type 6-12 with $P_{m0,max}$ = 2142 kN with 1680 mm² tendon area.



Picture 27: tendon geometry [1]

All other input is shown in the following picture. Important is the selection of the stressing sequence. In this example we use a stressing sequence from left and from right. Without any further input the program will use the maximum allowed values according to DIN-FB 102 including an overstress factor of κ = 1.5. As a result of calculation you will get a plot which contains the tendon geometry and the tendon forces.



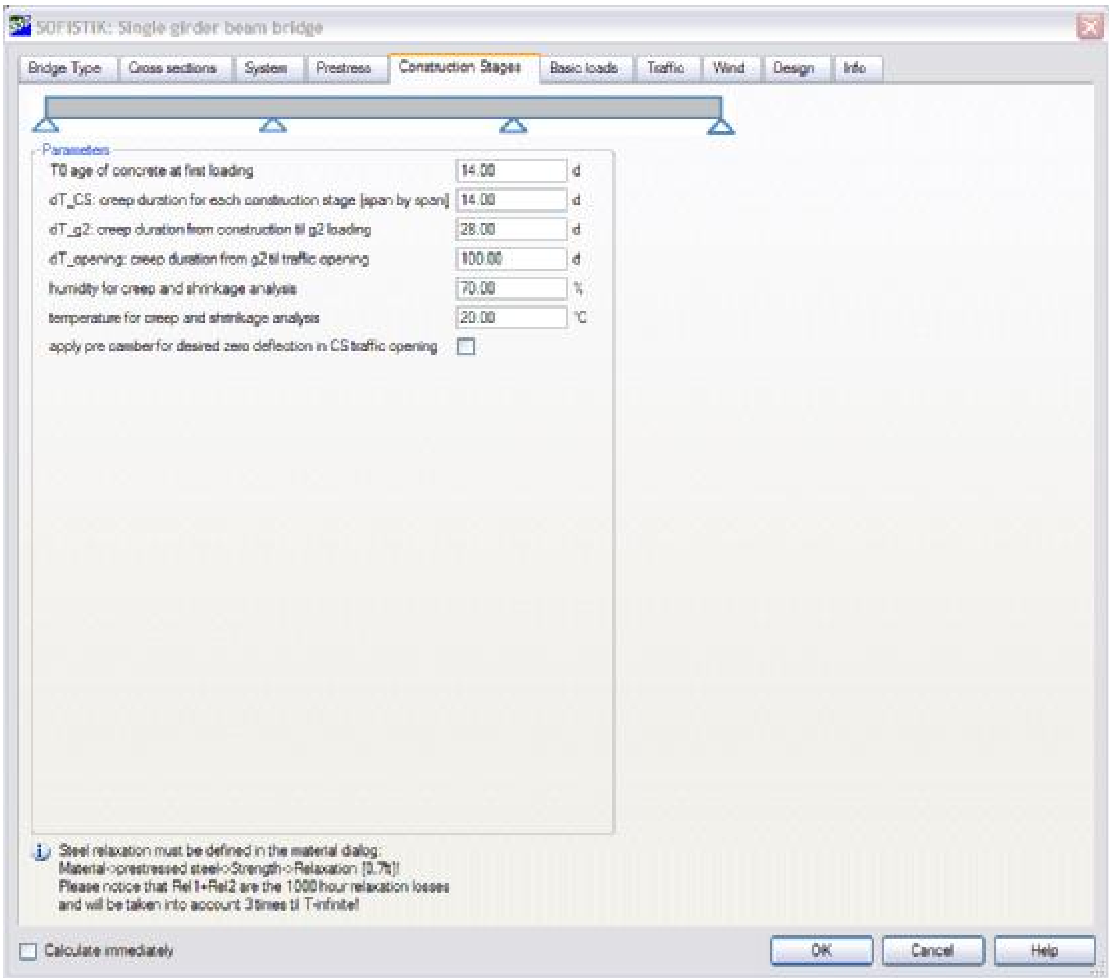
Picture 28: Input Prestress

3.4.5 Construction Stages

In this example no specific construction stages are possible. The bridge will be erected in one peace on a global framework.

The values for creep, shrinkage and relaxation are self explaining. We will use the default settings.

Also we don’t use the option for a precamber calculation.



Picture 29: Input construction stages

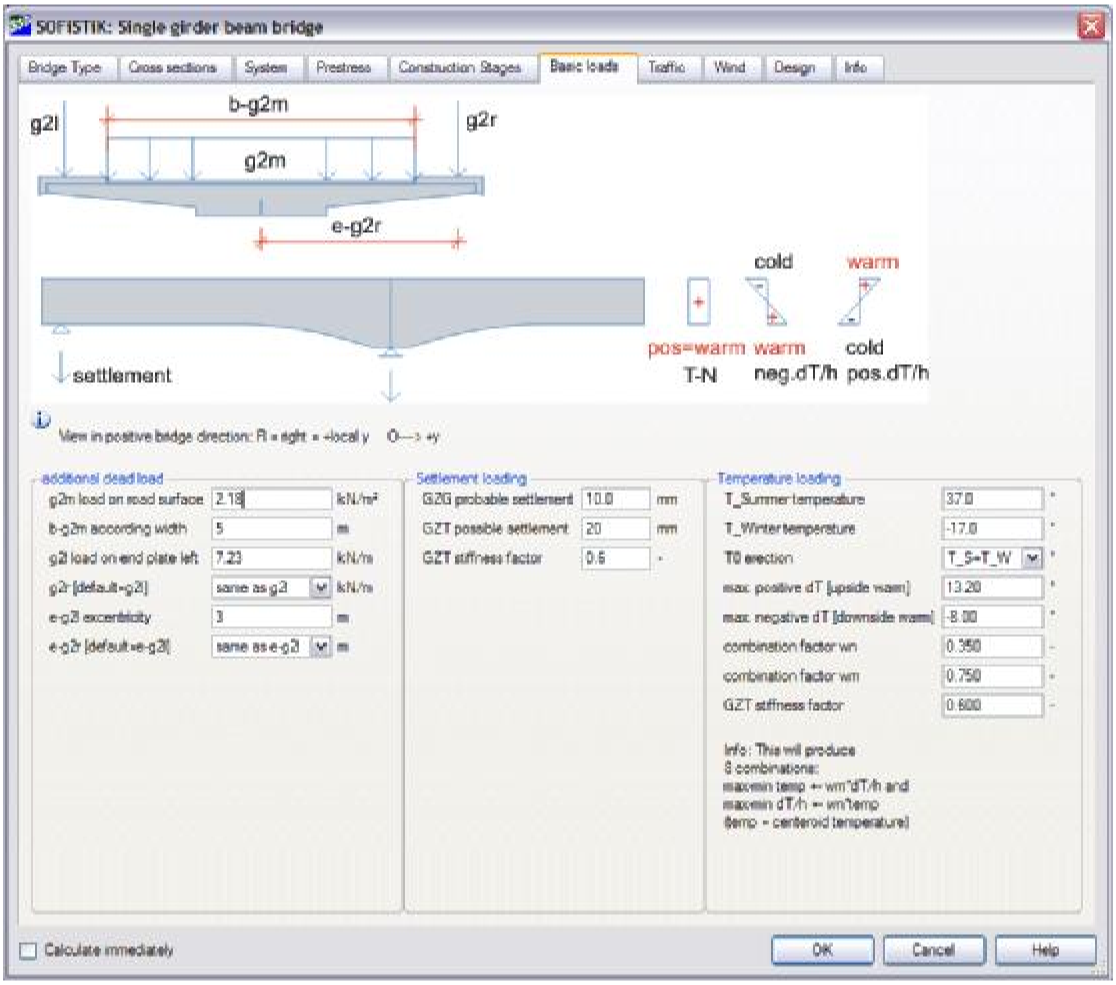
3.4.6 Basic Loads

Now we start to define loads on our system. Basic loads are: additional dead loads, settlement and temperature loads. The additional dead load equals to 2.18 kN/m² including an allowance for tolerance for the road surface of 0.50 kN/m². The dead load for the end plate equals to 7.23 kN/m for each side [1].

Probable settlement is 100 mm and possible settlement is 20 mm. We also use a stiffness factor of 0.6 according to ARS 11/2003 (7) for constraints caused by settlement and temperature, because these loads create internal forces and moments and we don't use a more specific analysis with cracked conditions.

Temperature loads are defined according to DIN-FB 101. The positive temperature difference (pos. dT/h) equals to 13.2 K including a factor K_{sur} for 7 cm pavement.

All other values are listed in the picture below.



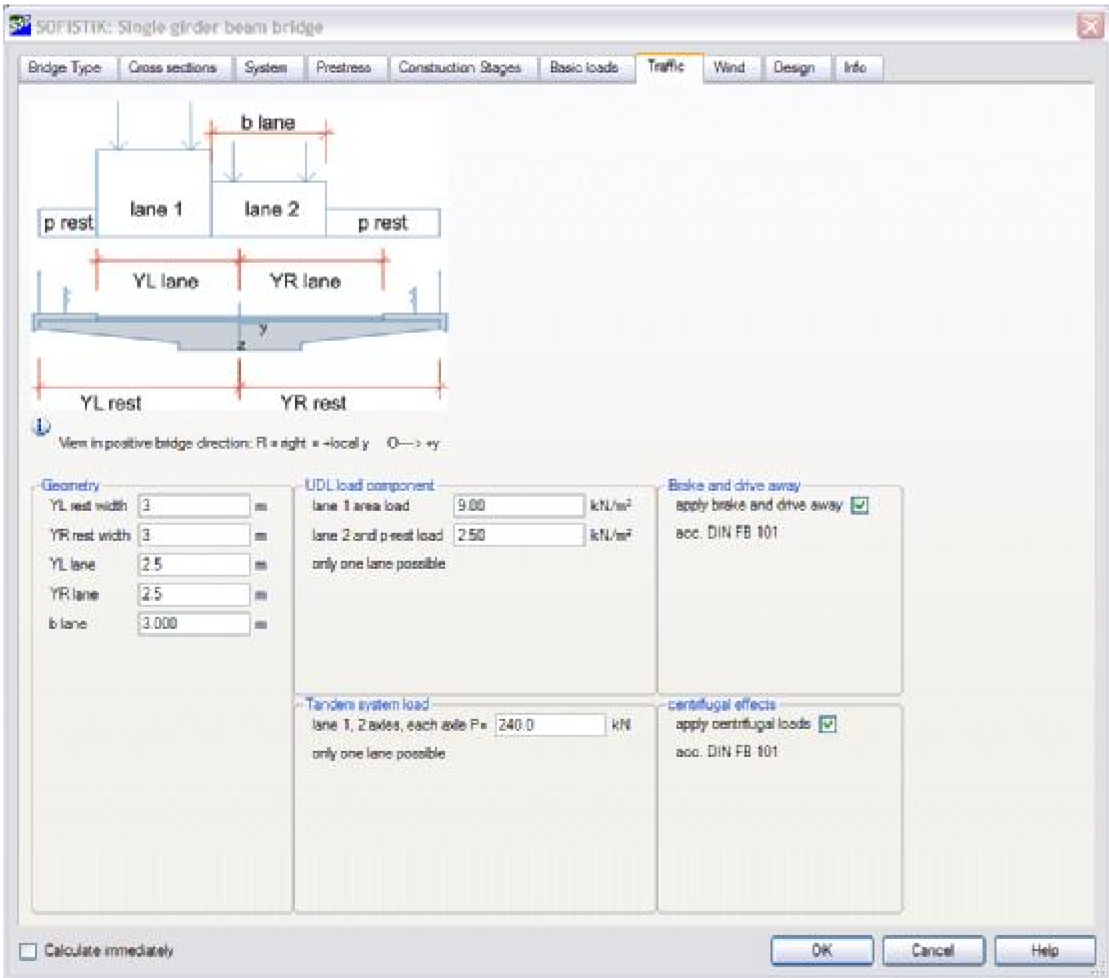
Picture 30: Input basic loads

3.4.7 Traffic Load

After finishing the basic load, we go on with the traffic loads. It is very important to define eccentric lanes on our bridge. For that we use geometric variables left and right to the bridge center line.

- **YL rest width+ YR rest width = width** in between the rest load must be applied. En general the width between the railings.
- **YL lane** + **YR lane** = **total width of roadway**; area where to apply the overload UDL, respectively the tandem system load TS
- **b-lane** = **width of lane**, en general 3.00 m in Germany

Please insert/modify all values for the area loads UDL, lane 1 area load and p-rest load, as well as the tandem system load. You may use the option for calculating brake and drive away loads, as well as centrifugal loads. The program automatically calculates the maximum number of lanes and the unfavourable load distribution.



Picture 31: Input traffic loads

All loads are beam loads with corresponding eccentricities. We recommend to check all loads using our program WINRAF.

3.4.8 Wind Load

In this example we don't apply any wind loads

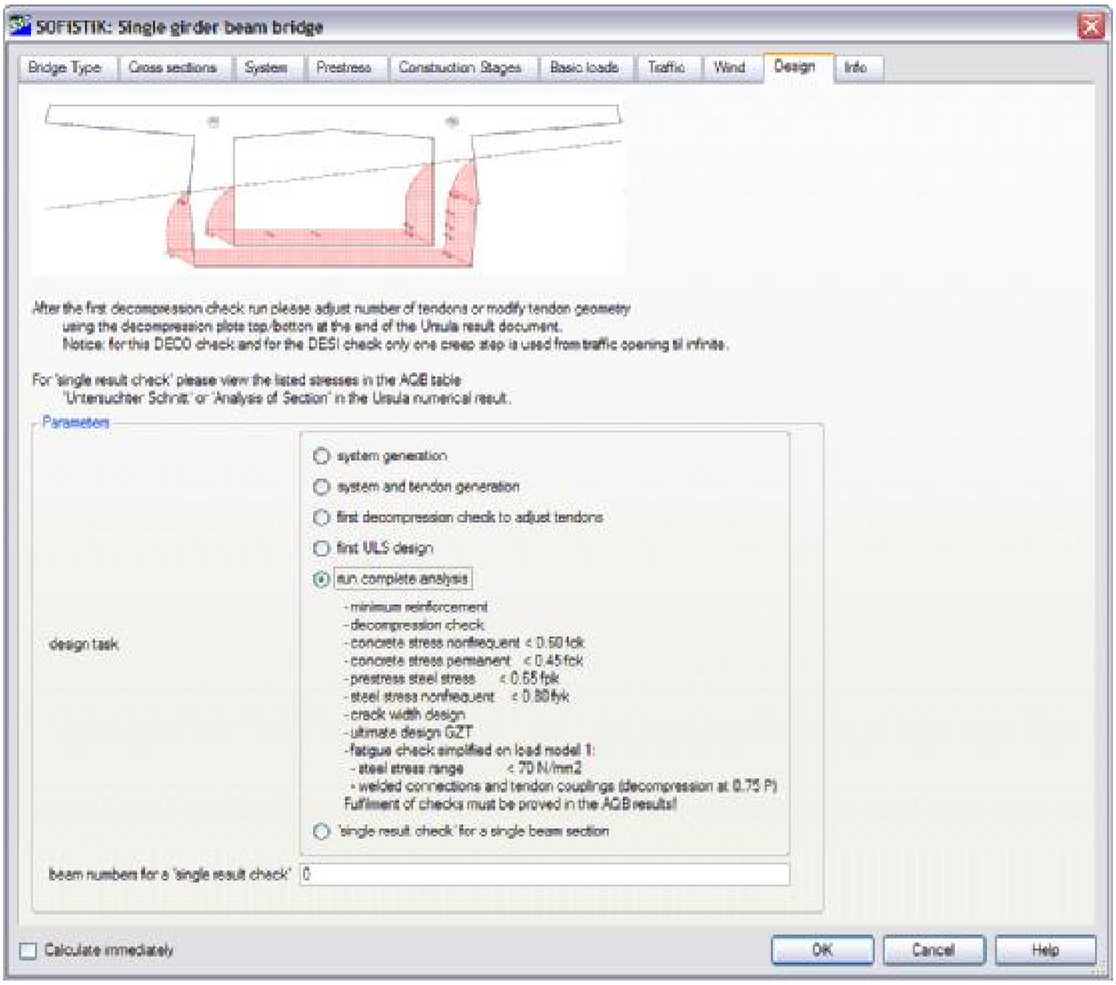
3.4.9 Design

After the input is complete, you can start with the design process. We recommend to work stepwise through the different design tasks. For the general information see chapter 2.2.9 .

After the concept design is finished you should have the values of all input variable to pass all design checks.

Then you can start the final design and run the complete analysis.

If you use the option “calculate immediately” the analysis starts automatically when you close the dialog with the OK button.

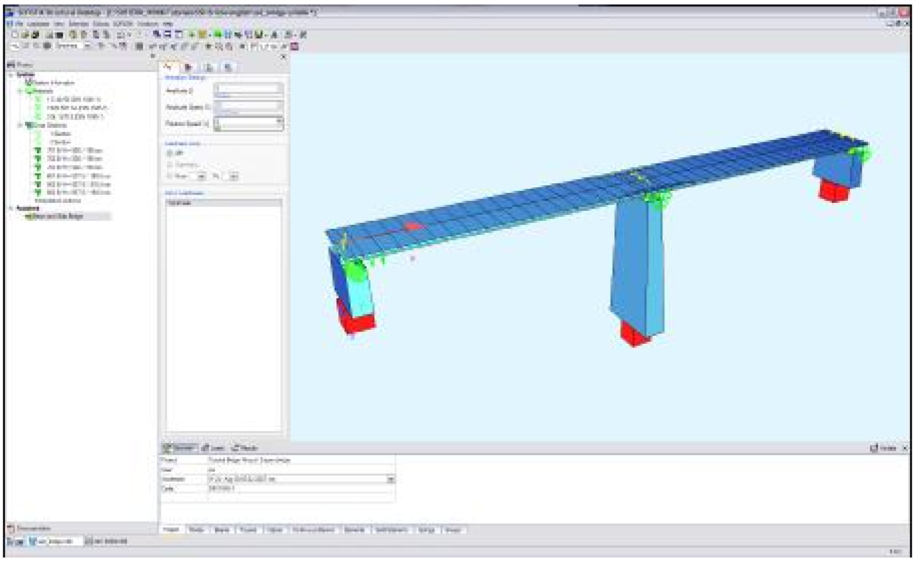


Picture 32: Input design

3.5 First Analysis and Optimisation of Tendons

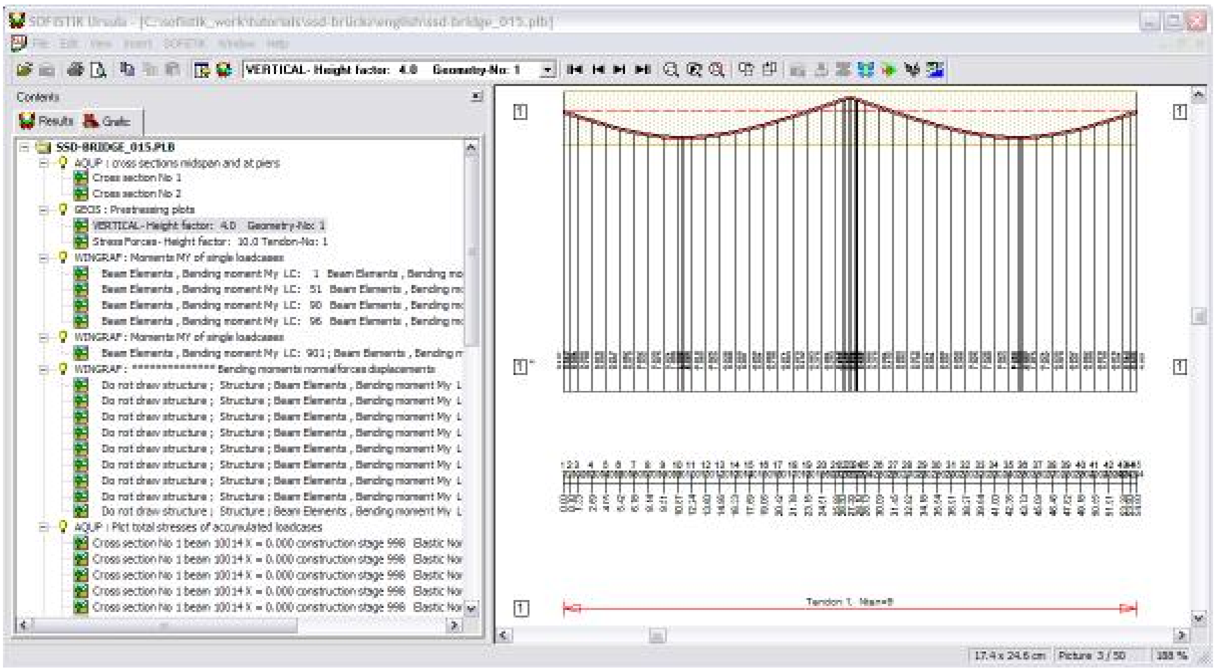
To start the calculation without the option “Calculate immediately” use the button out of the top menu inside the SSD window. In a first step we want to use the option

After the calculation is finished you will see your system on your SSD screen as shown blow.



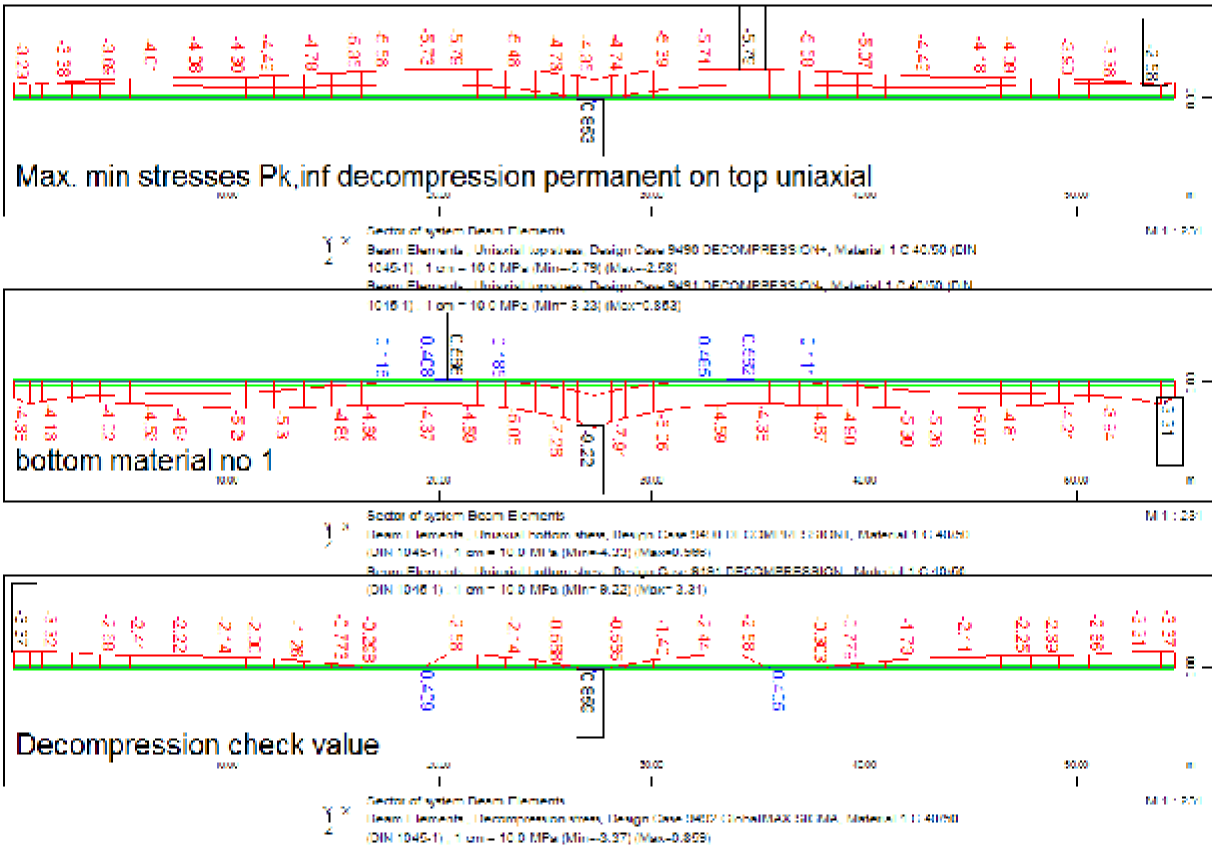
Picture 33: SSD main window ANIMATOR view of bridge

Please open the results using the URSULA button out of the top menu and select out of the GEOS block “VERTICAL – Height factor: 4.0 Geometry-No: 1”. You will see the plot of the tendon geometry.



Picture 34: URSULA view tendon geometry

If system an tendon geometry are correct you can go to the next step and start the calculation with the design task “first decompression check to adjust tendons”. Now the program creates all loads and starts a complex analysis including creep shrinkage and relaxation. Again please use the URSULA result browse to check the results. The decompression checks are at the end of the URSULA document.

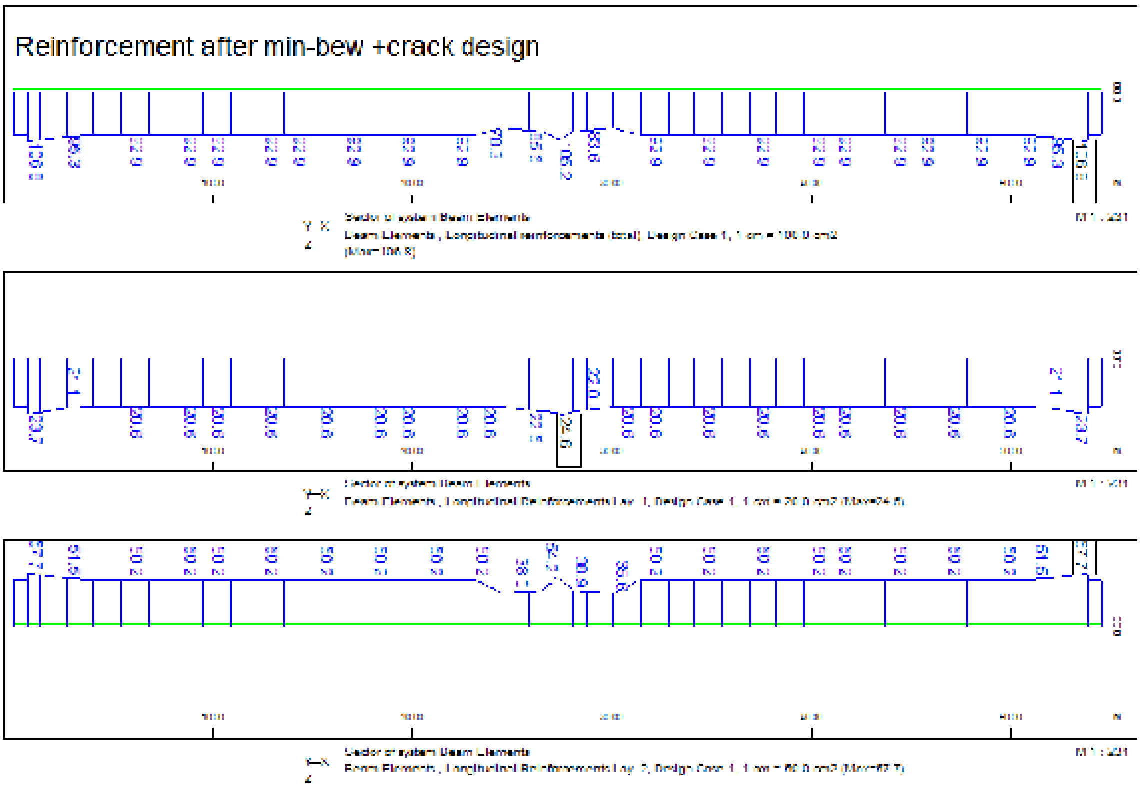


Picture 35: Result of decompression check

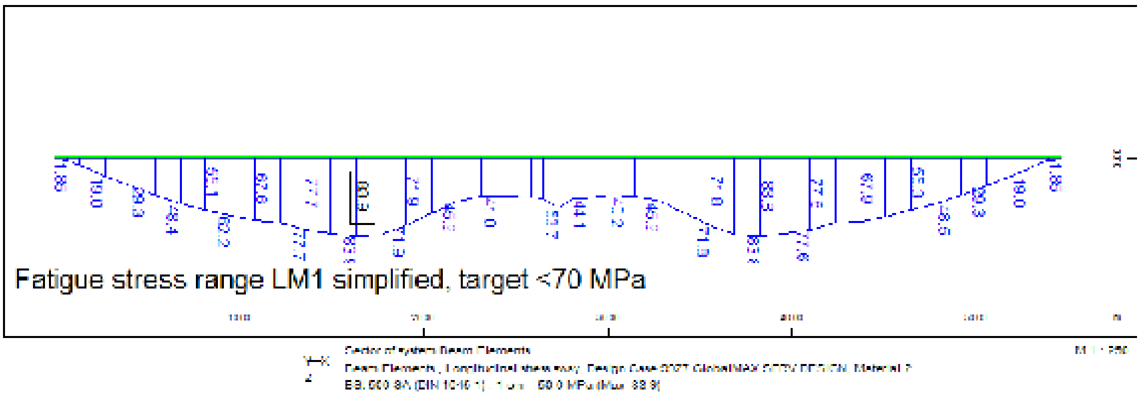
Using these result plots it and some repetitions of the analysis it is a comfortable way to optimise the tendons.

3.6 Complete Analysis and Design

In the end you start the design using the option “run complete analysis”. All necessary design checks according to DIN-FB 102 will be done and saved in an URSULA output file. Now check the complete analysis using all plots. Some plots will be shown below.



Picture 36: longitudinal reinforcement, total, layer 1 bottom, layer 2 top



Picture 37: range of steel stress due to fatigue check

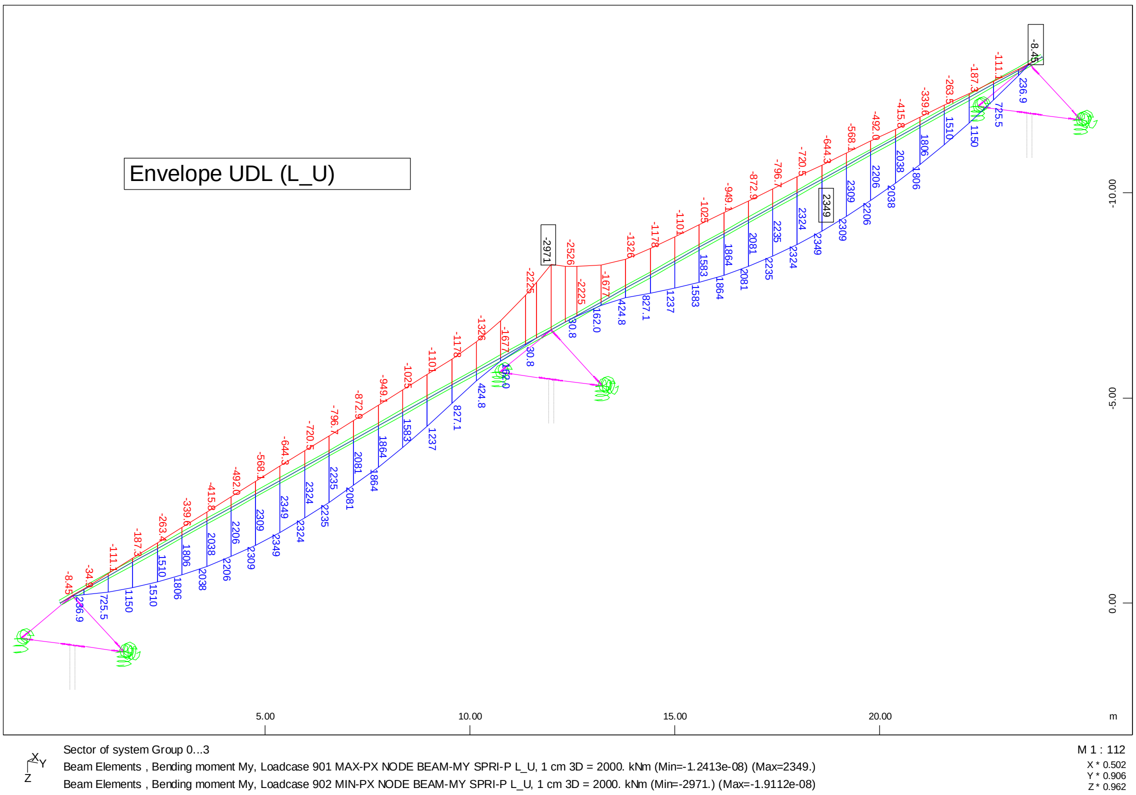
3.7 Validation of Results

A complete validation of all results according to the literature [1] will explode this tutorial. Therefore we focus on the envelopes of bending moment My according to the tandem system load (TS) and the uniform load (UDL).

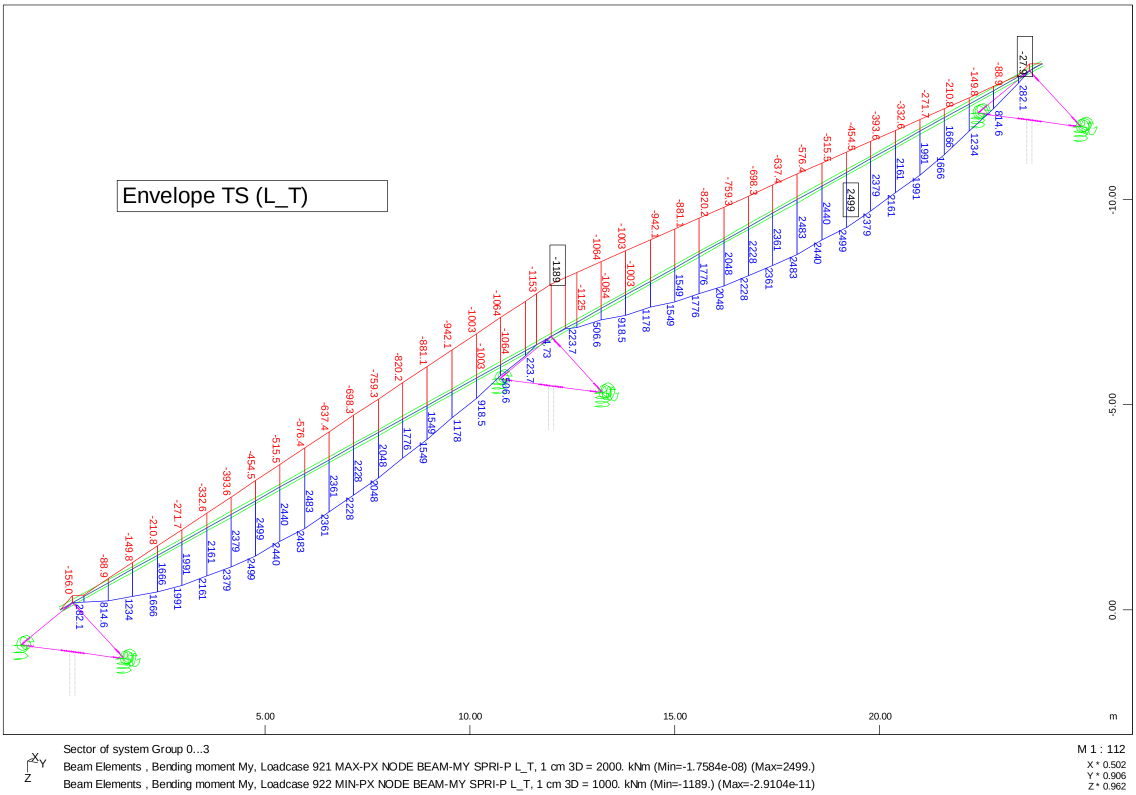
For that create a new task “Interactive Graphic” and select the relevant bending moments. The results are shown in the pictures below.

Picture 38: insert task "Interactive Graphic"

Results out of [1]	***TS***	***UDL***
Max. My midspan [kNm]	2505	2152
Min. My support [kNm]	-1226	-2830



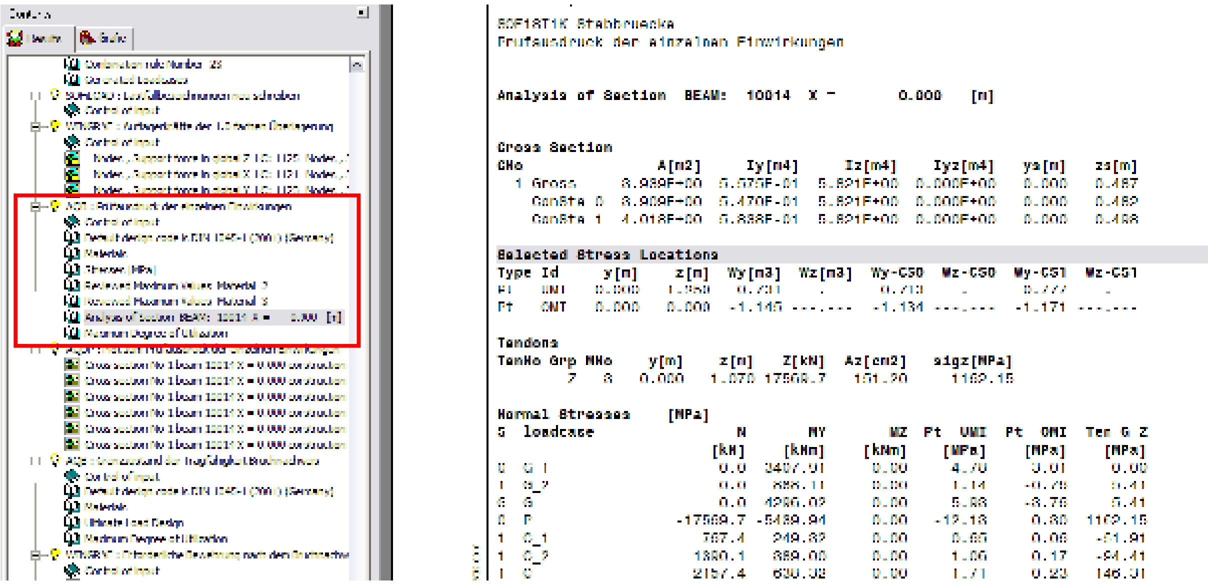
Picture 39: Results envelope My from UDL



Picture 40: Results envelops MY from tandem system

4 Additional Information

Very effective is to check the results out of the first AQB analysis „Prüfausdruck der einzelnen Einwirkungen“ = “Control plot single actions”. For that please open URSULA and navigate to the named node.



Picture 41: URSULA results

The table containing the results of all single actions is very important

Normal Stresses [MPa]

S	loadcase		N [kN]	MY [kNm]	MZ [kNm]	Pt UMI [MPa]	Pt OMI [MPa]	Ten G Z [MPa]
0	G_1		0.0	3407.91	0.00	4.78	3.01	0.00
1	G_2		0.0	858.11	0.00	1.14	-0.76	5.41
G	G		0.0	4266.02	0.00	5.93	-3.76	5.41
0	P		-17569.7	-5439.04	0.00	-12.13	0.30	1162.15
1	C_1		767.4	249.32	0.00	0.65	0.06	-51.91
1	C_2		1390.1	359.00	0.00	1.06	0.17	-94.41
1	C		2157.4	638.32	0.00	1.71	0.23	-146.31
0	****************		0.0	0.00	0.00	0.00	0.00	0.00
1	L_U	max-MY	0.0	2081.49	-0.00	2.60	-1.70	12.67
1	L_T	max-MY	0.0	2220.10	10.00	2.87	-1.90	13.56
1	T	max-MY	0.0	1617.38	0.00	2.06	-1.38	9.84
1	ZF	max-MY	0.0	422.25	0.00	0.54	0.36	2.57
1	SF	max-MY	0.0	506.70	0.00	0.65	0.43	3.08
1	ZL	max-MY	130.1	96.04	0.00	0.16	0.05	0.79
1	Y_1	max-MY	0.0	5702.26	1.20	7.34	4.87	34.71
1	Y_3	max-MY	0.0	3654.44	0.80	4.70	3.12	22.24
1	Y_4	max-MY	0.0	1284.18	0.24	1.65	1.10	7.82
1	Y_9	max-MY	130.1	6445.25	1.20	8.33	5.47	39.43
1	Y_D	max-MY	0.0	7844.48	1.80	10.10	-6.70	47.75
1	Y_G	max-MY	0.0	4309.58	1.20	5.55	-3.68	26.23
0	****************		0.0	0.00	0.00	0.00	0.00	0.00
1	L_U	min-MY	0.0	-872.80	8.90	-1.12	0.75	-5.31
1	L_T	min-MY	0.0	-690.32	10.19	-0.90	0.60	-4.25
1	T	min-MY	0.0	-995.05	0.00	-1.28	0.85	-6.06
1	ZF	min-MY	0.0	-422.25	0.00	-0.54	0.36	-2.57
1	SF	min-MY	0.0	-506.70	0.00	-0.65	0.43	-3.08
1	ZL	min-MY	-130.1	-96.04	0.00	-0.16	0.05	-0.79
1	Y_1	min-MY	0.0	-2596.01	-0.96	-3.34	2.22	-15.81
1	Y_3	min-MY	0.0	1600.66	14.31	2.06	1.37	9.74
1	Y_4	min-MY	0.0	736.49	3.82	0.95	0.63	4.48
1	Y_9	min-MY	130.1	3085.44	18.72	4.00	2.60	18.98
1	Y_D	min-MY	0.0	3401.33	1.92	4.38	2.90	20.70
1	Y_G	min-MY	0.0	1571.21	19.08	2.02	1.34	9.56
0	****************		0.0	0.00	0.00	0.00	0.00	0.00
0	G_1+P		17569.7	2032.03	0.00	7.35	2.70	1162.15
G	G+P		17569.7	1084.55	0.00	6.20	3.48	1167.56
G	G+P+C_1		-16802.3	-806.45	0.00	-5.55	-3.40	1115.65
G	G+P+C		-15412.3	-401.54	0.00	-4.49	-3.23	1021.25
G	DECOMPRESSION+		-13655.3	1418.68	0.24	-1.62	-4.36	912.85
G	DECOMPRESSION-		-13655.3	-601.97	3.02	-4.22	-2.63	900.55

Picture 42: Table of results for all actions

4.1 Open Input Data File

You can look at the automatic generated input data files using our text editor TEDDY. The relevant three files are:

- **project-name.dat**: main file for the analysis
- **project-name_000_csm.dat**: CSM file for creep, shrinkage and relaxation
- **project-name_000_desi.dat**: CSM file for the complete design process

Picture 43: Open text files

5 Literature

[1] *Bauer, T.; Müller, M; Blase, T.:* Straßenbrücken in Massivbauweise nach DIN-Fachbericht; 3. Auflage; Bauwerk Verlag, Berlin, 2005

[2] *Bellmann, J.:* SOFiSTiK und DIN-Fachberichte; latest version din_fb_sofistik.pdf for download on our SOFiSTiK Website, German language 'Referenzen/Beispiele' Example: DIN-FB 102: Berechnung einer Plattenbalkenbrücke .

[3] *Bergmeister, K.; Wörner, D. (Hrsg.):* Betonkalender 2004/I: Brücken Parkhäuser; Ernst&Sohn, Berlin, 2004

[4] *Rossner, W.; Graubner, C.-A.:* Spannbetonbauwerke Teil 3: Bemessungsbeispiele nach DIN 1045-1 und DIN-Fachbericht 102; Ernst&Sohn, Berlin, 2005

[5] *Hartmann, F.; Katz, C.:* Statik mit Finiten Elementen; Springer (VDI), 2002

[6] *Bellmann, J.; Maly, S.*: Precamber for Bridge Construction Stages, in Proceedings of the 6th Japanese-German Bridge Symposium, 2005