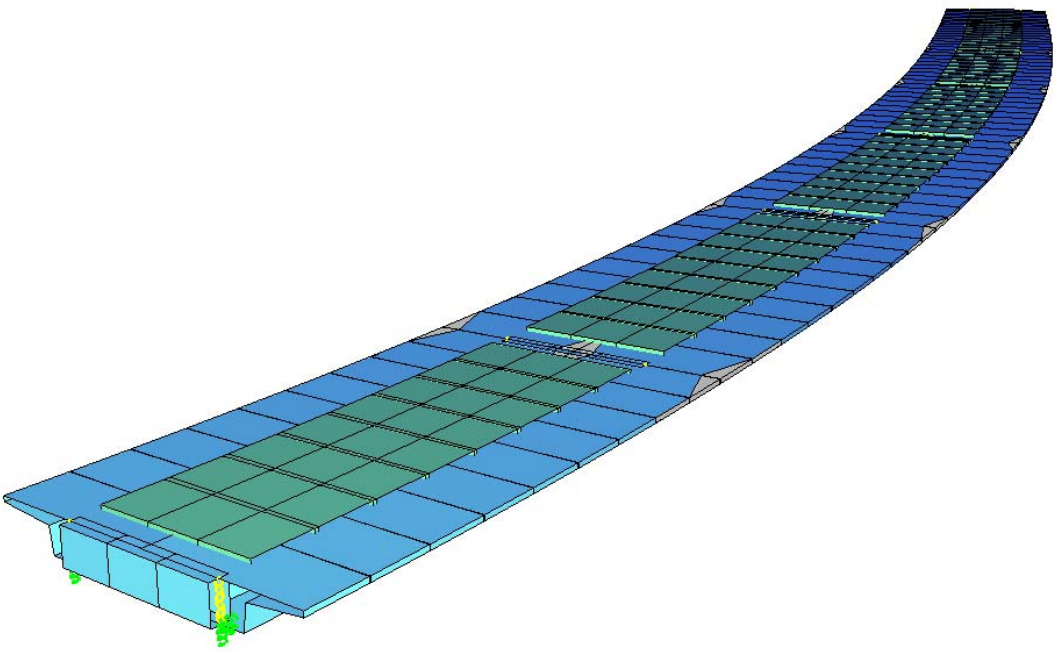
Bridge Design

CABD-Concept

SSD Version 2010-5



Contents

1 Basics

This tutorial shall explain the usage of the CABD (Computer Aided Bridge Design) concept which is available in the 2010 version of the SOFiSTiK FEA software.

All computations are controlled via the SOFiSTiK Structural Desktop SSD; the SSD can be started using the desktop icon depicted above. Alternatively you can start all programs directly from the installation directory: e.g. C:\Program Files\SOFiSTiK\2010\ANALYSIS_25

Prerequisites:

The CABD technology requires the CABD license; the influence line evaluation of traffic loads the ELLA license. Please contact your SOFiSTiK reseller or *info@sofistik.com* in case of vagueness.

Please refer to the basic SSD-SOFiPLUS (Version 2010) tutorial for general information on SSD and SOFiPLUS, the tutorial can be found in the Infoportal: http://www.sofistik.com/infoportal/

The software should be updated using SONAR before starting with this tutorial, the password for update has been send to all SOFiSTiK users having a valid maintenance contract. Please contact *support@sofistik.de* if you encounter any technical problems.

For installation and SONAR online update information please refer to the Administration manual: SSD Menu 'Help' -> 'SOFiSTiK Documentation…' 'administration_1.pdf'

1.1 Data structure of the SOFiSTiK software

- The control data of a computation is stored in a file called [project_name]**.sofistik**. Geometrical and structural information generated with the AutoCAD based preprocessor SOFiPLUS(-X) will be in a second file called [project_name]**.dwg**. These two files are **necessary** to reproduce the analysis and to generate the CDB.
- All data (elements, results etc.) is stored in the CDB = Central Data Base [project_name]**.cdb**
- The data written into the CDB can stem from several sources: SOFiPLUS, Revit Structure, ASCII-Input (CADINP).
- The ANIMATOR shows animated geometric and result information from the CDB.
- The SSD – SOFiSTiK Structural Desktop – controls all modules and allows running the calculation.

- Should data be prepared in ASCII-Input only, TEDDY is the alternative to the SSD. File name: [project_name]**.dat**.
- Further information can be found in the basic manual: SSD Menu 'Help' -> 'SOFiSTiK Documentation…' 'sofistik_1.pdf'

1.2 Important SOFiSTiK modules

Symbols resp. icons:

The symbol for TEDDY (Text Editor for data in ASCII format)

The ANIMATOR. File extension: *.cdb.

Graphic post-processing (WinGRAF). File extension: *.gra

Numeric post-.processing (DBview).

Reports – individual per module or complete for all modules. File extension: *.plb

Transforming data from the CDB (generated by graphic tools etc.) into ASCII.

Core module names and functionality (selection of the most important ones):

AQUA	Cross section definition (Cross section plots: AQUP)
SOFiMSHA, SOFiMSHC	Structural modelling
TENDON	Definition of Pre- an Post-tensioning
SOFiLOAD	Load and Action definitions
ASE	FE Analysis (Solver)
WING	Graphics
ELLA	Influence lines and envelopes
SOFiLOAD-V	Traffic load train library
MAXIMA	Combinations and envelopes
CSM (Construction Stage Manager)	Construction stage definitions
CSM- DESI	Design manager
AQB	Time dependent effects, Cross section checks for SLS and ULS for beam sections.
BEMESS	SLS and ULS design code check for shell elements.

1.3 Recommended loadcase numbering scheme

SOFiSTiK allows using any numbering scheme the user wants. However we recommend using the internal default set at least to ease the communication between customer and the support team. The internal default numbers will also avoid possible conflicts with double definitions and is considered by most of the SOFiSTiK users. The numbers used for the load combinations can be understood as “result cases”, not as typical loading cases.

The recommended (default) numbering scheme for loading cases is the following:

LCase 1 – 100:	free, usually used for selfweight/ deadload, pre-stressing, temperature, settlement, wind, braking...
LCase 101 – 999:	Used for the traffic loading
LCase 1001-1099:	Loadcases used for non-linear combinations
Result Case 1100-1999:	Envelopes of load combinations used for SLS checks (cf. MAXIMA manual)
Result Case 2100-2999:	Envelopes of load combinations used for ULS checks (cf. MAXIMA manual)
LCase 3000-3999:	Dynamics (mode-shapes etc.)
LCase 4001-4999:	Accumulated forces from the construction stage calculation e.g: LC 4001: situation after stage 1 (self weight) G LC 4005: situation after stage 5 (pre-stressing) P LC 4011: situation after stage 11 (creep&shrinkage) C_1 LC 4015: situation after stage 15 (self weight) G etc.
LCase 5001-5999:	individual result from the construction stage calculation. LC 5001: situation only for stage 1 LC 5005: situation only for stage 5 (i.e. LC 4005 minus 4001) LC 5011: situation only for stage 11 (i.e. LC 4011 minus 4005) LC 5015: situation only for stage 15 (i.e. LC 4015 minus 4011) etc.
LCase 6000 – 6999:	Creep and shrinkage loads and relaxation results for beams
LCase 7000 – 7999:	General stress results (generated optional by CSM)
LCase 8000 – 8999:	Reserved

9001 – 9999: Eigenmodes for response spectra analysis

1.4 Understanding SOFiSTiK

General workflow sequence in SOFiSTiK is as follows:

1. **Model creation and load definition:**
    a. Code
    b. Materials
    c. Cross Sections
    d. Geometry of the structure
    e. Tendon Layout, Prestressing
    f. Basic loads
    g. Moving Loads (traffic)
2. **Loadcase analysis (characteristic loads) and/or Influence line evaluation**
3. **Intermediate Superpositioning** (all variable actions/ loadcases) of inner forces related to the total cross section (final stage).
4. **Final-Superpositioning** (Dead load, superimposed dead load, prestress, creep&shrinkage&relaxation, envelopes of variable loads) of inner forces related to the partial cross sections.
5. **Design Code Checks**
    a. ULS Design for required reinforcement, bearing capacity calculation and other ultimate cases.
    b. SLS Design: Serviceability checks (fibre stress checks, crack width check, displacements of the structure, fatigue, dynamics etc.)

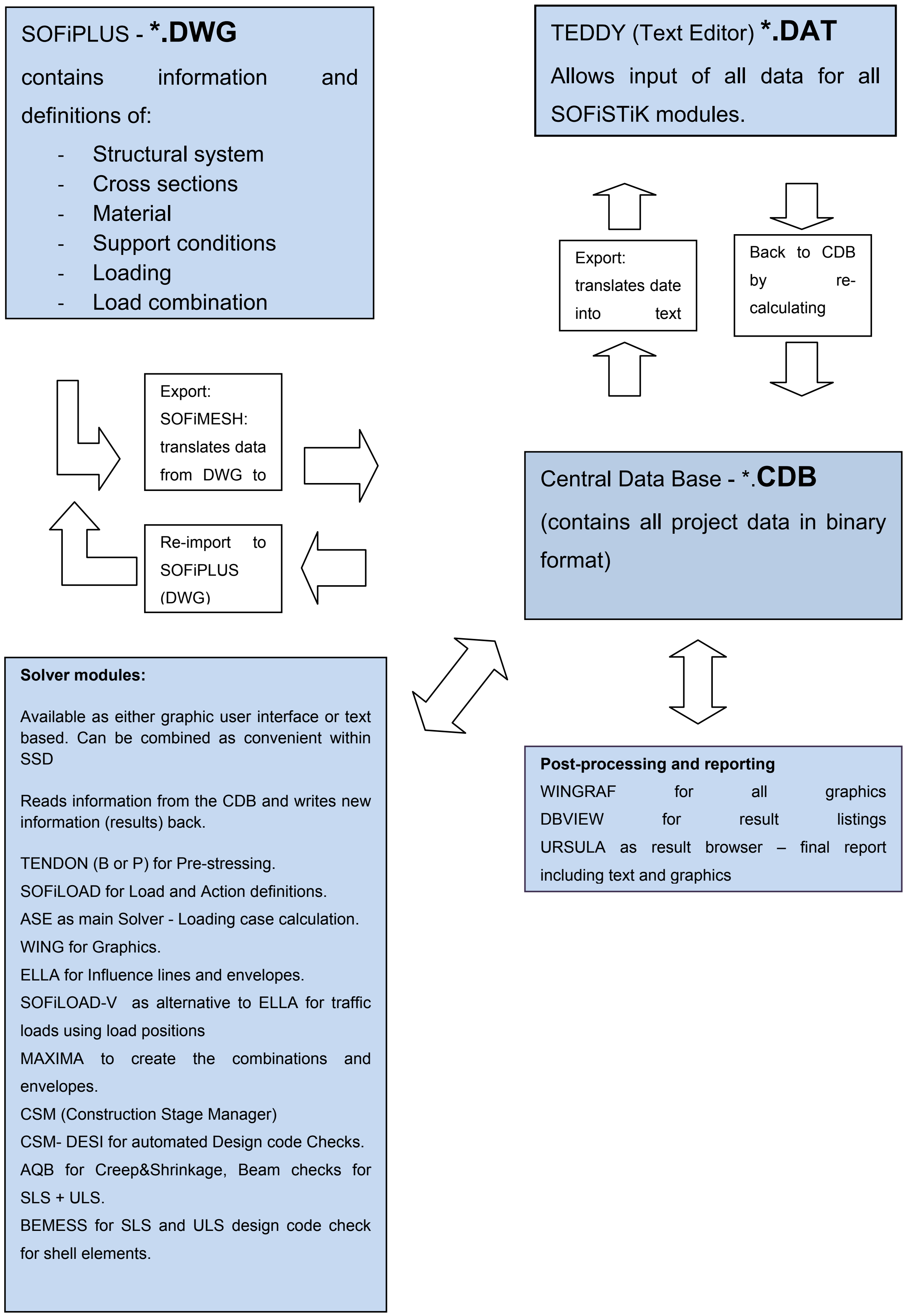


Figure 1: General workflow and data exchange

1.5 Components of the CABD technology

Start

System

Geometry / Definition of Axes
Primary Axis, Variables, Secondary Axis, Placements

Graphically: SSD Task ‚Definition of reference axis‘
Text Editor: SOFiMSHC; GAX-A/-H, GAXP, GAXV

.dwg
.dat

Cross Sections
Standard Sections, Template-Sections, Master-Sections

Graphically: Cross Section Editor in SOFiPLUS, SSD Task ‚Cross Sections‘, Template for cross sections (Cross-Temp by ABES)
Text Editor: AQUA.. TVAR, ‚=#Name‘

Top. Analy.

Substructure
Bearings, piers, abutments, foundations

Graphically: SSD Task ‚Supports‘
Text Editor: SOFiMSHC S-command (SPT,SLN,SAR), REF‘ :working in the infinite plane of any placement

.dwg

Substructure (gen.)
Arbitrary definition of Substructure (at the moment not related to axes)
Graphically: SOFiPLUS

Meshing

Tendon layout (opt.)
Prestressing system, Tendon layout, jacking sequence + stages.

Graphically: SSD Task ‚Tendons‘
Text Editor: TENDON

Analysis

Load definition and Analysis
Basic Loads, Traffic Loads

Graphically: SSD Tasks ‚Basic Loads‘ and ‚Traffic Loads‘
Text Editor: SOFiLOAD, ELLA, MAXIMA

Stage Analysis
Simulation of construction methods

Graphically: SSD Task ‚CSM‘
Text Editor: CSM, ASE

Design Code Checks
ULS/ SLS checks acc. Code,
Stresses checks, Stability checks,
Fatique, vibrations, earthquake

Design

Documentation/Output
Plots, Tables, indiv. Postprocessing (Excel etc.), Animation
Graphically: WinGRAF, GRAFiX
Text Editor: TEMPLATE (@KEY)

2 Cross Sections

There are three types of cross sections in SOFiSTiK, depending on the complexity of the design task we differ between:

- Standard Sections
- Thick walled sections (recommended for: polygonal cross sections e.g. R/C sections)
- Thin walled sections (recommended for: slender cross sections, welded sections, composite steel-concrete sections)

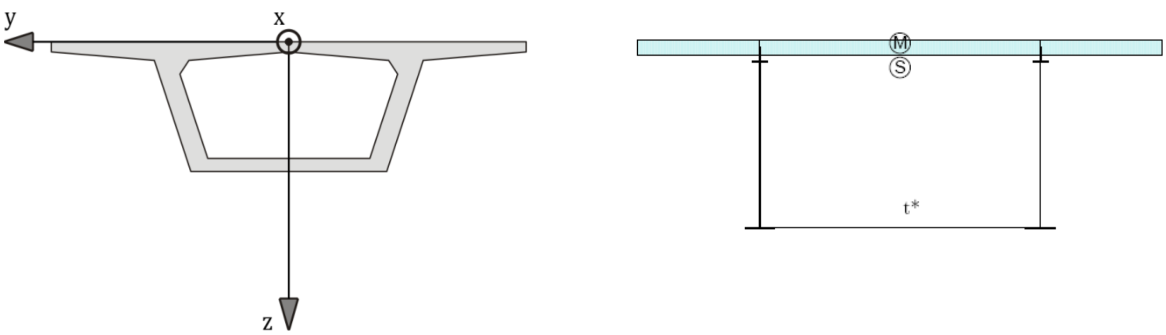


Figure 2: Thick walled and thin walled cross section – used coordinate system

2.1 General

Cross Sections can be defined within:

- TEDDY (Text Editor, Module AQUA) ***.DAT**
- CROSS SECTION EDITOR - SOFiPLUS - ***.DWG**

In SOFiSTiK cross sections are described in the local y−z coordinate system of the beam. The x−axis points in the longitudinal direction of the beam. The user is looking at the positive boundary of the section (from the end of the beam to the beginning) which means that he is looking backwards in the sense of the station value (chainage).
The coordinate system of the section is identical with the local beam coordinate−system, i.e. the local x−axis in longitudinal direction of the beam, the y- and the z-axis are right handed perpendicular to it. The z-axis defines the main bending direction and is in general oriented downwards in the gravity direction.

For the description of the forces, moments and the support conditions, three points along the beam have to be distinguished within a section:
• Beam axis (0,0)

This point may be given either by the center of gravity of the sections (centric beam) or it is defined by the origin of the sectional coordinate system (beam with a reference axis). Support conditions in the nodes thus are always specified relative to the beam axis position!

• Center of gravity (S)

This point is the reference for the normal force and the bending moments

• Shear center (M)

This point is the reference for the transverse shear force and the torsional moment. The section will rotate about that point in general.

2.2 Thick walled cross section

A freely defined solid cross section consists of any number of outer and inner perimeters using circular and polygonal shapes, as well as of reinforcement elements.

The general work sequence is as follows:

- Shape-description of the section: “Boundary”, “Opening”.
- Definition of appropriate design elements: “Stress point”, ”Point reinforcement”, “Line reinforcement”, “Shear cut”).
- For variable cross sections: Assignment of “Variables” and “references”: “Point to Point”, “Assign a Variable”, “Assign an Axis”, or “Point at line” are possible.
- For composite cross sections the definition of different parts for later activation in the stage definitions is necessary.
- Calculation and determination of the sectional properties.

A careful elaboration of the cross section within the Cross Section Editor or Teddy (AQUA) is enormously important for an accurate design as well as for a subsequent evaluation of construction stages.

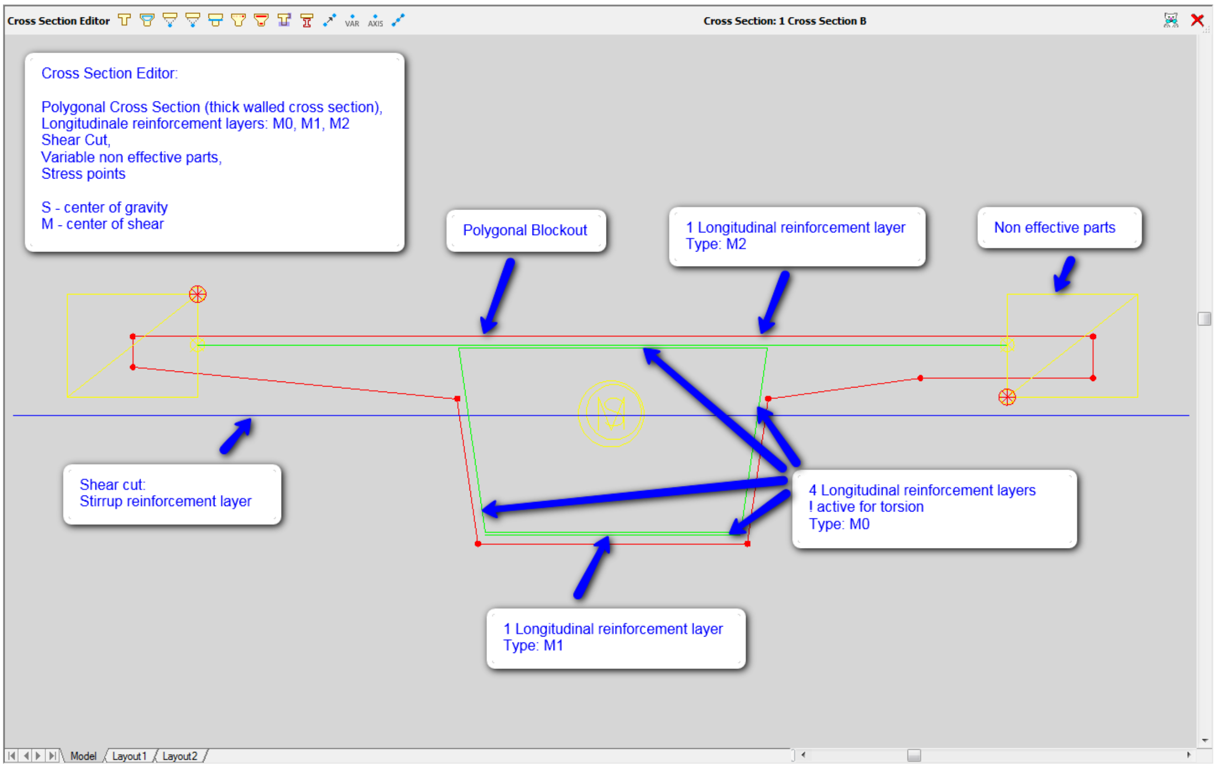


Figure 3: Cross Section Editor - Thick walled elements

2.3 Thin walled cross section

A freely defined thin−walled cross section may contain any number of thin walled elements, with a big ratio between element length and element thickness. A thin walled element assumes that the variation of the normal stress and most shear stresses over the thickness can be neglected. As a consequence the moment of inertia about the weak axis doesn't exist. Available elements are panels, standard steel shapes and welded joints, as well as reinforcement elements.

General work sequence is as follows:

- Shape-description of the section: "Thin walled element", "Welded Shear Connection".
- Definition of appropriate design elements: "Stress point", "Point reinforcement", "Line reinforcement".
- For variable cross sections: Assignment of "variables" and "references": "Point to Point", "Assign a Variable", "Assign an Axis", or "Point at line" are "possible"
- Calculation and determination of the properties

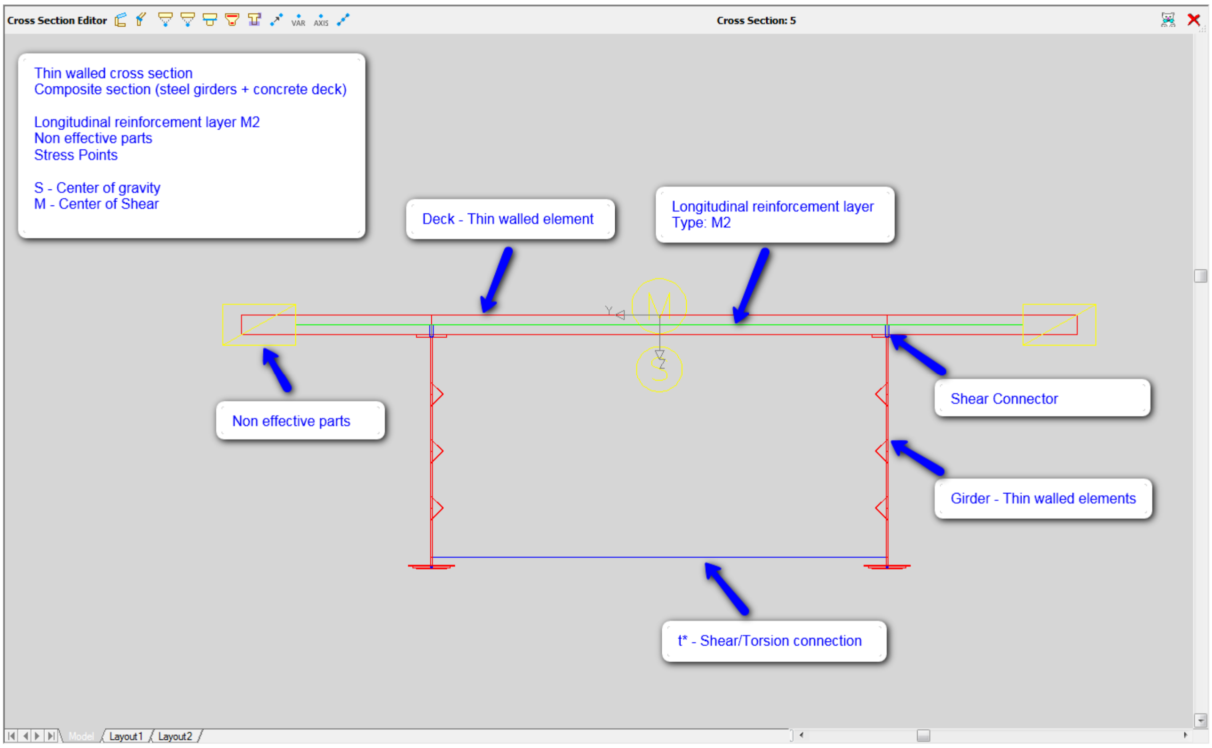


Figure 4: Cross Section Editor - Thin walled elements

The transmission of shear is only possible at interconnected elements. The program does not automatically recognise any intersection of thin− walled elements. In the determination of shear stresses, elements are automatically considered being connected with each other when their coordinate difference is less than 1 mm.

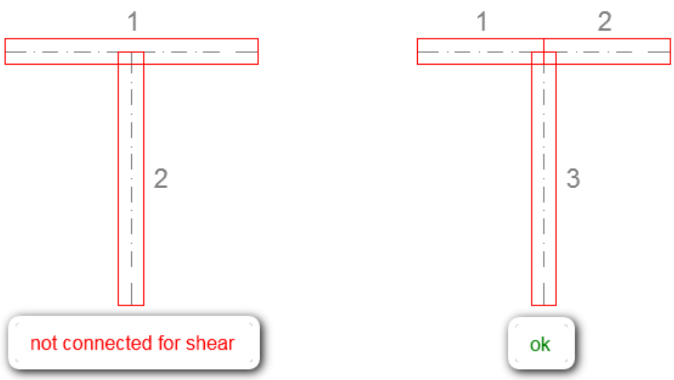


Figure 5: Connection of thin walled elements

2.4 Using the Cross Section Editor

The available commands in the Cross Section Editor and the command short descriptions are listed in the following table.

Icon	Sidebar	Short description
	Sidebar	
	New standard section	Creates a new standard cross section
	New thick walled section	Creates a new thick walled section
	New thin walled section	Creates a new thin walled section
	Delete all intpol. sections	Deletes all interpolated cross section in the database
	..	
	Toolbar/Sidebar	
	Boundary	Creates a new outer section boundary (=Polygonal Cross−Section Element / Block out).
	Opening	Creates a new inner section boundary (=Polygonal Cross−Section Element / Opening)
	Thin walled element	Creates thin walled elements (Uniform normal and shear stresses are generally assumed across the thickness of thin−walled elements)
	Welded Shear connection	Connects thin−walled section elements (Longitudinal seams of welded joints, Buckling fields of thin−walled sections, Shear bonds in composite sections, Trussed walls)
	Geometry point	Creates additional geometry points, MNO. 0
	Point of Stresses	Creates additional output points for normal and shear stresses or arbitrary reference points may be defined with
	Single Reinforcement	Defines an individual longitudinal reinforcement element (steel bar).
	Line Reinforcement	Defines a linearly distributed longitudinal reinforcement element (‘daubed’ steel bars).
	Shear Cut	Specifies a part of the section to be used for shear design and/or the minimum reinforcement or the crack width of a partial section
	Non effective parts	Defines non effective parts of the section for different types of forces or moments.

	rolled steel shapes	Insert rolled steel shapes, e.g. to define compound sections (thick or thinwalled)
	Point to point	Creates a constant reference between 2 vertexes (y-coordinate, z- coordinate, both the y- and z-coordinate).
	Assign variable	Assigns a predefined variable progress to vertices (y-coordinate, z- coordinate, both the y- and z-coordinate).
	Assign axis	Assigns a predefined axis to vertices (y- coordinate, z- coordinate, both the y- and z-coordinate).
	Point at line	Creates a reference point, following up to two other points within the cross section, this point may then a variable be assigned e.g.: variable height of inclined webs

2.5 Cross section properties and control options

Properties and control options of any cross section may be defined and modified in the general properties dialogue.

Properties: In the Tab ‚properties', number, name and material of the cross section may be specified.

The reference material number should, in general, be specified in this tab. The declaration of a material number with individual cross section elements is only appropriate for composite cross sections. In case of composite sections, ideal cross section values are calculated based on the material defined in the Tab properties:

$$A_i = \frac{(\sum A_i \cdot E_i)}{E_{ref}}$$

E_{ref}… Young-Modulus of the material specified in the Tab 'Properties'

E_i… Young-Modulus of the material used for the cross section elements

A_i… Area of the cross section element

Control values: Parameters for the analysis of the cross section might be specified in this tab. Depending on the type of cross section, different methods of cross section analysis will take place. Note: only for experienced users (for more information we refer to the manual of module AQUA).

Additional values: The tab might be used for input of additional cross section values, such as reduced torsion stiffness values, buckling strain curves, effective thickness, etc.

Layers: The user may specify up to ten longitudinal reinforcement layers with different types for each cross section. There are layers with the minimum reinforcement (M0 − M9) and extra layers (Z0−Z9).

- **M−Layers** have minimum reinforcement and, in the absence of any other instructions, they are laid by at least the specified AS values when doing a design.
- On the other hand, **Z−Layers** may be not activated at all. The layer number has no influence on the selection of a particular layer by the reinf. design module.

For ideal sectional values only the minimum values of the reinforcements will be used.
If, however, processing in the order of the layer numbers is desired, the layer numbers S0 − S9 should be used as a special case.

- **S−Layers** cannot be used in combination with M− or Z−layers. As an exception to this rule, a minimum reinforcement can be defined for the lowest layer by M0.

The ratios of the layers to each other are controlled by the layer type.

Variables: Variables assigned to vertexes and elements of the cross section are listed in this table.

Construction stages within the section: The user may specify up to 9 construction stages for each cross section. The construction stages are assigned to the individual elements of the section. The section is subdivided in as many parts as there are stages for composing the total section.
If a part of the section is active only temporarily, a CS “No.” for the “Expiry” of this part may be specified. This is then the last construction stage where this cross section part is active.

The individual parts and sectional values of the section with number “No.” may be addressed via a sub−number as:
Part No. is the complete section
Part No.1 is the first construction stage (even though if the first construction stage number isn’t 1)
Part No.2 is the second construction stage
Etc.

2.6 Parametric cross sections

It is very common, especially in the bridge design, that very similar sections are derived from a template (master cross section). The Cross Section Editor and module AQUA will therefore not only allow this parametric approach, but it will also store the parametric information of the cross section in the database CDB, in order to allow easy prototyping.

Primary solution for that task are formula expressions to be defined for any coordinate with up to 256 characters in the form of “=formula”. These formulas are stored together with the section and may be reevaluated for any section with different values along an axis or with explicit definitions locally.

The Cross Section Editor offers the following commands:

2.6.1 Point to point

The command creates a constant link between 2 coordinates (y- coordinate, z- coordinate, and both the y- and z-coordinate).

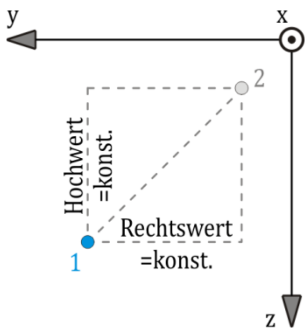


Figure 6: Point to point (Rechtswert/Hochwert = Right/Up-Value)

2.6.2 Assign variable

Assigns a formula expression to any coordinate (vertex or element: y- coordinate, z-coordinate, both the y- and z-coordinate).

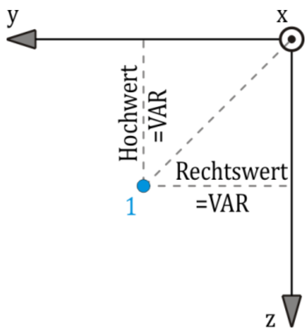


Figure 7: Assign variable (Rechtswert/Hochwert = Right/Up-Value)

2.6.3 Assign axis

Assigns the progress of a freely defined axis to any coordinate (vertex or element: y-coordinate, z- coordinate, both the y- and z-coordinate).

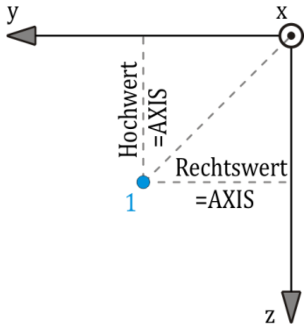


Figure 8: Assign axis (Rechtswert/Hochwert = Right/Up-Value)

2.6.4 Point at line

Creates a reference point (1) which follows up to two other control points ((2)-(3) = direction). A formula expression may then be assigned to this reference point. E.g.: variable height of inclined webs

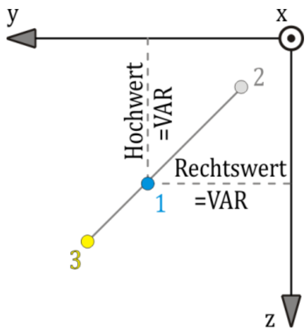


Figure 9: Point at line (Rechtswert/Hochwert = Right/Up-Value)

3 Geometry / Defintion of Axes

In SOFiMSHC (module for structural system definition), geometry axes represent general data structures which allow to define fully parametric input data sets. Structural elements are defined relative to these axes. If the geometry is changed, the structural system is updated automatically as references are managed automatically. In addition to its geometric shape, also variables and secondary axes are defined for every geometry axis, which can be used, for example, to describe varying sections, additional load lines or secondary axes.

An axis is basically independent from structural elements and is used for description:

- the geometry of superstructures,
- the geometry of road axes, when the axis of the superstructure doesn't coincide with the geometry of the road axis (the eccentricities will then be considered automatically)

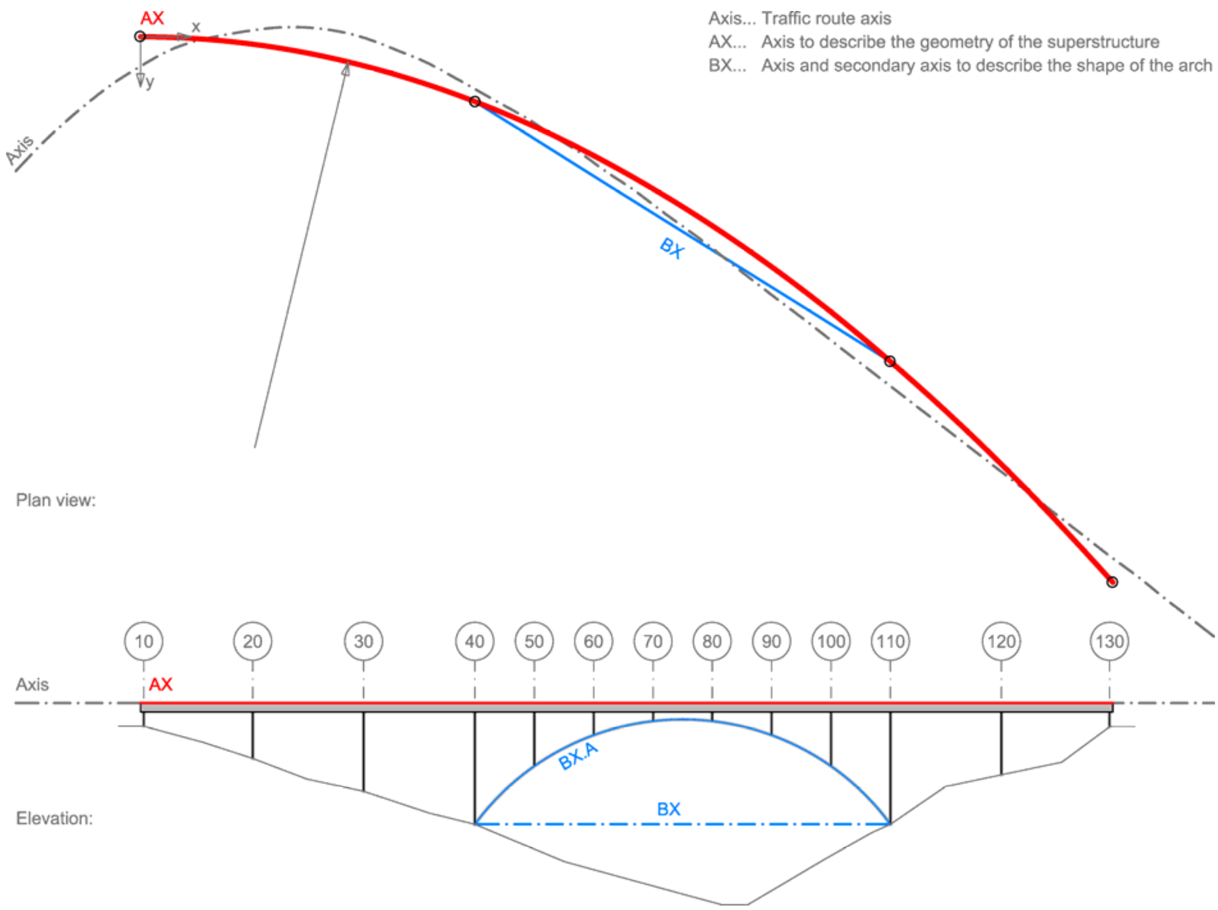


Figure 10: e.g.: Description of axes in plan view and elevation

Complicated road/rail layouts can be based on a reference axis, described in the plan view by straight, circular or spiral segments and in the elevation as straight or quadratic parabola elements.

The following commands are available in SOFiMSHC:

GAX	Definition of axes
*GAXA / GAXH	Alignment axes in plan view / and elevation
*GAXB	Straight lines and circular arcs in 3D
*GAXS	Secondary axes
*GAXP	Placements: special positions along an axis
*GAXV	Definition of variables along an axis

*command requires a valid CABD licence

An axis starts with a start point (station at start + coordinate) and a tangential direction, followed by any number of single alignment elements (straight, circular or spiral segments). The elevation (GAXH) of the alignment axis is defined at specific stations “S” along the axis with the corresponding height above datum value.

> Using the command GAXB straight or circular axes may be defined by coordinates (and not by starting coordinates and a tangential direction), so that the definition of vertical axes becomes possible as well.

Secondary axes (GAXS) are defined relative to a primary axis. The distance is defined either by a constant value in local y− and z−direction or by an axis variable specified with GAXV.

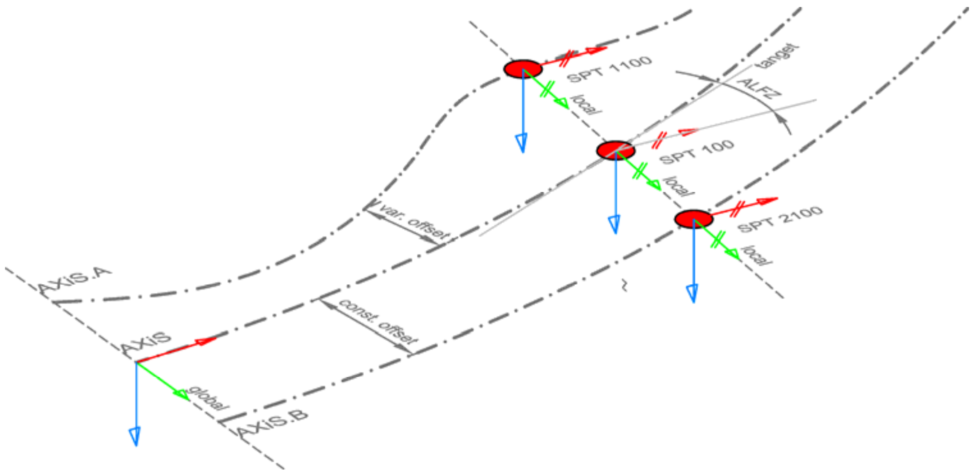


Figure 11: Definition of secondary axes

> The identifier of an axis may have up to 4 alpha-numeric characters. (e.g. A12D)
> The identifier of a secondary axis is one alphabetic character (e.g.: A12D.A)

So called placements (see table below) may be defined along a geometry axis GAX at a specific station “S”. Placements describe important locations and points along a geometry axis. Wherever structural changes and dependencies will take place a placement has to be introduced, by means of defining their geometric location and structural boundary conditions along the axis.

	Geometric information	Structural information
Placement	- Station value along the axis - Rotation about the global Z direction - Skew about the local x−axis - Skew about transverse y−axis - Skew about vertical z−axis - Cross fall to the right (+y) - Cross fall to the left (−y) see sign convention in 2.1	- Number of structural point to be created at the placement. - Group number of the following elements. - Cross−section number of the subsequent elements or number before and after the placement given as Literal ’ncs1:ncs2’.

For each placement a structural point is created on the axis. The node number of this structural point is set at SPT. Structural lines from point to point are created, which get the number of their preceding point assigned. The user may specify a group or section number for a specific placement on the axis which is then used for all subsequent beam elements created unless one of the numbers is changed. The generation of the structural elements basically starts at the first placement which has a cross−section number assigned and ends at the end of the axis or at a placement with type ’E = endface’.

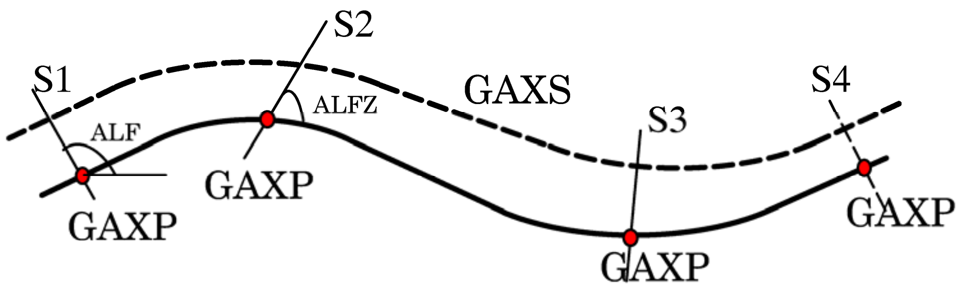
A placement defines an infinite plane at a given station S perpendicular to the axis tangent. Structural points and other placement properties on secondary axes are created at the intersection of this plane with the axis.

This infinite plane can be further rotated about the three local axis coordinates by setting ALFZ, ALFY and ALFX. The infinite plan can also be aligned within the global X−Y coordinate plane by setting an angle at ALF = {1−360 deg}.

The local coordinate system of the structural point created at the placement is orientated as default as follows:

local y: perpendicular to the axis at the given station

local x: tangential to the axis at the given station



We recommend to define an array for significant station's along the axis (e.g.: Support/Bearing lines) so that other placements before or after can be specified by an addition or a subtraction. Example for numeric input:

```
STO#SA 8, 0.00, 25.00, 57.00, 89.00, 121.00, 153.00, 185.00, 210.00

GAX 'AXIS' Titl 'Primary Axis'

GAXA  S 0 x 0 y 0 z 0 sx 1 sy 0      $ Start: Stat, coord., dir.
      $ L 210                        $ Straight element  L=210m
      $ L 210  R   300               $ circular element  L=210, R=300
        L 210  RA -300.000 RE 0.000  $ clothoide: L,Ra,Re -> A1,A2

$ Secondary Axis A with constant offset Y=-3.475
GAXS Axis A   Y -3.475   Titl 'Secondary Axis A'

$ Secondary Axis B with constant offset Y= 3.475
GAXS Axis B   Y  3.475   Titl 'Secondary Axis B'

$ Placements, using the array #SA

GAXP Axis IDS A S  #SA(1)         'S' SPT 110  NCS 2  GRP 101
GAXP Axis IDS A S  #SA(2)         'S' SPT 120
GAXP Axis IDS A S  #SA(2)+6.40    'J' SPT 122         GRP 102
GAXP Axis IDS A S  #SA(3)         'S' SPT 130
GAXP Axis IDS A S  #SA(3)+6.40    'J' SPT 132         GRP 103
GAXP Axis IDS A S  #SA(4)         'S' SPT 140
GAXP Axis IDS A S  #SA(4)+6.40    'J' SPT 142         GRP 104
GAXP Axis IDS A S  #SA(5)         'S' SPT 150
GAXP Axis IDS A S  #SA(5)+6.40    'J' SPT 152         GRP 105
GAXP Axis IDS A S  #SA(6)         'S' SPT 160
GAXP Axis IDS A S  #SA(6)+6.40    'J' SPT 162         GRP 106
GAXP Axis IDS A S  #SA(7)         'S' SPT 170
GAXP Axis IDS A S  #SA(7)+5.00    'J' SPT 172         GRP 107
GAXP Axis IDS A S  #SA(8)         'S' SPT 180
```

4 Structure and Substructure

As already mentioned, we differentiate between “geometry” and “structure”, although each placement may already contain structural information.

Based on the placement definition, all information is finally converted to structural elements (structural points, structural lines) when meshing. The meshing creates the final finite elements. For meshing we can use:

CTRL MESH	to define the bit pattern for mesh generation (beam or shell systems).
CTRL TOPO	to enforce a topological analysis.
CTRL HMIN	to specify the maximum element size.
CTRL NODE	to set up the automatic node numbering.

Module SOFiMSHC allows working in the infinite plane of any placement defined on the geometry axis. Coordinates of a structural point can be defined relative to a placement. The coordinates can be given in Euclidian coordinates x, y, z relative to the reference point and its direction.

This becomes important when parts of the substructure should be defined relative to the placement or oriented towards to it. In this way these structural elements are updated automatically if the geometry of the axis is changed.

We recommend making use of the input facilities in relative coordinates, related to a placement located on an axis.

The following figures should describe the proceeding:

Placement (as per default: y perpendicular to the axis at station 74.00m):

```
GAXP AXiS S 74.000 TYPE 'S' SPT 100
```

Structural Point (SPT):

```
SPT 1100 REF PT 100 X 0.00 y 3.00 z 4.00
```

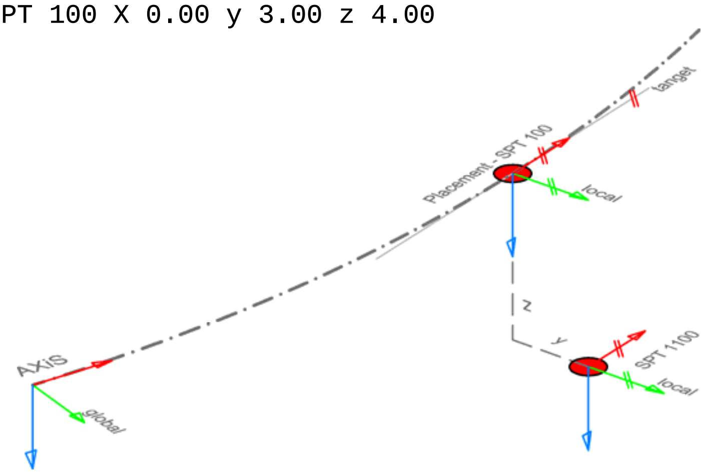


Figure 12: Structural point defined relative to a placement

Placement (skewed about local z / ALFZ=+25deg):

```
GAXP AXiS S 74.000 TYPE 'S' SPT 100 ALFZ 25
```

Structural Point (SPT):

```
SPT 1100 REF PT 100 X 0.00 y 3.00 z 4.00
```

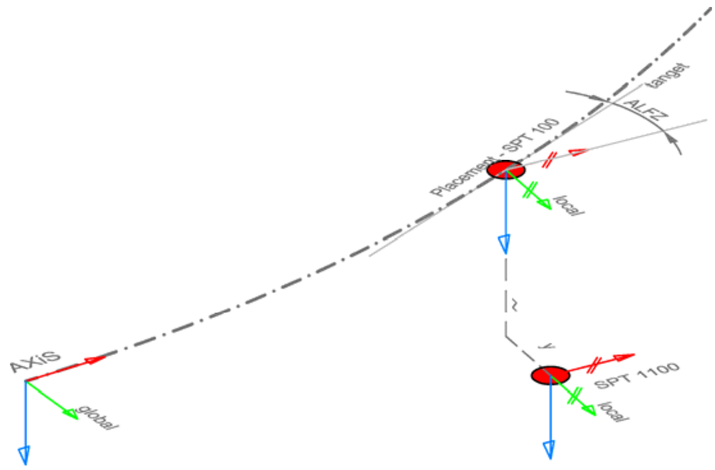


Figure 13: Structural point defined relative to a skewed placement (ALFZ)

Placement (skewed about global z / ALF=+5deg):

```
GAXP AXiS S 74.000 TYPE 'S' SPT 100 ALF 5
```

Structural Point (SPT):

```
SPT 1100 REF PT 100 X 0.00 y 3.00 z 4.00
```

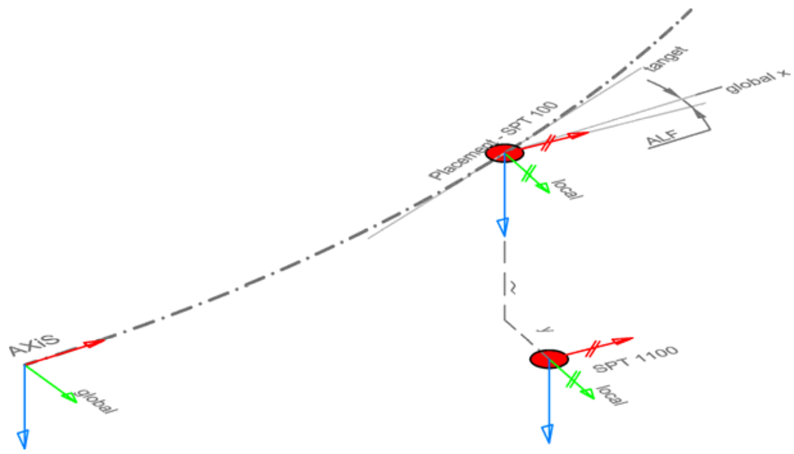


Figure 14: Structural point defined relative to a skewed placement (ALF)

Having defined the geometric location of a structural point, constraints and links to other structural points are defined within the command SPTP (Structural PoinT Properties).

Exemplary input (Structural point, Rigid link, Elastic link, Structural line):

```
SPT 1101 REF PT 100 X 0  Y 2.75  Z 3.5    ; SPTP KF ref 100  grp 10
  SPTP CX  ref 1103 val 1.0   grp 10
  SPTP CY  ref 1103 val 1.0e7 grp 10
  SPTP CZ  ref 1103 val 1.0e7 grp 10

SPT 1102 REF PT 100 X 0  Y 2.75  Z 3.5    ; SPTP KF ref 100  grp 10
  SPTP CX  ref 1104 val 1.0   grp 10
  SPTP NCY ref 1104 val 1.0   grp 10
  SPTP CZ  ref 1104 val 1.0e7 grp 10

SPT 1103 REF PT 100 X 0  Y 2.75  Z 3.5    ; SPTP KF ref 1105 grp 10
SPT 1104 REF PT 100 X 0  Y 2.75  Z 3.5    ; SPTP KF ref 1105 grp 10

SPT 1105 REF PT 100 X 0  Y 0.00  Z 3.5   fix F


sLN 1 NPA 1 2 SNO 1 KR DXLN 'Axis' 100.0
```

The structural point no. 100 is a placement defined on the axis. All structural points (1101, 1102, 1103, 1104) are then defined relatively to it. Support conditions, rigid links or elastic links may be described as property of a structural point (SPTP) respectively two SPT's.

Geometric axes may also be used as reference for inner and/or outer boundaries of structural areas; so that the shape of shell elements can be described with geometric axes as well.

Exemplary input (Structural area):

```
sln  1  710 610
sln  2  610 620     ref 'Axis.A' $Boundary secondary axis Axis.A
sln  3  620 720
sln  4  720 710     ref 'Axis.B' $Boundary secondary axis Axis.B

$ Structural area (SAR) with boundary sarb nl 1,2,3,4 (=Structural lines)
sar 1 mno 22 mrf 11 nra 7 t .32 Qref belo grp 51 h1 1.00 titl ‘slab’
sarb nl 1,2,3,4
```

The structural points (610,710,620,720) used for the defintion of structural lines (1,2,3,4) are placements on the secondary axes (Axis.A and Axis.B).

5 Tendon Layout

In SOFiSTiK the prestressing is defined only if a valid structural system is stored in the database.

The general work sequence for defining prestressung in Module “Tendon” is as follows:

- Prestressing system with material definitions in addition to the already existing prestressing material.
- Reference Axis and automatic search of a continuous beam series (no gaps)
- Duct and tendon geometry
- Tendon assignment to (construction stages, jacking sequence, friction loss, etc.)
- Calculation of losses (friction and wobble).
- Storage of prestressing as loading case (Primary and secondary effects).

Prestressing system:

First of all an appropriate prestressing system has to be defined (SSD-Task: Prestressing System). The prestressing system will be stored with a number (keyword NOPS) in the database and is selected within TENDON.

Reference Axis: a so called reference axis is the base of any tendon definition.

We recommend to either refer directly to a geometry axis (defined within SOFiMSHC),

```
AXES NOH 1 TYPE REFB Axis
```

... or to make use of the automatic beam search between start and end nodes.

```
AXES NOH 1 TYPE REFB Auto 10 210
```

In both cases TENDON will search for continuous beam series along this reference axis (AXES), to which the tendons finally belong. Top positions (keyword TOPP) may be defined along the reference axis, either by nodes or stations. The complete axis will then be divided into several ‘spans’ so that a definition via span-coordinates (Xi) becomes possible as well.

```
TOPP SP         S  NOH=1 KIND=REFB $ with S = station on axis in m
      0         0                  $ start face bridge deck
      1         1                  $ bearing line
      2      38.5                  $ bearing line
```

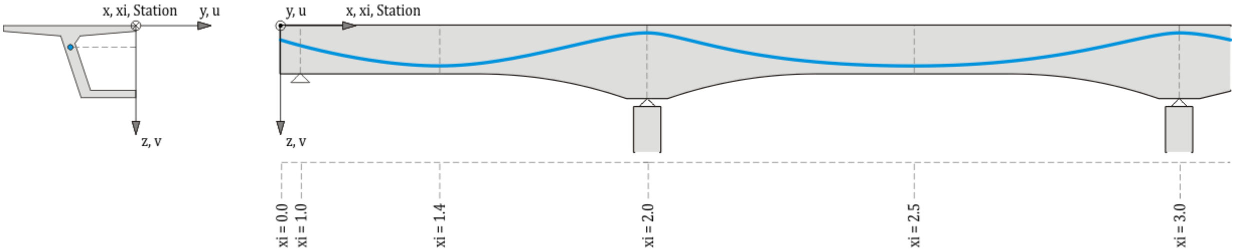


Figure 15: Top positions, span coordinates, tendon position (Xi/S, u, v)

Tendon geometry: When defining the layout of a tendon (keywords TGEO, PTUV) we refer to a predefined reference axis (AXES). Along this axis, at specific stations (Xi, S) the horizontal and vertical distance of the tendon (u, v) is defined. A minimum of three stations on the reference axis are mandatory for correct input.

```
TGEO NOG 1 NOH 1 NOPS 1 TITL 'Tendon geo 1'
PTUV TYPE     S     U      V   DUS   DVS    DL    DR    RH    RV    RL  KIND
     SPAN     0     -    0.4     -     -     -     -     -     -     -    -
     SPAN   1.4     -      3     -     0     -     -     -     -     -    -
     SPAN     2     -   0.15     -     0     -     -     -     8     1    -
     …
```

Two tendon geometries are available:

– Parabolic Prestress, Straight Prestress (keyword PTUV)
– External Prestress (keyword PTSV)

Tendon: The tendon has to be defined with its prestressing stages and jacking sequences. The following prestressing stages may be used:

ICS1= CS no. Tendon installed, stressed for the first time.

ICS2= CS no. Grouting of tendon (set ICS2 to 999 for unbounded tendon).

ICS3= CS no. Removal: last active number in construction sequence.

It is up to the user whether the input of the actual tendon stressing is carried out with stresses, factors or forces. (keyword PSIG, PDEF). We recommend carrying out the prestressing using stresses (PSIG).

Where Po,max is the max. jacking force and Pmo the remaining value of the force in the tendon immediately after removing the jack:

K1, K2	Po,max	= min(k1*fpk*..., k2*fp0,1k*...)
K3, K4	Pmo	= min(k3*fpk*..., k3*fp0,1k*...)

All default values are predefined acc. the chosen design code, e.g.:

Kapa	K1	K2	K3	K4	Code
0.00	0.80	0.90	0.75	0.85	EC−2−2004

The jacking sequence is set up within the following literals:

Type of the prestressing:		Prestressing procedure:
RI	prestressing from right	1 or T tensioning
LE	prestressing from left	2 or R release
RILE	first from right, then from left	4 or W restressing
LERI	first from left, then from right	8 or S slip

Finally the record TEND defines the tendon and starts the calculation of the prestressing forces considering all losses.

The geometry of the tendon is activated for the range “FROM, TO station”. With a given LC−number the load due to prestressing is calculated and stored under this load case number.

```
CS ICS1 11 ICS2 12 ICS3 0
PSIG KIND ri ANWS 'TS'
TEND NOT 1 NOG 1 NTEN 3 LC 11 TYPE REFB FROM 0.0 TO 48.5  TITL 'Tendon 1'
```

6 Classification of actions

The relevant type of load actions on bridges is classified in SOFiSTiK using module SOFiLOAD. It is possible to define all actions individually (user defined) or to make use of predefined actions of SOFiSTiK respectively the design code (for various design codes a so called initialisation file (.ini) exists, setting up all the safety- and combination factors for each action type in accordance with the chosen design code which is project).

A load case is always assigned to an action; the load case inherits all factors and combination rules from the action. In principal every action can be subdivided into sub-categories (appended with an underscore to the name of the action i.e. G_1 being a sub-category to action G). Each category has its own combination values and its own load cases and a default superposition rule within the action category. The hierarchy of actions types, categories and load cases is shown in the following table:

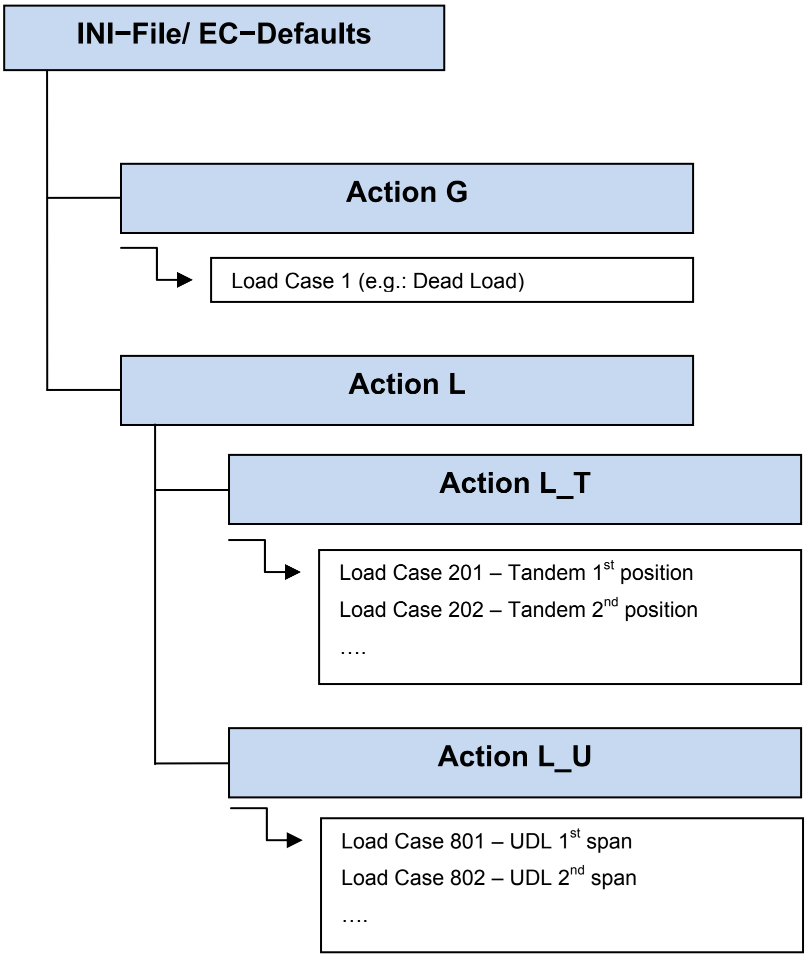


Figure 16: Structure of the actions, action categories and load cases

Exemplary input:

```
+prog sofiload
Head 'Classification of actions'

$ Traffic Load Actions
$ psi-values:   acc. to EN 1990/A1 Table A.2.1
$ gamma-values: acc. to EN 1990/A1 Table A.2.4(B)

ACT 'L_T' GAMU 1.35 0.00 0.75 0.75 0.00 PART Q EXCL TITL 'gr1a LM1 TS'
ACT 'L_U' GAMU 1.35 0.00 0.40 0.40 0.00 PART Q EXCL TITL 'gr1a LM1 UDL'
ACT 'L_F' GAMU 1.35 0.00 0.40 0.40 0.00 PART Q COND TITL 'gr1aLM1 Foot'
end
```

Exemplary roundup of defined actions in the output (module URSULA):

Actions

type	T	sup	Title	γ-u	γ-f	γ-a	ψ-0	ψ-1	ψ-2	ψ-1'
B	G	perm	erection load	1.35	1.00	1.00	1.00	1.00	1.00	1.00
C	G	perm	creep and shrinkage	1.00	1.00	1.00	1.00	1.00	1.00	1.00
C_1	G	perm	C+S till traffic opening	1.00	1.00	1.00	1.00	1.00	1.00	1.00
C_2	G	perm	C+S after traffic opening	1.00	1.00	1.00	1.00	1.00	1.00	1.00
G	G	perm	total dead load	1.35	1.00	1.00	0.85	1.00	1.00	1.00
G_1	G	perm	dead loads g_1	1.35	1.00	1.00	0.85	1.00	1.00	1.00
G_2	G	perm	other dead loads g_2	1.35	1.00	1.00	0.85	1.00	1.00	1.00
G_3	G	perm	deck surfacing g_3	1.35	1.00	1.00	0.85	1.00	1.00	1.00
G_9	G	perm	permanent earthpressure g_9	1.35	1.00	1.00	0.85	1.00	1.00	1.00
R	G	cond	earth pressure traffic	1.50	0.00	1.00	1.00	1.00	1.00	1.00
X	G	perm	single loadcase	1.00	1.00	1.00	1.00	1.00	1.00	1.00
P	P	perm	prestress	1.00	1.00	1.00	1.00	1.00	1.00	1.00
GR	Q	exex	groups without gr1a	1.35	0.00	1.00	0.00	0.00	0.00	0.00
GR_1	Q	exex	gr1a LM1	1.35	0.00	1.00	0.75	0.75	0.00	0.75
GR_2	Q	exex	gr2 Horizontal Forces	1.35	0.00	1.00	0.00	0.00	0.00	0.00
GR_3	Q	exex	gr3 Footways	1.35	0.00	1.00	0.00	0.00	0.00	0.00
GR_4	Q	exex	gr4 crowd load	1.35	0.00	1.00	0.00	0.75	0.00	0.75
GR_5	Q	exex	gr5 LM3 freq LM1	1.35	0.00	1.00	0.00	0.00	0.00	0.00
GR_B	Q	exex	gr1b LM2	1.35	0.00	1.00	0.00	0.75	0.00	0.75
L	Q	excl	gr1a LM1	1.35	0.00	1.00	0.75	0.75	0.00	0.80
L_G	Q	cond	gr1a LM1 Footway (comb-value)	1.35	0.00	1.00	0.40	0.40	0.00	0.70
L_T	Q	excl	gr1a LM1 TS	1.35	0.00	1.00	0.75	0.75	0.00	0.80
L_U	Q	excl	gr1a LM1 UDL	1.35	0.00	1.00	0.40	0.40	0.00	0.80
SF	Q	cond	possible settlement	1.00	0.00	1.00	1.00	1.00	1.00	1.00
SL	Q	excl	LM3 fatigue	1.50	0.00	1.00	0.00	0.00	0.00	0.00
T	Q	excl	temperature	0.81	0.00	1.00	0.60	0.50	0.00	0.60
W	Q	excl	wind loading	1.50	0.00	1.00	0.60	0.50	0.00	0.60
ZF	Q	cond	probable settlement	1.00	0.00	1.00	1.00	1.00	1.00	1.00
ZL	Q	excl	brake+zentrifugal	1.50	0.00	1.00	0.00	0.00	0.00	0.00

When working according to EN 1991 -2 we recommend to define a separate action type (e.g. GR_1, GR_2, …) for each group of loads. Each resulting envelope of a group of loads gets the corresponding action type for further and final combinations.

As given in the table above the header "PART" defines, to which action of the selected superposition equation the input action belongs. The actions in the superposition equations are distinguished according to their temporal variances:

- permanent actions: e.g dead load PART G
- prestress PART P
- variable actions (T, S, L, …) PART Q
- accidental actions PART A
- action earthquake PART E

The superposition (keyword SUP) of load cases within an action is handled as follows:

Superposition	Description
PERM	Always (permanent) selection of the safety factor action−wise (via the sum of the load cases). The same safety factor is used for all load cases of a permanent action (e.g. ACT G)
PERC	Always (permanent), with variable factors. The safety factors are considered load−case−wisely. Here one load case of the permanent action can get GAMU and another GAMF (gamma unfavorable and favorable).
COND	Conditional (only if unfavorable)
EXCL	Mutually exclusive but conditional (=”either – or”, only within the categories). An unfavorable load case from X_1 and an unfavorable load case from X_2 are used.
EXEX	Mutually exclusive but conditional inclusive categories (only within an action) Excludes only categories against each other within an action (no exclusion of different actions). If for example the categories X_1 and X_2 of the action X are defined with SUP EXEX, then only the most unfavorable load case is used either from X_1 or from X_2 and also only, if it has an unfavorable contribution.
UNSI	Conditional with unfavorable sign (e.g. earthquake)
USEX	Exclusive with unfavorable sign (e.g. wind)
ALEX	Always but exclusive (only within an action). Defines a permanent alternative group. Only the load cases of an action or all load cases of the categories of an action are used for the formation of the permanent alternative group. Here only a load case is considered for the superposition, also if it has a favorable contribution. Load cases of different actions do not exclude themselves mutually.

E.g.: The uniform distributed life load UDL and the Tandem system have different combination coefficients within one action L (live loading). Two categories will be defined (L_U, L_T). The UDL-system (L_U) will be applied span wise, to be superimposed conditionally, whereas the multiple load cases of the Tandem-system (L_T) exclude each other. All load cases however will be applied together as leading action.

7 Combination rules and superpositioning

The superposition according to the EC based codes is done with safety factors (GAMU, GAMF) and combination coefficients (PSI0, PSI1, PSI2, PS1S) for actions defined by arbitrary load cases. These load cases or combinations of them will be superimposed to define an action effect. A superposition contains all action effects which are defined by a combination rule. The most unfavorable variable action is designated as the dominant action. Module MAXIMA selects the most unfavorable one and considers it according to the chosen combination rule.

The actions and the load cases being part of the respective combination are to be input after the corresponding combination rule.

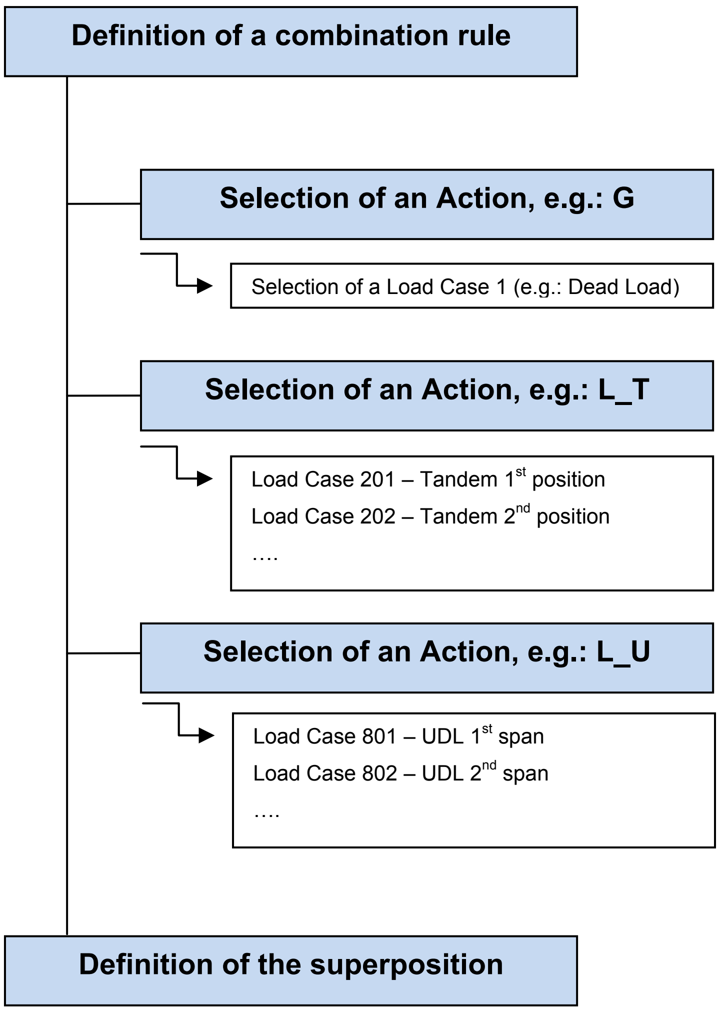


Figure 17: Superpositioning in SOFiSTiK - MAXIMA

Within module MAXIMA so called combination rules are defined.
The following predefined combination rules applies all the above mentioned factors and combination coefficients automatically to each action respectively to each selected load case.

For checks in ultimate limit state (ULS) we differentiate between:
DESI Ultimate design combination
ACCI Accidental design combination
EARQ Earthquake combination,

For checks in serviceability limit state (SLS) we differentiate between:
PERM Permanent combination
RARE Rare combination
FREQ Frequent combination
NONF Non−frequent combination

For intermediate superposition so called standard combination (**STAN**) without safety factors and combination coefficients might be used.

The goal of each superposition is to achieve an envelope for a certain combination rule (considering safety factors, combination coefficients for leading and accompanying or coexisting or associated actions). In SOFiSTiK the resulting envelope of each element and result component (force, displacement, stress,…) will be stored as a resulting load case ("load case" or "result case" number). Furthermore the resulting envelope may also be designated to a type (DESI,ACCI,EARQ,PERM,RARE,FREQ,NONF,STAN, action type, …), this becomes important when the envelope is subsequently used for a design situation using the modules AQB (design of beams) or BEMESS (design of shell elements).

The superposition is done for selected elements. An envelope is created for a leading result component (e.g. internal force) with all associated (resp co-existing) values and stored in a load case. The default load case numbering scheme is explained in the manual for module MAXIMA.

Note: A superpositioning in MAXIMA is only available for systems without changes due to construction changes. MAXIMA supports the load combinations for the final system only. Wherever construction stages are considered the superposition with

G+P+C is done using module AQB. This allows to also combine results due to stage changes.

8 Traffic Loading

The goal of the evaluation of moving load effects on bridge structures is to find the most unfavorable position of the loading for every single element and reaction. Complex load models and lane arrangements contribute to the fact that the governing loading cannot be found in an easy way. There a 2 principal approaches in SOFiSTiK to find the most unfavorable load position:

- Load Stepping: The first approach is to generate a number of representative loadcases explicitly. Each loadcase contains the load model at distinct positions along the axis. The loadcases are then calculated and the envelope is obtained with the superposition module MAXIMA. [Module: SOFiLOAD-V]
- Influence Lines: The second approach is to establish influence lines for forces and moments of all selected locations within the structure. In a second step the influence lines are evaluated with the module ELLA by applying the load models. The envelopes of the results can be directly obtained from this evaluation. [Module: ELLA]

Depending on the field of application, the type of structure and the number of load models and loading scenarios different evaluation methods can be advantageous. To help finding the optimal solution advantages and disadvantages of both the load stepping and the influence line method are listed in the following table.

	Advantages	Disadvantages
Load stepping	• Straightforward approach • Results can be checked easily as all load positions are explicitly generated Recommended for: Hybrid systems (beam and orthotropic shell deck), Slab systems (up to medium size), EN 1991-2 and DIN-FB 101 Road Loading	• Large number of single load cases. • More than 3 loaded lanes and unsynchronized load models increase the effort. • Manual superpositioning is necessary. • Combination for load groups might complicate the superpositioning • Not usable for special load trains, e.g. where load values depend on loaded length etc. (EN: LM3, BS, BRO)
Influence lines	• Complex load models and large number of lanes can be handled • Special features for the evaluation of combinations (load groups) etc. • Results from different directions of actions can be combined in one step • Enveloped results are obtained directly Recommended for: Single or twin-web beam systems, Slab systems	• Keeping track of result history is more difficult • Checks require additional effort (SOFiLOAD: COPY TYP ELLA CASE ..) • No corresponding nodal deformations and nodal support forces are evaluated.

Table 1: Load stepping method vs. influence line method

8.1 Moving Load Analysis acc. to EN 1991-2

In SOFiSTiK the basis of the traffic load process is the general road/rail axis, described in the plan view by straight, circular or spiral segments, and in the elevation as straight or quadratic parabola (c.f. chapter 3). Alternatively a user defined axis with the record GAX in the modules SOFiLOAD or ELLA can be used for traffic loads. Each axis may have up to 99 lanes with the lane numbers 1-99, defined with the record LANE.

Every single notional lane is loaded separately by a load train. Load trains are defined in SOFiLOAD. There are two different types of load trains:

- Standard load trains acc. to design codes (using record TRAI) or,
- User defined load trains (user defined using record TRPL, TRBL)

8.1.1 Subdivsion into LANES

Lanes are defined relative to a geometric axis (see above). The lanes may be created automatically by predefined standard subdivisions or with explicit coordinates. According to EN 1991-2 the carriageway is automatically subdivided into notional lanes as per Table 4.1 (EN 1991-2) using the TYPE EC in SOFiLOAD or ELLA.

SOFiSTiK: LANE lwrl+lwll

SOFiSTiK: LANE TYPE 'EC' =automatically subdivsion into Lanes

Carriageway width w	Number of notional lanes	Width of a notional lane w_l	Width of the remaining area
$w < 5{,}4\ \mathrm{m}$	$n_l = 1$	3 m	w - 3 m
$5{,}4\ \mathrm{m} \leq w < 6\ \mathrm{m}$	$n_l = 2$	$\frac{w}{2}$	0
$6\ \mathrm{m} \leq w$	$n_l = Int\left(\frac{w}{3}\right)$	3 m	$w - 3 \times n_l$

Figure 18: Number and width of notional lanes

Exemplary input:

```
+prog sofiload
Head ,Automatic subdivsion into lanes‘
LANE AXIS TYPE EC WR (14.05/2-1.65) WL -(14.05/2-0.90) YRA 14.05/2 YLA -
14.05/2
end
```

The carriageway width, w (= WR+|WL|), is measured between the curbs (WR, WL) or between the inner limits of the vehicle restraint systems.

The overall width of the bridge deck (including pedestrian areas, footways, cycle tracks, hard shoulders and/or strips) may be specified and measured with the values YRA and YLA. This becomes important when loads should be applied on these areas as well.

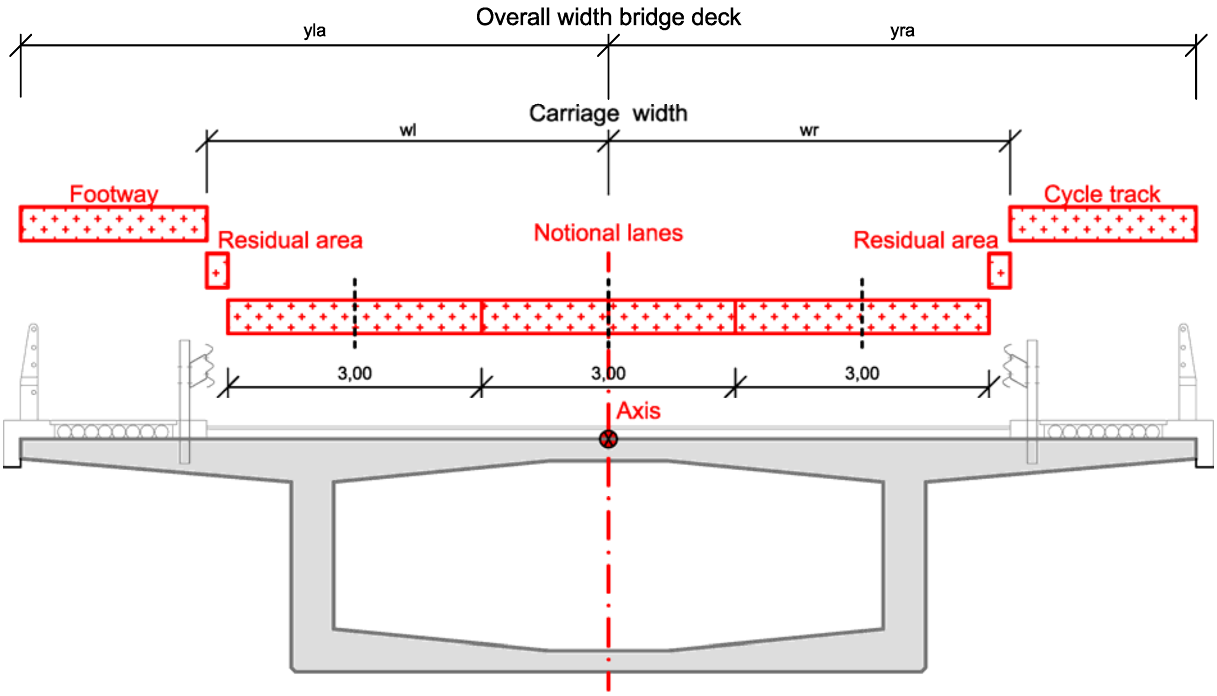


Figure 19: Notional Lanes, residual areas, footways and cycle tracks

'Right' means in direction of the positive local y-axis of the structure, looking in positive x-direction 'along' the bridge.

Note: The carriage width w=WR + |WL| may only be described by constant values. The overall width of the bridge deck YRA + |YLA| can be variable.

8.1.2 Location and numbering of the lanes for design

By choosing the automatic subdivision into lanes according to EN 1991-2 three different sets of lanes are generated by the program:

- **Centric** location on the carriageway and their numbering AXIS.1, AXIS.2, AXIS.3, … (This set will be the only one if the division 'w / lanewidth' has no remainder)
- **Rightmost** location on the carriageway and their numbering AXIS.10, AXIS.11, AXIS.12, …
- **Leftmost** location on the carriageway and their numbering AXIS.20, AXIS.21, AXIS.22, …

The Lane AXIS.0 is always generated and has the full width of the carriageway. Lane AXIS.1 is always the central lane.

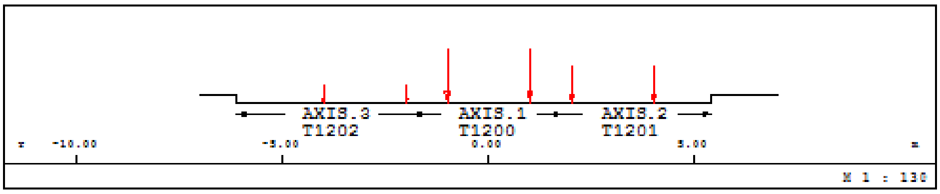


Figure 20: Centric location and numbering

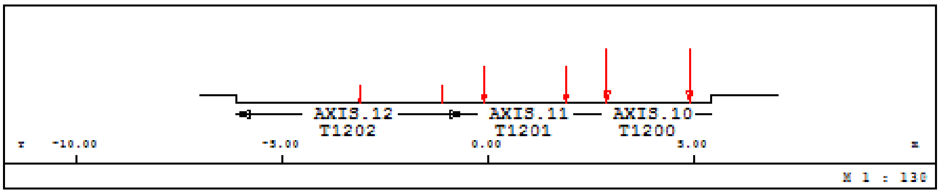


Figure 21: Rightmost location and numbering

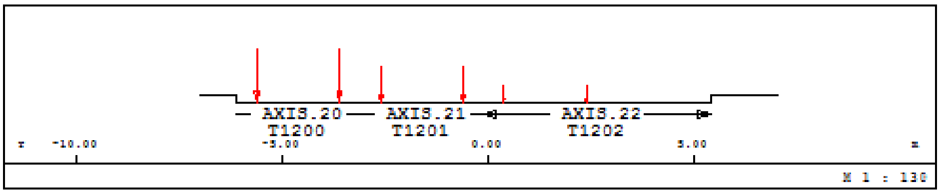


Figure 22: Leftmost location and numbering

The width of the individual lanes becomes important for the treatment of residual loading. The remaining area on the carriageway for each design situation is considered as part of the current adjacent notional lane. (e.g.: in Figure 21 notional Lane "Axis.12" includes the remaining area for the current design location as well).

The application of the load models on the individual notional lanes is treated in the following chapters.

Note: The identifier of a Lane is a number between 1 and 99 (e.g.: ABCD.12)

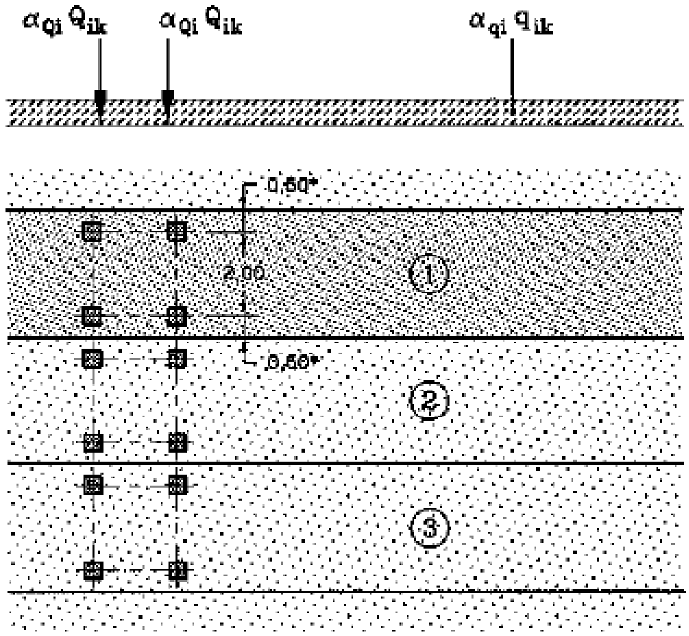
8.1.3 Load Model 1

Load model 1 consists of double-axle concentrated loads (tandem system: TS) and uniformly distributed loads (UDL system).

Location	Tandem system *TS*	*UDL* system
	Axle loads Q_{ik} (kN)	q_{ik} (or q_{rk}) (kN/m^2)
Lane Number 1	300	9
Lane Number 2	200	2,5
Lane Number 3	100	2,5
Other lanes	0	2,5
Remaining area (q_{rk})	0	2,5

TRAI LM1 300
TRAI LM1 200
TRAI LM1 100

Figure 23: Load Model 1 Characteristic values



(1) Lane Nr. 1 : Q_{1k} = 300 kN ; q_{1k} = 9 kN/m^2
(2) Lane Nr. 2 : Q_{2k} = 200 kN ; q_{2k} = 2,5 kN/m^2
(3) Lane Nr. 3 : Q_{3k} = 100 kN ; q_{3k} = 2,5 kN/m^2
(*) For w_l = 3,00 m

Figure 24: Application of LM1

For the different lane sets it is generally recommended to use different load trains with individual adjustment factors α_q. Each load train is activated within one load case. Load model 1 is described with the following input lines:

```
+PROG SOFILOAD
HEAD 'Defintion of LM1-Load Trains'

LC 101 TYPE NONE ; TRAI LM1 300
LC 102 TYPE NONE ; TRAI LM1 200
LC 103 TYPE NONE ; TRAI LM1 100

end
```

With these commands no loading will be applied to the structure yet. The loading definitions are saved under a free load case number.

> We generally recommend using different load trains with individual adjustment factors for every single notional lane.

8.1.4 Load Model 2

Load Model 2 consists of a single axle load being applied at any location on the carriageway. Exemplary input in SOFiLOAD:

```
+PROG SOFILOAD
HEAD 'Defintion of LM2-Load Train'

LC 104 TYPE NONE ; TRAI LM2 300

end
```

Load Model 2 is considered when local effects and verifications have to be treated. In this tutorial Load Model 2 is not analyzed in detail. (i.e. not be selected within ELLA).

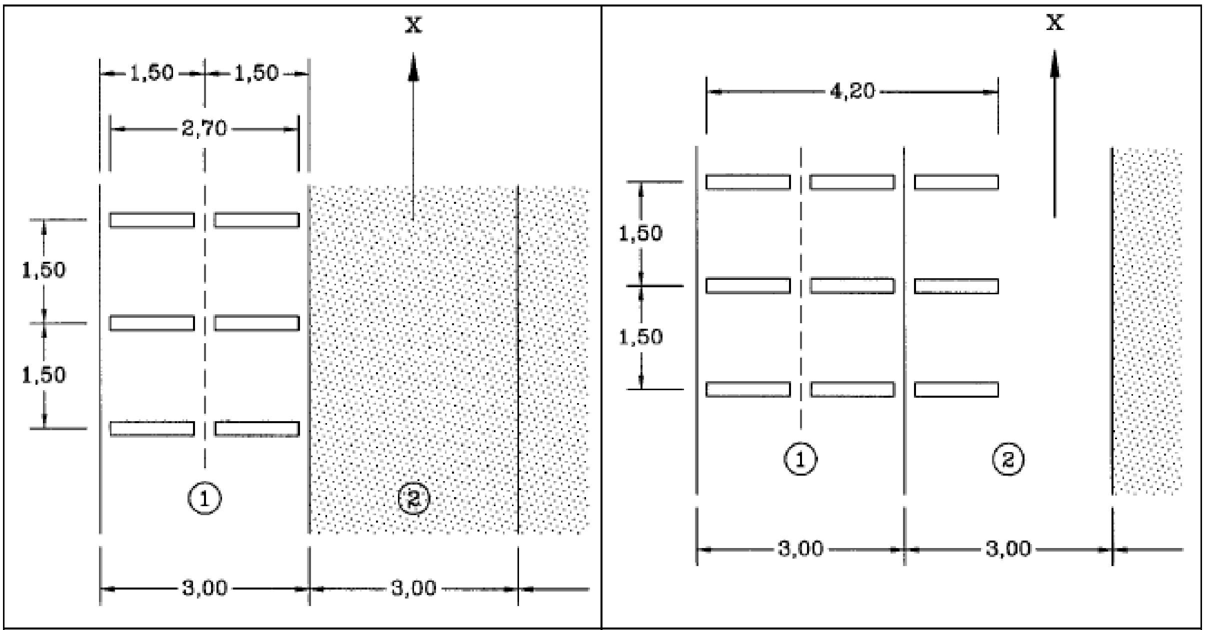
8.1.5 Load Model 3 (special vehicles)

EN 1991-2 defines basic models of special vehicles (EN 1991-2: Tables A1 and A2, and Figure A1).

Depending on the arrangement of axle-lines and the definition of wheel contact areas, EN 1991-2 differs according to classes of special vehicles composed of 150/200kN and 240kN.

According to EN 1991-2 special vehicle load models are applied on the carriageway:

- On one notional traffic lane for the models composed of 150 or 200 kN axle-lines, or
- on two adjacent notional lanes for models composed of 240 kN axle-lines.



Axle-lines of 150 or 200 kN (b = 2,70 m) X : Bridge axis direction (1) Lane 1 (2) Lane 2	Axle-lines of 240 kN (b = 4,20 m) X : Bridge axis direction (1) Lane 1 (2) Lane 2

Figure 25: Application of the special vehicles on notional lanes (EN 1991-2: FIGURE A2)

On the other lanes and the remaining area the bridge deck is loaded by Load Model 1 with its frequent values (ψ1) in addition. There is no loading within a distance of 25 m in front and behind that vehicle.

The dynamic amplification factor will be considered automatically according to EN 1991-2

$\varphi = 1.40 - 0.002 \cdot L$; $\varphi \geq 1.40$

Where the models are assumed to move at low speed, only vertical loads without dynamic amplification are taken into account. This is achieved by input of an explicit value phi=1.00 for the load train.

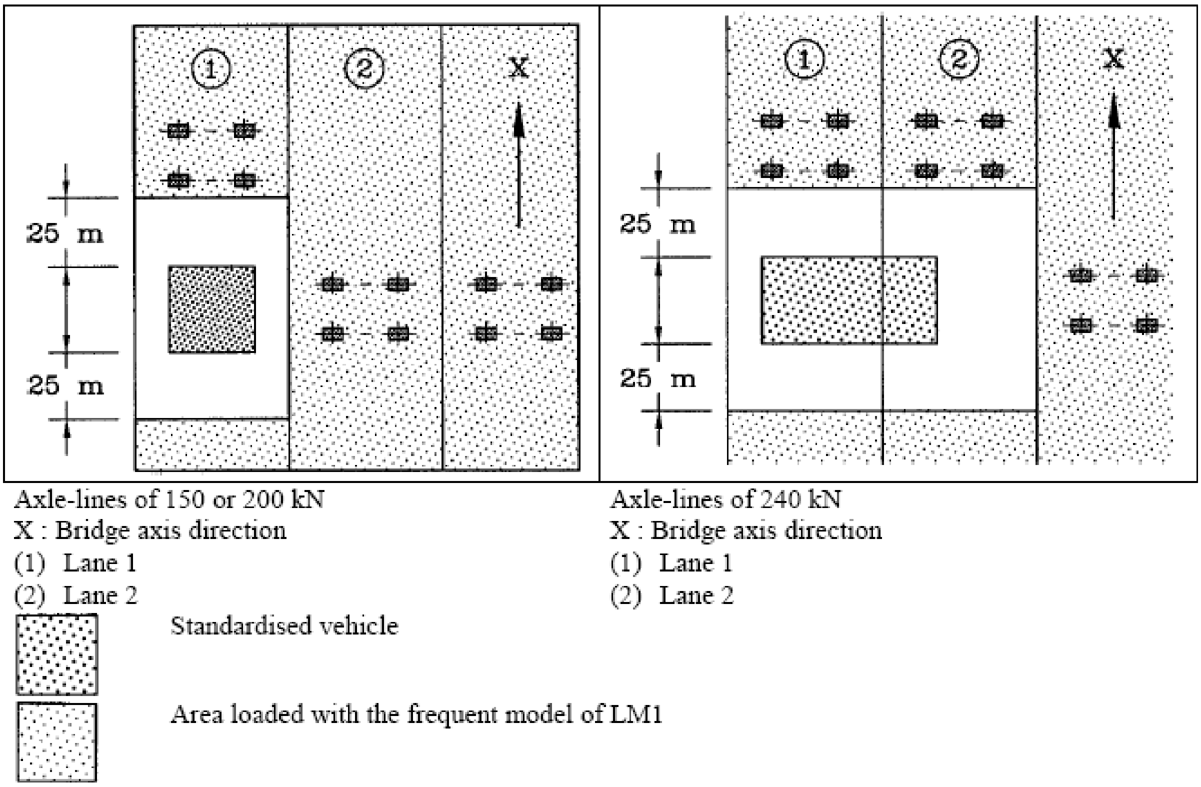


Figure 26: Simultaneity of Load Model 1 and special vehicles (EN 1991-2: FIGURE A3)

Exemplary input of LM3 variations:

```
+PROG SOFILOAD
HEAD 'Defintion of LM3-Load Trains'


LC 111 TYPE NONE ; TRAI LM3 3000 200     V +5 P4 101 $ 15x200kN / LM1 +
LC 112 TYPE NONE ; TRAI LM3 3000 200     V -5 P4 101 $ 15x200kN / LM1 -
LC 113 TYPE NONE ; TRAI LM3 3000 200 200 V +5 P4 101 $ 8x200+7x200kN  +
LC 114 TYPE NONE ; TRAI LM3 3000 200 200 V -5 P4 101 $ 8x200+7x200kN  -

LC 120 TYPE NONE ; TRAI LM3 3000 240     V  0 P4 102 $ empty space /LM1+
LC 121 TYPE NONE ; TRAI LM3 3000 240     V  5 P4 101 $ 1x120+12x240kN /+
LC 130 TYPE NONE ; TRAI LM3 3000 240     V -0 P4 102 $ empty space /LM1-
LC 131 TYPE NONE ; TRAI LM3 3000 240     V -5 P4 101 $ 1x120+12x240kN /-

End
```

In SOFiSTiK we have to distinguish between the narrow version fitting within one lane (2 axles) and the broader version for P2=240 (3 axles). While the vehicle itself is extending into the second lane, we have to specify the LM1 for the second lane as a special load train with V=0.

The speed of the load train is a very important parameter, it does not only control front and rear for some load trains, but it defines also if the load train is moving slowly (v = 5 km/h) or with a regular speed (v = 70 km/h). The dynamic amplification factor is then considered automatically according to EN 1991-2.

For three axles the speed defines also on which side the third axle is placed. Positive values will create them on the right side as in the picture above (Figure 26). A definition V=0 will create the empty space for the second lane only.
By specifying the load case number of a corresponding LM1 load train at P4 this load train is applied 25 m in front and behind the special vehicle.
To select the relative position to the vehicle in front or behind the sign of the P4 definition will be taken. As the traffic jam will be behind the vehicle in general positive values will select this position.

8.1.6 Load Model 4 (crowd loading)

Crowd loading, if relevant, is represented by a Load Model consisting of a uniformly distributed load (including dynamic amplification) equal to 5 kN/m².

```
+PROG SOFILOAD
HEAD 'Defintion of LM4-Load Train'

LC 108 TYPE NONE ; TRAI LM4

End
```

For the application of this load model the definition of the load group is mandatory:
GR3 or GR4 for the area loading and GR0 for the service vehicle.

8.1.7 Fatique Load Model 1

This model is identical to LM1, but the axle loads is reduced by a factor of 0.7, and the distributed loading by a factor of 0.3.
Exemplary input of FLM1:

```
+PROG SOFILOAD
HEAD 'Defintion of FLM1-Load Train'

LC 140 TYPE NONE ; TRAI FLM1 300

End
```

8.1.8 Fatique Load Model 2

Fatigue Load Model 2 consists of a set of idealized lorries, called "frequent" lorries:
Exemplary input of FLM2:

```
+PROG SOFILOAD
HEAD 'Defintion of FLM2-Load Train'

LC 141 TYPE NONE ; TRAI FLM2 3

End
```

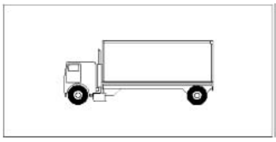
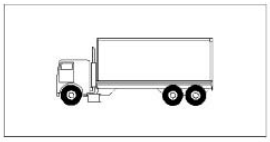
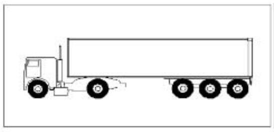
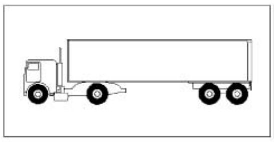
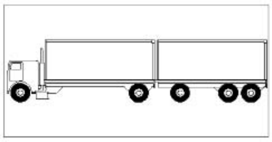
	4,5	90 190	A B
	4,20 1,30	80 140 140	A B B
	3,20 5,20 1,30 1,30	90 180 120 120 120	A B C C C
	3,40 6,00 1,80	90 190 140 140	A B B B
	4,80 3,60 4,40 1,30	90 180 120 110 110	A B C C C

Figure 27: Fatigue Load Model (EN 1991-2: Table 4.6)

8.1.9 Fatique Load Model 3

This model consists of four axles, each of them having two identical wheels.

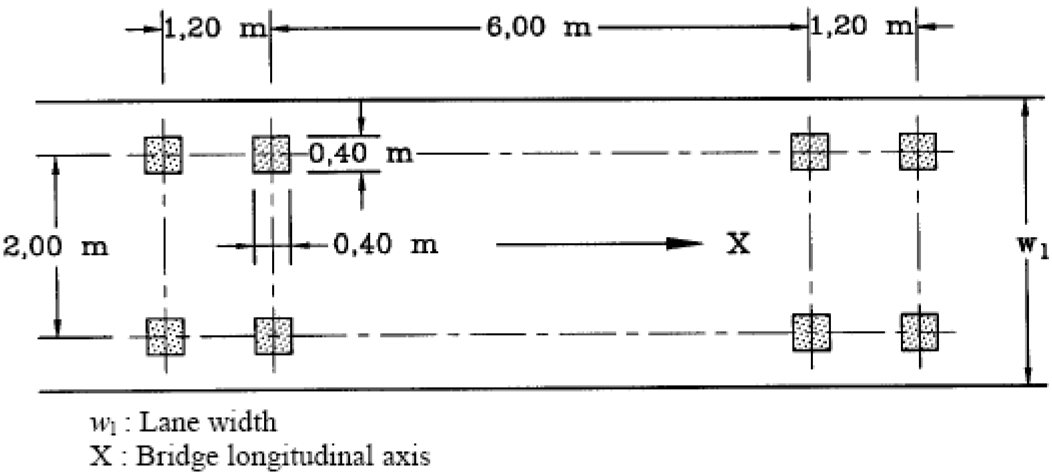


Figure 28: Fatigue Load Model 3 (EN 1991-2: Figure 4.8)

Exemplary input of FLM3:

```
+PROG SOFILOAD
HEAD 'Defintion of FLM3-Load Train'

LC 142 TYPE NONE ; TRAI FLM3

end
```

8.1.10 Fatique Load Model 4

The fatigue Load Model 4 consists of sets of standard lorries which together produce effects equivalent to those of typical traffic on European roads.

The value P1 selects the standard lorries according to table 4.7 of EN 1991-2 with the following axle loading:

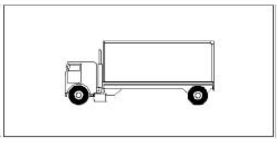
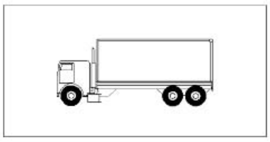
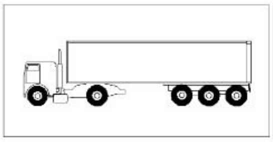
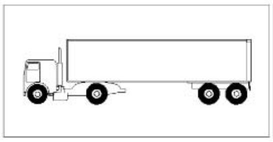
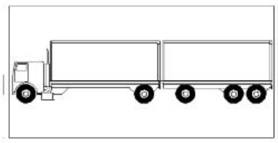
	4,5	70 130
	4,20 1,30	70 120 120
	3,20 5,20 1,30 1,30	70 150 90 90 90
	3,40 6,00 1,80	70 140 90 90
	4,80 3,60 4,40 1,30	70 130 90 80 80

Figure 29: Fatigue Load Model 4 (EN 1991-2: Figure 4.7)

Exemplary input of FLM4:

```
+PROG SOFILOAD
HEAD 'Defintion of FLM4-Load Train'

LC 144 TYPE NONE ; TRAI FLM4 3

end
```

8.1.11 Groups of traffic loads

The simultaneity of the loading systems defined (Load Model 1, Load Model 2, Load Model 3, Load Model 4, horizontal forces, and the loads for footways) are taken into account by considering the groups of loads defined in EN 1992-1 Table 4.4a.

Each of these groups of loads, **which are mutually exclusive**, are considered as (are defining) a characteristic action for combination with non-traffic loads.

		CARRIAGEWAY						FOOTWAYS AND CYCLE TRACKS
Load type		Vertical forces				Horizontal forces		Vertical forces only
Reference		4.3.2	4.3.3	4.3.4	4.3.5	4.4.1	4.4.2	5.3.2-(1)
Load system		LM1 (TS and UDL systems)	LM2 (Single axle)	LM3 (Special vehicles)	LM4 (Crowd loading)	Braking and acceleration forces	Centrifugal and transverse forces	Uniformly Distributed load
Groups of Loads	gr1a	Characteristic values				(*)	(*)	Combination value (**)
	gr1b		Characteristic value					
	gr2	Frequent values(**)				Characteristic value	Characteristic value	
	gr3 (****)							Characteristic value (***)
	gr4				Characteristic value			Characteristic value (**)
	gr5	See Annex A		Characteristic value				
	Dominant component action (designated as component associated with the group)							

Note :
(*) If specified, may be defined in the National Annex.
(**) May be defined in the National Annex.
(***) See 5.3.2.1-(3). One footway only should be considered to be loaded if the effect is more unfavourable than the effect of two loaded footways.
(****) This group is irrelevant if gr4 is considered.

Figure 30: Assessment of groups of traffic loads (EN 1991-2: Table 4.4a)

We strongly recommend making use of the type respectively group of traffic loads when analyzing and evaluating load trains within SOFiSTiK (quod vide 8.2 and 8.3):

'GR0'	gr1	- Tandem-System-LM1 only
'GRU'	gr1	- UDL-System-LM1 only
'GR1'	gr1	- LM1: TS + UDL + Footways and cycle tracks (comb. value)
'GR2'	gr2	- freq. LM1 + Braking and acceleration forces + Centrifugal forces
'GR2N	gr2	- non-freq. variant of gr 2
'GR2F'	gr2	- freq. variant of gr 2
'GR2L'	gr2	- only longitudinal loads of gr 2
'GR20'	gr2	- only horizontal loads of gr 2
'GR3'	gr3	- Footways and cycle tracks
'GR4'	gr4	- Crowd load + Footways and cycle tracks (comb. value)
'GR5'	gr5	- Freq. LM1 + Special vehicles LM3

Note: Load trains in SOFiSTiK do not consist only of vertical loads like the tandem system or the UDL system. Hence, there are different combination coefficients which should be applied for the load components of a load train, the selection of the group assigns individual factors for the components of a load train definition
Every load train can contain braking loads, wind loads, centrifugal loads as well. The user decides when evaluating the loads (in SOFiLOAD or ELLA) which load types respectively load groups is evaluated.

8.2 Load stepping method

The procedure is characterized by the analysis of explicit loadcases (respectively load positions) along a geometric axis. Via the SOFiLOAD 'COPY' command the load trains are transformed into individual loadings. With reference to a geometric axis and an associated lane the following transformations take place:

- The eccentricity is applied relative to the lane center
- The load train is aligned to the geometry of the lane

The proceeding is as follows:

- Subdivision into lanes (quod vide 8.1.1).
- Definition of load trains (quod vide 8.1.3).
- Generation of loadcases within a loop.
- Analysis of the loadcases.
- Superposition of resulting forces of every single load step.

The following input shows in general the functionality of the COPY command within SOFiLOAD, when by instance the Tandem-System of a load train at a certain station along the lane is applied:

```
+PROG SOFILOAD
HEAD ‚Explicit Load cases‘

LC 201 TYPE none TITL 'LM1-TS : Position 1’

COPY NO 101 TYPE GR0 REF Axis.10 DX 20.0

end
```

With this input the load case 101 (defined as load train – Load Model 1) is applied on the structure with reference to the geometric axis 'Axis' and an associated lane (Axis.10) and

then stored under load case no. 201 and action type none (the action type L_T for Tandem systems is applied later when doing the Superpositioning within MAXIMA).
DX is again the station value along the axis. With type GR0 only vertical single loads (TS) of the load train are applied on the structure.

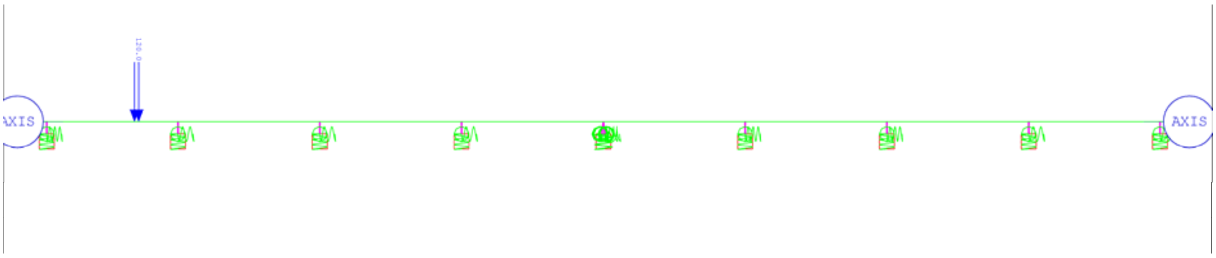


Figure 31: Application of the TS at station DX

Whenever the UDL-System of a Load Train is applied span wise the following input is required:

```
+PROG SOFILOAD
HEAD ‚Explicit Load cases‘

LC 801 TYPE none TITL 'LM1-UDL : 1st span’

COPY NO 101 TYPE GRU REF Axis.10 FROM 1 TO – INC 0

end
```

In the same way the UDL-system of a load train are applied to the structure. The values (FROM, TO and INC) select the segments (spans) of a lane which is loaded. With this input the span no. 1 of the Lane no. 10 is loaded. This becomes important when UDL-loads are applied span by span, which is obviously necessary to obtain max/min-values for inner forces.

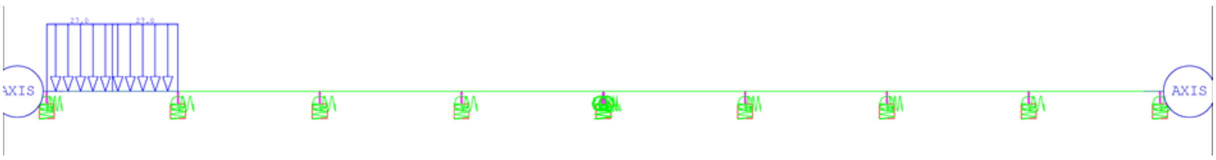


Figure 32: Application of the UDL system in the 1st span

The following input highlights a span wise application of uniformly distributed loads on footways and/or cycle tracks.

```
+PROG SOFILOAD
HEAD ‚Explicit Load cases‘

LC 901 TYPE none TITL 'LM1-Foot : 1st span’

COPY NR 101 TYPE GR3 REF Axis.0 FROM 1 TO – INC 0

end
```

This input is necessary in case that lane no. 0 has the total width of the carriage way so as remaining area the area of cycletracks and footways (|yla|+|yra|)-(|wl|+|wr|) is loaded.

Loads are copied as default with a factor of 1.0. As an alternate or additional mean one may select (record TYPE) combinations for load groups according to the EN 1991-2 (see chapter 8.1.11). While without entry all loads of the load train are taken into account.

In this way multiple load cases are created in a loop. Afterwards the single load cases are combined to envelopes for each single action (L_T, L_U,) within the superposition tool MAXIMA and then in a second step combined in a final superposition with all the other actions for ULS and SLS design combinations.

8.3 Influence Line Method

The evaluation of influence lines becomes necessary for an accurate analysis of larger bridge systems with complex multi lane traffic loading schemes. Within this method influence lines for forces and moments of all selected locations within the structure are established. The influence lines are evaluated for several load trains, possible lane configurations and required groups of loads.

The general work sequence using the Influence Line method is as follows:

- Determination of the stiffness matrix (Analysis of one LC, i.e. dlz).
- Subdivision into lanes (quod vide8.1.1).
- Definition of the load trains (quod vide 8.1.3).
- Activation of influence lines for forces and moments.
- Evaluation of different load train -and lane configurations.
- Saving of evaluation cases in envelopes (load groups).
- Eventually intermediate combination of evaluation cases.

Already defined lanes are activated for evaluation with the record LSEL. Without a lane number all lanes of that axis are selected and the influence lines are established. Obviously the user can define the way how the influence Line is established for every single lane.
So, the input for INT becomes very important, depending on the system a different transversal load distribution is considered:

- 0 for single beam systems.
- 1 for two beam systems, assuming a hinged distribution between the 2 girders.

- 2 for two beam systems, assuming a rigid distribution between the 2 girders.
- 3,5,7,9 for slab systems a transverse influence lines on shell elements are established, where the numbers of load points in transverse direction are specified.

For spatial systems we strongly recommend to make use of the parameter DZ, which defines the depth under the lane which will be investigated for node-sequences. Nodes above the lane axis (negative z axis) are never considered.
One may specify for which inner forces we get an envelope. For each inner force the load case no. for the maximum and minimum values are defined. The resulting LC no. of the envelope in the Database = BASE no. + LMAX/LMIN.
In the same way as for the load stepping method different cases (multi lane traffic loading schemes) are investigated within ELLA. The envelope of several evaluation cases is stored with the record SAVE in a so called saving case, by specifying base-number and action type for the results. Each case gets an identification number and assigns a load train to an individual lane. With input of GRP one specifies the groups of traffic loads according to EN 1991-2, which is evaluated from the load train.
All evaluation cases in a saving case are mutually exclusive; the maximum of these will be carried out by ELLA.

The command SHOW is used to represent influence lines for inner forces and moments with the corresponding load position in a lane for a given element (e.g. beam element 40040).

The synchronization with secondary lanes should not be applied; this is controlled for each lane with the record SYNC OFF in POSL.
The double axle is applied only in total. Thus all loads of the axles are set to be applied even if favorable.

Exemplary Input:

```
+PROG ELLA urs:16.1
HEAD 'AUTOMATIC EVALUATION OF LOAD TRAINS'
SIZE URS 0 HDIV 3
SHOW SNO Axis.10 TYPE ALL NO 300002 ETYP ALL
ECHO LPOS

LSEL Axis INT 0 DZ 0.1

CALC MY 1  2
CALC VZ 3  4
CALC MT 5  6
CALC MZ 7  8
CALC VY 9  10
CALC P  11 12


SAVE LCB 900 TYPE L_T  TITL 'Gr1a LM1:TS'  $ Saving CASE, Envelope L_T -
> LC's: 1001ff
CASE 1 GR0 $ TS:centric
   POSL Axis.1  TRAI 101  SYNC OFF
   POSL Axis.2  TRAI 102  SYNC OFF
   POSL Axis.3  TRAI 103  SYNC OFF
CASE 2 GR0 $ TS:rightmost
   POSL Axis.10 TRAI 101  SYNC OFF
   POSL Axis.11 TRAI 102  SYNC OFF
   POSL Axis.12 TRAI 103  SYNC OFF
CASE 3 GR0 $ TS:leftmost
   POSL Axis.20 TRAI 101  SYNC OFF
   POSL Axis.21 TRAI 102  SYNC OFF
   POSL Axis.22 TRAI 103  SYNC OFF


SAVE LCB 920 TYPE L_U  TITL 'Gr1a LM1:UDL' $ Saving CASE, Envelope L_U -
> LC's: 1021ff
CASE 1 GRU $ UDL:centric
   POSL Axis.1  TRAI 101  SYNC OFF
   POSL Axis.2  TRAI 102  SYNC OFF
   POSL Axis.3  TRAI 103  SYNC OFF
CASE 2 GRU $ UDL:rightmost
   POSL Axis.10 TRAI 101  SYNC OFF
   POSL Axis.11 TRAI 102  SYNC OFF
   POSL Axis.12 TRAI 103  SYNC OFF
   POSL Axis.13 TRAI 103  SYNC OFF
CASE 3 GRU $ UDL:leftmost
   POSL Axis.20 TRAI 101  SYNC OFF
   POSL Axis.21 TRAI 102  SYNC OFF
   POSL Axis.22 TRAI 103  SYNC OFF
   POSL Axis.23 TRAI 103  SYNC OFF
CASE 4 GR3 $ UDL:footways and caycletracks
   POSL Axis.0  TRAI 101  SYNC OFF

COMB A0   1:3  1.0  4  1.0
```

The input above generates individual cases (CASE) for each set of possible arrangements (i.e. leftmost rightmost and centric here). Alternatively a generic approach using a uniform load model can be employed, an example can be found in the CABD example file: cabd_beam_1.sofistik (B: Influence line evaluation).

8.4 Highway bridge loading acc. to BS 5400 (BD37/01)

The basic features of a moving load analysis described in the previous sections is be applied to various other traffic loading scenarios. In the following the traffic loading according to the composite version of BS 5400 (Volume 1 Section 3 and Appendix A) is considered.

> Due to dependency of the HA type UDL loading on loaded length and variable overall length of the HB vehicle a correct evaluation is only possible using influence lines in module ELLA.

8.4.1 Subdivision into notional lanes

Similar to the subdivision scheme of the Eurocode, an automatic subdivision of the carriageway acc. to BS 5400 clause 3.2.9 is activated with LANE TYPE BS. The exemplary input for a carriageway width of 12m and a total width of 13.2m reads:

```
$ wr-|wl|= Width between curbs
$ yra-|yla|= total width
$ Here a basic LANE with LANE distribution type BS is generated:
LANE AX_1 TYPE BS WR 12.0/2 WL -12.0/2 YRA 13.2/2 YLA -13.2/2
```

Following clause 3.2.9.3.1 between 10.95 and incl. 14.60m 4 notional lanes are generated, each with the width 12.0m/4=3m. The plots of the transverse loading scheme in ELLA show designation and arrangement of these four lanes.

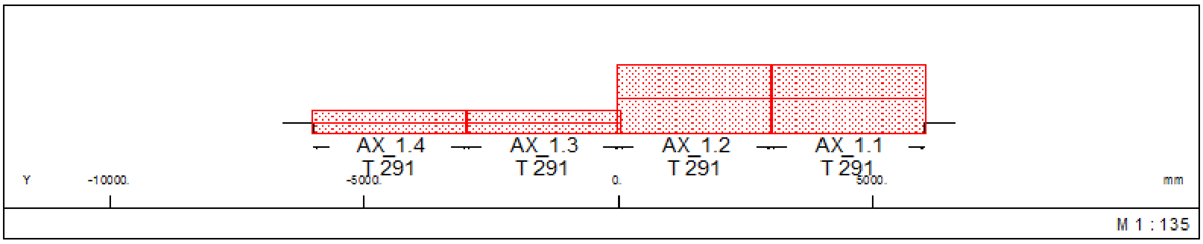


Figure 33: Notional lanes AX_1.1-4 for a 12m carriageway

8.4.2 Definition of loadmodels (HA,HB)

Sorry … Under Construction

9 Construction stages

Time dependent effects of creep, shrinkage and relaxation are investigated using the 'Construction Stage Manager' (CSM), which enables the simulation of all kinds of construction methods. The construction process is managed in a table using an abstract timeline for the activation or deactivation of groups and loads and their properties.

The general work sequence is as follows:

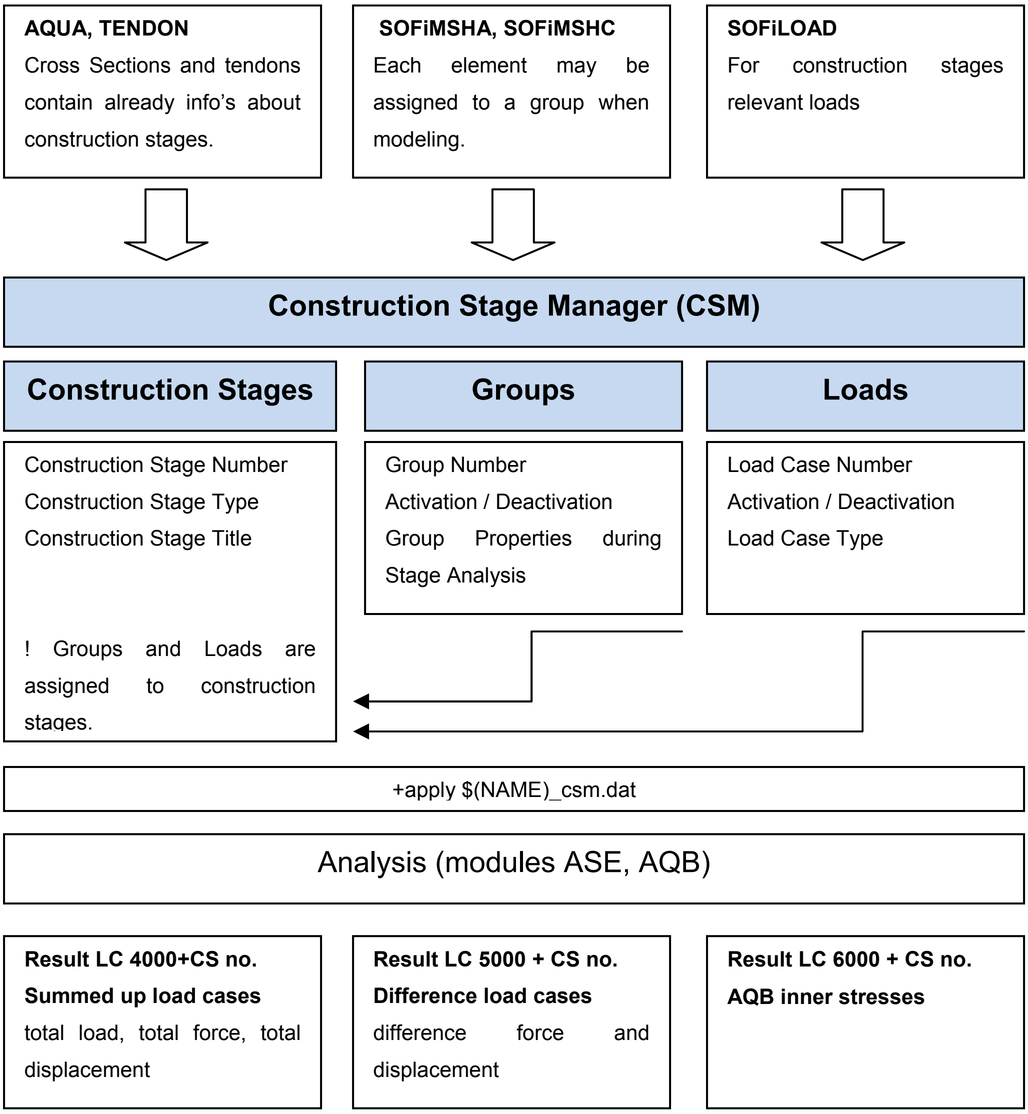


Figure 34: General workflow for Construction Stages Analysis

9.1 Table: Construction Stages

The table construction stages contains all required construction stages (CS that cause changes of stresses have to be considered).

For the numeration we recommend to differ between ‘main Construction Stages’ and ‘Intermediate Construction stages’.

Due to the fact that intermediate construction steps (e.g.: prestressing, grouting, temporary loads, creep&shrinkage) are carried out between two main construction Stages it is useful to store these main stages in every 10th construction stage number. So there are enough numbers in between to store the intermediate steps properly without shifting the numbers of the main stages, even if the intermediate steps are inserted subsequently.

Proposed Numeration:

Stage No.	Stage Type
10	Activation of a new group
11	Prestressing
12	Grouting
13	Temporary load (e.g.: cantilever carriage)
15	Creep−step
20	Activation of a new group

Where a part of a cross section should be activated the definition of a new construction stage is required.

9.2 Table: Groups

Sequence and properties of the activation/deactivation of elements (assigned to a group) are described in the group table.

9.3 Table: Loads

The load case table defines the sequence and properties of the activation/deactivation of additional or temporary loads.

10 Design Code Checks

The most time consuming part of a bridge analysis is the set-up of the appropriate design code checks and assessments. According to different codes and requirements the design code related input varies in dependence on the situation and bridge type under consideration. The SOFiSTiK module CSM offers code check manager functionality when used with the input records ACT and DESI. The module generates an auxiliary file called [projectname]_desi.dat which contains a proposal of design check module input using MAXIMA, AQB and BEMESS. This file should be carefully checked and adjusted to individual requirements. This chapter of the tutorial will be enhanced in the future to cover the bridge design following several design codes. In the exemplary input the envelopes of loadgroups 1,2 and 3 are declared for the usage in SLS and ULS checks according to the EN 1992-2004.

```
+PROG CSM
HEAD Bridge Design
$
ACT  TYPE    FOR    $ Definition of additional actions:
$     G,P,C          $ GPC are activated automatically also
                    $ without this line!
     T      SLS,ULS $ SLS,ULS
     ZF     SLS     $ SLS
     SF     ULS     $ ULS

    GR_1   SLS,ULS
    GR_2   SLS,ULS
    GR_5   SLS,ULS

 DESI DECO         $ DECO    first decompression check
$ DESI ULTI              $ ULTI    first short ULS design
$ DESI STAN              $ STAN    all usual checks

SELE BEAM 400001 X 0          $ Stäbe für die AQB-AQUP Spannungsausgabe
END

$ Overview created loadcases and superposition actions see CSM-manual
Theoretical Principles

+apply $(project)_desi.dat
```

Annex A: Examples

cabd_beam_1.sofistik

Example for a road beam bridge acc. t. EN

This SSD Example shall illustrate the basic usage of the CABD concept. It employs text-based (CADINP) input of:

- Master Cross-Sections
- Axis, Geometry and Variables
- Superstructure and Bearings
- Prestressing Systems
- Basic loading
- Traffic Loading

<u>Summary:</u>

8 span bridge (37.5+40.5+40.5+40.5+40.5+40.5+40.5+37.5)

1-Beam-System, prestressed box girder (variable non effective parts)

Span by span erection

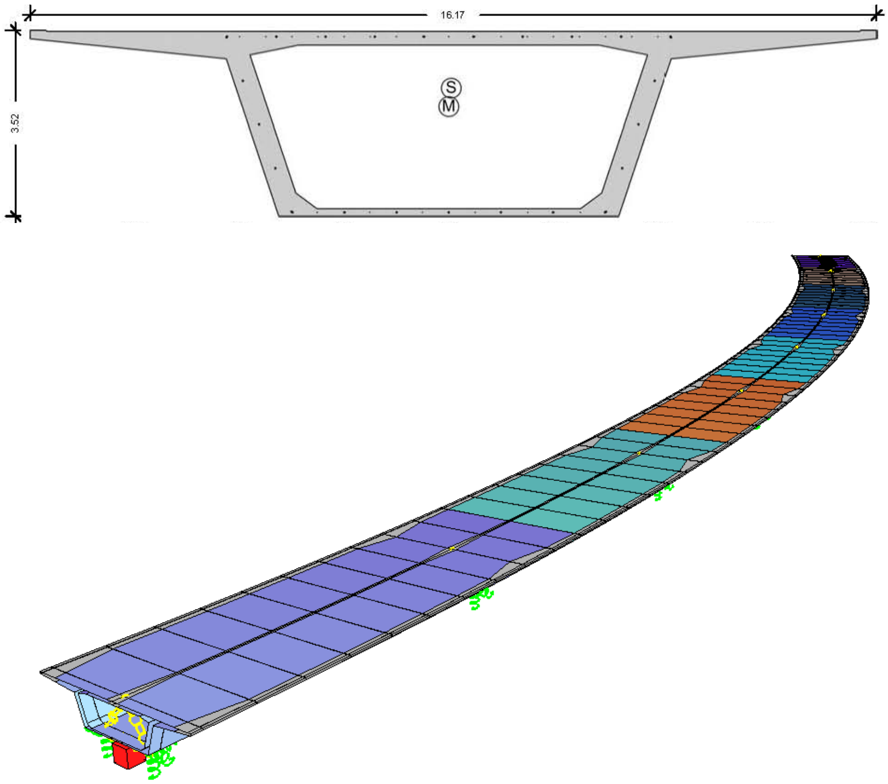


Figure 35: Prestressed box girder, 8 spans

cabd_gird_1.sofistik

Example for a road beam bridge acc. t. EN

This SSD Example shall illustrate the basic usage of the CABD concept. It employs text-based (CADINP) input of:

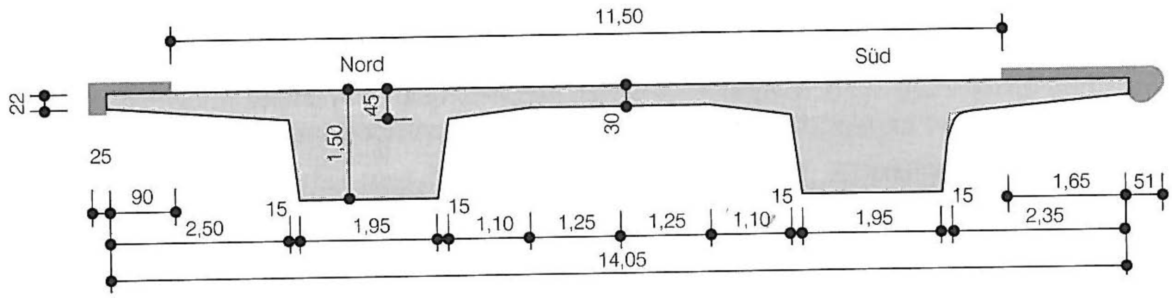
- Master Cross-Sections (Cross Section Editor)
- Axis, Secondary Axis, Geometry and Variables
- Superstructure and Bearings
- Prestressing Systems
- Basic loading
- Traffic Loading

<u>Summary:</u>

7 span bridge (25.0 + 32.0*5 + 25.0)

2-Beam-System with transversal beams, prestressed box girder (variable non effective parts)

Span by span erection



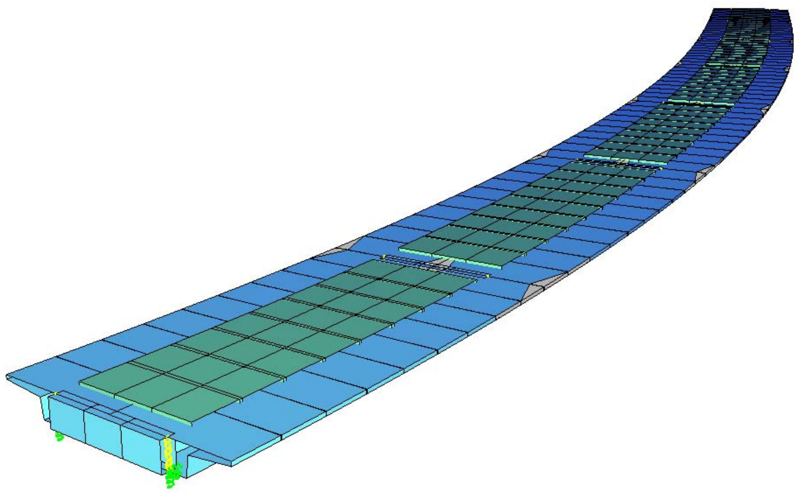


Figure 36: Two Beam-System with transversal beams, 7 spans