1 FE-model

The FE- model is made through a MATLAB- script that accepts certain input parameters to describe a suspension bridge. This makes it easy for the analyst to change parameters in specific analyses. The model may easily be modified to describe a variety of suspension bridges. This is in big contrast to the traditional way of making FE-models of bridges, where it would be time consuming to make large changes of the model geometry. The suspension bridge is very applicable for such a generic modelling technique as the geometry is easily described by a small number of parameters.

The model has been programmed in MATLAB, where a script that accepts certain input parameters automatically creates an Abaqus model of a single-, double-, or triple- girder suspension bridge, according to the input parameters. Abaqus is an advanced tool for performing finite element simulations within several engineering- and mechanical- disciplines. The Abaqus Solver uses an input text file that describes the entire model and type of simulation, to run an analysis. The mentioned MATLAB script writes this input file from the selected input paremeters.

master_script.m

The master script is the user inter-phase of the MATLAB program that has been made. In this script the analyst must specify the necessary parameters to describe the geometry and the mechanical properties of the bridge.

Table 1: Input parameters suspension bridge code

L	Length of mid span
L_side	Length of side span
d	Horizontal c-c distance between main and side girders
e_vert	Vertical distance between main- and side- girder
D	Horizontal distance between cables mid span
D_pylon	Horizontal distance between cables over pylons
subheight	distance from pylon- bottom to girder
hangers	Numbers of hanger- pairs in the main span
number of girders	Number of girders for the bridge. The values 1, 2 and 3 are accepted
girder_sag	Positive sag of girder ("Girder height") before gravity is applied
h	Vertical distance from horizon to cable ("Cable height")before gravity
sag	Cable sag before gravity is applied.

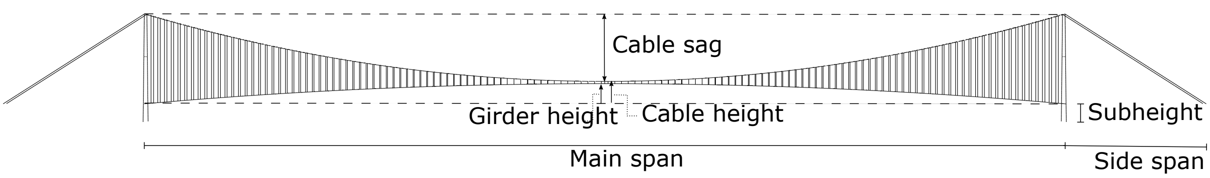

Figure 1: Main parameters of the suspension bridge. Undeformed geometry.

The parameters in Table 1 creates the undeformed geometry the suspension bridge illustrated in Figure 1. This is the geometry of the bridge before it is subjected to gravity loads. The analyst must also specify the mechanical properties of the bridge elements, listed in Table 2.

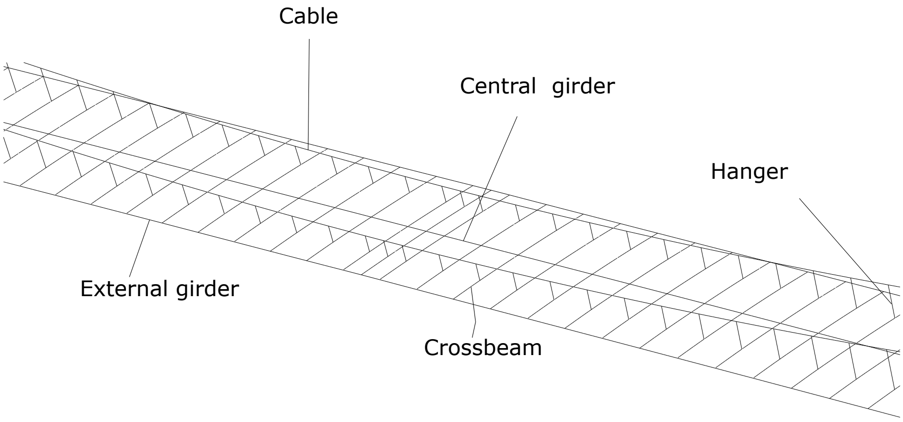

Figure 2: Main elements of the triple- girder suspension bridge.

Table 2: Mechanical properties that must be defined by the analyst for each main structural part of the bridge. See Figure 3 for axis definition.

A	Area of cross section
I_y	Moment of inertia for bending about the n1-axis
I_zy	Moment of inertia for cross bending
I_z	Moment of inertia for bending about the n2-axis
I_t	Torsional constant
E	Young's modulus
G	Torsional shear modulus, G
m	Mass per unit length

The program is created to accept all values in SI-units. The master script accepts these inputs

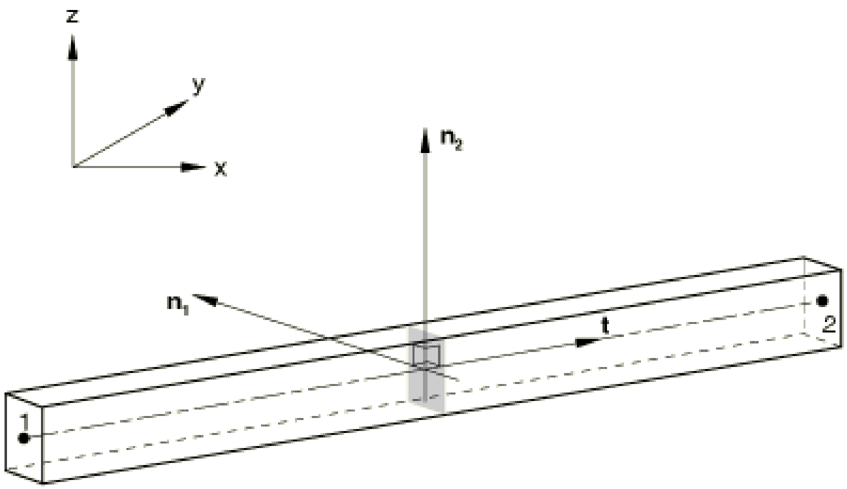

Figure 3: Element axis definition in Abaqus. Facsimile from Abaqus Keywords Reference Guide [?].

from the analyst, and passes them on to a set of functions that creates the necessary vectors and matrices to describe the system.

geometry.m

The geometry- function reads all the geometric input from the master script. It uses this input to create all the nodes of the model and specifying the spacial coordinates of every mode, as well as giving each mode a unique number. The geometry created by the input parameters is the undeformed geometry of the bridge, before the gravity loads are applied.

elements.m

This script arranges the nodes into elements. Every element is given a unique number. The elements are grouped into element sets for each of the different structural members in the model.

writefile.m

After running geometry.m and elements.m, all the neccecary data to create the Abaqus model is saved in the workspace. This script writes the input text-file by using the built-in function *fprintf* in MATLAB. The Abaqus input file processor requires a certain syntax of the input-file in order to perform a calculation. The matrices in the workspace are now written to the text-file in a specific order.

1. The spacial coordinates of all nodes in the model. Each node must also be given a unique number.
2. The elements and element type. Each element must be given a unique number, and it must be specified which nodes the elements contains. The elements are arranged in element sets that differ between the different structural members
3. The sections. Element sets are given mechanical properties and orientation.
4. Static step. Adding gravity loads on all members.
5. Frequency step. Eigenvalue extraction to calculate the natural frequencies and corresponding mode shapes.

Documentation for the requirements of the Abaqus input file can be found in Abaqus Keywords Reference guide [**?**].

Single, double, and triple-girder principle

The model can be implemented with one, two, or three bridge deck girders. See Figure 4. For the single girder case the model disregards the external girders. The crossbeams is given negligible mass, and very high stiffness. The double girder bridge is created by disregarding the central girder. The girders is now connected to the hangers by large cross-beams. The position of the hangers on the crossbeams is controlled by the cable distance and the distance between the girders. The triple girder has the same structural principle as the double girder except that the central girder is included.

The analyst only needs to select the number of bridge girders in the analysis. The excess girders will be automatically disregarded, and the BC's will be updated the writefile-script.

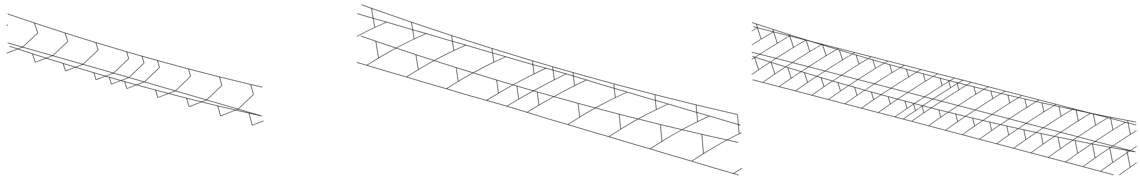

Figure 4: Bridge cross section for one, two and three girders

Static tensioning step

It is important to add the gravity loads before doing the frequency step. This is because the geometrical stiffness is of particular importance for suspension bridges. This is especially for the cables, which rely very much on the geometrical stiffness. To account for the geometric non-linearities, and the large-deflection theory that occurs especially in the cables, the *NLGEOM option in Abaqus is enabled.

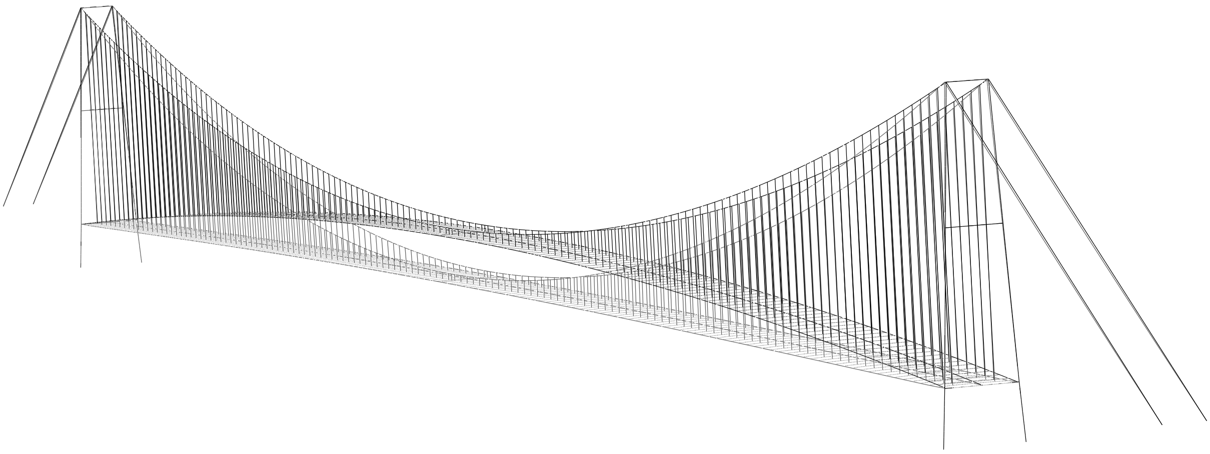

Figure 5: Geometry before and after the *STATIC tensioning step. Example setup: Girder height mid span before tensioning = 45m. Girder height after tenioning = 8.5 m.

The parameters *Girder_sag*, *h* and *sag* in Table 1 must be tested by the analyst. These three parameters describes the sag of the girder and cable before the tensioning step. Their size must be changed in order to obtain a reasonable equilibrium position for the bridge after gravity loads are applied. In other words: Since the geometry must be developed before gravity loads are applied, the analyst must try different settings for the sag to achieve the geometry desired after gravity is applied. In this thesis, a cable sag of $\frac{L}{10}$ and a positive girder sag of 10-20 meters is desired, after gravity is applied.

Elements and boundary conditions

This model uses two element types. The main- and side-girders, together with the crossbeams and the pylons are modelled as B32 beam elements. The main cables and the hangers are B31 beam elements. B31 is a beam element in space, with linear interpolation. It has one node in each end. B32 has quadratic interpolation, and hence also three nodes over the element length. The documentation can be found in the Abaqus Analysis User's Guide [?].

The cables are fixed in all translational and twisting degrees of freedom at the ends. Towers are also fixed in the same way at ground level. The main girder BC's is changed after the *STATIC tensioning step described in Chapter 1. This is because the girder needs symmetric BC's in the tensioning procedure in order to deflect symmetric. This is an important condition to obtain symmetric mode shapes in the *FREQUENCY- step. In the *STATIC step the girders- ends has slide-bearings in the longitudinal direction, but they are fixed against vertical- and lateral-translation, as well as rotation about the longitudinal axis. The BC's are changed slightly after the *STATIC load step: One of the girder-ends are fixed against longitudinal translation, in order to prevent the modes obtained in the *FREQUENCY- step to include large oscillations in the longitudinal direction. This restraint will reflect the physical behaviour of a real suspension bridge, that is not able create large longitudinal deflections due to the abutments.

Pylons

The master script that generates the FE-model and analysis accepts input parameters for the pylon geometry, stiffness and material properties. As argued later in this paper, the code is not customized for decreasing area and stiffness over the height of the pylons. This decision is based on the results obtained in Chapter **??**.

The pylons has three stiffening girders over the height. The top girder is located at the pylon top. The lowest stiffening girder is 10 cm beneath the support of the bridge girder. Note that these are not connected as the boundary conditions of the bridge girder prevents them from making contact.

The model has an option that accepts different distance between the cables mid- span and over the pylons, which will make the towers A- shaped. This is a much used solution for many suspension bridges world wide. This option may not be used unconditionally, as it induces large compression forces in the crossbeams in the mid-span. For wide setups this will cause buckling in the crossbeams.

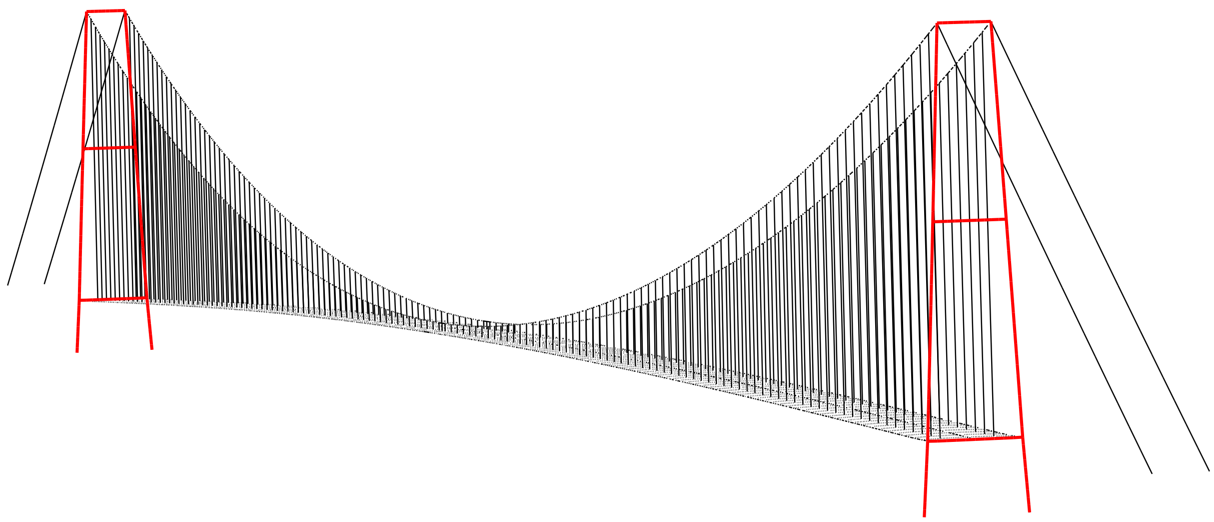

Figure 6: Setup 3 with pylons highlighted

2 Geometric and mechanical properties Setup 1-3

Table 3: Geometrical- and material- properties for Setup 1-3

Parameter	Central deck	External deck
Area $[m^2]$	0,5391	1,225
Mass [kg/m]	3300	3050
Vertical second moment of inertia I_y $[m^4]$	0,4919	0
Lateral second moment of inertia I_z $[m^4]$	14,015	
Torsional inertia $I_T[m^4]$	0,15	
Youngs Modulus E $[N/mm^2]$	2,05e11	2,05e11
Shear Modulus G $[N/mm^2]$	7,8e10	7,8e10
	Hangers	Cables
Area $[m^2]$	0,01	See Table **??**
Youngs Modulus E $[N/mm^2]$	2,05e11	2,05e11
	Crossbeams	Pylons
Area $[m^2]$	0.073286	17.181
Mass [kg/m]	575.2	-
Vertical second moment of inertia I_y $[m^4]$	0.034401	
Lateral second moment of inertia I_z $[m^4]$	0.008386	
Torsional moment of inertia $I_T[m^4]$	0.15e-4	224.90
Youngs Modulus E $[N/mm^2]$	2,1e11	4,0e11
Modulus G $[N/mm^2]$	80,8e9	1,2e10

3 Geometric and mechanical properties Hardanger Bridge

The geometrical and mechanical properties of the approximated Hardanger FE-model in Chapter **??**.

```
L=1310;           %Total span-length
L_side= 250;      %Side span length
d= 9;             %distance from main girder to side girders (center-center)
D=18;             %Distance between cables mid span
D_pylon=D;        %Distance between cables over pylons
subheight=20;     %Length of pylon legs under the girder
e_vert= 1.8;      %vertical distance from side to main girders

hangers= 67;      %must be an odd number

girder_sag= 20;   %positive curvature of girder before gravity
h=25;             %vertical distance from cable to girder before gravity
sag=110+h;        %Sag of cable before gravity loads are applied
```

Table 4: Geometrical- and material- properties og the Hardanger Bridge

Parameter	Central deck	
Area $[m^2]$	0,5813	
Mass [kg/m]	8080	
Vertical second moment of inertia I_y $[m^4]$	0,972	
Lateral second moment of inertia I_z $[m^4]$	16,448	
Torsional moment of inertia $I_T [m^4]$	4,298	
Vertical mass moment of inertia $I_{m,y}$ $[m^4]$	217020	
Lateral mass moment of inertia $I_{m,y}$ $[m^4]$	12515	
Youngs Modulus E $[N/mm^2]$	2,10e11	
Shear Modulus G $[N/mm^2]$	8,08e10	
	Hangers	Cables
Area $[m^2]$	0,0032	0,22132
Youngs Modulus E $[N/mm^2]$	1,6e11	2,00e11
	Crossbeams	Pylons
Area $[m^2]$	1	17.181
Mass [kg/m]	1	-
Vertical second moment of inertia I_y $[m^4]$	1000	172,65
Lateral second moment of inertia I_z $[m^4]$	1000	124,66
Torsional moment of inertia $I_T [m^4]$	0	224.8967383
Youngs Modulus E $[N/mm^2]$	2,1e11	4,0e10
Modulus G $[N/mm^2]$	80,8e9	1,2e10