Tutorial:
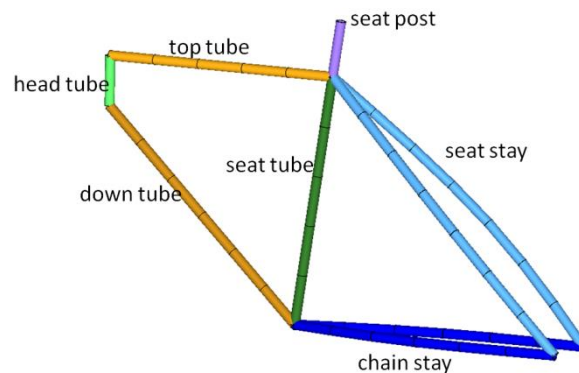# Conceptual Design of a Bicycle Frame

Based on a report partly written by Lena Hartung*, Nicolas Torgau*,
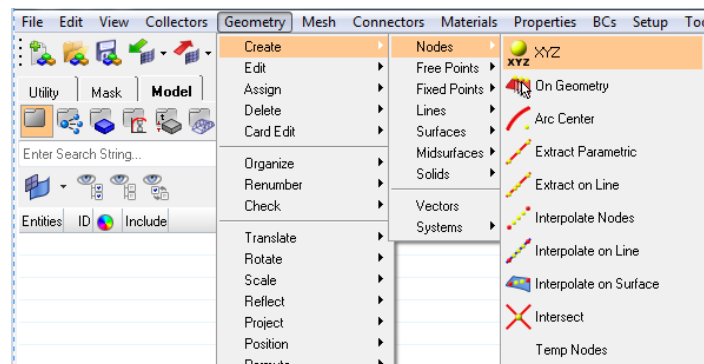Thomas Lehmann, Kristian Holm and Matthias Goelke (*Students at Altair)

Updated to HW 2017.2 by Sanjay Nainani

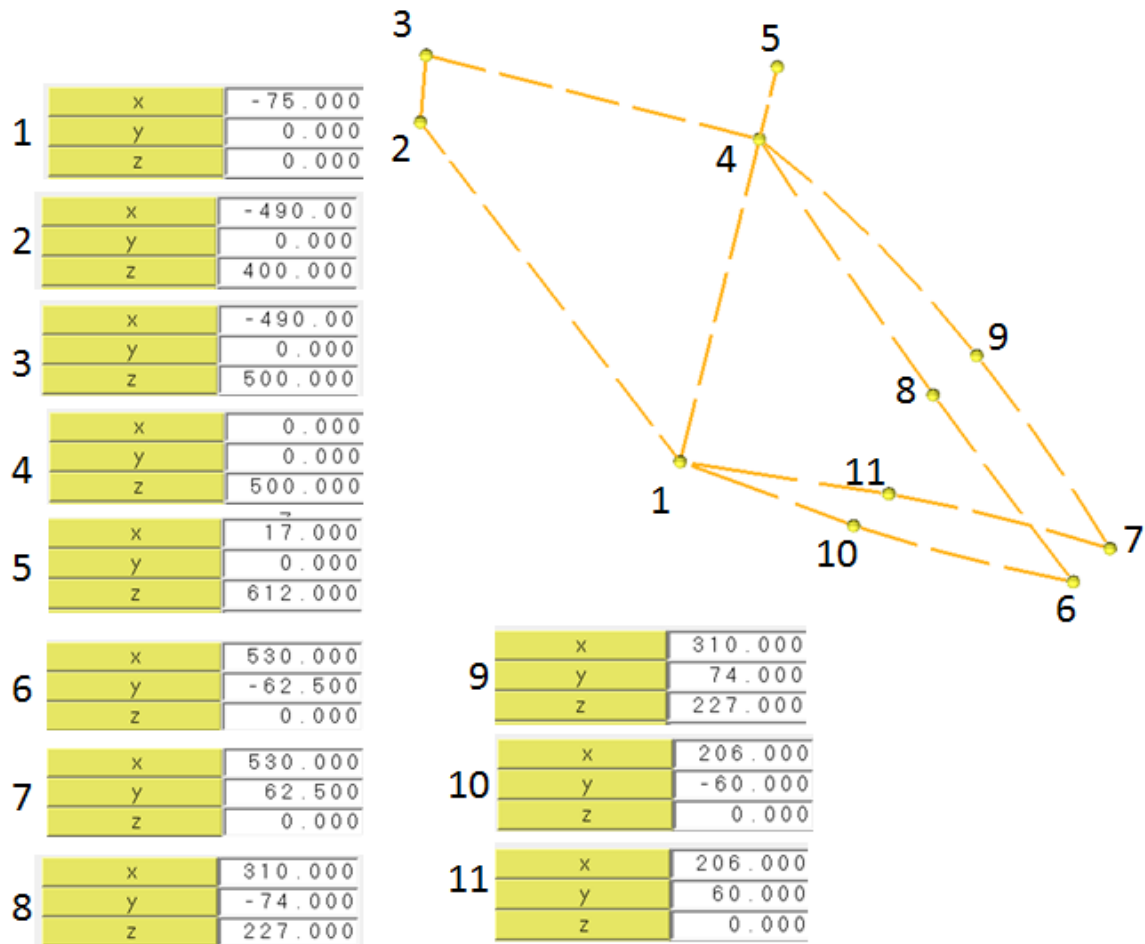## Part 1: Bike frame construction

In this section we are going to construct a bicycle frame as shown below in HyperMesh

Step 1: Open a new HyperMesh session and define temporary nodes
Go to Geometry → Create → Nodes →XYZ

The coordinates of the bounding nodes are shown below.

Node	x	y	z
1	-75.000	0.000	0.000
2	-490.00	0.000	400.000
3	-490.00	0.000	500.000
4	0.000	0.000	500.000
5	17.000	0.000	612.000
6	530.000	-62.500	0.000
7	530.000	62.500	0.000
8	310.000	-74.000	227.000
9	310.000	74.000	227.000
10	206.000	-60.000	0.000
11	206.000	60.000	0.000

Step 2: Build lines

First, create a Component Collector (i.e. storage container of the lines / geometry), then start building lines with respect to the previously defined temporary nodes
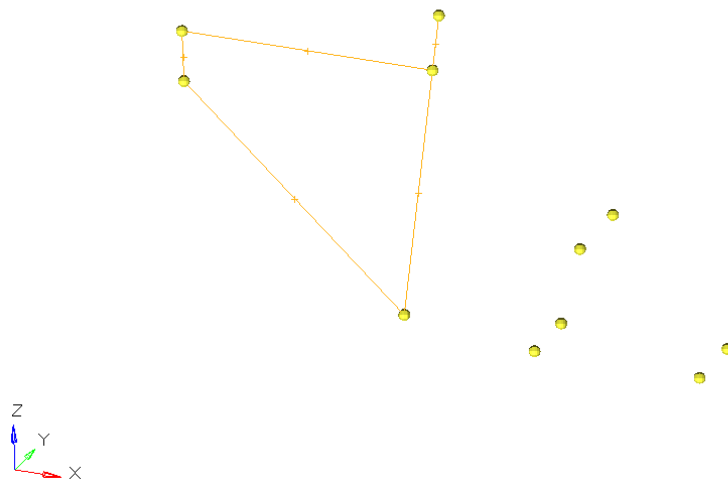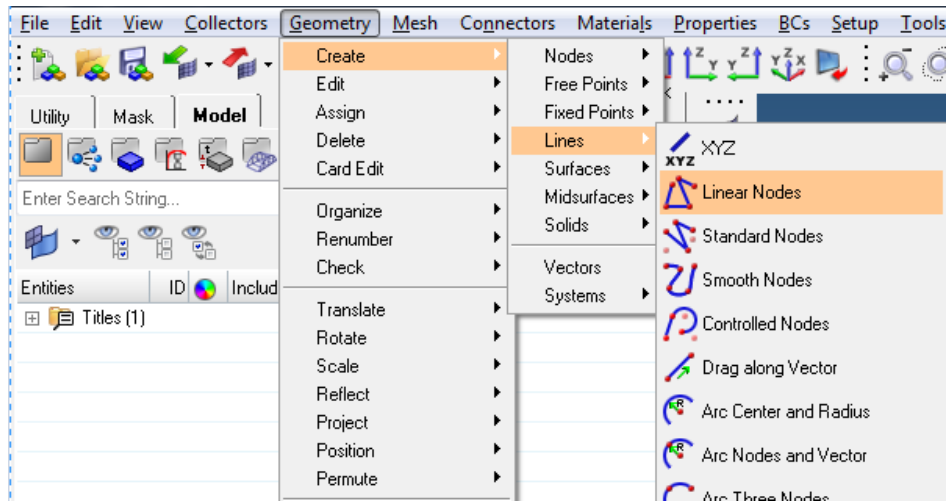
The geometry (lines) regarding chain stay and seat stay are build by employing "Smooth Nodes"
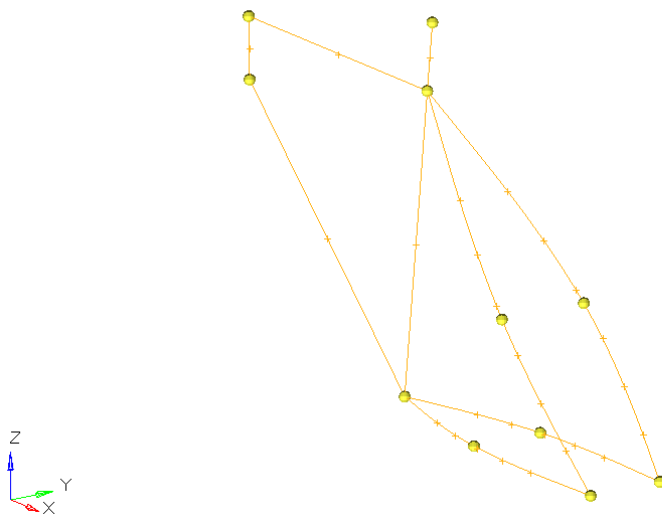
Finally, delete all temporary nodes as they may confuse you later on.
Save the geometry (lines) as an *.hm file (→ bike_frame_1.hm).

Make sure the User Profile ![icon] OptiStruct is selected
Load the hm file from the previous step: bike_frame_1.hm

Part 2: 1D Beam element creation

We assume that the frame is made out of aluminum tubes which will be
modeled as 1D elements (i.e. CBEAM elements). The seat post is made out of
steel.
Note: Beam element allow for axial loads, shear forces, bending and torsion
moments. In beam elements the neural axis and shear center are non-
coincident.
It is recommend to make use the predefined 1D element cross-sections
available with the OptiStruct library.

The beams cross-section is best defined via HyperBeam.
Go to Properties → HyperBeam
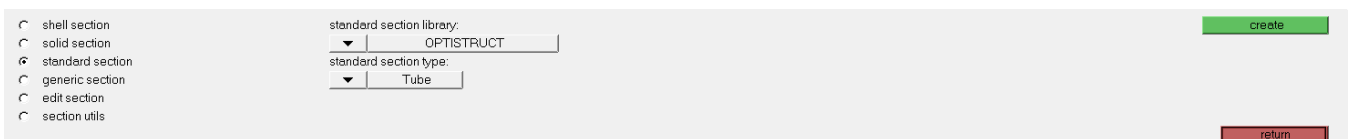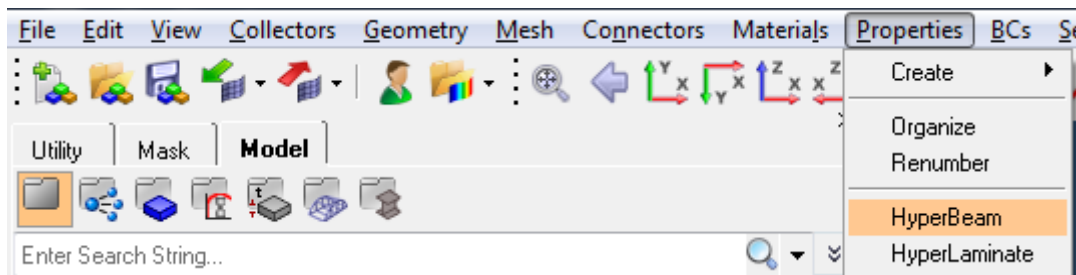
As "standard section library" select OPTISTRUCT, as section type choose
"Tube", then click "create" which opens up the GUI of HyperBeam.
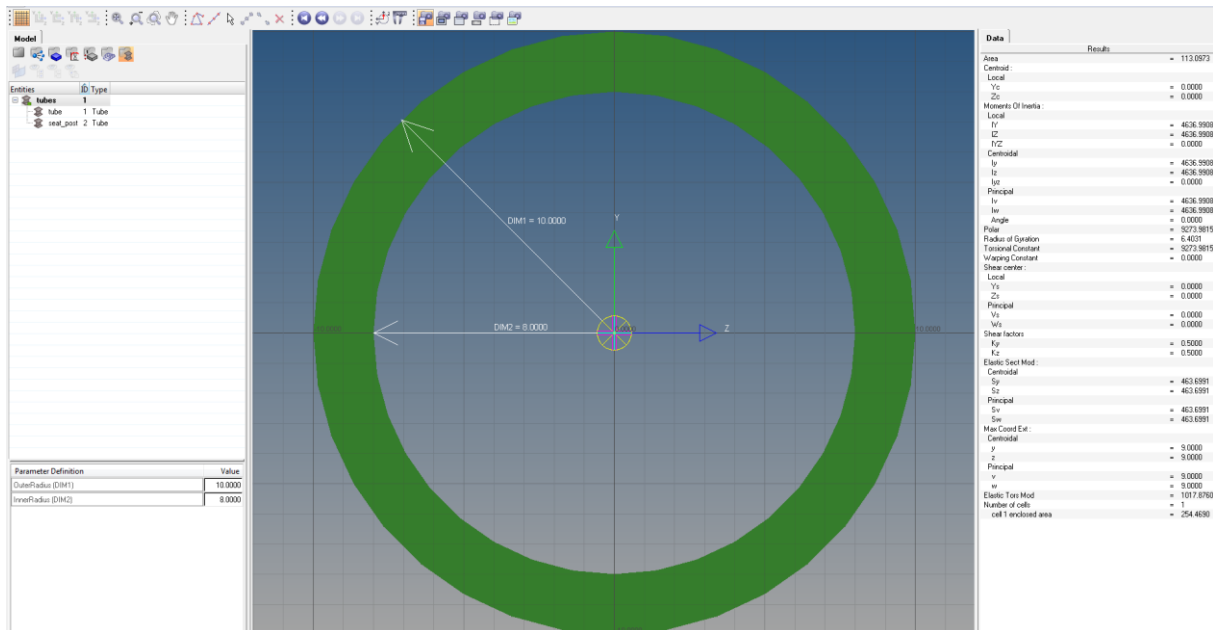
Define / create two beamsections (here named: tube and seat_post):

- tube: outer radius (DIM1) is 10 mm, inner radius (DIM2) is 8 mm
- seat_post: outer radius (DIM1) is 10 mm, inner radius (DIM2) is 8 mm

Both beamsections will be stored in a beamsection collector, here named "tubes".

Note, that all the relevant cross-sectional data such as area, moments of inertia are automatically calculated and listed in the side bar of HyperBeam. The beamsections will be referenced inside the corresponding Property collectors, later.

Exit HyperBeam by clicking the model view icon in the top left ⬜ corner. Next we define the material collector and property collector.

Create Material
Right click in the Model Browser: Create → Material
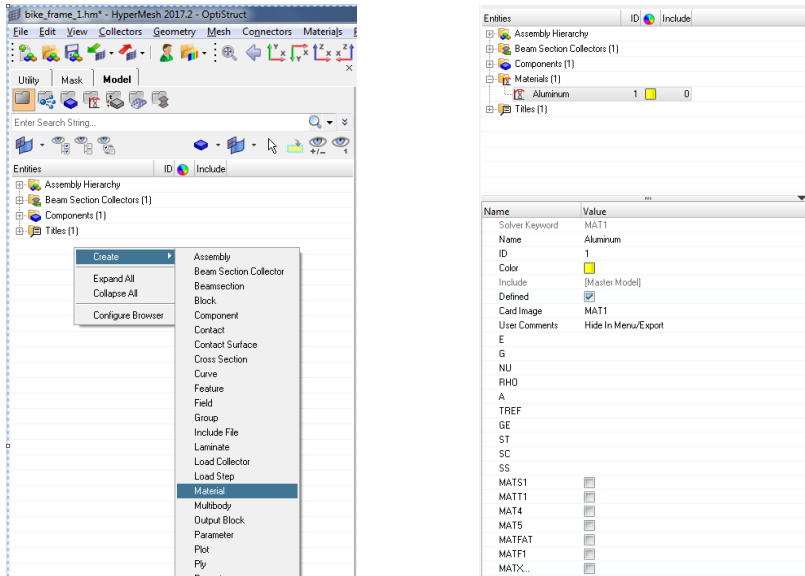
Enter the name (Aluminum) and choose as Card image MAT1. You can enter the below values directly here in the entity editor or you can right click on the material and click Card Edit option and enter the values there.
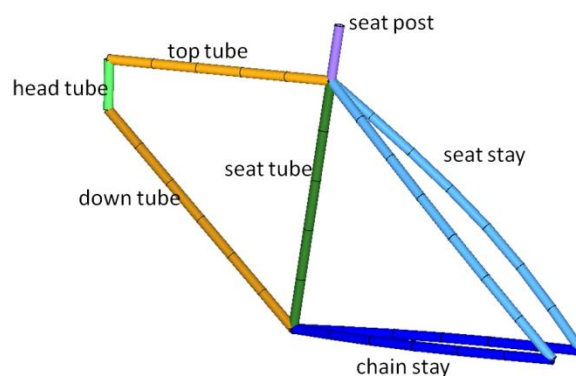In the material card enter the values below:

RHO = 2.7E-9 t/mm³   E = 70000 N/mm²   Nu = 0.3

The bikes seat post will be made of steel. Thus an additional material collector must be defined (using standard values):

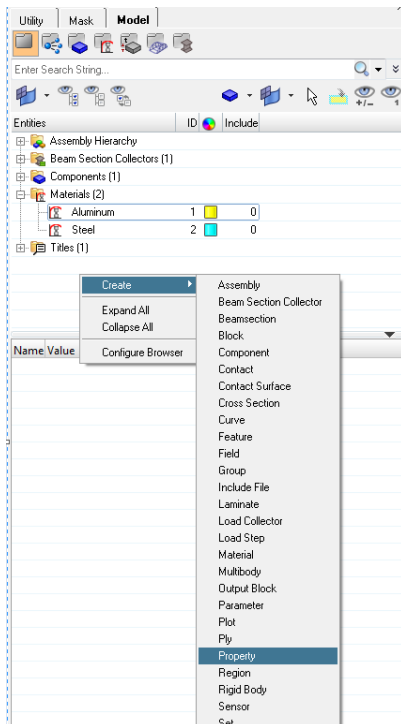RHO = 7.9E-9 t/mm³   E = 210000 N/mm²   Nu = 0.3

Create Property
Later on, the bicycle frame will be partitioned into 7 "domains" with different radiuses, respectively:

(Note: In the image the model is already meshed; meshing will be discussed further below).

Right click in the Model Browser: Create → Property and define its name e.g. "top_tube"

As Card image select PBEAML. Also select the appropriate material for top tube it is Aluminum.
Thus, the property collector contains the information about PBEAML (i.e. the properties of a beam element by cross-sectional dimensions) and the associated material.
"Inside" the property collector the beamsection of interest needs to be referenced. Click on "beamsec" to select the beamsection of interest here which would be "tube".
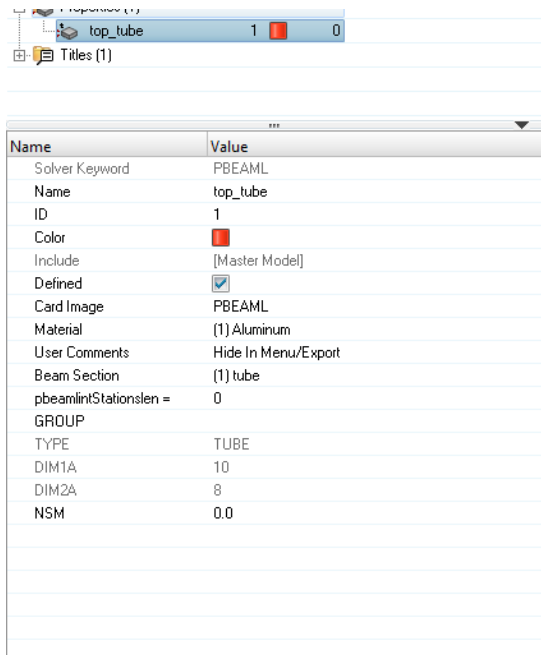
All the other property collectors are created in just the same way.

| Name of Property	Material	BeamSection
Head Tube	Aluminum	tube
Down Tube	Aluminum	tube
Seat Tube	Aluminum	tube
Seat Post	Steel	seat_post
Seat Stay	Aluminum	tube
Chain Stay	Aluminum	tube

Create Beam elements

There are several ways to model CBEAM elements in HyperMesh.
Here the Line Mesh subpanel is used.
Go to: Mesh → Create → Line Mesh
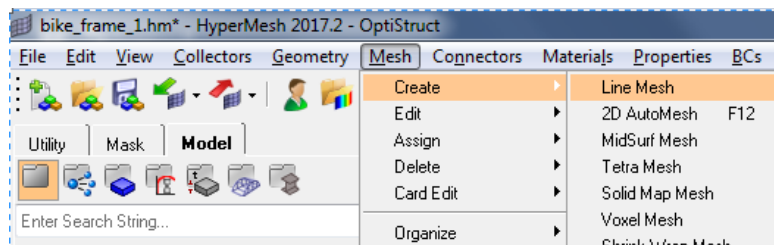
Depending on your very personal working process & philosophy you may mesh all lines in a single step (note: then all elements would be assigned the same property which then needs adjustments later on). Alternatively, the different "domains" are meshed individually (step by step) which allows to assign the correct property directly.
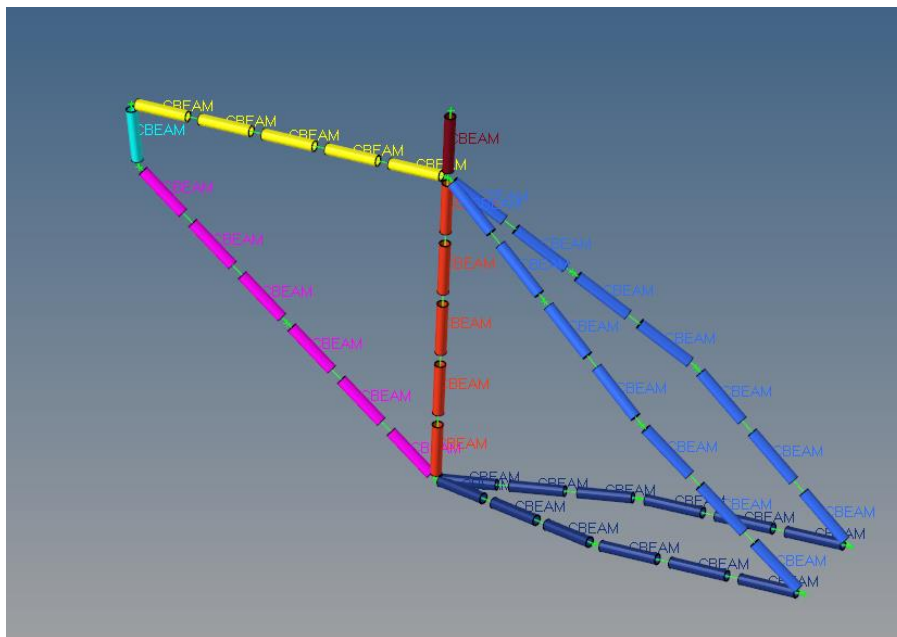
The frame is meshed with an element size 100 mm.
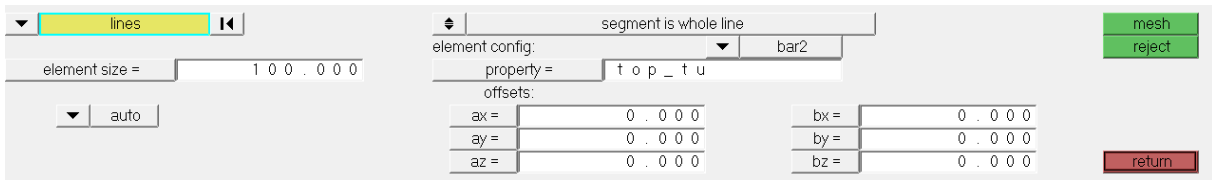
Changing the visualization mode of the element display to "1D Detailed Element Representation"
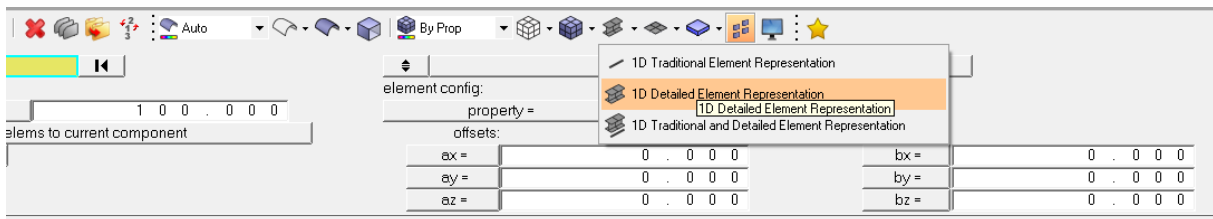

The frame may look like:

Somehow it looks like meshing of the head tube failed.

This problem may be related to a corrupt line (problem with geometry) or to a wrong orientation of the element cross-section. Reviewing (see panel below) the orientation of the elements cross section confirms the latter.

Thus, we manually adjust the local xy-plane of the 1D element by defining an orientation vector with respect to its global coordinates (see help menu for details).
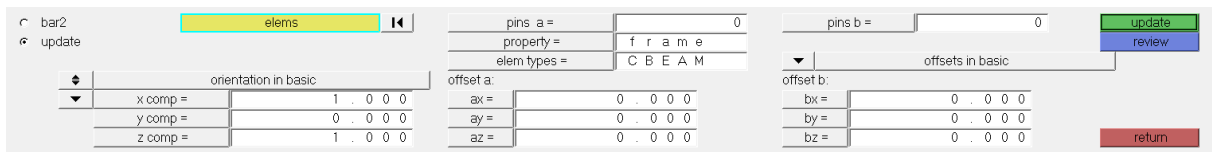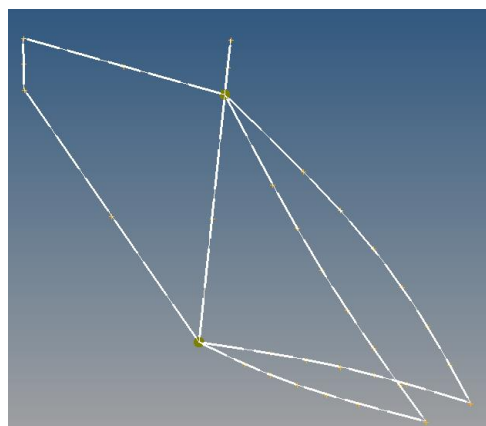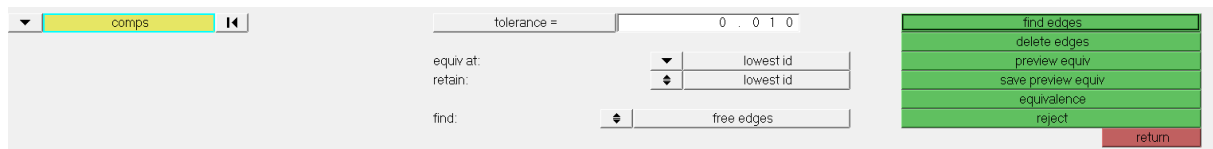

After updating / correcting the element orientation, the frame can be displayed as:
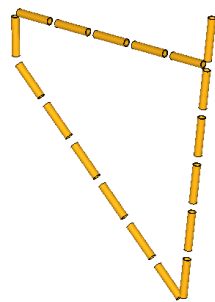
Before proceeding with the definition of loads and constraints, make sure that the mesh is compatible i.e. that all elements are connected to each other.
Tools→edges→preview equiv

The highlighted nodes indicate that the mesh is not merged there (i.e. location of multiple nodes). Making use of "equivalence" (with an appropriate "tolerance" value) helps to overcome this issue.
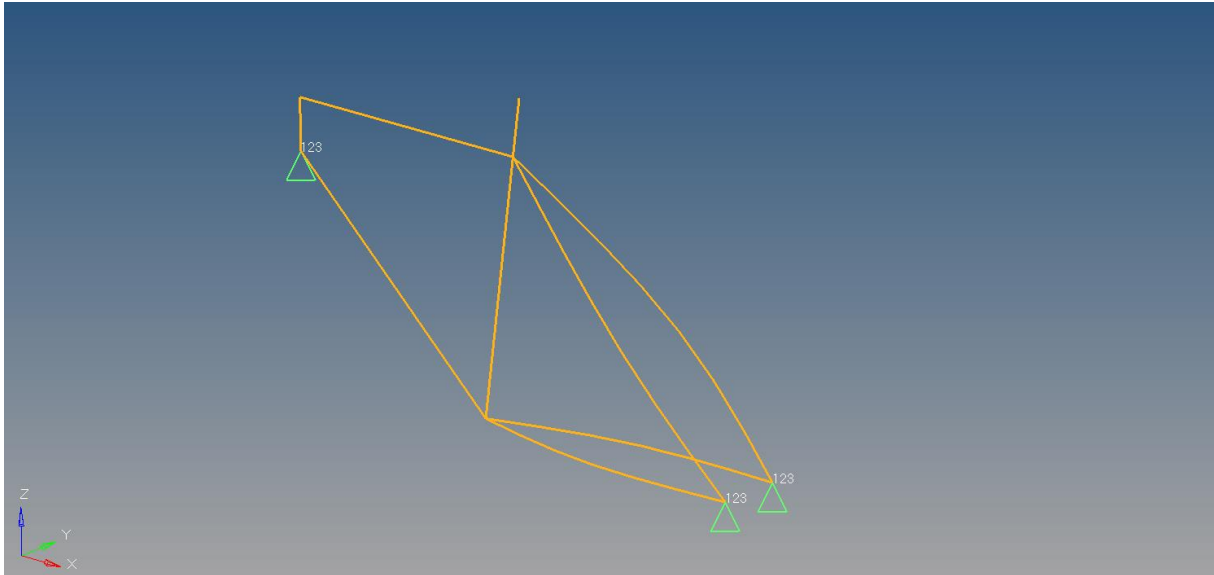
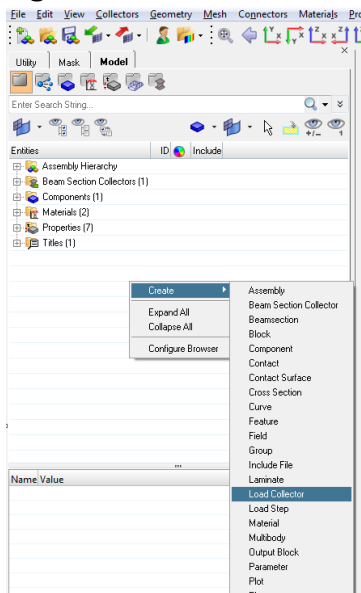Save the model as an HM binary file: bike_frame_2.hm

Part 3: Analysis set up

Load the hm file from the previous step: bike_frame_2.hm

Create constraints
The frame is (arbitrarily) constraint as shown below (translational dof's are assumed to be fixed)

Create a load collector named "constraints"; no Card image needed:
Right click in the Model Browser → create → Load Collector

Once the load collector exists, the constraints can be applied.
Go to BCs → Create → Constraints
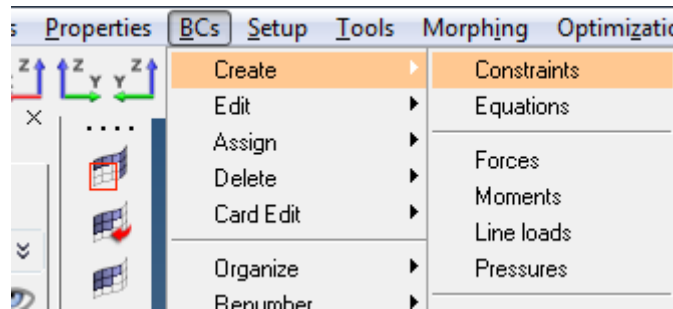
Create loads

We arbitrarily assume that the biker's weight is 120kg. In addition, the pedals are "loaded" by another200 N (acting in negative z-direction, respectively). Moreover, a safety factor of 2 is applied.
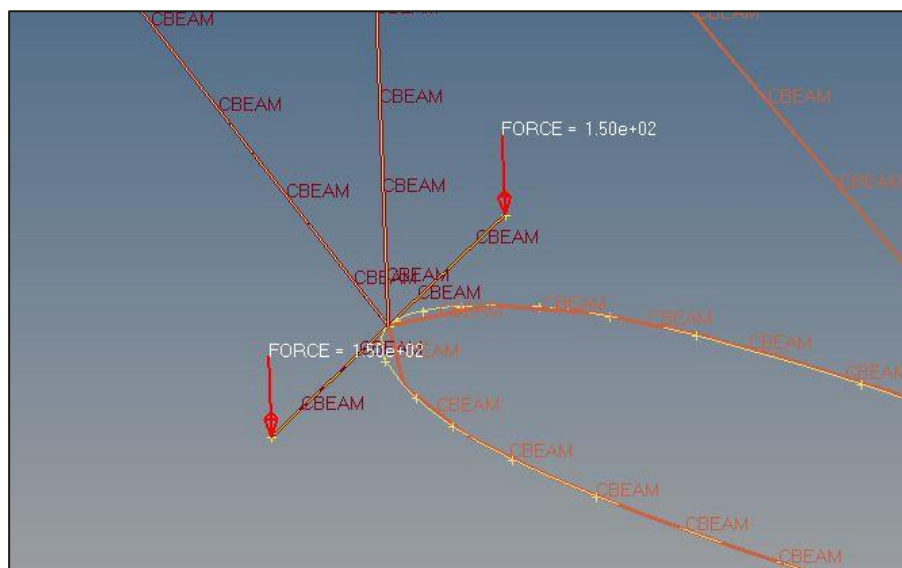
Create a load collector named "force"; no Card image needed.
Right click in the Model Browser → create → Load Collector
Enter the Force subpanel to create the load: BCs → Create → Forces

Altair
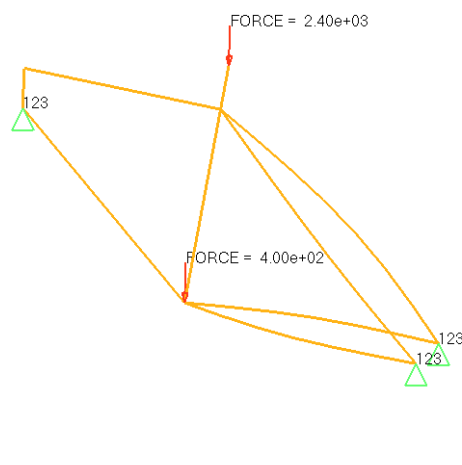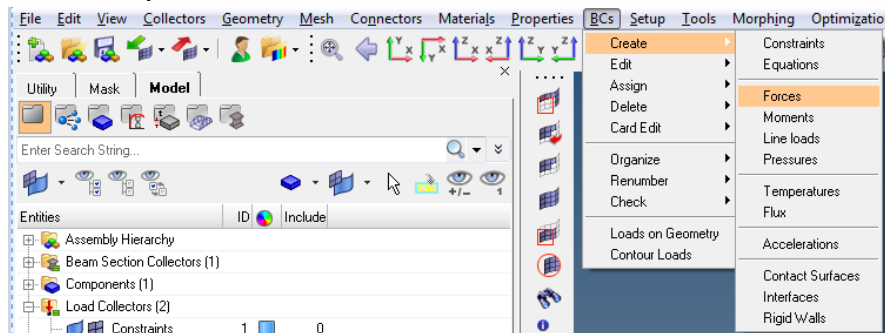
Of course, the way the force is applied to the "bearing" is overly simplified. In a somewhat more detailed study you may improve the model as depicted below …
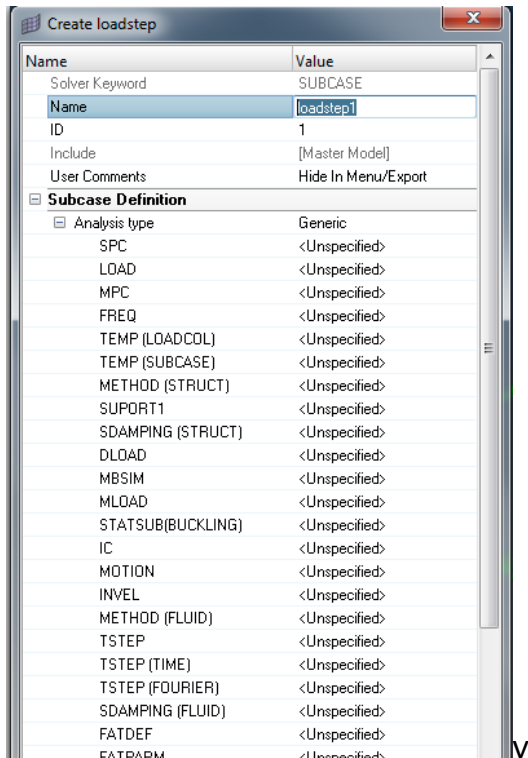

Create a LoadStep

In the LoadStep the forces and constraints from above are referenced i.e. these loads and constraints must be taken into account during the analysis.
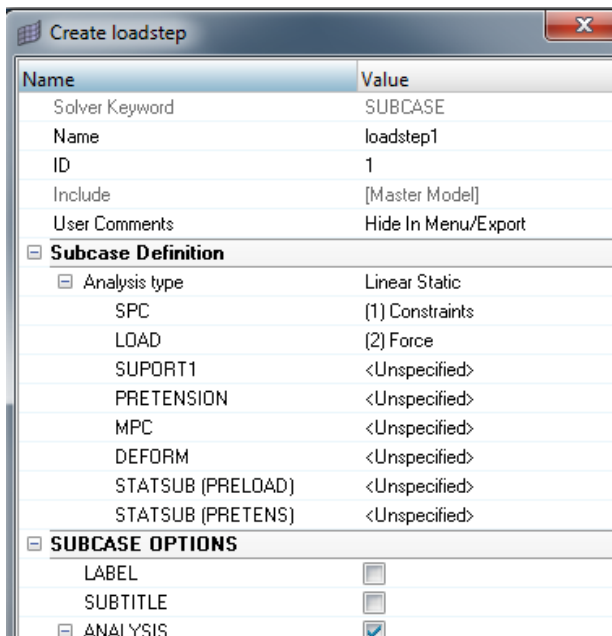Go to Set up → Create → Load Step
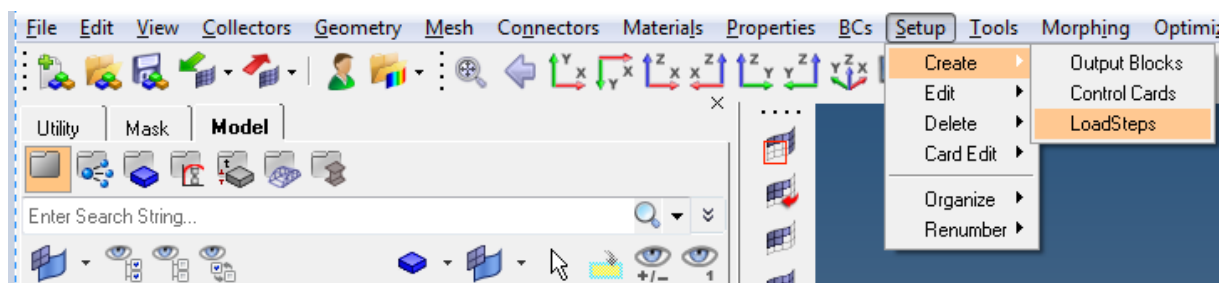
This will pop-up a new window

In this enter the desired name and select for type "linear static".
Check "Single Point Constraint (SPC)" and select the load collector containing the constraints.
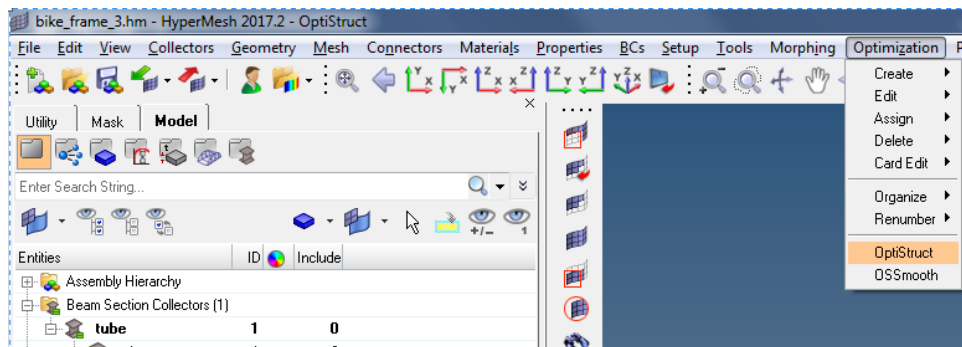Check "Load" and select the load collector containing the loads (forces).

The model-set is completed, the model browser in HyperMesh lists the following entities (note, the order of the entities may be different in your model)

Save your model as bike_frame_3.hm

Start the Analysis

Load the model bike_frame_3.hm and start the analysis in the OptiStruct subpanel

In the OptiStruct panel make sure that "run options" is set to "analysis" and "export" option is set to "all". Then run/start the analysis.
Activate "view out" for more information about the analysis run i.e. demanded memory, model information, warnings, errors etc.
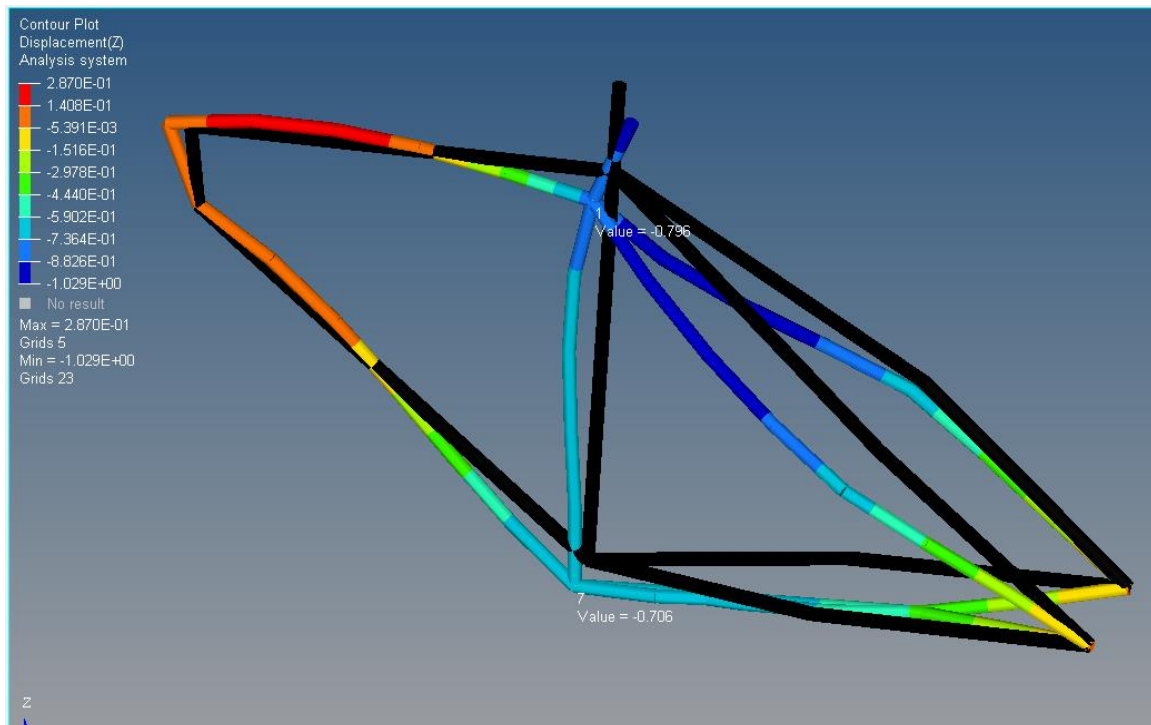
Postprocessing with HyperView

In the *.out file we can read that the mass of the "base" design is 1.43 kg

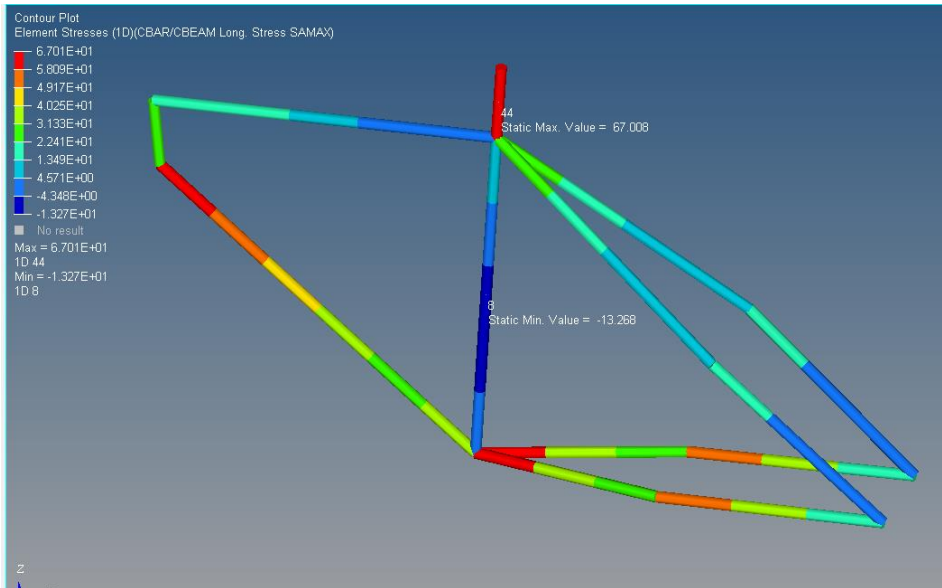Z-Displacement Contour Plot
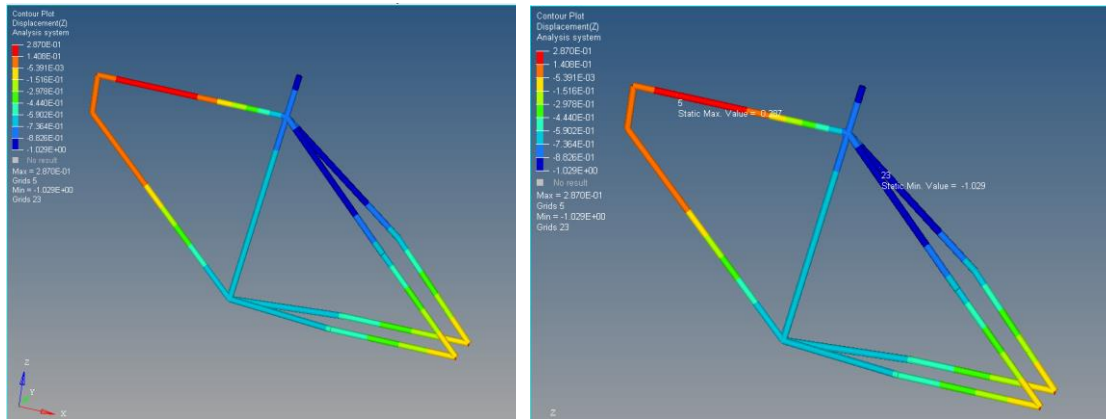
The z-displacement at the "bearing" (node 7) is -0.706 mm, at the top end of the "seat tube" (node 1) is -0.796 mm.
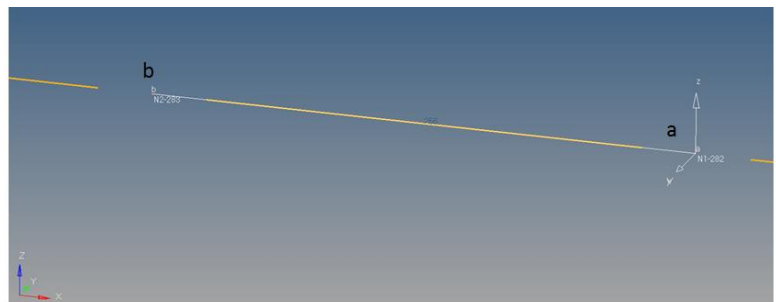

Contoured z-displacement plot (scaled by factor 50) with undeformed frame (in brown color)
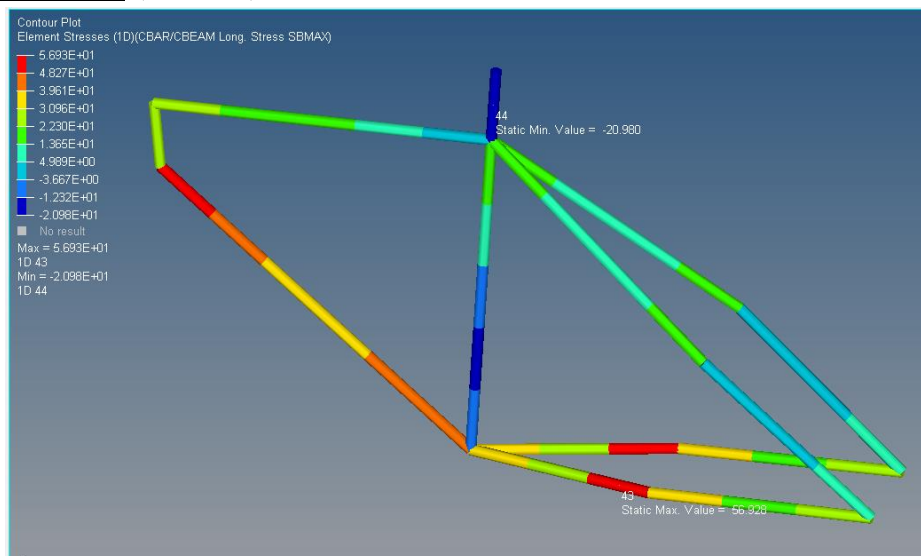
Element stresses (SAMAX)

Maximum element stress (SAMAX) is 67 MPa (N/mm2 (i.e. SAMAX = element stress at point A of the Beam-elements is plotted for the entire element)

Element stresses (SBMAX)

Maximum element stress (SBMAX) is 57 MPa.

Now, you may say: Job Done …