

Tutorial:

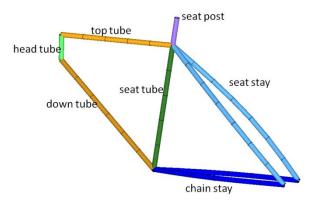
Conceptual Design of a Bicycle Frame

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

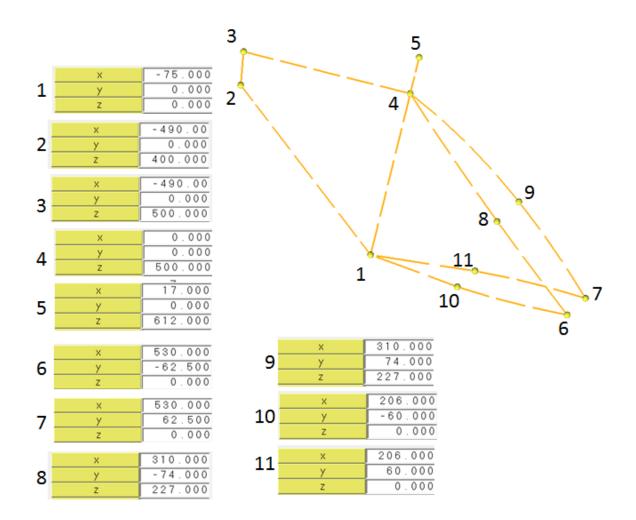
In this section we are going to construct a bicycle frame as shown below in HyperMesh



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

File Edit View Collectors	Geometry Mesh	Connecto	ors Materials	Properties BCs Setup	Тос
12 📂 🗟 🖌 🍊	Create	•	Nodes 💦 🕨	xYZ xYZ	
	Edit	•	Free Points 🕨		
Utility Mask Model	Assign	•	Fixed Points 🕨	🦚 On Geometry	
	Delete	•	Lines 🕨 🕨	🖊 Arc Center	
	Card Edit	•	Surfaces 🕨 🕨	1	
Enter Search String	Organize	•	Midsurfaces 🕨	/ Extract Parametric	
🚽 - 🐃 🖏	Renumber	• <u> </u>	Solids 🕨 🕨	🖊 Extract on Line	
	Check	•	Vectors	· · · · · · · · · ·	
Entities ID 😯 Include	Translate	•	Systems 🕨 🕨	Interpolate Nodes	
1	Rotate	E E		/ Interpolate on Line	
	Scale			Interpolate on Surface	
I	Reflect				
1	Project			X Intersect	
	Position	- + -		Temp Nodes	
1	Permute			romp reades	





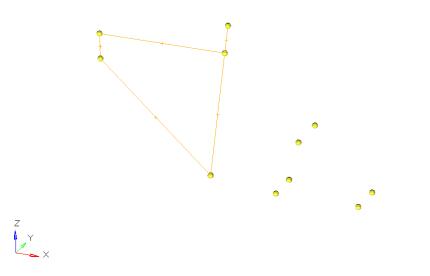
The coordinates of the bounding nodes are shown below.

Step 2: Build lines

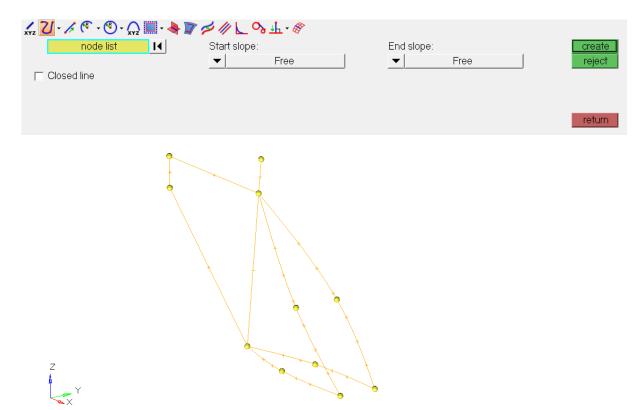
First, create a Component Collector (i.e. storage container of the lines / geometry), then start building lines with respect to the previously defined temporary nodes



<u>File Edit View Collectors</u>	Geometry Mesh	Connectors	Materia <u>l</u> s	<u>P</u> roperties	<u>B</u> Cs <u>S</u>	etup <u>T</u> ools
🐍 🐹 🗟 🖆 🍊	Create Edit		odes 🕨	t ^z y y ^z	Y 🔭 🚬	
Utility Mask Model	Assign Delete Card Edit Organize	► Fix ► Lin ► Su	ked Points 🕨	XYZ XYZ Linear N	odes	
Function ID 💊 Includ	Renumber Check	► ► Ve	olids	Standar 71 Smooth		
Endes 10 Titles (1)	Translate Rotate Scale	• Sy	vstems 🕨	Controlle		
	Reflect Project Position)))		Arc Cen		
	Permute	•		Are Thre	e Noder	



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 (\rightarrow bike_frame_1.hm).



Make sure the User Profile S OptiStruct is selected Load the hm file from the previous step: bike_frame_1.hm

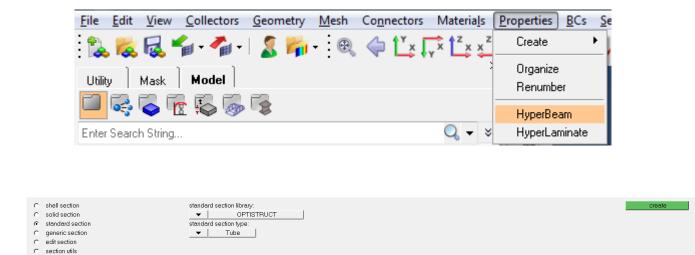
Part 2: 1D Beam element creation

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 noncoincident.

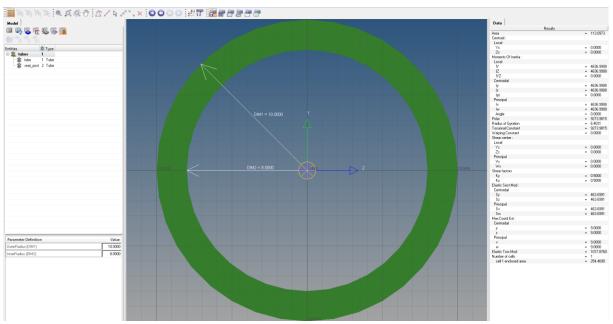
It is recommend to make use the predefined 1D element cross-sections available with the OptiStruct library.

The beams cross-section is best defined via HyperBeam. Go to Properties \rightarrow 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 Next we define the material collector and property collector.

corner.



Create Material

Right click in the Model Browser: Create \rightarrow Material

bike_frame_1.hm* - HyperMesh 20	17.2 - OptiStruct	Entities	ID 💽 Include
Eile Edit View Collectors Geor	netry <u>M</u> esh Co <u>n</u> nectors Materia <u>l</u> s [🕕 🔂 Assembly Hier	
		🕒 🙀 Beam Section	
🔊 😽 🖓 🗐 🖓 👘	📫 . : 🛷 📥 🗖 thu rà 🗂	🗄 🛜 Components (
Utility Mask Model	^	🖨 🙀 Materials (1)	
		Aluminum	n 1 0
📕 🗟 🖕 🕆 🖉 🦉		E III Titles (1)	
nter Search String	Q - ×		
• • • •	🗢 • 🎽 • 🔓 🏄 🖤 🥰		
	Notude		
- 💫 Assembly Hierarchy			
Beam Section Collectors (1)			
- Components (1)		Name	Value
- D Titles (1)		Solver Keyword	MAT1
3. dm (1992 [1]		Name	Aluminum
Create 🕨	Assembly	ID	1
	Beam Section Collector	Color	
Expand All	Beamsection	Include	[Master Model]
Collapse All	Block	Defined	
Configure Browser	Component	Card Image	MAT1
0011000000000	Contact	User Comments	Hide In Menu/Export
	Contact Surface	E	
	Cross Section	G	
	Curve	NU	
	Feature	BHO	
	Field	A	
	Group	TREF	
	Include File	GE	
	Laminate	ST	
	Load Collector	SC	
	Load Step	SS	
	Material	MATS1	
	Multibody	MATT1	
	Output Block	MAT4	
	Parameter	MAT5	
	Plot	MATFAT	
	Ply	MATF1	
	Fly	MATX	

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:

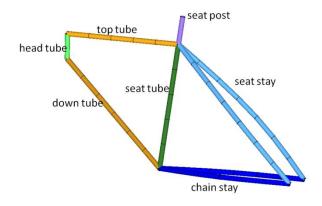
RHO = $2.7E-9 \text{ t/mm}^3$ E = 70000 N/mm² Nu = 0.3

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

 $RHO = 7.9E-9 t/mm^3$ $E = 210000 N/mm^2$ Nu = 0.3

Create Property

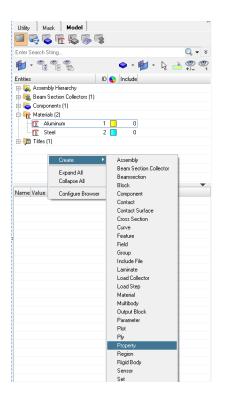
Later on, the bicycle frame will be partitioned into 7 "domains" with different radiuses, respectively:





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

Right click in the Model Browser: Create \rightarrow 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".



ame	Value
Solver Keyword	PBEAML
Name	top_tube
ID	1
Color	
Include	[Master Model]
Defined	
Card Image	PBEAML
Material	(1) Aluminum
User Comments	Hide In Menu/Export
Beam Section	(1) tube
pbeamlintStationslen =	0
GROUP	
TYPE	TUBE
DIM1A	10
DIM2A	8
NSM	0.0

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 \rightarrow Create \rightarrow Line Mesh

bike_frame_1.hm* - HyperMesh 2017.2 -	OptiStruct	
<u>File Edit View Collectors G</u> eometry	Mesh Connectors	Materia <u>l</u> s <u>P</u> roperties <u>B</u> Cs
🗽 🕵 🕵 🖆 🍊 - 🔏 🎁	Create	Line Mesh
	Edit	 2D AutoMesh F12
Utility Mask Model	Assign	 MidSurf Mesh
T R C R C C C	Delete	 Tetra Mesh
	Card Edit	 Solid Map Mesh
Enter Search String	Organize	Voxel Mesh
·	Organize	Shrink Wran Mesh

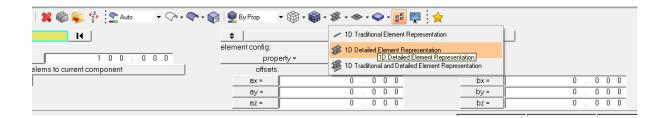


Depending on your very personal working process & philosophy you may mesh all lines in a single step (note: then all elements would be assigned the same property which then needs adjustments later on). Alternatively, the different "domains" are meshed individually (step by step) which allows to assign the correct property directly.

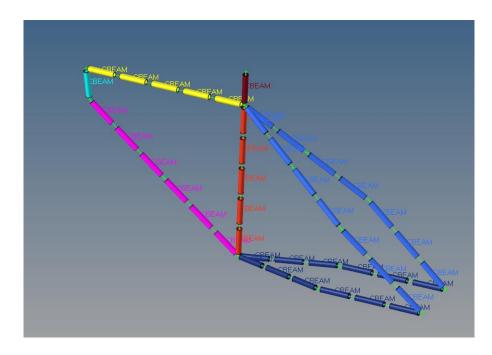
The frame is meshed with an element size 100 mm.

✓ lines I4	♦ seg	ment is whole line		mesh
	element config:	▼ bar2		reject
element size = 1 0 0 . 0	0 0 property = t	op_tu		
	offsets:			
▼ auto	9X =	bx =	0.000	
	ay =	0.000 by=	0.000	
	az =	0.000 bz =	0.000	return

Changing the visualization mode of the element display to "1D Detailed Element Representation"



The frame may look like:





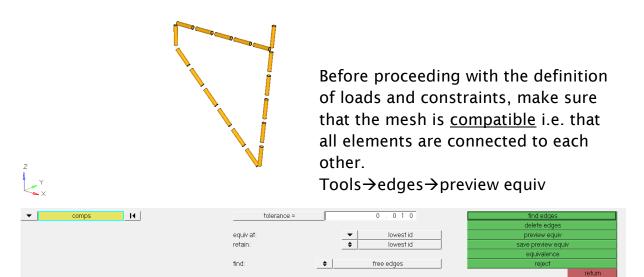
Somehow it looks like meshing of the head tube failed.

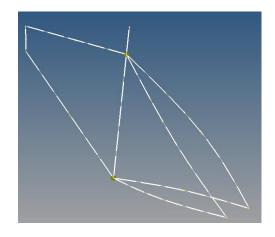
This problem may be related to a corrupt line (problem with geometry) or to a wrong orientation of the element cross-section. Reviewing (see panel below) the orientation of the elements cross section confirms the latter.

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).

c ba	ar2		elems	I	pi	nsa=	0	Ī	pin	sb=	0	update
📀 up	odate				pro	operty =	frame	_				review
					eler	n types =	CBEAM		•	offsets in ba	віс	
	\$	orie	entation in basic		offset a:				offset b:			_
	-	x comp =	1.0	0 0	ax =		0.000		bx =	0.0	0 0	
		y comp =	0.0	0 0	ay =		0.000		by =	0.0	0 0	
		z comp =	1.0	0 0	az =		0.000		bz =	0.0	0 0	return

After updating / correcting the element orientation, the frame can be displayed as:







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

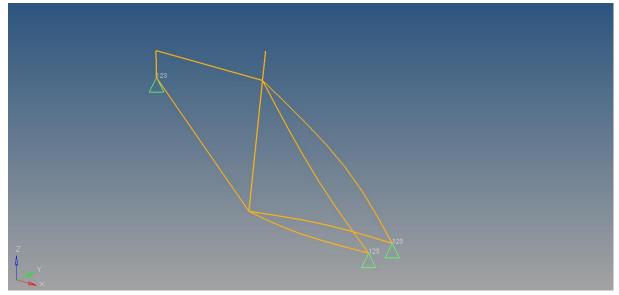


Part 3: Analysis set up

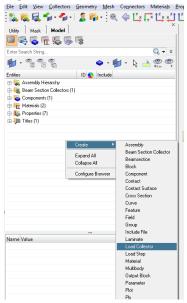
Load the hm file from the previous step: bike_frame_2.hm

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 \rightarrow create \rightarrow Load Collector





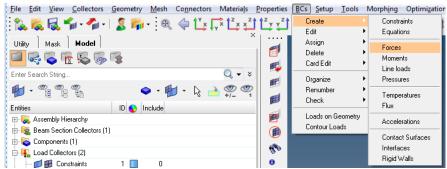
Once the load collector exists, the constraints can be applied. Go to BCs \rightarrow Create \rightarrow Constraints

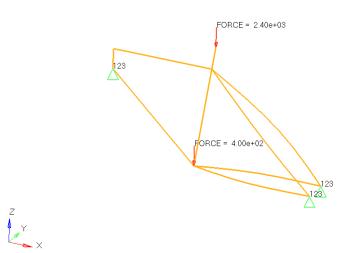
	<u>Properties</u>	<u>S</u> etup	<u>T</u> ools Mo	rph <u>i</u> ng (Optimi <u>z</u> atio		
	z z z y z z	Create	•	Constrain			
	×	Edit	•	Equation	s		
	1	Assign Delete Card Edit	+ + +	Forces Moments Line load			
	* •	Organize	•	Pressure	s		
		Benumber	F				
C create C update relative size		50.000	🔽 label constraints	dof1 dof2 dof3			create create/edit reject
conste	int value		☐ fixed	│ dof4 │ dof5 │ dof6 │ dof6	= = 	0 . 0 0 0 0 . 0 0 0 0 . 0 0 0	review

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.

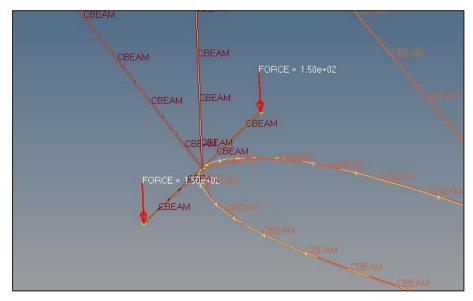
Create a load collector named "force"; no Card image needed. Right click in the Model Browser \rightarrow create \rightarrow Load Collector Enter the Force subpanel to create the load: BCs \rightarrow Create \rightarrow Forces





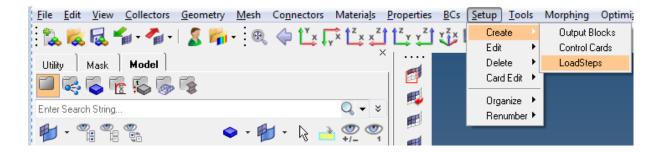


Of course, the way the force is applied to the "bearing" is overly simplified. In a somewhat more detailed study you may improve the model as depicted below ...



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 \rightarrow Create \rightarrow Load Step





Create loadstep		x
Name	Value	_ <u>^</u>
Solver Keyword	SUBCASE	
Name	loadstep1	
ID	1	
Include	[Master Model]	
User Comments	Hide In Menu/Export	
Subcase Definition		
🖃 Analysis type	Generic	
SPC	<unspecified></unspecified>	
LOAD	<unspecified></unspecified>	
MPC	<unspecified></unspecified>	
FREQ	<unspecified></unspecified>	
TEMP (LOADCOL)	<unspecified></unspecified>	=
TEMP (SUBCASE)	<unspecified></unspecified>	
METHOD (STRUCT)	<unspecified></unspecified>	
SUPORT1	<unspecified></unspecified>	
SDAMPING (STRUCT)	<unspecified></unspecified>	
DLOAD	<unspecified></unspecified>	
MBSIM	<unspecified></unspecified>	
MLOAD	<unspecified></unspecified>	
STATSUB(BUCKLING)	<unspecified></unspecified>	
IC	<unspecified></unspecified>	
MOTION	<unspecified></unspecified>	
INVEL	<unspecified></unspecified>	
METHOD (FLUID)	<unspecified></unspecified>	
TSTEP	<unspecified></unspecified>	
TSTEP (TIME)	<unspecified></unspecified>	
TSTEP (FOURIER)	<unspecified></unspecified>	
SDAMPING (FLUID)	<unspecified></unspecified>	
FATDEF	<unspecified></unspecified>	
FATPARM	/Unspecifieds	

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).

Create loadstep	×
Name	Value
Solver Keyword	SUBCASE
Name	loadstep1
ID	1
Include	[Master Model]
User Comments	Hide In Menu/Export
Subcase Definition	
🖃 Analysis type	Linear Static
SPC	(1) Constraints
LOAD	(2) Force
SUPORT1	<unspecified></unspecified>
PRETENSION	<unspecified></unspecified>
MPC	<unspecified></unspecified>
DEFORM	<unspecified></unspecified>
STATSUB (PRELOAD)	<unspecified></unspecified>
STATSUB (PRETENS)	<unspecified></unspecified>
■ SUBCASE OPTIONS	
LABEL	
SUBTITLE	
ANALYSIS	



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)

Entities		ID 📀	Include
🖽 💫 Assemt	oly Hierarchy		
🗄 🙀 Beam 9	Section Collectors (1)	l i	
🖻 🤱 tu	be	1	0
- 1	tube	1	0
	seat_post	2	0
🖨 😂 Compo			
- 🖉 🖪	auto1	1 📘	0
🗄 👯 Load C			
	Constraints	1 📘	0
	Force	2 📕	0
📄 📑 Load S	teps (1)		
loa 📥 🚽	adstep1	1	0
🖻 🙀 Materia	ıls (2)		
😰 Ali	uminum	1 🗌	0
- 😰 St	eel	2 📘	0
🗄 💫 Propert	ies (7)		
🎲 top	o_tube	1 🗌	0
ta he	ad_tube	2 📘	0
🏷 da	wn_tube	3 📘	0
🏷 se	at_tube	4 🚺	0
to se	at_post	5 📗	0
🏷 se	at_stay	6 🚺	0
🛁 😓 ch	iain_stay	7 🔳	0
🗄 📁 Titles (1	l)		

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

<u>File Edit View Collectors</u>	<u>G</u> eometry	<u>M</u> esh C	o <u>n</u> nectors	Materia <u>l</u> s	<u>P</u> roperties	<u>B</u> Cs <u>S</u>	etup <u>T</u> ools	Morph <u>i</u> ng	Optimization	P
🕵 ቘ 🗟 🖆 - 🐔 -	2 🎁	- 🔍 <	¢ Ľ₄ [Y 🐉 📮	, QQ	+ 🖑	Create Edit	1
Utility Mask Model									Assign	
🗖 🗟 🖉 🖗 🖏	2								Delete Card Edit	21
Enter Search String	*			Q ×	- 🗾					<u> </u>
_		<i>.</i>			Ø				Organize	•
🔁 • 🐂 🐂 🖏	•	> - 📂	- 🗟 📄) 🖤 🤍 +/_ 🖤					Renumber	<u>•</u>
Entities	ID 💽 I	nclude			I				OptiStruct	
🖽 🛜 Assembly Hierarchy	•								OSSmooth	
🖶 🙀 Beam Section Collectors (1)									
🗄 🤹 tube	1	0								
🗢	-	0			60					
inputfile; < < 0 1 7	. 1	icvcl			i k o f		3.fem		ave as	OptiStru
inputine: < < 0 1 7	/ I . D	i cyci	e r r	ame/b	I K U _ T	ane.	e m	s	ave as	
export options:	1	run optic	ns: optimiza	tion	memo	ry options: memor	y default	J		HyperVi

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.

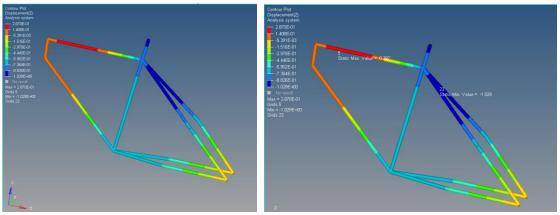
Activate "view out" for more information about the analysis run i.e. demanded memory, model information, warnings, errors etc.



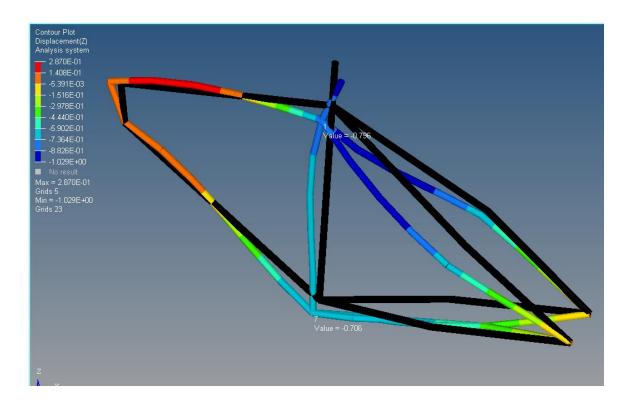
Postprocessing with HyperView

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

Z-Displacement Contour Plot



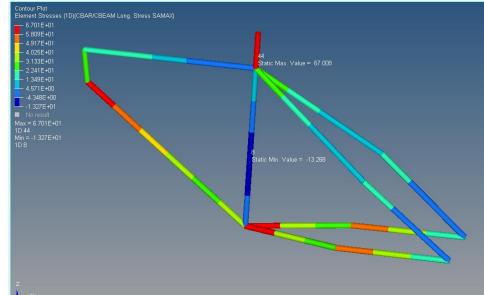
The z-displacement at the "bearing" (node 7) is -0.706 mm, at the top end of the "seat tube" (node 1) is -0.796 mm.



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

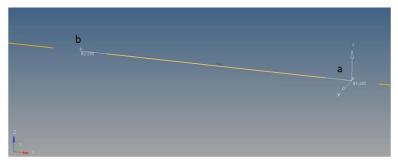


Element stresses (SAMAX)

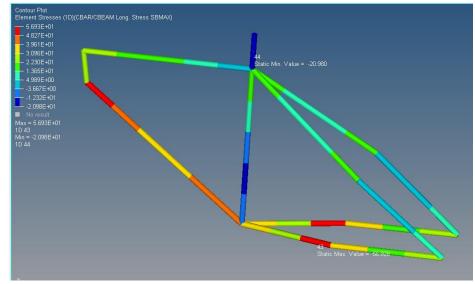


Maximum_element stress (SAMAX) is 67 MPa (N/mm2 (i.e. SAMAX = element

stress at point A of the Beamelements is plotted for the entire element)



Element stresses (SBMAX)



Maximum element stress (SBMAX) is 57 MPa.

Now, you may say: Job Done ...