

FEM Calculation of a Crane Hook as a single Part and as an Assembly



(C) 2020 by Ing.Büro HTA-Software Germany-Rheinau

> <u>www.femcad.de</u> <u>www.fem-infos.com</u>

# Part 18: Crane Hook with MEANS-LITE and Crane Hook Assembly with MEANS HIGH-END

**Crane Hook Part:** The Crane Hook is a single part with a point load of 15 KN. The FEM Model consists 92 343 Tetrahedral elements and the FEM calculation can therefore be done with **MEANS-LITE up to 100,000 elements**.



#### **Material Datas:**

The Crane Hook is loaded with a hook load of 15 KN and consists of the tempered steel 42CrMo4 with the following material datas:

Tensile Strength Rm = 1100 MPa Young's modulus E = 210,000 MPa Yield Strength Re = 900 MPa

Crane Hook Assembly: The Assembly consists of Hook and Bolt and is loaded at the bolt end with 15 KN. The Assembly is calculated using a Contact Analysis and consists 350,000 TET4 elements and 350 MPC elements and can only be calculated with MEANS HIGH-END and Add On CONTACT.



# **Crane hook with MEANS-LITE**

First an FEM mesh with over 99,000 tetrahedral elements is generated from the crane hook CAD model and is calculated with the light version MEANS-LITE which is limited until 100,000 elements and nodes.



Select the "Files" tab and select "New" to create a new FEM project.

Select "3D Tetrahedral Meshing with STL, STEP or IGES for the following formats:

- STL this 3D model consists of a triangular outer shell for the 3D Mesh generation, this flexibel format can also be imported and exported. STL Files with holes or gaps, it can also be repaired before meshing.
- STEP today it is the Standard format, note that no CAD assemblies but only single parts can be meshed. Assemblies can be combined via the "Boolean operations" or with "Screw Models" into one part.
- IGES like STEP format but is not as common anymore

Use the "Browser" button to select the "hook.step" STEP file and click on menu "Start Mesh Generator No. 2 with CAD File" to display it in the mesh generator.

🚽 Mesh (	Generation			-		×
Directory:	C:\projekte\a		Browse	r		
۲	STEP	O IGES	O STL / AST (ASCII)		Default	
		Ū.	0		Help	
esticator.s esticator_ esticator_ esticator_ esticator hook_step Hook_Eye -STL Opt	step cf.stp hook 1.stp hook 2.stp hook 2_cf.stp e_bolt_assemb imization and f timize STL File	l <b>y.STEP</b> Repair • before starting Mesh Gen on with all tests	eration	_		
Mesh Ge	ener <mark>ator</mark> No. 2	with OpenGL-Interface				
Star	t Mesh Genera	tor No.2 with CAD File	with error messages		Help	
Mesh Ga	enerator No. 3 Start Mesh (	with automatic repair funct Generator No. 3	ion Mesh Density: 0		Help	
		Ca	ncel			

The model can now be seen in the mesh generator and can be rotated as required.



First select the "Mesh" menu and the "General" tab and select the mesh density "fine".

<u>G</u> eneral	<u>M</u> esh Size	STL Charts	<u>O</u> ptimizer	<u>I</u> nsider	<u>D</u> ebug
	Mes	h <mark>g</mark> ranularity :	fine		
	First Step :		very coarse		
		Last Step :	coarse moder	ate	
	Dr	int Messages :	fine		_1
		Parallel me	very fi user d	ne efined	

#### Mesh size grading

Then select the "Mesh Size" tab and generate with the mesh-size-grading = 0.225 and the main menu "Generate Mesh" an FEM mesh with 99214 tetrahedral elements so that the number of elements is just below the MEANS-Lite limit of 100,000 elements.



#### Export Mesh

.

.

After generating the mesh, the FEM mesh with the name "test.fem" must be exported to MEANS V11. Select the menu "File" and "Export Mesh" and save the mesh "test.fem" in the specified debug mesh path.

File Geometry Mesh View	v Refinement	
Load Geometry <i><g> Save Geometry</g></i>		
Recent Files	•	
Load Mesh <i><m></m></i>	•	
Recent Meshes	•	
Save Mesh <s><m< td=""><td>· .</td><td></td></m<></s>	· .	
Merge Mesh		
Import Mesh		Select the menu "File" and "Export Mesh"
Export Mesh		and save the mesh "test.fem" in the
Export Filetype	•	specified debug mesh path.

#### **Generate Surface Model**

After exporting "test.fem", MEANS V11 is started automatically and first creates the Surface Model so that surfaces, edges and nodes can be selected for loads, boundary conditions or element groups.

		• ц х
) normal	🔘 viele	🔘 sehr viele
ilung= 0.91	Hinwei	is
◯ gesamt m	iit V8 ⊖ m	it Modellbereich
Neuee Flär	hanmodall array	
	normal ilung= 0.91	normal Oviele      ilung= 0.91     Hinwe      Ogesamt mit V8 Om

## Crane hook clampled fixed

The crane hook is firmly clamped at the top. To do this, select the "Edit FEM project" tab and "Boundary conditions".

Choose "Select Surface" and "Create RBs" button and click on surface 6 to display it in the select box. The clamping is created there with "Generate".

Ansicht Netzgenerier	rung FEM-Projekt bearbeiten	FEM-Analyse	Ergebnisauswertung	Training
1. Knotenbelastung +	Randbedingungen	edingungen 🛛 👻 dingungen darstellen Flächen-Modus aktivi	Elementgruppen ert - Fläche= 6	Materialdaten
🖳 Randbedingungen		- 🗆 X		
Anzahl Randbedingungen A Wert der Randbedingung: Freiheitsgrad sperren: (Achsen-Farben: SCHWAR2 Selectieren	aktuell: 99 Neu 1E-10 in X-Richtung in Z-Ric in Y-Richtung Einspan Z: X-Achse; BLAU: Y-Achse; ROT: Z-Achse O Rechteck aufspanne icken alle angezeigten Knot	htung inung ie) n ten wählen aces wählen	MALLINE.	
Randbedingungen darste	llen: 00034 normal	~		
Cancel Edit	itor RBs erze	ugen		
	RBs lõso	:hen	1	
Surface 6 CLE EDI Knoten Pläche EBemente Kanten CANCEL ERZEUG	× ETE AR IT n 1 SEN			

#### Hook loaded with a Point Load

The hook is loaded with 15 KN in the Y direction. First, a selectable node area must be created in the Node-Modus by spanning a rectangle over the load. Now the model is rotated and zoomed so that the node is not covered and can be clicked on.

Knotenbereich erzeugen Bitte mit der Maus ein Rechteck aufspannen oder einzelner Knoten anklicken! Anzahl Knotenbereich = 0 Neu Knotenbereich aus Knotenbereich erzeugen Help Selection Rechteck aufspannen O Knoten picken O Koordinatenbereich Cancel Knotenbereich erzeugen	Anzahl Eckknoten = 1413
Bitte mit der Maus ein Rechteck aufspannen oder einzelner Knoten anklicken!         Anzahl Knotenbereich = 0       Neu         Knotenbereich aus Knotenbereich erzeugen       Help         Selection       Methods (Knoten picken)         Rechteck aufspannen       Knoten picken         Cancel       Knotenbereich erzeugen	
Anzahl Knotenbereich = 0 Neu Knotenbereich aus Knotenbereich erzeugen Help Selection © Rechteck aufspannen O Knoten picken O Koordinatenbereich Cancel Knotenbereich erzeugen	Kanten von: 1 his: 93
Knotenbereich aus Knotenbereich erzeugen     Help      Selection     Rechteck aufspannen     Knoten picken     Knotenbereich erzeugen      Cancel     Knotenbereich erzeugen	Knoten anzeigen
Knotenbereich aus Knotenbereich erzeugen Help      Selection     Rechteck aufspannen O Knoten picken O Koordinatenbereich     Cancel Knotenbereich erzeugen	Knotenbereich erzeug
Selection  Rechteck aufspannen Knoten picken Cancel Knotenbereich erzeugen	Knotenbereich erzeuge
Rechteck aufspannen     Knoten picken     Koordinatenbereich     Cancel     Knotenbereich erzeugen	Flächenknoten
Cancel Knotenbereich erzeugen	Flächen-Randknoten
Cancel Knotenbereich erzeugen	Knotenbereich lösche
Cancel Knotenbereich erzeugen	Knotenbereich löscher
Cancel Knotenbereich erzeugen	Knotenbereich änder Koordinaten-Faktor
	Knoten: 100112 EI
	X: 2.150728
	7: -15.56149
	Elementgruppen nume
	Lastwerte anzeigen
	Knoten-Size editieren:
	Größe= .01
	Size= normal

Now you can click on node "Edit FEM project" and "Point Load" of node 329 with a load of -15000 N in the Y direction

	Ansicht Netzgenerierung	g FEM-Projekt bearbeiten FEM-Analyse Ergebnisauswertung Training
1.	Knotenbelastung	1. Randbedingungen ·
1.	Knotenbelastung	ndbedingungen 🗹 Randbedingungen darstellen Elementgruppen Materialdaten Editor Temper
2.	Linienbelastung	aktueller Knoten 329: X-Koord.= 1.305765; Y-Koord.=-15.44374; Z-Koord.= 0
3.	Flächenbelastung	
4.	Gravitationsbelastung	
5.	Fliehkraftbelastung	
6.	Temperaturbelastung	
7.	Ungleichmäßige Radiallast	
8.	Lastfall einstellen	🖼 Knotenlast erzeugen — 🗆 🗙
9.	Editor	
		Aktueller Lastfall: 1 +
		Anzahl Lastwerte: 1 Neu
		(Einheit z.B. in N)
		Frahaterrad
		Carlottagidu. O X-Richtung O Z-Richtung
		Y-Richtung
		Selektion:
		O Rächenmodus O Rechteck aufspannen
		einzelne Knoten anklicken         O alle angezeigten Knoten
		Koordinatenbereich definieren     alle angezeigten Surfaces
		Knotenlasten darstellen: 0004 normal V
		Cancel Editor Belastung erzeugen
		Belastung löschen

#### Model scaled

So that the Hook is comparable to another FEM model, the node coordinates should be multiplied by a factor of 0.25.



Select the tab "Edit FEM Project" and "Node coordinates" and multiply the coordinates by a coordinate factor of "0.25". The node check can be carried out.

 	Knoten	-Modus aktiviert			1. 2. 3.	Elementgruppen Elementknoten Knotenkoordinaten	تعالی المراجع ا
Nr.    Nr.	en X-Koordinaten  0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Y-Koordinaten           380           380           210           210           210           190           190           190           75.78242           87.63221           75.78242           90.6013           380           380           380           380	Z-Koordinaten 26 -26 26 26 37.5 -37.5 -37.5 -37.5 -37.5 0 0 0 0 -14.43947 0 14.43947 0 24.02087 18.09373 9.949769 -245041 en-Faktor	× ^	3. 4. 5. 6. 7. 7. 8.	Knotenkoordinaten Materialdaten Randbedingungen Belastungen Formoptimierung Löschen	Koordinaten-Faktor Faktor setzen : Imultiplizieren addieren addieren ersetzen Achsen vertauschen X-Wete mit Y-Wete vertauschen X-Wete mit Z-Wete vertauschen Y-Wete mit Z-Wete vertauschen Y-Koordinaten III Faktor verändem X-Koordinaten III Faktor veränden Inur die angezeigten Knoten im Knotenmodus verwenden von Knotenpunkt: 1 bis Knotenpunkt: 20123 Koordinatenfaktor: 0.25 Nullpunktsverschiebung durch Knotenpunkt: 1 Koordinaten mit Faktor verändem Verformungen mit Faktor zu den Koordinaten addieren: Verformunge-Faktor: 1
							CANCEL

## Postprocessing

After the FEM analysis the node stresses can be evaluated with the "Postprocessing" tab.

Frankrissa sinladan	
O Verformungen auswerten	Lastfall: 1 🗸
Knotenspannungen gemittelt	🔿 Auflagerkräfte auswerten
O Elementspannungen ungemittelt	O Ergebnisdatei anzeigen
Legende	
Raster-Genauigkeit:	Verformungsfaktor/Wertebereich
[ [	Legende und Farben einstellen
1 3 4	Knotenwerte picken, suchen, sichem
Ergebnis-Komponente wählen	
v.Mises-Vergleichsspannung Normalspannung Sigma x Normalspannung Sigma y	
Can Nomalspannung Sigma z Schubspannung Tau xy Schubspannung Tau xz	

The max. v.Mises-Stress is 637.17 MPa.



## **Connect the Crane Hook Assembly**

The crane hook assembly consists of the previous Hook and Bolt, the latter is meshing with "fine" with 26 254 tetrahedradral elements and saved as Bolt.FEM.



### Merge Hook and Bolt Mesh to a single Part

The following two FEM meshes are now combined with an FEM-Merge:

Mesh 1: Hook.FEM with 21 948 nodes and 99 214 tetrahedral elements

Mesh 2: Bolt.FEM with 6100 nodes and 26 254 tetrahedral elements

First load the larger FEM model Hook.FEM into MEANS V11, then add the second mesh using the "File" and "FEM-Merge" tabs.



The hook and bolt are now combined to a single FEM model, but unfortunately the two parts do not touch, so the bolt must be moved by -6.5 mm without scaling or -1.62 m with scaling in the Y direction.

Use the "View" and "Node Modus" tabs to set the node mode menu and show all nodes of element group 1. Select "Coordinate factor" and add the Y coordinates of the displayed ones Nodes with a coordinate factor of "-6.5" mm.



#### **MPC-Contact**

The two parts are now combined in a single FEM file, but they must also be connected at the point of contact via MPC elements (MPC elements are discussed in detail in Chapter 10 - MPC Analysis with MEANS V11).

Use the "View" and "Node Modus" and "Create a Range of Nodes" tabs to create a rectangle over the two contact areas.

Double-click on the blue marking rectangle to display all nodes in this area.

🖳 Knotenbereich erzeugen			
Bitte mit der Maus ein Rechteck aufspar Anzahl Knotenbereich = 0	nnen oder einzelner Knoten an	klicken!	
Selection  Rechteck aufspannen  Knotenbereich aus Knotenbe	n picken O Koordi	natenbereich	
Cancel	Knotenbereich erzeug	en	

# Then select the "File" and "MPC Contacts" tab and start the contact calculation to automatically generate 297 MPC elements with the node area above.

Calculate M	PC-Contacts	Step 1		acts automatically		
Stop and check t	he contact points		MPC Contacts automatically			
lumber of Contacts = 84 are	saved in the mpc.con File!		MPC-Contacts with two surfaces     MPC-Contacts with a Range of Node			
Show Range	e of Nodes A	Step 2	MIPCCONLACES WILL A Hange of Hit			
Take over Range of Nodes A			Contact-Angle	42		
Range of Nodes A: 77			Contact-Distan	ice: () <= 0.1	<= 0	
Show Range of Nodes B		Step 3	max. MPC-Len	nath: 8.492378		
Take over Ra	nge of Nodes B				]	
Range of Nodes B: 26			Surface A	1		
Save Contact-Nodes	Load Contact-Nodes	Step 4	Surface B:	2		
Create MP	C-Contacts	Step 5				
MPC-Elemente prüfen	Delete MPC-Contacts					

# Hook clamping

The hook is firmly clamped at the top, select the "Edit FEM project" tab and "Boundary conditions" to clamp surface 12 in the X, Y and Z directions.



#### Load the bolt end with surface load

At the end, the bolt is loaded with -15,000 N in Y direction. Select the tab "Edit FEM Project" and "Surface Load" to load Surface 6 with -15000 N vertical to the surface.

As of the July 2020 Update, the surface load can either be entered in MPa or in N, with the latter, the value is automatically calculated from the selected areas and divided by the N value:



211

# Postprocessing

After the FEM analysis with the quick solver, the node stresses are evaluated.



The max. v.Mises Stress of the Assembly is 644 MPa and almost exactly matches the v.Mises Stress of the part with 637 MPa.

### Bolt evaluation

Select "Edit FEM project" and "Element groups" and only show element group 1. Switch back to postprocessing to display the 1st principal Stresses S1 and the 3rd principle Stresses S3 with the maximal tensile and minimal compressive stresses.

