Part 28: Heavy duty trailer with a weight and inertial force

An excavator with a dead weight of 15t is transported with a heavy-duty trailer. How high are the deformations and stresses when braking hard on the beam girders of S355 structural steel.

15t excavator with heavy-duty trailer

Dimensions for the center of gravity SP and for the middle part



Weight and Inertial Force

Emergency braking in the direction of travel with a forward mass inertia of 2g and a downward weight of 1g.



FEM mesh generation

First the CAD assembly is converted into a part since only a part and not an assembly can be meshed and export it in STEP format



Start MEANS V12 via the desktop icon and select "New" to use the 3D mesh generator "Mesh Generator MEANSMES V2 for complex Structures" available from August 1st, 2022 and select the desired STEP file.

	Files	View	Mesh Ge	neration	Edit FEM-	Project	FEM-A	Analysis	F
New			6	FEM-Merg MPC-Cont	e Impo act Expo	ort: STL ort: DXF	- - -		1. (
INEW	Tari	Open 19	Save 13	Unit		CAD	rau	1 12	
		NEW PROJ	ECT		107	- C	x c		
		 3D Mesh 3D Mesh Mesh Gei 2D/3D Bi 3D Shel I 	Generator MI Generator GM nerator MEAN eam Model wi Model with a C	EANSMES V1 ISH (STEP) ISMES V2 for th Line-Modus Container Mest	(STEP, IGE comlex Stru 1 Geneneral	:S, STL) Ictures	Install		

Select the "Start mesh generator with CAD file" menu to display the STEP model in the 3D mesh generator in a new frame window.

Directory:	C:\projekte\		Browser			
(STEP		○ STL / AST (ASCII)		Defa	ult
anhaeng anhaeng	jerbaugruppe.st jerpart.step	ер				
carframe car_traile FRAC T/ FRAC T/ p1.step	step er_part.step ANK.STEP ANK_part.step					
STLO	otimization and I	Repair				
SILO	TI -Datei vorbei	r ontimieren lassen (emn	foblen z B. hei Absturz in NGSol	vel		
S	TL-Datei vorhei	r optimieren lassen (emp Tests	fohlen z.B. beï Absturz in NGSol	ve)		
Start 3	TL-Datei vorhei	r optimieren lassen (emp Fests tor	fohlen z.B. bei Absturz in NGSol	ve)		
Start 3	TL-Datei vorhei	r optimieren lassen (emp Fests tor Start 3D Tetral	fohlen z.B. bei Absturz in NGSol nedral Mesh Generator	ve)		
Start 3	TL-Datei vorhei	r optimieren lassen (emp Fests tor Start 3D Tetral	fohlen z.B. bei Absturz in NGSol nedral Mesh Generator	ve)		

Since the trailer consists of very thin U-profiles and a moderate FEM mesh of approx. 450,000 tetrahedrons is to be generated, the following setting must first be set with the Mesh/Meshing Options menu:

General	Mesh Size	STI Charts	Optimizer	Debug		
generar		ore enores	opennizer	Debug		
		max m	esh-size 10	00 🚖		
		min m	esh-size 0			
		mesh-size	grading 1.0	0		
		C 1				
	mesh-size	file:		1		
	-			Brow	wse	
CSC						
CSG me	sh-size		0.0 FI			
			0.2 Elen	nents per cu	irvature rad	lius
			0.2 Elem	nents per ed	lge	
	ch cizo					
	511-512E	E.				
			J.2 M SI	IL - chart di	stance	
			0.2 🗹 ST	IL - line len	gth	
			3.0 Ø ST	I /IGES/STE	P - close e	dae
			0.2 <u>!</u> _ SI	L - surface	curvature	
			0.2! 🗌 ST	IL - edge an	igle	
				-	-	
		Ľ		re - sunace	mesh curv	
	C CTI D	ecalc mesh	size for sur	face optimi	zation	
-	M SIL - K					
	M 21L - K	Ca	Ic New H			

Select "Generate Mesh" to create a mesh with 166054 nodes and 457400 elements.



Export the FEM mesh either in Neutral format with *.FEM or in Abaqus format with *.INP. Then switch back to MEANS V12 and open this FEM mesh with the "Open" menu. The surface model will only be needed later for the middle section.



So that the dimensions can be adopted on page 1, a "Node of Range" must first be created in the lower right corner with the icon from the View tab in order to be able to read corner node 87, then carry out a zero point shift with the "Coordinate Factor" menu.

💀 Koordinaten-Faktor — 🗆 🗙	Rächen Knoten Linien
Faktor setzen	Anzahl Eckknoten = 166054
	 Kanten 🗸
	von: 1
	bis: 58827
Achsen vertauschen	Knoten anzeigen
O X-Werte mit Y-Werte vertauschen	Knotenbereich erzeuger
X-Werte mit Z-Werte vertauschen	Knotenbereich erzeugen
O Y-Werte mit Z-Werte vertauschen	Flächenknoten
Koordinaten mit Faktor verändem	Flächen-Randknoten
X-Koordinaten Y-Koordinaten Z-Koordinaten	Knotenbereich löschen
	Knotenbereich löschen
🗹 nur die angezeigten Knoten im Knotenmodus verwenden	Knotenbereich ändern
von Knotenpunkt: 1	Koordinaten-Faktor
bis Knotenpunkt: 166054	Knoten: 87 EDI
Koordinatenfaktor:	X: -3082
	Y: -175
Nullpunktsverschiebung durch Knotenpunkt: 87	Z: 712.4
Kaanlinstan nii Esktar unrändam	Knoten numerieren
Roordinaten mit raktor verandem	Elemente numerieren
	Elementgruppen numerie
Verformungen mit Faktor zu den Koordinaten addieren:	Lastwerte anzeigen N
Verformungs-Faktor: 1	Knoten-Size editieren:
CANCEL	×

📮 — 🗆 X

MEANS version

Unfortunately, this model is too large for the user with a **MEANS LITE** version with up to 400,000 nodes and elements. In the CAD the user would have to try to decimate the trailer in non-relevant parts or switch to a higher MEANS version such as **MEANS DESIGN** up to 999,000 nodes and elements or **MEANS HIGH END** with unlimited nodes and elements.

Create Element Group 2

Only the most heavily loaded center section from X= 1392 mm to X= 4208 mm is now used. To do this, element group 2 is created from the middle part and the rest of the trailer is hidden. Select the "Edit FEM project" tab and the "Element groups" menu and create element group 2 using a coordinate range.

Datei Ansicht Netzgenerierung FEM-Projekt bearbeiten FEM-Analyse Ergebnisauswertung Train	ning
1. Knotenbelastung - 1. Randbedingungen -	
Belastungen 🗹 Belastungen darstellen Randbedingungen 🗹 Randbedingungen darstellen Elementgruppen Material	Idat
Fiachen-Modus aktivient - Fiache= 42	
EG = 2 X	
□ ON EG=1	
ON EG= 2	
Elementgruppe erzeugen: 2	
O EG mit aufgespannten Rechteck und einer Tiefe erzeugen	
C EG mit allen angezeigten Knoten erzeugen	
Classe Classe	_
ON EG=6 EG=6 Elementgruppe mit Koordinatenbereich erzeugen	_
☑ ON EG=7	
Für neue Farbe auf Farbrahmen klicken	
✓ als Drahtgitter sichtbar von: -100000 bis: 100000	_
Refresh Hidden-Line	
Gruppen 1 - 7	
Reue Elementgruppen erzeugen	
Anzahl Elementgruppen andem: 2	
Elementgruppe andem von: auf:	
Andem	
Cancel	

von X:	1391.5	bis X:	4208.5	
von Y:	-75.5	bis Y:	244	
von Z:	-398	bis Z:	2123	

Define the coordinate range with the following limits values plus a tolerance of 0.5:

Then first hide element group 1 and create the surface model of element group 2 with the "View" tab and Menü "Surface Modus".



Create a Surface Load

The weight load of 15t can be generated as a surface load. Select the tab "Edit FEM project and "Surface Load" and enter the value of the loading with "150000" N and select menu "Create Load" and click on the surface 33.

Eelas	Province Ansicht Netzgen Ansicht Netzgen Ansicht Netzgen Ansicht Netzgen Belastungen darsteller	erierung FEM-Projekt bearbeiter	FEM-Analyse bedingungen + edingungen darstellen	Ergebnisauswertung	Training	6. Belastungen	Temperatur
			Infozeile				Γ <u>ν</u>
Ī	🛃 Flächenlast erzeugen	– 🗆 X					
	Aktueller Lastfall: 1 Anzahl Lastwerte: 9259	- +		_			
	Wert der Flächenlast: .2208386	○ N/mm²				<u> </u>	
	Wert Belastung in N: 150000	Info					
	Freiheitsgrad: O X-Richtung	O Z-Bichtung				<	
	O Y-Richtung	 Senkrecht zur Fläche 			\ge		
	Flächenmodus	O Rechteck aufspannen		~~~~			
_	O einzelne Knoten anklicken	🔘 alle angezeigten Knoten			and the second s		~ //
	Rächenlast-Wert (N/mm ²) aus Bela	astung (N) FL-Farbe:				N	N
	Cancel Editor	Belastung erzeugen Belastung löschen	×				
				Ì		\checkmark	
	Y	and the second se					
	×						

Create Boundary Conditions

The trailer is clamped fixed on the lower surfaces. Select the "Edit FEM Project" tab and create the Boundary Condtions on the Surface 38 and 64.



Material Data

The material steel S355 is always preset with a modulus of elasticity of 210 000 MPa and a Poisson number of 0.3 like all other types of steel.

FEM Analysis

After saving the FEM model under any name, select the "FEM Analysis" tab and the "Statics" menu to use the Quick Solver to calculate the deformations and stresses.

O MEANS-Solver Quick-Solver			
Schritt 1: FEM-S	oolver starten		
Schritt 2: Postproo	cessing starten		
Schritt 3: Nach	verfeinerung		
wählen	Ergebnisgrößen eins	stellen	
	Schritt 1: FEM-S Schritt 2: Postproc Schritt 3: Nach	Schritt 1: FEM-Solver starten Schritt 2: Postprocessing starten Schritt 3: Nachverfeinerung wählen Ergebnisgrößen ein:	Schritt 1: FEM-Solver starten Schritt 2: Postprocessing starten Schritt 3: Nachverfeinerung wählen Ergebnisgrößen einstellen

Postprocessing

The displacements and stresses are evaluated graphically with the "Postprocessing" tab.

Displacements

max. Displacements in Y direction = -4.0713 mm



v.Mises-Stresses

The max. v.Mises stress is 245 MPa and is below the Yield Strength of 355 N/mm².



Generate mass inertia forces

The mass inertia forces act in the center of gravity SP and must be evenly projected onto the center part of the trailer using MPC elements. The downward Y inertia forces are 1g with FGy = -150,000 N and the forward X inertia forces are 2g with FGx = 300,000 N.



First, the surface load is shown with the View tab and Node-Modus menu, as well as the side menus "Load" and "Show Nodes" to create an "Range of Nodes" of the load.



With the "Edit FEM-Project" tab and the "Bending and Torsional Moment" menu



all nodes of the "Range of Nodes" are listed. With "Inner Nodes" only the 4798 inside nodes are shown. The internal nodes must then be reduced to 100 nodes using the "Reduce" menu, since only 100 nodes can be calculated with an MP node in the FEM solver.

Enter a "Point Load" in Y direction of "-150000 N" and a MP distance in Y direction = 1000 mm and select menu "Step 2 - Calculate MP Coordinates" and menu "Step 3 - Create a Point Load without Lever Arm" to generate a Point Load.



A Point Load of -150000 N is generated at the MP node, which is connected to the 100 selected nodes via MPC elements.



However, the MPC elements can also be created individually in the Line-Modus by connecting the 100 nodes with lines to the MP node.

Repeat the input and use a second MP node to create load case 2 with a Point Load of "300,000 N" in the X direction and carry out an FEM analysis with the Quick Solver.

The result is load case 2 with superimposed load cases 1+2 with the following v.Mises stress contour:

