Part 27: CAD Assembly with Node Imprint calculate

Part 1: Steel-Rubber Buffer

The CAD Assembly of a Steel-Bracket with an E-Modulus = 210 000 MPa and a Rubber-Buffer with an E-Modulus = 15 MPa cannot be meshed in one step because of the different materials.

With MEANS V12, such structures can be meshed with different materials by first separately meshing all components with uniform materials and reuniting them with a node imprint of the contacting mesh.

In this way, even complex welded constructions or thin container structures can be meshed up to a certain degree of difficulty.

Unfortunately, numerically unstable tetrahedra with a negative Jacobi determinant can also arise when generating the node imprint, but these can be corrected, displayed or deleted in MEANS V12.



Rubber Buffer generated separatly with GMSH:



Steel Bracket generated separatly with NETGEN:



Steel Bracket with the Node Imprint of the Rubber Buffer:



FEM mesh combined with Steel Bracket and Rubber Buffer



Generate Node Imprint

Either the Node Imprint of the rubber buffer can be inserted into the mesh of the steel bracket or the Node Imprint of the steel bracket can be inserted into the mesh of the rubber buffer. A FEM file with all node coordinates or a faster Add List with a range of nodes can be used to generate the Node Imprint.

Create the Node Imprint of the rubber buffer

First load the FEM mesh of the steel bracket and select the "Mesh generation" and "Local Refinement" tabs as well as menu "Step 1: Create a new Point-List and Surface-List of the actual FEM-Mesh" to save the nodes of the actual mesh. Then select menu "Load a Point-List from a FEM-File" to load the coordinates of the rubber buffer. Then select the menu "Generate with the additional Point-List" to add the Add-List to the steel bracket mesh. The refined FEM mesh shown above is obtained.

step 2: Create a Range of Nodes	Step 3: Create Add-List
Step 4: Generate with Add-List	Select Add-List: Actual Add-List Center-Point of TET-Volume Center-Point of TET-Volume
Generate with the additional Point-List	from Node: 51298 until Node: 67135
air and Refine Tools	
Load a FEM mesh from ELE-File	Refine all Elements to 8x TET Mesh Refine TET Mesh with V11

Download the Video-MP4, FEM-, INP- and FRD-File for the Steel-Rubber-Buffer

Postprocessing

LASTFALL= 1

With Register "Postprocessing" and the Icon and stresses can be evaluated.

Max. Displacements in X Direction = 0.912 mm

Max. v.Mises-Stresses = 6.51 MPa



the results of displacements

Part 2: Steel Angle with Plastic Block

A CAD assembly consisting of a Steel Angle with an E-Modulus = 210 000 MPa that was embedded in a Plastic Block with an E-Modulus = 1200 MPa cannot be meshed in one step because of the different materials.



The Steel Angle and Plastic Block must first be meshed separately with GMSH:



FEM-Mesh Plastic Block from above and below:



Create a Range of Nodes of the Steel Angle

Load the FEM mesh of the Steel Angle and create a Range of Nodes of all surfaces that contact the Plastic Block with the Register "View" and "Node-Modus" as well as the menu "Surface Nodes" except the upper and lower ones.



Then you create an Add List with the Register "3D Mesh Generators" and "Local Refinement" as well as menu "Save Add-List" or "Create an additional Point-List".

💋 Local Refinement		J-Meshes, Refine, Delete.	Jacobi-Determinante	Vessel Gene	
3D Mesh Generation 🕞 2D Mesh Ge	eneration 🗔	Manipulate Meshes	r₃ Check Meshes	r⊊ Spec	
3D-Mesh-Generation					
Barris and					
Mesh Parameters Refining Jacobi-Test Info	0\$				
0-10	Distantia		FEM Mark		
Step 1: Ch	eate a new Point-List a	ind Surrace-List of the actua	I FEIVI-IVIESN		
Create and refine a Add-List					
Step 2: Cmate a Paper of Nodes	Stan 2	Crosto Add List	Center-Points of TET-Edge	es	
Step 2. Create a hange of hodes	Step 5.	Create Add-Ust	Center-Points of TET-Surfa	aces	
Step 4: Generate with Add-List	Select Add-List:	Actual Add-List 🗸 🗸	Center-Point of TET-Volum	ie	
		Actual Add-List			
Create an existing Add-List or load a Add-list f	rom a FEM-File	Save Add-List			
		Split Add-List			
Create an additional Point-List	Load a Point	-List from a FEM File	Load a Point-List from a No	de-File	
	from Node:	until	Node:		
Generate with the additional Point-List					
Generate with the additional Point-List					
Generate with the additional Point-List Repair and Refine Tools					

Load the FEM mesh of the Plastic Block. Select in the same dialog box menu "Step 1: Create a new Point-List and Surface-List of the actual FEM-Mesh" and then select the menu "Load a Point-List from a Node-File" to load the Add-List of the Steel Angle and check it on the model.

Then select the menu "Generate with the additional Point-List" to create a Node Imprint of the Steel Angle in the Plastic Block.

Step 1. Cit	sale a new round use and Sunace Las of the actual right-mean
Create and refine a Add-List	
Step 2: Create a Range of Nodes	Step 3: Create Add-List
Step 4: Generate with Add-List	Select Add-List: Actual Add-List Center-Points of TET-Volume
Create an existing Add-List or load a Add-list fr	rom a FEM-File
Create an additional Point-List	Load a Point-List from a FEM File Load a Point-List from a Node-File
Generate with the additional Point-List	from Node: 18113 until Node: 20296
Descioned Defect Table	
Repair and Refine Tools	
Load a FEM mesh from ELE-File	Refine all Elements to 8x TET Mesh Refine TET Mesh with V11
Load a FEM mesh from ELE-File	Refine all Elements to 8x TET Mesh Refine TET Mesh with V11
Load a FEM mesh from ELE-File	Refine all Elements to &x TET Mesh Refine TET Mesh with V11
Load a FEM mesh from ELE-File	Refine all Elements to & TET Mesh Refine TET Mesh with V11
Load a FEM mesh from ELE-File	Refine all Elements to &x TET Mesh Refine TET Mesh with V11
Load a FEM mesh from ELE-File	Refine all Elements to &x TET Mesh Refine TET Mesh with V11 rel OK
Load a FEM mesh from ELE-File	Refine all Elements to &x TET Mesh Refine TET Mesh with V11 rel OK
Load a FEM mesh from ELE-File	Refine all Elements to &x TET Mesh Refine TET Mesh with V11 rel OK
Load a FEM mesh from ELE-File	Refine all Elements to &x TET Mesh Refine TET Mesh with V11
Load a FEM mesh from ELE-File	Refine all Elements to &x TET Mesh Refine TET Mesh with V11
Load a FEM mesh from ELE-File	Refine all Elements to &x TET Mesh Refine TET Mesh with V11
Load a FEM mesh from ELE-File Cance	Refine all Elements to &x TET Mesh Refine TET Mesh with V11
Load a FEM mesh from ELE-File Canc	Refine all Elements to &x TET Mesh Refine TET Mesh with V11 rel OK



Then the following FEM mesh is obtained with the Node Imprint of the Steel Angle:

This is followed by an FEM Merge of the Steel Angle with Register "File" and menu "FEM-Merge" as well as a Node Check with Register "Mesh Generation" and menu "Check Node Numbering" to delete the 2202 node overlay.



Create a Point Load and Boundary Conditions

The Steel Angle is loaded with -500 N in Y Direction. Select the Register "Edit FEM-Project" and menu "Point Load" and load the upper surface with -1 N. Then enter a load value of -3.56 N with "Editor" which is calculated from 500/149.

9	Files	View	Mesh Generation	Edit FEM-Project	FEM-Analysis	Postpro	ocessing	Training		
F 🖕	1. Po	int Load	- D	→ 1. Bc	oundary-Condition:	-			6. Loads	
oads	Sho	w Loads	Boundary	-Conditions 🗹 Sho	ow Boundary-Condi	tions E	lement-Gro	oups Material-Datas	Editor	
						Info Line				
					1111111	11111	mun	1		
					++++++++	****	HINN	ł.		
)		an -
		the same							1	
	7					-			المرتجنين	
								<u></u>	F. Willing	
	1									1
	1								- +1.5041.501015	
	-								。 标准的 化	
		Belastunge	n		- 🗆	×			· Latta	
		Nr.	Knoten	FHG	Wert	^	🖳 Las	tfall	— 🗆 X	
	•	1	8	2	-3.56					
		2	9	2	-3.56	-11	Alet	ueller Lastfall: 1		
		3	15	2	-3.56	_				
		4	16	2	-3.56	-11		Faktor= -3.56		
	_	5	117	2	-3.56	-11		-		
		6	118	2	-3.56	-11			O dividieren	
		7	119	2	-3.56	_		 addieren 	ersetzen	
		8	248	2	-3.56	_				
	_	9	249	2	-3.56	- 1		CANCEL	ОК	
		10	250	2	-3.56	_				
	_	11	293	2	-3.56	-11			· Inter	
		12	294	2	-3.56					
				N A						
	AKTL	leller Lastrall:		 Anzani La 					1.	
	Anz	ahl Lasten/p	ro Lastfall: 149	Lasttyp: 1	Knotenlast					
		Neuer I	astfall erzeugen	Last	fälle überlagern					
		Las	tfall löschen	Lastfälle a	ddieren und kopieren					
		La	stfall-Faktor	Tempe	eraturlast einlesen					
		Flächen	last->Knotenlast	Freihe	eitsgrade ändern					
		Kester	فمدامونونا خامدا	1						

The Plastic Block is fixed clamped on the left and right side. Select "Boundary Conditions" to create the BCs.

Enter the Material Datas

Select Register "Edit FEM project" and menu "Material-Datas" and enter the following material datas:

Element Group 1 - Steel Angle: E-Modulus = 210,000 MPa, Poisson's Value = 0.3

Element Group 2 - Plastic Block: E-Modulus = 1,200 MPa, Poisson's Value= 0.38

<u>.</u>	Aaterialdaten				\times	🖳 I	Materialdaten				\times
	Bezeichnung	Materialwerte					Bezeichnung	Materialwerte			
•	E-Modul	210000				•	E-Modul	1200			
	Poisson-Zahl	.3					Poisson-Zahl	.38			
	Dichte	7.8E-06					Dichte	7.8E-06			
	Waermekoeffizient	1.2E-05					Waermekoeffizient	1.2E-05			
Be	mentgruppe: 1	Elementtyp: TET10		<	>	Đe	amentgruppe: 2	Bementtyp: TET10		<	>
-	 Isotrop 	O Anisotrop					Isotrop	Anisotrop			
	Material-Datenbar	nk	OK				Material-Datenbar	nk 🔜 🕴	OK		
	Materialdaten kopie	ren					Materialdaten kopie	eren			

FEM Analysis

Before starting the FEM calculation, save the FEM model under the name "Steelangle-Plasticblock.FEM" and select Register "FEM Analysis" and start the Quick Solver to calculate the displacements and stresses.

Download Video-MP4, FEM, INP, FRD Files for the Steel-Angel and Plastic-Box

Postprocessing

With Register "Postprocessing" and the Icon and stresses can be evaluated.



Max. Displacements in Y Direction = - 0.0178 mm

Max. Normal Stresses SIGxx of the Steel Angle = 26.6 MPa Min. Normal Stresses SIGxx of the Steel Angle = - 31 Mpa



the results of displacements