Part 3 - The first FEM Project - Eccentric Bolt with an Axial Load

(C) 2021

Engineering Office HTA-Software Maiwaldstraße 24 77866 Rheinau-Germany Tel. 0049-7844-98641 <u>www.fem-infos.com</u> info@fem-infos.com

Table of Contents:

1.	Create the 1.1 Exact 1.2 CAD I	e Eccentric Bolt Result Model	2 2 2
	1.3 Gener		3
	1.4 Bound	dary Conditions	5
	1.5 Axial I	Load	6
	1.6 FEM-/	Analysis	7
	1.6.1	MÉANS-Solver	7
	1.6.2	Quick-Solver with Tetrahedral element TET4	8
	1.6.3	Quick-Solver with Tetrahedral element TET4X8	8
	1.6.4	Quick-Solver with Tetrahedral element TET10	8
	1.7 Resul	ts	9

Further Parts:

Part 23 - Eccentric Bolt with inserted hole and torsional moment

1

Side

1. Create the Eccentric Bolt

The eccentric bolt made of steel consists of a cylinder with D = 60 mm and L = 192 mm which is pressed with an axial force of 100t onto a quader with the dimensions 124 mm x 124 mm x 48 mm.

1.1 Exact result

The axial stresses in the cylinder can be calculated exactly with the cross-sectional area A.



Axial Stress = F_A / A_Z = 1 000 000 N / 3.1416 * D² / 4 = 1 000 000 N * 4 / 3.1416 * 60² mm² = 353.7 N/mm²

1.2 CAD Model

In the CAD system, the eccentric bolt is generated with a cylinder and a quader. After united both solids to a part, the eccentric bolt is saved in STEP format.



1.3 Generate the FEM Mesh



Start MEANS V12 with the desktop icon **Start** and select the "File" register and the "New" menu and select "3D Mesh Generator GMSH (STEP)" to generate a FEM-Mesh from the STEP-File "Eccentric_Bolt.STEP" with the mesh generator GMSH.

	NEW PROJECT			×		
	○ 3D Mesh Generator NE	TGEN (STEP,)	GES, STL)			
	 3D Mesh Generator GM. 	SH (STEP)				
	○ 2D/3D Beam Model with	Line-Modus				
	3D Shel Model with a Co	ontainer Mesh G	ienenerato	r		
	O 2D Plane Model with a t	Seanng Mesh G	enerator			
	○ 2D axialsym. Model with	a Screw Mesh	Generator			
	NEW PR	OJECT				
	NEW FO	OJECI				
🖷 3D Mesh Ger	neration			<u> </u>		×
Directory: C:\p	niekte\Exzenterbolzen\				Browse	er
		O STL/A	ST (ASCII)		Defau	t
U SIE	· · · · · · · · · · · · · · · · · · ·	-				
evzenterbolzen	sten					
exzenterbolzen.	step					
exzenterbolzen.	step					
exzenterbolzen.	step					
exzenterbolzen .	step activate the mesh tab and select the	3D menu and th	ne Refine by	r splitting	menu.	
In GMSH, first a	step activate the mesh tab and select the a with File/Export in INP-Abaqus fon	3D menu and th nat in order to di	ne Refine by splay it in M	splitting EANS V	menu. 12!	
In GMSH, first a Export the mesh Start 3D Mesh	step Inctivate the mesh tab and select the In with File/Export in INP-Abaqus fon Generator	3D menu and th nat in order to di	ne Refine by splay it in M	[,] splitting EANS V	menu. 12!	
In GMSH, first a Export the mesh Start 3D Mesh	step activate the mesh tab and select the n with File/Export in INP-Abaqus fon Generator Start 3D Tetrahedr	3D menu and th nat in order to di al Mesh Generat	ne Refine by splay it in M or	splitting EANS V	menu. 12!	
In GMSH, first a Export the mest Start 3D Mesh	step activate the mesh tab and select the a with File/Export in INP-Abaqus fon Generator Start 3D Tetrahedr	3D menu and th nat in order to di al Mesh Generat	ne Refine by splay it in M or	, splitting EANS V	menu. 12!	

In GMSH, the eccentric bolt is first shown in a blue wireframe. In the menu on the left, first select "3D" and then "Refine by splitting" to refine the FEM mesh 8 times more finely. After 2 clicks you get an FEM mesh with 31 808 tetrahedra.



Please also always look at the info line below, if no tetrahedra can be created "Warnings" and "Errors" are displayed. In such cases, unfortunately, GMSH must be terminated and an attempt must be made to mesh with NETGEN.



Now export the generated FEM mesh with the menu "File" and "Export" and select the format "Mesh - Abaqus INP (* .inp)".

▲ G	imsh - fil	e.stp	
<u>F</u> ile	Tools	Window	Help
New.			Ctrl+N
Open	۱		Ctrl+O
Open	Recent		•
Merg	e		Ctrl+Shift+O
Wato	h Patter	n	
Clear	r.		
Rena	me		Ctrl+R
Delet	e		
Rem	ote		•
Save	Mesh		Ctrl+Shift+S
Save	Model C	Options	Ctrl+J
Save	Options	As Defaul	t Ctrl+Shift+J
Expo	rt		Ctrl+E
Quit			Ctrl+Q
	5000	0000 211	

If the mesh is saved in the same directory with the name "file", it is automatically imported and displayed by MEANS V12.

F	Guess From Extension (*.*)
Export	Geometry - Gmsh Options (*.opt)
Firmer	Geometry - Gmsh Unrolled GEO (*.geo_unrolled)
$- \rightarrow - \uparrow$	Geometry - OpenCASCADE BRep (*.brep)
	Geometry - STEP (*.step)
Organisieren 💌	Mesh - Gmsh MSH (*.msh)
organisieren	Mesh - Abaqus INP (*.inp)
Neuer Ord	Mesh - LSDYNA KEY (*.key)
	Mesh - CELUM (*.celum)
Oracle	Mesh - CGNS (Experimental) (*.cgns)
Dack	Mesh - Diffpack 3D (*.diff)
FOCK	Mesh - I-deas Universal (*.unv)
PerfLogs	Mesh - Iridum (*.ir3)
-	Mesh - MED (*.med)
ProgramD	Mesh - INRIA Medit (*.mesh)
Programm	Mesh - CEA Triangulation (*.mail)
	Mesh - Matlab (*.m)
Programm	Mesh - Nastran Bulk Data File (*.bdf)
nrojekte	Mesh - Plot3D Structured Mesh (*.p3d)
projekte	Mesh - STL Surface (*.stl)
100t	Mesh - VRML Surface (*.wrl)
-	Mesh - VTK (*.vtk)
beamtest	Mesh - Tochnog (*.dat)
behaelte	Mesh - PLY2 Surface (*.ply2)
	Mesh - SU2 (*.su2)
dino	Mesh - GAMBIT Neutral File (*.neu)
Evzenter	Post-processing - Gmsh POS (*.pos)
	Post-processing - X3D (X3D) (*.x3d)
	Post-processing - MED (*.rmed)
Dateiname:	Post-processing - Generic TXT (*.txt)
Dateityn	Guess From Extension (* *)
e aren j pr	
Ordner aushlende	2
oruner ausbienue	

After the GMSH export, select "Generate Surface Model" to be able to select the 8 main surfaces of the eccentric bolt for the surface load and clamping.

ன FEM System MEANS V12	2 - FEM Structure File C:\projekte\exzenterbolzen\mesh.fem	
C 0 1 0 =		
Files View	Mesh Generation Edit FEM-Project FEM-Analysis Postprocessing Training	
	FEM-Merge MPC-Contact Import: STL * 2. C:\projekte\exzenterbolzen\tet4.fem *	○ German● English
New 🕞 Open 🕞	Save 🕞 Unit CAD Path 🕞 Last opened FEM-Projects 🕞	Language 🖓
Mesh Generator V12		
CENEDATE	🖳 Create Surface Model — 🗆 🗙	
1.08 ~	○ very course	
Elements: 11573 Nodes: 2546	Surface Accuracy: 0.91 Help	
with Mesh	complete V12 complete V8 model region	
Meshing Options	with more Edge-Checks but need longer runtime	
Create Surfaces	Cancel Generate Surface Model	
Refining		
Coarsing		
New		
Cancel		

1.4 Boundary Conditions

Use the "Edit FEM-Project" register and the menu "Boundary-Conditions" to create with the selection "Surfaces" the BCs in the Z-direction on the front side of the quader with the Surface 1. In the selectbox choose "Create" to create 144 Boundary Conditions.

	Files	View	Mesh Generation	Edit FEM-Project	FEM-Analysis	Postprocessing	Training					
Loads	3. Su	rface Load w Loads	Boundar	y-Conditions	undary-Condition: w Boundary-Condi Surface Mod	* Element-Gr us is active - Surface	roups Ma	aterial-Datas	Editor	6. Loads	*	Tempera
Surface	1 ^	DELETE CLEAR EDIT	Bound	lary Conditions rof Boundary Conditions	144	- D	×					
✓ Node ✓ Elemin CANCE	ents 🔽] Surfaces] Edges CREATE	Value o Constra	f Boundary [1 ints displacement in in in in	E-10 X Direction [Y Direction [☐ in Z Direction ☐ Clamped fixed 						
			Selection Select	on: ect Surfaces ect Nodes ine a coordinate range	O Dragging a O Select all sh	model region iowing nodes iowing surfaces					-	
			Edit BC	-Symbols-Size	ymbols Color	s:						
			Can	cel Editor		Create BCs Delete BCs						
	Ľ	Z X								- J.4 - J.4	and the second	

1.5 Axial Load

Choose the "Edit FEM project" register and the "Surface Load" menu.

In the next dialog box select Load case "1" and the option "or in N" and enter the value "1000000" and the degree of freedom "vertical to Surface".

With the selection "Surfaces" and the "Create a Surface Load" menu, generate a surface load by clicking the front of the cylinder with the Surface 5. In the selectbox choose "Create" to create 286 loads.

1.6. FEM-Analysis

Choose the Register "FEM-Analysis" and the Icon to start a FEM-Calculation with the MEANS-Solvers or the Quick-Solver.

U	Files View	Mesh Generation	Edit FEM-Proj	ect FEM-Analys	sis Pos	tprocessing
	1. Statics	✓ Sel	ect FEM-Solver	Structure Model Model Dimensions	s	0
	FEM-Analysis	; Fy	Select FEM 🕞	Structure Info	Fig FEM	-Assistent 🕞
	Finite Ele C:\pro Select	ement Analysis jekte \exzenterbolzen \m Solver	esh.fem olver OQu	→ □	×	
		Step 1.	Statung FEM Analys	5		
		Step 2:	Starting Postprocess	ing		
			ment of the Element			
		Select FEM Solvers		Define Results		
			Cancel			
						8

1.6.1 MEANS-Solver

The MEANS-Solver developed by HTA-Software, which needs a longer time to calculate than the Quick-Solver, is started.

III means1k	1		\times
File Edit View State Window Help			
MEANS Eingabedate: C:\projekte\EXZENT~1\tet4.TEM Ergebnisdatei Temperatur C:\projekte\EXZENT~1\tet4.TEM Pfad fuer temporaere Dateien (*.tmp) C:\Users\info\AppData\Local\Temp		-)(
Struktur mit 11573 Elementen und 2546 Knoten			
Statikberechnung besetzte Freiheitsgrade Statik VX VY VZ Zahl der Freiheitgrade 7638 Maximale Frontbreite 474 Zahl der Eliminations-Operationen 136000 Speicherplatz Gleichungssystem: Festplatte: 22 MB Hauptspeicher: 1 MB Hauptspeicher fuer alle dynamischen Felder: 6 MB path_scratch:Ci\Users\info\AppData\Local\Temp scratch_file:C:\Users\info\AppData\Local\Temp\means-0.tmp			
Struktur: Volumen: .123E+07 mm^3 Masse: .963E+01 kg			
Liste Elementgruppen Anzahl Anzahl Anzahl Nummer 1-D-Elemente 2-D-Elemente 3-D-Elemente 1 0 0 11573			
Lastfall: 1 Iteration 0 Verschiebungen (m) VX VY VZ Max .151700E-01 .152478E-01 .339998E+00 Min161633E-01158505E-01125325E-02	code 0	×	
Lastfall: 1 Iteration 0 Summe Auflagerkraefte (N) FX FY FZ Normal .161390E-08 .103850E-08100000E+07	Vein		
Lastfall: 1 Iteration 0 maximale Spannungen Volumenelemente SIG-XX SIG-YY SIG-ZZ SIG-XY SIG-YZ Mises Max 1.35E+03 .15E+03 .316E+03 .942E+02 .323E+03 .288E+03 .809E+03 Min438E+03383E+03774E+03514E+02228E+03232E+03 .703E+00			
Lastfall: 1 Iteration 0 Mittelwerte in den Eckknoten SIG-XX SIG-YY SIG-Z SIG-XY SIG-YZ SIG-XZ Mises Max .334E+02 .329E+02 .972E+02 .397E+02 .105E+03 .144E+03 .505E+03 Min _ 154E+03 _ 173E+03 _ 558E+03 _ 365E+02 _ 113E+03 _ 144E+03 Min _ 154E+03 _ 173E+03 _ 558E+03 _ 365E+02 _ 113E+03 _ 144E+03			

1.6.2 Quick-Solver with Tetrahedral Element TET4

The Quick-Solver with the linear Tetrahedral Element TET4 is started.

Normal Precision	 show and solve with C3D4 (4-node linear tetrahedral elements) show C3D4 and solve intern with a refining mesh of 8 x C3D4 convert C3D4 -> C3D10 and show and solve with C3D10
Path for INP-Solver:	C:\Program Files\FEM-System_MEANS_V12\Debug\inpsolver\inpsolver64bit.e Browser
Path for INP Files:	C: vprojekte vexzenterbolzen vexzenter I.INP Select Solver In-Core-Solver Out-of-Core-Solver
	Start FEM-Solver with INP-Interface
	Settings Help + Infos Cancel

1.6.3 Quick-Solver with Tetrahedral Element TET4X8

The Quick-Solver with the Tetrahedral Element TET4X8 developed by HTA-Software is started and can calculate 8 times more accurately than TET4.

Middle Precision	 show and solve with C3D4 (4-node linear tetrahedral elements) show C3D4 and solve intern with a refining mesh of 8 x C3D4 convert C3D4 -> C3D10 and show and solve with C3D10 	- Contraction Contraction
Path for INP-Solver:	C:\Program Files\FEM-System_MEANS_V12\Debug\inpsolver\inpsolver64bit.c	ser
Path for INP Files:	C:\projekte\exzenterbolzen\exzenter1.INP	
	Select Solver O Out-of-Core-Solver	
	Start FEM-Solver with INP-Interface	
	Settings Help + Infos Cancel	

1.6.4 Quick-Solver with Tetrahedral Element TET10

The Quick-Solver with the quadratic Tetrahedral Element TET10 is started.

High Precision	 show and solve with C3D4 (4-node linear tetrahedral elements) show C3D4 and solve intern with a refining mesh of 8 x C3D4 convert C3D4 -> C3D10 and show and solve with C3D10
Path for INP-Solver: Path for INP Files:	C:\Program Files\FEM-System_MEANS_V12\Debug\inpsolver\inpsolver64bit.e Browser C:\projekte\exzenterbolzen\exzenter1.INP
	Select Solver In-Core-Solver Out-of-Core-Solver
	Start FEM-Solver with INP-Interface
	Settings Help + Infos Cancel

1.7 Results

displacement or stress distribution.

to display the results in color as a Choose the Register "Postprocessing" and the Icon

😽 FEM System MEANS V12 - FE	M Structure File C:\projek	te\exzenterbolzen\tet	4.fem	
🔼 🛛 † 🔘 📼				
Files View N	lesh Generation Edit FE	M-Project FEM-A	Analysis Postprocessir	ng Training
Show Results	Displacement-Factor Pick, Search Values	Legend 1 + Diagram 1 +	DXF-Postprocessing Value-Animation 👻	List FEM-File List STA-File
Postprocessing 🕞	Factor/Values	Legend/Diagram 🕞	Animations 🕞	List Files 🕞
	Postprocessing		- 0	×
	Results: Contour of Displacemer Nodal Stress Contour Element Stress Contour Accuracy: Edit Accuracy: 1 3 4	nt Load Ca Reau Cont Dia Edit Pick,	Ise: 1 ction Forces our of Forces splacement Factor Colours for Legend Search, Save Values	
	Select Result Component: von Mises Stress von Mises Stress Normal Stress Sign Normal Stress Sign Normal Stress Sign Normal Stress Sign Shear Stress Tau Shear Stress Tau	ma x ma y ma z xy yz zx s S1 ss S2 s S3		

A comparison of the results with the different tetrahedral elements of MEANS V12 follows:

- TET4
- TET4X8
- TET10

shows that the accuracy of TET4 to TET10 is getting more and more better, but the computing time is getting longer and longer.

v.Mises Nodal Stresses of TET4

v.Mises Element Stresses of TET4

v.Mises Nodal Stresses of TET4X8

v.Mises Nodal Stresses of TET10

