Part 14: FEM-Analysis of a CAD Assembly with MEANS V11

A CAD Assembly from Inventor is loaded with 7 bar internal pressure, how high are the displacements and stresses with FEM System MEANS V11.

CAD-Assembly from Inventor



Create a Model Part

Now delete all unimportant components such as screws, left flange, sockets in the CAD system and combine the model into a large part and export it as a STEP file.



STEP Interface

Now close the CAD System and start the FEM System MEANS V11 to generate a FEM Mesh with the STEP-Model.

🖳 FEM-Sys	stem M	IEANS V11	für DirectX11						
0	3 8	() =							
	Datei	Ansicht	Netzgen	erierung FEM-	Projekt bearbeiter	FEM-Ana	alyse Erg	gebnisauswertung Training	
				Zuladen Vereinen	Importieren Exportieren	MEANS- SHELL		1. C:\projekte\kugelventil\Kugelventil.fem	
Neu 15	Eir	nladen 🕞	Sichern 🖓	Baugruppen 🖓	Schnittstelle 🕞	Extern 🕞	Pfade 🕞	Zuletzt geöffnete FEM-Projekte	, Fa

Select the "Mesh Generation" tab and "New" to create a new FEM project.

ROJECT	—	×
③ 3D Tetrahedral Meshing with STL, STEP or IGE	s	
○ 2D Triangle Meshing with DXF		
○ 2D QUAD Meshing with STEP or Triangles		
O 2D Extrudates in Z direction to 3D Structures		
O Mesh manipulations or edit Element groups		
O Create a new Model with Beam-Line-Modus		
O Mesh Generation for Container or Silos		
NEW PROJECT		

Select "3D Tetrahedral Meshing with STL, STEP or IGES

A dialog box appears, showing the following CAD formats:

STL consists of a triangular outer shell for 3D mesh generationSTEP consists of solid elements and is the most suitable 3D formatIGES like STEP format but not so common anymore

Select the STEP file from the FEM-Projects/New-Directory or with "Browser" and click on the button "Start Mesh generator No. 2 with CAD File" to display it in the mesh generator.

3D-Netzgenerator			-	
Directory: C:\projekte\oebk	.e\neu\			Browser
STEP			SCILL	Default
() STE	U IGES	O SIE/ASI (A	Jon y	Help
vereinfacht_ohne_schraube	n_part_cf.stp			
STL-Optimierer	antara tanan karat	Line Dilet Alexandri Al	00000	
mit allen Tests	mieren iassen (empro	nien z.b. dei Absturz in N	GSOIVe)	
3D-Netzgenerator Nr. 2 mit	STI -/STEP-/IGES-F	ile starten		
Netzgenerator Nr. 2 mi	it CAD-File starten	mit Fehleranzeige/R	ecent File	Hinweis
- 3D-Netzoenerator Nr. 3 mit	STI -File starten			
- 3D-Netzgenerator Nr. 3 mit Netzgenerator N	STL-File starten	Netzdichte: 0	~	Hinweis
3D-Netzgenerator Nr. 3 mit Netzgenerator N	STL-File starten Ir. 3 starten	Netzdichte: 0	~	Hinweis

The STEP File can be seen in the mesh generator and can be rotated as required.



Select the menu "Mesh" and "Meshing Options" and generate with the mesh density "moderate" and the menu "Generate Mesh" a FEM Model with tetrahedral elements.



After the mesh generation, the mesh must be exported to MEANS V11 with name "**test.fem**". Select "File" and "Export Mesh" and save it into the Debug/Mesh Path.

File Geometry Mesh View	Refinement
Load Geometry <l><g> Save Geometry Recent Files</g></l>	•
Load Mesh <i><m> Recent Meshes</m></i>	•
Save Mesh <s><m> Merge Mesh Import Mesh</m></s>	please export the generated
Export Mesh Export Filetype	model with the name "test.fem" into the default Mesh-Debug-Pa

Surface model

After mesh generation, the surface model is created with the option "very course" with 119 faces. The surface model now makes it possible to select the surfaces for the boundary conditions and surface loads or to view and edit the model from the inside.



Create Boundary Conditions

Select register "Edit FEM Project" and click on the "Boundary Condition" icon to clamped fix the model to his flanges.

Select "Create BC's" and click on the clamped surfaces. Finally, select "Create" in the Selectbox to create the RBs



Create the Pressure Loads

The casting is loaded with a pressure load of 7 bar or 0.7 MPa in the inside. First, the surfaces for the pressure must be determined. Select the register "View" and menu "Switch Surfaces ON/OFF"

	0 1	() =								
2	Files	View	Mesh Generation	Edit FEM-Project	FEM-Analysis	Postprocessin	g Trainin	9		
Hidden-	Line (● without № ● with Mes	Aesh ○ Wireframe h ☑ Edges	Light 10% + Hidden-Line new	31 1. M	lain View		1.	Surface-Modus	Zo
					Surfac	e Modus is active		2.	Node-Modus	
								2	Line-Modus	
									Line modus	
								4.	Create Surface Mod	lel

In the new left dialogbox select "Hide all Surfaces" to switch OFF all surfaces. Then switch ON the surfaces 3, 10, 20, 26 and 40 and note this 5 surfaces for the next step. Then select again "Show all surfaces".



Select register "Edit FEM-Project" and "Surface Load" to create the surface load in the inside.

	Files	View Mesh G	eneration	Edit FEM-F	Project	FEM-Ana	alysis	Postprocessing	Trainii
	3.	Surface Load		→	1. Bou	undary-Con	dition: +		
ads	1.	Point Load	Boundary	-Conditions	Show	w Boundary	-Conditio	ons Element-G	roups
	2.	Line Load	1				Surface	Modus is active	
	3.	Surface Load	1						
	4.	Centrifugal Load	🖳 1	oads					×
	5.	Gravitation Load						-	
	6.	Temperature Load	Ci	irrent Loadcas	e 1		• +	1	
	7.	Nonuniform Loads	Nu	imber of Loads	0		New		
	8.	Edit Load Case	V=	lue of Load	7				
	9,	Editor	ve	ide of Load	./		(for	example: MPA)	
			(Co	our of Axis: BLA	CK: X-Axis;	BLUE: Y-Axis;	RED: Z-Ax	is)	
) Select Surfac	ces	C) Draggin	g a model region	
			C) Select Node	s	0) Select a	Il showing nodes	
			C) Define a coo	rdinate ran	ige 🤇) Select a	II showing surfaces	
				Calcula	ate Surface	e Load Value	e (MPA) froi	m Load (N)	
				Cancel	Edi	tor	Create	Surface Load	1
						Г	Del	lete Loade	1

Enter the value "0.7" and select "Create Surface Load"

Now click on the surfaces in the right window one by one on 3, 10, 20, 26 and 40. These are displayed in the selectbox, where "Generate" generates the surface load.



Material Datas



Select register "Edit FEM-Project" and select the icon Material-Datas to enter the Material Datas such as Young's modulus and Poisson's Ratio for Steel is always preset.

🙀 De	mo FEM System ME	ANS V11 - FEM Struct	ure File C:\projekte\	bebke\ne	eu\netz8.fem		
0	0 ? 🕑	÷					
U	Files View	Mesh Generation	Edit FEM-Proje	ct F	EM-Analysis	Postprocessing	Training
Load	3. Surface Loa ds Show Loads	ad + Bound	I.,	Bound Show B	ary-Condition: oundary-Condi Surfac	tions Element-Gro e Modus is active	Dups Material-Datas
🔜 E	dit Material Datas		- 🗆	×			
	Name	Material Datas	8				
•	Youngs modulus	161500					
	Poisson Ratio	.3					
	Density	7.8E-06					
	Heat Coefficient	1.2E-05					
Ele	ement Group: 1 Isotrop Material Data Ba Copy Material Da	Element TET4 O Anisotrop se	с ОК	>			

FEM-Analysis

Before the FEM Analysis, always save the model with "Files" and "Save" under a name in the project directory.



Select FEM-Analysis and "1. Statics" and click on the solver icon to start the FEM-Analysis with the Dr.Kühn-Solver or with the Quick-Solver.

	View	Mesh Genera	tion Edit FEM-Proj	ect FEM-Ana	alysis Pos	tprocessing	Training	
1	. Statics		Select FEM-Solver	Structure Mode				
	1. Statics			woder Dimensi	UIIS			
	2. Dynamic	s	Select FEM IS	Structure Inf	O GI FEM	Assistent 🕞		
3	3. Tempera	ture						
4	4. Buckling							
1	5. Geometr	ic Nonlinear						
	6. Contact-	Analysis						
	7. Material	Nonlinear						
8	8. Formopt	imization	🔡 Finite Elemer	nt Analysis		~		
				Step 1: 5	Starting FEM An	alysis]
				Step 2: S	tarting Postproc	essing		
				Step 3: Refiner	ment of the Elem	ent Stresses		
			Selec	t FEM Solvers		Define Res	ults	

For larger FEM structures over 100,000 elements, choose the faster Quick-Solver with three options for precision.

In the Quick-Solver, select "convert C3D4-> C3D10 ..." to perform the FEM analysis with the very accurate quadratic TET10 tetrahedral elements.

🖶 INP-Interface for FE-	Solvers —		×
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 	$\langle \rangle$	5.
Path for INP-Solver:	C:\Program Files\FEM-System_MEANS_V11\Debug\inpsolver\inpsolver64bit.c	Browser	
	Select Solver In-Core-Solver Out-of-Core-Solver		
	Start FEM-Solver with INP-Interface		
	Settings Help + Infos Cancel		

FEM-Projekt: C:\pro Bitte warten: FEM-Ar	njekte∖300x300_IPE-Profil\statik\tet4 nalyse von 122329 Elementen und 220089 Knotenpunkten	
	Postprocessing MEANS V11 wieder starten	
	Rechenzeit: 0:0:0:8:838	Abbruch
plastic data orientations: amplitudes: data points i print request transformatio property card	points per material: 0 0 2 n all amplitudes: 2 cs: 2 ons: 0 ls: 0	
STEP Static analysis Decascading the Determining the	1 was selected MPC's structure of the matrix:	

After a calculation time of a few minutes or even longer with the "Out-of-Core-Solver" a short tone signal can be heard, now the menu "Start Postprocessing MEANS V11" is enabled again and you can start the postprocessor for the result evaluation.

Postprocessing

Select the tab "Postprocessing" and click on the Icon Postprocessor for the result evaluation automatically.



to start the

🖊 Files View N	lesh Generation	Edit FEM-Project	FEM-Analysis	Postprocessing	Training	
Show Results	Displacement-Fac Pick, Search Value	ctor Legend 1 es Diagram 1	• Value-A	Animation - Li	st FEM-File st STA-File	FKM-Richtlinie f
Postprocessing 🕞	Factor/Values	s 🗔 Legend/Dia	agram 🗐 🖌 Ani	mations 🕞	List Files 🕞	Fatique-Anal
	🖳 Postp	rocessing			X	1
	Results: Cor Noi Ele	ntour of Displacement dal Stress Contour ment Stress Contour	Load C 〇 Rea 〇 Con	ase: 1 v inction Forces tour of Forces		
	Accurac	y:	D	isplacement Factor		
	Į		Edi	t Colours for Legend		
	Select F	Result Component: von Mises Stress Normal Stress Sigma Normal Stress Sigma Shear Stress Tau xy Shear Stress Tau zy Shear Stress Tau zx Shear Stress Tau zx	x y z 1 2			

The following Result Evaluations are available:

- Contour of Displacments
- Nodal Stress Contour
- Element Stress Contour
- Reaction Forces
- Contour of Forces

Displacements in Z-Direction

Max. Displacements in Z-Direction = 0.33 mm (with a Displacement Factor = 500)



v.Mises Stresses

Max. v.Mises-Stresses = 157 MPa



Model-Cut

Create a Model-Cut with a new Element Group 2 and with a define range.



v.Mises Stresses

Max. v.Mises-Stresses = 157 MPa

