Part 33 - Tensile Test Mesh Generator according to ISO 6892 / DIN 50125

With the new Tensile Test Mesh Generator of the FEM System MEANS V12 (<u>www.fem-infos.com</u>) it is possible to generate tensile test models create of linear or quadratical hexahedrons, pentahedrons or tetrahedrons fully automatically with a few key data and to calculate them non-linearly with a stress-strain diagram.

Round Tensile Samples:



Flat Tensile Samples:



Tensile Sample:



Entering the key data in the tensile test dialog box:

Select the "Mesh Generation" tab and the "TENSILE TEST" submenu and enter the following key data for the tensile test shown above:

Mesh Ger	neration Edit FEM-Pro	ject FEM-Analysis Postproc	essing Training	
erators nent		QUAD-Meshes, Refine, Delete	Check Node Numbering Jacobi-Determinante	SOLIDGEN VESSEL TENSILE TEST FEMM
1 Fa	2D Mesh Generation 🕞	Manipulate Meshes 🕞	Check Meshes 🛛 🖓	Mesh Generators 🖓

d_0	Probendurchmesser	=	8
d_1	Kopfdurchmesser	=	20
h	Kopfhöhe	=	30
r	Übergangsradius	=	3.5
Lo	Anfangsmesslänge	=	50
$L_{\rm c}$	Parallele Länge	=	88
Lt	Gesamtlänge	=	155
d_2	Durchmesser des Ansatzes	=	0
g	Länge des Ansatzes	=	0
a ₀	Probendicke	=	0
b_0	Probenbreite	=	0
В	Kopfbreite	=	0

Select the "Create Tensile Sample with SOLIDGEN" button

Create Tensile Sample with SOLIDGEN

to generate the tensile sample using the SOLIDGEN mesh generator with 6 cylinders and mesh density:

Number of nodes in the radial direction X-ND = 6 Number of nodes around the circumference Y-ND = 24 Number of 2D meshes in the Z direction Z-ND = 10

Part 33 - Tensile Test Mesh Generator according to ISO 6892 / DIN 50125

Mesh-Density:	X-ND-C	YL:	6 Y-N	D-CYL:	24	Z-ND-CYL	: 10		X-ND-QU:	0	
Number of Elem	ent Groups	s: 6	✓ Star	t-Angel:	0	End-Angel	360		Y-ND-QU:	0	
Zylinder 1											
Di:	0	Da:	8	X-MP: [0	Y-MP:	0	X-V4:	0	NGR:	1
Z-MP:	0	Z- L:	30	X-V3: [8	Y-V3:	0	Y-V4:	0	Name	
Zylinder 2											
X-V1:	8	Y-V1:	20	X-V2:	0	Y-V2:	0	X-V4:	0	NGR:	1
Z-MP:	0	Z-L:	30	X-V3:	20	Y-V3:	0	Y-V4:	0	Name	
Zylinder 3											
Di:	0	Da:	8	X-MP:	0	Y-MP:	0	X-V4:	0	NGR:	1
Z-MP:	30	Z-L:	3.5	X-V3:	8	Y-V3:	0	Y-V4:	0	Name	
Zylinder 4											
Di:	8	Da:	20	X-MP:	0	Y-MP:	0	X-V4:	0	NGR:	1
Z-MP:	30	Z:	3.5	X-V3:	8	Y-V3:	0	Y-V4:	0	Name	
Zylinder 5											
Di:	0	Da:	8	X-MP:	0	Y-MP:	0	X-V4:	0	NGR:	1
Z-MP:	33.5	Z-L:	19	X-V3:	8	Y-V3:	0	Y-V4:	0	Name	
Zylinder 6											
Di:	0	Da:	8	X-MP:	0	Y-MP:	0	X-V4:	0	NGR:	2
Z-MP:	52.5	Z-L:	25	X-D:	8	Y-D:	0	Y-V4:	0	Name	

Select menu "HEX8 Meshing" to create a linear FEM model from 6480 HEX8 and 923 PEN6 elements as well as 6637 nodes and 3 element groups.



		HEX8 MESHI	NG
MIRROF	۶ [HEX8->TET4	REFINE
	Save	load	Cancel

Mirror model

Select "Mirror" menu to create a FEM model from 12,960 HEX8 elements and 13,153 nodes and 3 element groups.



TET4 Model

Select the menu "HEX8->TET4" to convert the FEM model into a tetrahedral mesh.

Refine

Select the "Refine" menu to create an 8x finer mesh from the tetrahedral mesh

Increase in accuracy

For higher accuracy, the FEM model can later be converted into TET10 or HEX20 and PEN15 mesh in the quick solver.

Generate Load Cases

In the tensile test dialog box, define an increasing axial load in the Z direction with a starting load of 6000 N and a load increase of 500 N and 15 load increments at the top of the tensile test at Z = 0 mm.

Select "Insert" to be able to examine the area of the yield point in more detail with finer load case steps, for example.



Generate boundary conditions

At the end of the tensile sample, define a clamping in the X, Y and Z directions with a clamping depth of 25 mm.



Generate material data

Define with an elastic modulus of 210,000 N/mm² and a Poisson's ratio of 0.3 for the material steel, which is always preset.

Nonlinear FEM Analysis

Select the "FEM Analysis" tab and "Material Nonlinear" menu.



Select the FEM analysis "Nonlinear with Hardening Isotropic" and the stress-strain diagram "STEEL NORMAL".

Plastische Verformungen				×
Material Law:				
🔘 Linear Elastic				
Nonlinear with Hardening Isotrop				
O Nonlinear with Hardening Kinem	atic			
O Nonlinear with Hardening Combi	ned			
Select a Stress-Strain-Curve from	the Database		Edit	
Select a Stress-Strain-Curve from Plot Stress-Strain-Curve	the Database Plot Stress-Disp	lacem	Edit ient-Curve	
Select a Stress-Strain-Curve from Plot Stress-Strain-Curve Material: STEEL NORMAL	the Database Plot Stress-Disp	lacem	Edit ent-Curve	

Select "Plot Stress-Strain Curve" to display the stress-strain curve:



STEEL NORMAL

The displacements and stresses increase linear up to the yield strength of 190 N/mm² (= Hook's straight line), then the plastic deformations begin with a flattening of the curve, i.e. the strains increase more than the stresses.

The curve can also be edited using any text editor and "Edit" menu:

STEEL NORMAL

0.00000000E+00	1.9400000D+02
0.077700000E+00	3.11210000D+02
0.155500000E+00	3.59950000D+02
0.233000000E+00	3.96540000D+02
0.311000000E+00	4.23890000D+02
0.388000000E+00	4.45580000D+02
0.466000000E+00	4.63940000D+02
0.544000000E+00	4.80260000D+02
0.622000000E+00	4.94950000D+02
0.699000000E+00	5.08310000D+02
0.777000000E+00	5.20260000D+02
0.855000000E+00	5.31420000D+02
0.933000000E+00	5.41870000D+02
1.011000000E+00	5.51710000D+02
1.088000000E+00	5.60910000D+02
1.166000000E+00	5.69560000D+02
1.244000000E+00	5.77820000D+02
1.322000000E+00	5.85720000D+02
1.399000000E+00	5.93280000D+02

-END-

Start Quick Solver

Select the "START NONLINEAR FEM ANALYSIS" button to start the quick solver with the setting "C3D8 (8-node linear isoparametric element)".

	🔛 Quick-Solver		- 🗆 X	
	Normal Precision	 C3D8 (8-node linear isoparametric element) show C3D4 and solve intern with a refining mesh of 8 x C3D4 C3D20 (20-node quadric isoparametric element) 		
	Path for INP-Solver:	C:\Program Files\FEM-System_MEANS_V12\Debug\inpsolver\inpsolver64bit_	Browser	the step step step step step step step
E	Path for INP Files:	C:\projekte\nonlinear\solidgen\probe8a.INP]	
-		Select Solver In-Core-Solver Out-of-Core-Solver		
		Start FEM-Solver with INP-Interface		
		Show and edit last INP Resolve last INP		
		Setting Help + Infos Cancel		

Stress-strain diagram for element group 2

Use the "Edit FEM project" tab to only show element group 2 so that only the results of the Gage Length L0 and no secondary stresses are evaluated.



Show Stress-Strain diagram

After the nonlinear FEM calculation, you can first display the important stress-strain curve using the "Postprocessing" and "Stress-strain diagram" tabs.

E	dit FEM-Project	FEM-A	nalysis	Postprocessing		
T	Legend 1	Dis	Displacement-Factor			
	List Table	1	ctor/Ani	mation 12	List File	
	Diagram 1 Stress-Strain-Dia New Diagram	gramm				

Set Gage Length L0

Enter the Gage Length = 50 mm in the dialog box and select the "Start" button so that the Displacements, v.Mises Stress and the Load per load increment are read and clearly displayed in the table.

In addition, the strains are output from the displacements divided by L0.

Anzahl La Ausgangs	ung eines Spannu Istfälle = 15 Ilänge L0 = 50	ngs-Dehnungs-Diag Knoten für Knoten für	ramms Verfomungen = M Spannungen = M	ax ∨ ax ∨ v.Mises	 ✓ FEM-File: ✓ Result-File: 	C:\projekte\non C:\projekte\non	linear\solidgen\probe8 linear\solidgen\probe8	fem FRD		- 0	×
LF 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15	Knoten 1002 520 519 519 517 1016 519 527 283 283 283 283 283 283 283 283 283 283	X-Verformung 0.00 0.01 0.01 0.01 0.01 0.01 0.01 0.01 0.01 0.01 0.01 0.01 0.01 0.02 52 -0.03456	Y-Verformung 0.00 0.01 -0.01255 -0.012555 -0.034556	Z-Verformung 0.04025 0.04361 0.04596 0.05332 0.05332 0.05379 0.05379 0.05379 0.05379 0.05401 0.05401 0.05401 0.04001 0.34601 0.34601 0.34601 2.38949 2.48945 2.88215	Dehnung=Verf./L0 000872174 000872174 000872174 00106354 00107836 001278722 00522028 00522028 00532028 00532028 00532028 00532028 0053204 0053204 0053204 0053204 0053204 0053204 00530805 00557643	Knoten 6990 283 6906 6905 6905 6908 6908 6908 6908 6908 6982 482 482 489 489 489 490 490	Spannung 120,74 130,80 140,86 150,92 160,98 171,04 181,10 191,17 201,18 211,20 231,26 241,30 251,35 261,39	Last FX 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.	Last FY 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.	Last FZ 6000.01 6590.02 6999.98 7999.98 8999.99 999.99 10000.00 10499.98 11009.00 11499.98 12000.03 12499.99 13000.04	
Fert	ig, die Diagramme kö	innen nun dargestellt v	verden! Anhalten	Diagr	am-Beispiele	Diagramm (darstellen und auswähle	en mit Stre	ss-Strain	~	

Stress-Strain diagram

Select the "Plot Diagram with Selection" button as well "Stress-Strain" to display the diagram with GNUPLOT.



The plastic deformations begin at the yield strength of 190 N/mm²



Load-Strain diagram

Select the "Plot Diagram with Selection" button as well "Load-Strain" to display the diagram with GNUPLOT.



The plastic deformations begin at an axial load of 9500 N



Stress-Displacement diagram

Select the "Plot Diagram with Selection" button as well "Stress-Displacement" to display the diagram with GNUPLOT.



Stress-Displacement-Diagram of the FE-Project: C:\projekte\nonlinear\solidgen\probe8.fem

Load-Displacement diagram

Select the "Plot Diagram with Selection" button as well "Load-Displacement" to display the diagram with GNUPLOT.



Stress Distribution

Select the "Postprocessing" and "Nodal Stresses" tab to display the v.Mises stress or displacement distribution for load cases 1-15.

Load Case 1

v.Mises Stress = 128 N/mm²



Nonlinear-Animation

Select the "Postprocessing" and "Nonlinear-Animation" tabs to animate all load cases one after the other.

E	Edit FEM-Project FE	EM-Analysis	Postpro	cessing
	Legend 1 - Stress-Strain-Di -	Displacement Value-Animat	-Factor	FEM INI STA FR
	Legend/Diagram 🕞	Value-Anima Scale-Anima Nonlinear-Anima Temperature DXF-Postpro	ition tion nimation -Animatic cessing	List Files

Load Case 15

v.Mises Stress = 261 N/mm²

LASTFALL= 15	🔡 Nonlinear-Animation	- 🗆 X	
Vergleichsspannung v.Mises		Cuit	
261.40			
221.45			
184.10			
146.76			
109.42			
72.073			
34.729			
0.0000			
Bearbeiten : +			

Displacements in Z direction = 3.73 mm

