Part 26: Contact-Analysis of a Glass-Balustrade and a Laminated Glass Plate

Part 1: Contact-Analysis of a Glass-Balustrade

The glass balustrade assembly consists of a glass-plate, aluminium-bracket and steel-uprofil. What displacements and stresses are there when the left side is loaded with 1000 N. The bracket is clamped fixed in the uprofil. The uprofil is not calculated because there are low stresses.



Glass-Plate

consists of 45 864 tedrahedral elements and 10 804 nodes are generated with NETGEN with E-Modulus = 75 000 N/mmm² and Poisson-Zahl = 0.17



Aluminium-Bracket

consists of 44 928 tedrahedral elements and 10 354 nodes are generated with NETGEN with E-Modulus = 71 000 N/mmm² and Poisson-Zahl = 0.34



FEM-Merge

Select register "Files" and menu "FEM-Merge" in order to combine both structures to one FEM mesh with 90 792 tetrahedral elements, 21 158 nodes and 2 element groups.



Create Load Case 1

The glass balustrade is loaded with 1000 N on the left side.

At first create a Surface Model with Register "View" and the icon



C C	Training
Rendering O Wireframe Light 10% - with Mesh with Edges - Hidden-Line new View Preview Zoom 1. Main View - Surface Modus is active - Surface 21	Surfaces Nodes 2. Node-Modus •
	Surface Nodes Lines
	Number of Surfaces = 48
Create Surface Model — X Very course moderate Very fine Very much Surface Accuracy: 0.91 Help Complete V12 Complete V8 O Ruick with a Bance	Surface 5 Surface 6 Surface 7 Surface 7 Surface 8 Surface 10 Surface 11 Surface 11 Surface 12 Surface 14 Surface 14 Surface 16 Surface 17 Surface 18 Surface 19 Surface 19 Surface 19 Surface 10 Surface 17 Surface 18 Surface 19 Surface 19 Surface 20 Surface 22 Surfa
with more Edge-Checks but need longer runtime	Hidden-Line erzeugen
	Create a Surface Model
Cancel Generate Surface Model	Sort and Hide Surfaces
	Hide some Surfaces

Select register "Edit FEM-Project" and "Point Load" and create a Point Load with Value of Load = 1 in x direction in order to select Surface 14 on the left side.

	Thes view wesh	Edici Elvi-Fi					
F	1. Point Load		1. Boundary-Condition:				1
Loads	1. Point Load	Boundary-Conditions	Show Boundary-Condit	ions Ele	ement-Groups	Material-Datas Edi	tor
	2. Line Load		Surface Modu	s is active -	Surface= 14		
	3. Surface Load						
	5. Gravitation Load		🛃 Load	s		- 0	×
	4. Centrifugal Load						
	6. Temperature Load		Curren	Loadcase:	1	- +	
	7. Nonuniform Loads		Numbe	r of Values:	0	New	
	8. Edit Load Case		Vehice	d Lands			
	9. Editor		Value	JF LOAU.		(for example: N)	
	· · · · · · · · · · · · · · · · · · ·		Degree				
			Degree		_		
			Degici		(in X Direction	on 🔿 in Z Direction	
			Logic		 in X Direction in Y Direction 	on () in Z Direction	
	- n x]		Selectio	n:	 in X Direction in Y Direction 	on () in Z Direction	
	×		Selection	n: lect Surface:	 in X Direction in Y Direction 	on () in Z Direction on () Dragging a model reg	ji <mark>o</mark> n
Magazari Surface	×		Selectio Selection S	n: lect Surface: lect Nodes	 in X Direction in Y Direction s 	on O in Z Direction on Dragging a model reg O Select all showing no	gion ides
Ref Surface	- X 14 DELETE CLEAR		Selectio Selection S	n: lect Surface: lect Nodes fine a coordi	 in X Direction in Y Direction s inate range 	on O in Z Direction on Dragging a model reg Select all showing no O Select all showing su	gion Ides Ifaces
Surface	- X 14 DELETE CLEAR EDIT		Selectio Selection S	n: lect Surface: lect Nodes fine a coordi	 in X Direction in Y Direction s inate range 	on O in Z Direction on Dragging a model reg Select all showing no Select all showing su	gion Ides Ifaces
Surface	e 14 DELETE CLEAR EDIT des Surfaces		Selection Selection	n: lect Surface: lect Nodes fine a coordi	in X Directic in Y Directic in Y Directic s inate range	on O in Z Direction on Dragging a model reg Select all showing no Select all showing su	gion Ides Ifaces
Surface	e 14 DELETE CLEAR EDIT des Surfaces ments Z Edges		Selectic Select	n: lect Surfacer lect Nodes fine a coordi	in X Directic in Y Directic s inate range	on O in Z Direction on Dragging a model reg Select all showing no Select all showing su	gion Ides Ifaces
Surface	e 14 DELETE CLEAR EDIT des Surfaces ments Edges		Selectic Select	n: lect Surfacer fine a coordi Edit Lo	in X Directic in Y Directic in Y Directic inate range inate range ad-Size	on O in Z Direction on Dragging a model reg Select all showing no Select all showing su Color:	gion Ides Infaces
Surface	e 14 DELETE CLEAR E DIT des Surfaces ments Edges CEL CREATE		Selection Selection	n: lect Surfacer lect Nodes fine a coordi Edit Lo	in X Directio in Y Directio in Y Directio s inate range ad-Size	on O in Z Direction on Dragging a model reg Select all showing no Select all showing su Color:	jion ides ifaces
Surface	e 14 DELETE CLEAR EDIT des Surfaces ments Zeges CEL CREATE		Selectic Selectic Selectic De De Canv	n: lect Surface: fine a coordi Edit Lo	in X Directio in Y Directio in Y Directio s inate range ad-Size Editor	on O in Z Direction on Dragging a model reg Select all showing su O Select all showing su Color:	gion ides ifaces

Select "Editor" and multiply the load values with the Load Factor 2.82 which is calculated with 1000 N / 355.

	SHOW LOads	Boun	dary-Conditions	Show Boundary-Con	utions	Element-Groups	Material-Datas	Editor		
				Surface Mo	dus is a	tive - Surface= 14				
	Edit Loads			- 0	×					
	Nr.	Node	FHG	Value	^					
	1	10356	1	2.82						
	2	10357	1	2.82						
	3	10359	1	2.82						
	4	10360	1	2.82						-
	5	10409	1	2.82						
	6	10410	1	2.82						
	7	10411	1	2.82	-	Edit Load O	Case	<u></u>		
	8	10412	1	2.82						
	9	10413	1	2.82		Actual Load	(Case: 1		>	
	10	10414	1	2.82		Actual Edac		•	- 1	
	11	10415	1	2.82		Fa	ctor: 2.82			
	12	10416	1	2.82						
						۲) multiply	O divide		
a	d Case:	1 <	> Load C	ases: 1		C) add	⊖ replace		
m	ber of Loads/Loa	d Case: 355	Load Type: 1	Point Load		_				
					~~		CANCEL	OK		
	New Loa	ad Case	Co	mbine Load Cases						
Delete Load Case		. (Copy Load Case							
Load Factor Convert		Convert Te	mperature to a Load Case							
	Pressure->F	Point Load		Change FHG						
	Point Load-:	>Line Load								



Create Load Case 2 with a Master-Surface Load

Load case 2 is the Master-Surface-Load for contact between plate and bracket. Select register "Edit FEM-Project" and "Surface Load" to create a surface load "vertical to Surface" with the Surfaces 3 and 6, the load value is zero because it is not used.

Files View Mesh Generation Edit FEM-Project FEM-Analysis Postprocessing Training	
Image: Surface Load Image: Show Boundary-Conditions Image: Show Boundary-Conditions Image: Show Boundary-Conditions Image: Show Boundary-Conditions Image: Surface Module is active - Surface Image: Surface Module is	
Sufface 3 DELETE Sufface 3 Sufface 3 Sufface 3 Sufface 3 Sufface 4 Sufface	



Create Load Case 3 with a Slave-Surface Load

Load case 3 is the Slave-Surface-Load for second contact between plate and bracket. Before the next step you must hide Element Group 2 with menu "Element Groups" so that only show the bracket part.



Select register "Edit FEM-Project" and "Surface Load" to create a surface load "vertical to Surface" with the 9 bracket contact surfaces (see below in black).



Boundary Conditions

The Bracket is clamped fixed in the uprofil. Select register "Edit FEM-Project" and "Boundary Conditions" and clamped fixed the 6 surfaces of the bracket.

Surface Load Show Loads Show Loads Show Loads Show Soundary-Conditions Eler	ment-Groups Material-Datas Editor 6. Loads
Info Line	
🖁 Boundary Conditions — 🗆 🗙	
Number of Boundary Conditions 7590 New	
Value of Boundary 1E-10	
in X Direction In Z Direction	
in Y Direction I Clamped fixed	
Selection:	
Select Surfaces Dragging a model region	HTTTP://
Select Nodes Select all showing nodes	
O Define a coordinate range O Select all showing surfaces	Little and the second
Edit BC-Symbols-Size	
Tum BC-Symbols Colors:	taletee .
Cancel Editor Coasta PCa	
Delete BCe	

Material Datas

Select register "Edit FEM-Project", "Material Datas" and "Material Data-Base" to enter the material datas for the Glass-Plate with E-Modulus = 75000 N/mm^2 and Poisson-Value = 0.17 and for the Aluminium-Bracket with E-Modulus = 71000 N/mm^2 and Poisson-Value= 0.34.

		÷									
9	Files View	Mesh Generation	Edit FEM-Project	FEM-Ar	alysis Postprocessing	g Training					
F Load	3. Surface Loa s Show Loads	ad 🔹 月 Boundar	y-Conditions	indary-Co v Boundar	ndition: - y-Conditions Element	-Groups Material-Datas	Editor	6. Loads		emperature	
-					Into Line					(Ja)	
🔛 Ed	dit Material Datas		- 0 ×								
	Name	Material Datas	🖳 Material Data B	ase							
•	Youngs modulus	75000									
	Poisson Ratio	.17	Material:	Alumini	um	Density (kg/mm³):		2.700002621480	73E-06	Specific Heat	at Ca
	Density	2.200002E-06	E-Modulus (MPa):	71000		Heat Coefficient:		0.0000239		i i	
	Heat Coefficient	5.4E-06	Poisson Patio:	0.34		Heat Conductivity (W/r	nmk).	0 230000010924	414	Sort	t hv
•		10	Poisson Rado.				inin yr	0.20000010024	141		,
			Add Delete	Save	Take Up Material	Load Data Base		Save Data	a Base	O Meter (۲
			Material		Young Modulus	Poisson-Value		Density	Heat	Coefficient	
			Aluminium		71000	0.34	A 2 700	003E-06	• 0.0000239		23
Ber	ment Group: 2 Jeotrop Material Data Ba Copy Material Da	Bement TET10 O Anisotrop se ta	Bronze Cadmium Celluloid Ceramic Clinker Concrete Concret	1 FR 5 GF-11	110000 51000 2500 2500 210000 210000 20000 163000 20000 163000 75000 78000 78000 78000 108000 9600 211000 123000 4910 2500 4910 2500 4910 2500 4910 2500 103000 103000 100000 2000 100000 3000 5500 110000 3000 5500 170000 3100 3200	0.44 0.3 0.29 3 0.28 3 0.28 0.3 0.28 0.13 0.28 0.27 0.25 0.42 0.25 0.42 0.371 0.22 0.33 0.28 0.371 0.25 0.43 0.28 0.33 0.28 0.37 0.25 0.44 0.37 0.25 0.44 0.37 0.25 0.32 0.35 0.32 0.32 0.35 0.32 0.35 0.35 0.35 0.35 0.35 0.35 0.35 0.35 0.35 0.35 0.35 0.35 0.35 0.32 0.35 0.32 0.32 0.32 0.32 0.32 0.32 0.32 0.32 0.32 0.32 0.32 0.32 0.35 0.32	8,700 0 0 2,100 2,600 2,600 2,600 2,600 2,200 2,200 2,200 2,200 2,200 2,200 2,200 2,5	0008-06 0028-06 0038-06 0038-06 0028-06 0028-06 0028-06 0028-06 0028-05 0018-05 0078-06 0028-07 0079-06 0038-07 0079-06 0038-06 0008-06 0088-06 0088-06 0088-06 0088-08 0088-08 0088-08 0088-0	0,000017 0 0,00003 0,000012 0,000058 0,000012 0 0,000014 0,000014 0,000014 0,000014 0,000016 0,000016 0,000016 0,0000183 0,0000183 0,0000183 0,0000183 0,0000183 0,0000183 0,0000088 0,0001 0,0000088 0,0001 0,0000088 0,00001 0,0000088 0,0000183 0,0000088 0,0000183 0,0000183 0,0000183 0,0000183 0,0000183 0,0000183 0,0000088 0,0000183 0,0000183 0,0000183 0,0000088 0,0000183 0,0000183 0,0000183 0,0000183 0,0000183 0,0000183 0,0000183 0,0000183 0,00000000000000000000000000000000000		058 0 0 0.002 0.084 0.000 0.002 0 0.001 0.001 0.001 0.001 0.001 0.002000 0.00200000000

Contact-Analysis

Select register "FEM-Analysis" and "Contact-Analysis" to calculate the displacments and stresses with the Quick-Solver.

G Files	View	Mesh Generation	n Edit FEM-Proj	ect FEN	1-Analysis	Postprocessing	Traii
	6. Contact-A	nalysis 🔹 S	elect FEM-Solver	Structure Model Dir	Model nensions		
	FEM-Analysis	F <u>5</u>	Select FEM 🕞	Structu	re Info 🕞	FEM-Assistent	
	🖷 Contac	t-Pressure				×	
	C:\proje	te\vitroplena\neu\g	lass1.fem				
	PRE	SSURE-OVERCLOS	URE=LINEAR				
		SSURE-OVERCLOS	URE=EXPOTENTIAL				
		Swap Mastersurf	ace <-> Slavesurface	(Loadcase 2	- 3)		
		Start Conta	ct-Calculation with Qui	ck-Solver			
		Start	t Contact-Postprocessi	ng			
			Cancel				

The Contact-Analysis needs about 15 iteration for the calculation. The runtime for the linear TET4 elements is 2 minutes and for the more precise quadratical TET10 elements is about 30 minutes.

Runtime:	0:0:1:0:532	Cance
224 40		
ements=32140		
e stress calculati e svmmetric stiffr	ion. ness/mass contributions.	
ations using the s stress calculati	symmetric spooles solver ion.	
e stress calculati	ion.	
	stress calculat: symmetric stiff ntions using the s stress calculat: stress calculat:	stress calculation. symmetric stiffness/mass contributions. ntions using the symmetric spooles solver stress calculation. stress calculation.

Postprocessing

Select register "Postprocessing" and the icon and stresses.

Files View M	esh Generation Edit FEM-	Project FEM-A	Analysis Postprocess	ing Training	
Show Results	Displacement-Factor Pick, Search Values	egend 1 + iagram 1 +	DXF-Postprocessing Value-Animation 👻	FEM INP STA FRD	FKM-Richtlinie for wel +
stprocessing 🗔	Factor/Values 🕞 L	egend/Diagram 🗔	Animations 🕞	List Files 🕞	Fatique-Analysis 🕞
	4				
	🖳 Postproces	ing		- 0	×
	Results: O Contour o Nodal Str O Element S Accuracy: Edit Accu 1	f Displacement ess Contour itress Contour racy:	Load Case: 1 O Reaction Force O Contour of Force Displacement Edit Colours for Pick, Search, S	es ces it Factor ir Legend ave Values	
	Select Result (Dist COT CSL Car Dist Dist Dist Dist Dist	Component: lacement in x directio VEN IP1 lacement in x directio lacement in x directio lacement in x, y and lacement in x, and y c lacement in x and y c lacement in x and y c	n n z direction lirection lirection		

Max. Displacements in x-direction = 74 mm (exactly 77 mm)

🚮 Demo FEM System MEA	NS 172 - FDM Structure File C projekte/vitroplens/glassifiem	-	o ×
	Net Assessed as the Particle Assessed as a second		
Show Result	Digitacementan April Carryon Community State Community Composition (Community Composition Community Composition Community Composition Community Composition Community Composition Community Communit		
LOAD CASE= 1			
Displacements in x direction			
74,569 63,915 53,261 42,606 31,952 21,298	Exactly: vx = f * L ¹ /E *1 = 1000 N * 1250 ³ mm ³ * 12 / 3 * 75000 //mm ² * 400 mm * 15 ³ mm ³) = 77 mm		
10.644 -0.0100			
y y z z			

to evaluate the displacements

Max. v.Mises-Stress of the Glass-Plate = 88 N/mm² (exactly 83 N/mm²)

Max. Tensile-Stress of the Alu-Bracket = + 57 N/mm² Min. Compressive Stress of the Alu-Bracket = -71 N/mm²



Part 2: Linear Statics-Analysis of a laminated Glass Plate

The previous glass plate 1333 mm x 400 mm x 15.33 mm is now calculated as laminated glass with a three-layer glass structure and the following material data:

Glass Pane 1: Thickness = 6 mmModulus of Elasticity = 75000 N/mm²P = 0.17PVB Film:Thickness = 3.33 mmModulus of Elasticity = 3 N/mm²P = 0.498Glass Pane 2: Thickness = 6 mmModulus of Elasticity = 75000 N/mm²P = 0.17



Create Layer 1 with 3D Mesh Generator GMSH

Glass pane 1 with the dimensions 1333 mm x 400 mm x 6 mm is first entered in the CAD system and saved as a STEP file. In MEANS V12, select the "File" tab and the "New" menu and select "3D mesh generator GMSH (STEP).



Like NETGEN, GMSH is a very powerful STEP tetrahedron mesh generator that is very easy to use. Select menu "3D" to first generate a moderate tetrahedral mesh of 256 elements and 112 nodes, whereby the upper and lower mesh density should be the same.



≡ O X Y Z ♀ 1:1 S III ≪ I ▷ II> Done optimizing mesh (Wall 0.0156248s, CPU 0.015625s)

Refine by splitting

The mesh refinement will only be carried out later in MEANS V12. In GMSH, however, the "Refine by splitting" menu can also be used to mesh 8 times more finely.



Export in INP Abaqus format

Select the menu "File" and "Export" as well as "Mesh - Abaqus INP (* .inp)" and export the INP file with the same name so that it is automatically loaded and displayed in MEANS V12.



The glass pane is automatically displayed in MEANS V12 and has a Z depth of 0 mm to 6 mm.



Smooth the mesh

So that the FEM mesh can be clamped from 1250 mm to 1333 mm, create the following Range of Nodes with a Coordinate Range from X = 1150 to X = 1300 and

	Surface Nodes Lines Number of Nodes = 112 Edges from: 1
🔜 Create a Coordinate Range 💴 🗆 🗙	until: 56
from X: 1150 until X: 1300	Show Nodes Create a Range of Nodes Create a Range of Nodes
until Z: 0 from Z: 6	Surface Nodes Edge Nodes Surface
Create a Range of Nodes	Delete a Range of Nodes Delete Range of Nodes Edit a Range of Nodes Coordinate-Factor
	Node: 1 EDIT X: 0 0 0 Y: 0 0 0 0 Z: 0 0 0 0 0 Node numbering 0
Z	Element numbering Element groups Show Load Values N
Y X	Edit Node-Size:

replace all X values of the Range of Nodes with 1250 mm with "Coordinate Factor".

🛃 Coordinate Factor		-		\times
Input Factor				
O Multiply	🔘 Divide			
⊖ Add	Replace			
Change Axis				
Change x-values	with y-values			
O Change x-values	with z-values			
O Change y-values	with z-values			
Change Coordinates				
X Coordinates	Y Coordinates	🗌 Z Co	ordinates	
Using the select	ed nodes			
from node:	1			
to node:	112			
Coordinate Factor:	1250			
Moving point 0 /	0 / 0 through the node:	[1	
Cha	nge Coordinates with a f	Factor		

The mesh can later be clamped exactly from 1250 mm to 1333 mm. Save this model under the name "Layer_1.fem".



Create Layer 2

To move the PVB film to the center, select the "View" tab and

🖳 Coordinate Factor		Ø <u>-</u>		×
Input Factor				
O Multiply	O Divide			
Add	⊖ Replace			
Change Axis				
O Change x-values	with y-values			
O Change x-values	with z-values			
O Change y-values	with z-values			
Change Coordinates				
X Coordinates	Y Coordinates	Z Co	ordinates	
Using the select	ed nodes			
from node:	1			
to node:	112			
Coordinate Factor:	6			

"Node Modus" and "Coordinate-Factor" and add 6 mm in the Z direction. Then select the "FEM analysis" tab and "Model Dimensions" to check the new Z depth of 6 mm to 12 mm.

🛃 Measures)		×
<u>Measures of the</u>	e activated El	ement Groups		
Width: from	0 mm	until	1333	mm
Height: from	0 mm	until	400 n	nm
Deep: from	6 mm	until	12 m	n

Move Range of Nodes

The nodes of the upper surface must be subtracted from 12 mm to 9.33 mm. To do this, create a Coordinate range of 11.8 mm to 12 mm in Node Modus.



Then select the Coordinate-Factor and add the value -2.67 mm in the Z direction to get the value 9.33 mm. With the "Model Dimensions" menu, the input must be checked again. Save the FEM-Mesh with "Layer_2.fem".

Create Layer 3

Load the first model "Layer_1.fem" again and add all Z values with a Coordinate-Factor of 9.33 mm. Save the mesh with a Z depth of 9.33 mm to 15.33 mm as "Layer_3.fem".

🔛 Abmessungen		-		×
Abmessungen	der eingeblende	eten Element	gruppen	
Breite: von	0 mm	bis	1333 m	m
Höhe: von	0 mm	bis	400 mn	1
Tiefe: von	9.33 mm	bis	15.33 n	nm

Merge Layers together

Load the first mesh "Layer_1.fem" again with the "File" tab and load the mesh "Layer_2.fem" and "Layer_3.fem" one after the other with the "FEM-Merge" menu, so that a main mesh with 3 layers is created.

Select the "Edit FEM Project" tab and "Element Groups" to display the mesh made up of 3 element groups.



Node-Check

Carry out a Check Node Numbering with the snap radius "3.2" in the register "Mesh Generation" in order to delete all double nodes. The structure now consists of 768 tetrahedral elements and 226 nodes.

9	Files	View	Mesh Generation	Edit FEM-Project	FEM-Analysis	Postproc	essing	Training	
	J 3D	Mesh Ger al Refiner	nerators ment	QU	AD-Meshes, Refine, I	Delete	Check N Jacobi-E	lode Numberi Determinante	ng
	3D Mesh	Generatio	n 🕞 2D Mes	h Generation 🕞	Manipulate Mesh	es 🖓	Ch	eck Meshes	Fa
		🖳 Bi	tte warten		-		×		
		Ohn	e Überprüfung:						
		Stru	ukturmodell hat 336 K	notenpunkte und 768	Elemente sowie 3 E	lementgrupp	pen		
		Mit	Oberprüfung:						
		Str	ukturmodell hat 226 K	notenpunkte und 768	Elemente sowie 3 E	Bementgrup	pen		
		Far	ngradius: 3.2	2					
			Einzelne Knotenpunk	te ohne eine Element-	Verbindung löschen				
		[Cancel	only Hidden-Line	Numerierun	ng prüfen			

Refinement

Finally, select "Mesh Generation" tab and menu "Local Refinement" and twice the menu "FEM Mesh 8x finer" to get the laminated glass pane with 49 152 tetrahedral elements, 9541 nodes and 3 element groups.



Create Load Case 1

The panel is loaded with 1000 N on the left side. First the surface model must be created with the "View" tab. Then select the "Edit FEM Project" tab and "Point Load" and generate a Point Load in the Z direction with the load value = 1 N by clicking on the left Surface 4.



Select "Editor" and multiply the load value by the load factor 5.92, which can be calculated from 1000 N / 169.

edi	ingungen 🗹 Ran	dbedingungen d	arstellen Elemer	ntgruppen Materia	aldaten	6. Belastungen - Editor Temperatur
		Flächen-Mo	dus aktiviert - Fläche	e= 4		la l
_						
0	Belastungen			- 0	×	
	Nr.	Knoten	FHG	Wert	^	
•	1	1	3	-5.92		
	2	2	3	-5.92		
	3	3	3	-5.92		
	4	4	3	-5.92		
	5	9	3	-5.92		
	6	10	3	-5.92		🖳 Lastfall — 🗌
	7	11	3	-5.92		
	8	12	3	-5.92		Alduraliza Lastfalla 1
	9	113	3	-5.92	1	
	10	114	3	-5.92		Faktor= 5.92
	11	117	3	-5.92		
	12	118	3	-5.92		multiplizieren dividieren
						🔿 addieren 🔿 ersetzen
Akt	tueller Lastfall: 1	<	Anzahl La	stfälle: 1		
Ana	zahl Lasten/pro Lastf	all: 169 I	asttyp: 1	Knotenlast		CANCEL OK
			1			
	Neuer Lastfall (erzeugen	Last	fälle überlagem		
	l astfall lös	chen	Lastfälle a	ddieren und konieren		
	1. 10 1. 5	1.	T		= 7	
	Lastrall-Fa	iktor	Tempe	eratunast einiesen	_	
	Flächenlast->K	notenlast	Freihe	eitsgrade ändem		
	Knotenlast->Li	inienlast				
		-	014			

Clamped fixed

The laminated glass plate is fixed clamped from 1250 mm to 1333 mm. Select the "Edit FEM Project" tab, "Boundary Conditions" and "Define Coordinate Range" to fixed clamp the plate.

				>
ndbedingungen aktuell:	1125	Neu		
Randbedingung: 1E	-10			
ad sperren:				
in >	K-Richtung	in Z-Richtun	g	
🗌 in Y	(-Richtung	🗹 Einspannung	3	
n				
enmodus	◯ Rechtec	k aufspannen		
ne Knoten anklicken	🔘 alle ange	zeigten Knoten wä	ihlen	
natenbereich definieren	🔘 alle ange	zeigten Suifaces v	vählen	
Editor		RBs erzeuge	en	1
Editor		RBs erzeuge	en	1
I Editor	x	RBs erzeuge	en]
I Editor stenbereich – 1250 bis X:	×	RBs erzeuge	en	1)
I Editor etenbereich – 1250 bis X: 0 bis Y:	- × [1333] [400]	RBs erzeuge	en])
I Editor atenbereich – 1250 bis X: 0 bis Y: 0 bis Z:	-	RBs erzeuge	en])
I Editor	-	RBs erzeuge	en	

Finally, the input of the material datas of the 3 element groups and the Statics-Analysis with the Quick-Solver follow as already described in Part 1.

Results

This is followed by a comparison of the results between a monolithic glass pane and the 3-layer laminated glass plate with 2 glass panes and a PVB film.

Max. Z-Displacements monolithic Glass Plate = -72.23 mm (exactly = 72 mm)









Max. v.Mises-Stress = 85.2 N/mm² with a monolithic Glass Plate

Max. v.Mises-Stress = 222 N/mm² with a 3-layer laminated Glass Plate



Laminated Glass Effect

The linear statics analysis is only valid up to the yield point of glass - therefore unfortunately it cannot be taken into account that after the break the glass splinters stick to the film and thus increase the load-bearing behavior again. Research into this effect is currently being carried out on car windshields in particular.

Part 3: Calculate glass plate with table legs

The tetrahedron glass plate mesh from Chapter 26 is connected to a mesh with 4 pentahedron aluminum table legs.



The node imprint of the table legs at Z = 0 must be inserted into the glass plate mesh so that the glass plate and table legs have the same node structures.



Step 1: Create a Range of Nodes

Load the table legs mesh and select the "View" tab and the "Node-Modus" menu and "Surface Nodes" to create a range of nodes for surfaces 3, 5, 7 and 13 at Z = 0.



Step 2: Create a new Add-List

Select the "Mesh Generation" tab and "Local Refinement" as well as the "Create an additional Point-List" menu and save the range of nodes under "tab14.Node".

dges urfaces lume
lume
Node-File

Step 3: Create the node list of the Glass plate

Reload the glass plate mesh and select menu "Local Refinement" and "Step 1: Create a new Point-List and Surface List of the actual FEM-Mesh" to save the number of nodes and elements before inserting the add list.

Step 4: Insert the nodes from the .Node file

Then load the node list of the table legs with the menu "Load a Point-List from a Node-File" and check on the model whether the nodes are displayed correctly.

Step 5: Generate with Add-List

Now use the "Generate with the additional Point-List" menu to call up the mesh generator in order to insert the additional nodes of the table legs into the glass plate.

You get a new glass plate mesh with the knot imprint of the table legs



Step 6: Convert a pentahedral mesh into a tetrahedral mesh

If the table legs are made of extruded pentahedra, the pentahedron mesh must first be converted into a tetrahedron mesh with register "Mesh Generation" and menu "Quad Meshing / Refine ..." and menu "HEX8 / PEN6 -> TET4" before loading.



Step 7: FEM Merge

Then reload the the table legs mesh to the glass plate mesh with the register "File" and "FEM Merge" and finally delete the overlaid nodes from the structure with the register "Mesh generation" and the menu "Node Check".

The glass plate and aluminum table legs are now combined in one FEM mesh with 2 element groups.



Calculate CAD Assemblies

In this way, moderate CAD Assemblies with different Material Datas like E-Modulus can be united together and calculated step by step.



Postprocessing

Select register "Postprocessing" to obtained the results when the glass plate is loaded with 100 kg weight and the table legs are fixed clamped.



Maximal Displacements in Z-Direction = -7.17 mm

Maximal v.Mises-Stresses = 41 N/mm²





Minimal Principal Stresses S3 = - 48 N/mm²

Maximal Shear-Stresses TAUxy = 9.3 N/mm² Minimal Shear-Stresses Tauxy = - 11.25 N/mm²

