Part 37: Drop Test FEM Simulation of a 10-Ton Rock falls onto Car Roof

The deformations and stresses are to be simulated using the Finite Element System MEANS V13 (<u>www.femcad.de</u>) when a 10-ton rock falls onto a car roof.

The car body is made of lightweight aluminum, which is used in almost all electric cars today due to the additional battery weight of 100 kg.

Disadvantages, however, are the higher plastic deformations and the lower yield strength.

What are the:

- Impact force
- Impact speed
- Fall height of the rock
- Fall time of the rock

when the A-pillar has deformed by -90 mm?



FEM model of the car and FEM model of the rock for a nonlinear contact analysis





v. Mises stress distribution of 76 N/mm² in the A-pillar area.

Calculating the impact force

The impact force can be read from the Load-Displacement Diagram for a deformation in the Y direction of -90 mm at node 9515.



The impact force of the rock is 3,600,000 N or 3,600 kN.

Calculating the impact velocity

First, the mass of the rock is calculated using a gravitational load, with a modulus of elasticity of 19,000 N/mm² and a density of 2,000 kg/m³. After the calculation, the sum of the support forces in the Y direction can be read as 12,188 N, resulting in a rock mass of 1,219 kg.

With the impact force, the mass, and the deformation of -90 mm, the impact speed can be calculated using the braking distance formula:

$$F = \frac{mv^2}{2*S}$$

F = impact force

m = rock mass

S = distance traveled during an impact

v = average speed of the rock and car

This is solved for v:

$$v = \sqrt{\frac{F * 2 * S}{m}} = \sqrt{\frac{3600 \text{ kN}^2 * 0.09 \text{ m}}{1219 \text{ kg}}}$$
$$= 23 \text{ m/s} = 82 \text{ km/h}$$

The impact speed of the rock is twice as high due to the stationary car:

$$v = (v_{Auto} + v_{Fels}) / 2$$

The impact speed of the rock is therefore 46 m/s or 166 km/h.

Calculating the Fall Height and Fall Time

Using the speed of the rock, the fall height and fall time can be calculated using the free fall formula:

$$V = \sqrt{\frac{2 \cdot g \cdot H}{2 \cdot g}} = \frac{46 \text{ m/s} \cdot 46 \text{ m/s}}{2 \cdot 9.81 \text{ m/s}^2} = 105 \text{ m}$$
$$t = \sqrt{\frac{2 \cdot H}{g}} = \sqrt{\frac{2 \cdot 105 \text{ m}}{9.81 \text{ m/s}^2}} = 4.6 \text{ s}$$

The fall height of the rock is 105 m with a fall time of 4.6 s.

FEM Model Car

The FEM model Car is generated from a STEP file from <u>www.grabcad.com</u> using the 3D mesh generator NETGEN. However, six "bad edges" or six unconnected edges must first be optimized using the repair tool Transmagic.

Number of analytics made from splines: 0 Pranes, 0 Spheres, 0 Cornes, 0 Tori 40 Interves, 28 Interves, 28 Interves, 28 Interves, 4 Elipses, GEOMBUILD FIX RESULTS : The deaded body after geombuild fix : mode deges = 187 mode dedges = 3 mod coedges = 374 mod bad edges = 4 mod vertices = 6 mod bad tangent edges analytic = 1 mod bad tange	eometrie-Informationen		
Statistics of the healed body after geombuild fix: no. of edges = 187 no. of bad edges = 3 no. of bad coedges = 374 no. of bad coedges = 374 no. of bad tangent edges = 3 no. of bad tangent edges analytic = 1 no. of 61 bad tangent edges analytic = 1 no. of bad tangent edges analytic = 1 no. of bad tangent edges uv_unv = 2 no. of bad tangent edges uv_unv = 2 no. of bad tangent edges uv_unv = 0 no. of bad tangent edges noruv_nonuv = 0 no. of bad tangent edges analytic = 0 percentage of good geom = 98	Number of analytics made from splines: 0 Planes, 0 Spheres, 0 Cylinders, 0 Cones, 0 Tori 40 Inteurves, 28 Inteurves, 8 Straights, 0 Circles, 4 Ellipses, GEOMBLIED FIX BESTIETS -	~	
ОК	Statistics of the healed body after geombuild fix : no. of edges = 187 no. of bad edges = 374 no. of bad coedges = 374 no. of bad coedges = 4 no. of vertices = 122 no. of bad tangent edges = 3 no. of bad tangent edges analytic = 1 no. of G1 bad tangent edges analytic = 1 no. of bad tangent edges uv_uv = 2 no. of bad tangent edges uv_uv = 2 no. of bad tangent edges vu_nonuv = 0 no. of bad tangent edges vu_nonuv = 0 no. of bad tangent edges 3_4.sided = 0 no. of surfaces = 57 no. of discontinuous surfaces = 0 percentage of good geom = 98		bad edges
	ОК	•	\mathbf{x}

After mesh generation, you get an FEM model consisting of 197,248 tetrahedra and 41,070 nodes. Save the FEM model as Car.FEM.



FEM Model Rock

Create a CAD cube measuring 1000 mm x 500 mm x 1000 mm and create an FEM model consisting of 19,136 tetrahedra and 4,053 nodes. Perform a zero point shift through node 3 using the "Coordinate Factor" menu in Node Mode and save the FEM mesh as "Rock.FEM."

		🖷 – 🗆 X
		Flächen Knoten Linien
		Anzahl Eckknoten = 4053
		Kanten 🗸
		von: 1
	Roordinaten-Faktor – L X	bis: 204
a da anti-a da anti-	Faktor setzen :	
		Knoten anzeigen
		Knotenbereich erzeugen
	O addieren O ersetzen	Knotenbereich erzeugen
	Achsen vertauschen	Flächenknoten
	O X-Werte mit Y-Werte vertauschen	Rächen-Randknoten
	○ X-Werte mit Z-Werte vertauschen	Knotenbereich löschen
	O Y-Werte mit Z-Werte vertauschen	Knotenbereich löschen
	Koordinaten mit Faktor verändem	Knotenbereich ändern
		Koordinaten-Faktor
	🗹 X-Koordinaten 🗹 Y-Koordinaten 🗹 Z-Koordinaten	Kooten: 3 EDIT
No. 199	🗖 nur die angezeigten Knoten im Knotenmodus verwenden	
A. A		X: 0
	von Knotenpunkt: 1	r: 0
	bis Knotenpunkt: 4053	Z: 0
	Keesterbefelten (Knoten numerieren
	Noordinatemattor.	Bemente numerieren
	Nullpunktsverschiebung durch Knotenpunkt: 3	Elementgruppen numerieren
		Lastwerte anzeigen N
/	Koordinaten mit Faktor verändem	
Knoten 3 in den		Knoten-Size editieren:
Nullpunkt verschleben		
	Verformungen mit Faktor addieren:	
	. New state of the second se	

Merge FEM Models

First, load the FEM model Auto.FEM normally. Then, select the "File" tab and the "FEM Merge" menu to add the second FEM model Rock.FEM.



You will receive an FEM model consisting of 216,384 TET4 elements, 45,123 nodes, and two element groups.

Coordinate Move of the Rock



Display all nodes of element group 2 in Node Mode

and use "Coordinate Factor" to move the rock by three coordinate moves at the following distances:

410 mm in the X direction
315 mm in the Y direction
500 mm in the Z direction

First coordinate move of element group 2 by - 410 mm in the X direction:



Third coordinate move of element group 2 by 500 mm in the Z direction:

🖳 Koordinaten-Faktor	8.—		×
Faktor setzen :			
O multiplizieren O dividie	ren		
addieren O ersetze	en		
Achsen vertauschen			
O X-Werte mit Y-Werte vertauschen			
O X-Werte mit Z-Werte vertauschen			
O Y-Werte mit Z-Werte vertauschen			
Koordinaten mit Faktor verändem			
X-Koordinaten Y-Koordinater	Z-Ka	pordinaten	
✓ nur die angezeigten Knoten im Knote von Knotenpunkt.	nm <mark>odus verv</mark>	venden	
bis Knotenpunkt: 45123			
Koordinatenfaktor: 500			
Nullpunktsverschiebung durch Knoter	npunkt: [1	
Koordinaten mit Faktor v	erändem		

Material Data

Using the "Edit FEM Project" tab and the "Material Data" menu, enter the following material data:

Element group 1: Car body Material: Aluminum Young's modulus: 71,000 MPa Poisson's ratio = 0.34 Density = 2,700 kg/m³

Yield strength = 70 MPa -> the nonlinear stress-strain curve for Alu 999 must be selected in the FEM analysis "Material-Nonlinear"

Element Group 2: Rock Material: Natural hard rock granite Young's modulus: 100,000 MPa Poisson's ratio = 0.25 Density = 3100 kg/m³

Create boundary conditions

Create a fine surface model in Surface mode with the "Very Many" setting and use "Edit FEA Project" and "Boundary Conditions" to firmly constrain the soil in the X, Y, and Z directions.



Create prescribed boundary conditions

The upper and lower surface of the rock are fixed in the X and Z directions, and a prescribed displacement of "-200" mm is specified in the Y direction.



Create any desired, but very small, nodal load, as the external load is already specified by the boundary conditions above.

Create Load Case 2 with Master Contact Surface

Create the master contact surface by hiding element group 2 using Elementgruppen and then displaying the surface nodes of surface 5 in Node Mode.



Reduce this node of range again with "Create a Node Range" with a second node range from -420 mm to 620 mm in the X direction. .

		Max			
1100		1		4	
		1		-	
				Creation of the second s	
			相联/人		2
			14		
Knotenbereich erzeugen	- 0	×	- A		
Bitte mit der Maus ein Rechteck aufspannen oder einzelne	Knoten anklicken!		Koordinatenbereich	- 0	
A			1 10 10	-	
	Neu		von X: -420	bis X: 620	
Knotenbereich aus Knotenbereich erzeugen	Help		von Y: -278.779	bis Y: 815	_
Selection			von 7: 500	bie 7: 500	
Rechteck aufspännen Knoten picken	Koordinatenbereich				
			Knotenb	ereich erzeugen	
Knotenbereich sichem Knotenbere	tich erzeugen				
Knotenbereich einladen Car	ncel				



Finally, a surface load is created with load case 2 and the "all displayed nodes" selection without a load value.

Datei Ansicht Netzg <u>F</u> Belastungen Belastungen darstel	enerierung FEM-Projekt bearbeiten Image: serierung	FEM-Analyse I edingungen + dingungen darstellen	Ergebnisauswertung	Training De Materialdaten	6. Bela
		Knoten-Modus al	ctiviert		
🖳 Flächenlast erzeugen	×				l
Aktueller Lastfall: 2 Anzahl Lastwerte: 1272	· +				
Wert der Flachenlast: .01	O N/mm ² (oder in N	<u>}</u>			Ц
Wert Belastung in N: 1000	Info		Souther	States	
Freiheitsgrad: O X-Richtung O Y-Richtung	 Z-Richtung senkrecht zur Fläche 				
Selektion:	-/-				
O Flächenmodus) Rechteck aufspannen				714
🔿 einzelne Knoten anklicken	alle angezeigten Knoten	States	a se		
Koordinatenbereich definieren) alle angezeigten Surfaces				
Flächenlast-Wert (N/mm²) aus Belastur	ng (N) FL-Farbe:				
Cancel Editor	Belastung erzeugen				
	Belastung löschen		<u> </u>		

Create load case 3 with a Slave Contact Surface

Create the slave contact surface by redisplaying element group 2 and hiding element group 1, and create a surface load with load case 3 and the "surface mode" selection on the lower surface of the rock.

🖳 Flächenlast erzeugen	– 🗆 ×	V V V V V V V V V V V V V V V V V V V
Aktueller Lastfall: 3	• +	
Anzahl Lastwerte: 800	Neu	
Wert der Flächenlast: .01	O N/mm² () oder in N	
Wert Belastung in N: 0.01	Info	
Freiheitsgrad: O X-Richtung	O Z-Richtung	
O Y-Richtung	senkrecht zur Fläche	
Selektion:		
Flächenmodus	O Rechteck aufspannen	
🔿 einzelne Knoten anklicken	🔘 alle angezeigten Knoten	
O Koordinatenbereich definieren	🔿 alle angezeigten Surfaces	
Flächenlast-Wert (N/mm²) aus Bela	stung (N) FL-Farbe:	
		There is at all be be at at at an at a day day day day day day day
Cancel Editor	Belastung erzeugen	
	Belastung löschen	
	Ashrin a she had been been	

Nonlinear Contact Analysis

The FEA assembly consisting of the car body and the rock can now be calculated using a nonlinear contact analysis. Select the "FEA Analysis" tab and the "Material - Nonlinear" menu and define:



Use the "Select a Stress Curve from the Database" menu to select the stress-strain curve "ALUMINIUM PURE 99 996 ANNEALED."

Use the "With NLGEOM Solver and Time Steps" option to configure the solver for large deformations and adopt the number of increments, initial time, step time, minimum time, and maximum time from the default settings.

Use the "With Contact Analysis" option to enable contact analysis.

			(777)	
Material Law:				
O Linear Elastic				
Nonlinear with Hardening	Isotrop			
O Nonlinear with Hardening	Kinematic			
O Nonlinear with Hardening	Combined			
Select a Stress-Strain-Cur	ve from the Databa	se	Edit	
Plot Stress-Strain-Curve	Plot S	tress-Displa	acement-Curve	e
	and the second second			
Mit NLGEUM-Solver und ∠e Anzahl Inkremente: Anfangszeit: Solvettaeit.	200			
Mit NLGEUM-Solver und Ze Anzahl Inkremente: Anfangszeit: Schrittzeit:	200			
Mit NLGEOM-Solver und ∠e Anzahl Inkremente: Anfangszeit: Schrittzeit: Mindestzeit:	200 0.3 1 0.1E-7			
Mit NLGEUM-Solver und ∠e Anzahl Inkremente: Anfangszeit: Schrittzeit: Mindestzeit: Maximalzeit:	200 0.3 1 0.1E-7 1		Default	
 mit NLGEUM-Solver und Ze Anzahl Inkremente: Anfangszeit: Schrittzeit: Mindestzeit: Maximalzeit: mit Kontakt-Analyse 	200 0.3 1 0.1E-7 1		Default	
mit NLGEUM-Solver und Ze Anzahl Inkremente: Anfangszeit: Schrittzeit: Mindestzeit: Maximalzeit: mit Kontakt-Analyse Belastung: Lastfal	200 0.3 1 0.1E-7 1 1 1 1		Default	
 mit NLGEUM-Solver und Ze Anzahl Inkremente: Anfangszeit: Schrittzeit: Mindestzeit: Maximalzeit: mit Kontakt-Analyse Belastung: Lastfal MASTER-Surface: Lastfal 	200 0.3 1 0.1E-7 1 1		Default	
mit NLGEOM-Solver und Ze Anzahl Inkremente: Anfangszeit: Schrittzeit: Mindestzeit: Maximalzeit: mit Kontakt-Analyse Belastung: Lastfal MASTER-Surface: Lastfal SLAVE-Surface: Lastfal	200 0.3 1 0.1E-7 1 1 1 1 1		Default	
mt NLGEUM-Solver und Ze Anzahl Inkremente: Anfangszeit: Schrittzeit: Mindestzeit: Maximalzeit: mit Kontakt-Analyse Belastung: Lastfal MASTER-Surface: Lastfal SLAVE-Surface: Lastfal CAN/CEL	200 0.3 1 0.1E-7 1 1 1 1 2 1 1 2 3 5 5 1 5 1 1 1 1 1 1 1 1 1 1 1 1 1		Default	2127

Postprocessing

Select the "Postprocessing" tab and click the icon



to evaluate the results.

Activate "Read all load cases" to first determine the number of load cases from the FRD file, as the FEM solver often converges slowly towards the end with the error message "too many cutbacks" and aborts the calculation.

MEANS V13	×
Number of Loadcases = 45 FRD-File has Lines = 10722690	
ОК	

Select "Model with Results Evaluation" from the menu with the desired load case to display the stress or deformation distribution.

Ergebnisse einladen	
O Verformungen auswerten	
Knotenspannungen gemittelt	O Auflagerkräfte auswerten
O Elementspannungen ungemittelt	O Knotenkräfte auswerten
Legende	
Raster-Genauigkeit:	Verformungsfaktor/Wertebereich
	Legende und Farben einstellen
1 3 4 8	Knotenwerte picken, suchen, sichem
	Schnitt-Surfaces mit Knotenbereich
Ergebnis-Komponente wählen	
v.Mises-Vergleichsspannun	g ~
Cancel	Modell mit Ergebnigeuswertung

Stress and deformation distribution

For the final load case 45, the following plastic deformations and nodal stresses with a stress-strain curve for aluminum result:



Maximum plastic deformation in the Y direction = -117.20 mm

v.Mises nodal stresses = 76 MPa



Sum of impact force = 3,800,795 N = 3,800 kN = 380 tons

Select the "Postprocessing" tab, "Pick Node Values," and "Sum of Reaction Forces" to display the sum of reaction forces.

×
äfte = 3800795 N
ОК

Load-Displacement-Diagram

In addition, the impact force on the A-pillar at node 9518 can be determined from the load-deformation diagram.



Select the "Postprocessing" tab, then the "Stress-Strain Diagram" menu to open an 11-column dialog box.



Enter node 9518 and select "Start" to list the deformations, stresses, and forces at that node for each load case.

Anzahl Lastfälle =	45	Knoten für Verformungen =	9518	\sim	in Y	\sim
Ausgangslänge L0 =	50	Knoten für Spannungen =	9518	~	v.Mises	\sim

After a few minutes, select "Display diagram and select with" and "Load-Displacement" to display the load-deformation diagram.

	Knoten für Spannungen = 95	i18 🗸 v.Mises 🗸	Result-File: C:\projekte\auto	\car\carfels2.FRD	
LF Knoten 9518 9558	X-Verformung Y-Verformung 7-732820 -41.443 7-73826 -46.56 800758 -46.008 8207758 -46.008 8207758 -46.908 820775 -47.511 954167 -51.051 954167 -54.162 973322 -56.313 -10.142 -57.423 -10.548 -56.625 -10.544 -56.625 -11.130 -65.017 -11.131 -62.393 -11.132 -65.017 -11.133 -65.017 -11.130 -67.150 -12.994 -69.237	Z-Verformung Dehnung=Verf./L0 77 14169 ∩ 77 38260 0.09024081 77 38260 0.09024081 77 38260 0.09124081 800758 1.01657 800758 1.10748 8,34553 1.1187618 8,34453 1.1187618 9,3932 1.44061 9,3932 1.53976 101.345 1.155784 10.5848 1.1057784 11.1.14 1.1557784 11.1.141 1.3228916 11.1.130 2.000348 11.1.93 2.200384 11.1.93 2.232438 7.24500 2.232438	Knoten Spannung 9518 76<429 9518 76<427 9518 76<427 9518 76<426 9518 76<426 9518 76<426 9518 76<426 9519 76<425 9518 76<424 9519 76<424 9519 76<424 9519 76<424 9510 76<424 9513 76<424 9513 76<424 9513 76<424 9513 76<421 9513 76<421 9513 76<421 9513 76<419 9513 76<419 9513 76<416 9513 76<416 9513 76<416 9513 76<400 9513 76<400	Last FX Last FY 0.00 0.00 0.00 0.00 0.00 0.00 0.00 0.	Last FZ Last FZ

Load-Displacement Diagram

A plastic deformation of -70 mm results in an impact force of approximately 350 tons at node 9815. This is a plausible force when compared to the pressing force of an automobile press, which is over 500 tons.



Stress-Strain Diagram

Up to the aluminum yield strength of 75 MPa, stresses and strains are elastic. After that, plastic deformation begins, meaning the strains increase but the stresses change only slightly. The material begins to flow, and irreversible permanent deformation occurs.

