AUTODESK INVENTOR Trial Projects

Dynamic Simulation Analyze front loader assembly



In Inventor, click the 'Projects' icon in the ribbon. Navigate to where you saved the project files and select *Assy, Chassis, Front DS.ipj*. Then open *Assy, Chassis, Front DS.iam*.

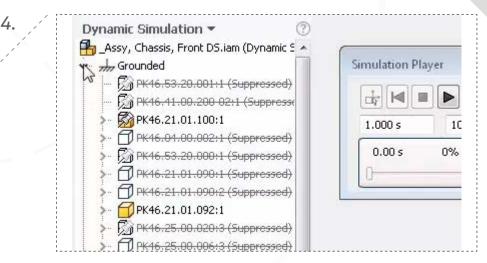
+ □ + ₩}			\$	*	(
Dynamic Simulation	Stress Analysis	Frame Analysis	Inventor Studio	Cable and Harness	Tu
				Begin	
			2	<	
Model -			(0	
	lodelina V	iew 🔻 🌢	4		
₹ ° @ M	second .				

Select 'Concentric' from the drop-down for the placement.

✓ 🔓 Level of Detail : Master Master All Components Suppressed 占 All Parts Suppressed 👝 All Content Center Suppresse Default Dynam Repeat Update Origin Fasteners Delete PK46.21.01.100 Activate PK46.80.00.000 Copy PK46.21.01.200 Copy to View Rep PK46.25.02.010 PK46.25.03.000 How To

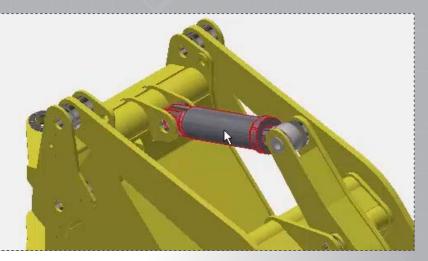
2.

Right click the 'Level of Detail Dynamic Simulation' in the browser and select 'Activate'.



In the browser, collape the 'Grounded' item.

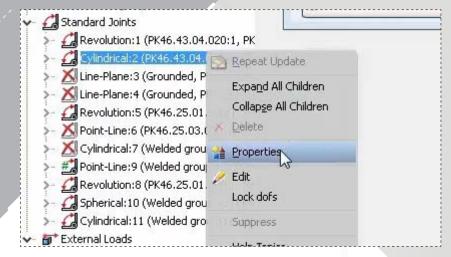




In the Graphics Window, select the hydraulic cylinder assmebly, 'PK46.43.04.0202:1 (~1)'.

			2
<u> </u>	Position:		
	400.000 mm	C Locked	
	Velocity:		
	0.000 mm/s	Computed	
Bounds	Value:	Stiffness:	Damping:
🔲 Min.	400.000 mm	0.000 N/mm	0.000 N s/mr 🕨
Max.	400.000 mm 🕨	0.000 N/mm 🕨	0.000 N s/mr +

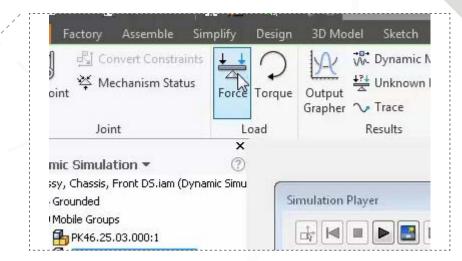
In the 'Joint Properties' dialog box, on the dof 2 (T) tab, enter initial condition value of 400 mm. Then click 'OK'.



6.

8.

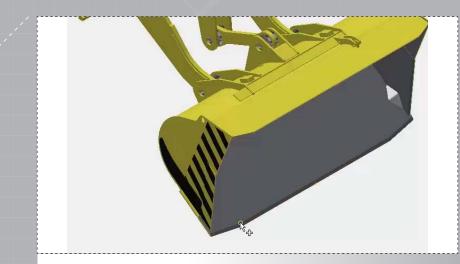
In the Browser, under 'Standard Joints', right click 'Cylindrical:2' and select 'Properties' from the short cut menu.



Apply load force 'Dynamic Simulation > Load > Force'.



page: 4

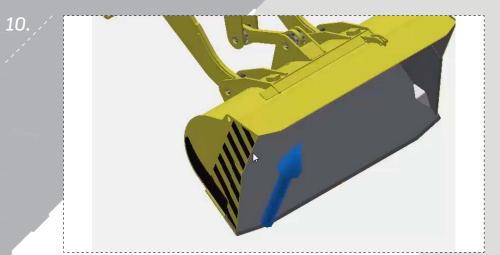


In the graphic window, select a point on the bottom of the bucket as shown.

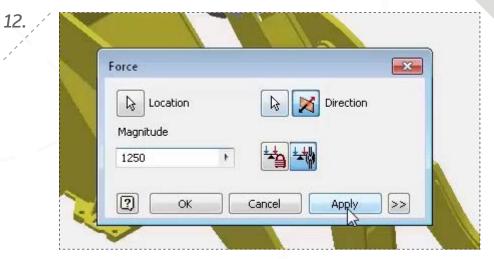
11.

Force		×
Location	Direction	
Magnitude		
0.000 N		
	Cancel Apply	

Flip the load direction so that it is pointing down.



Select the edge shown to specify the load direction.

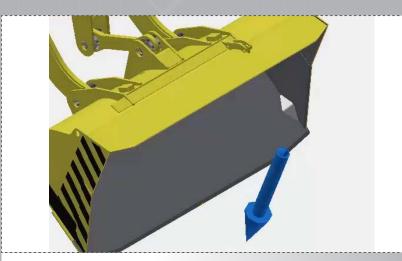


Switch to 'Associative Load Direction', and enter a value of 1250 N, then click Apply.



13.

15.



Repeat step 9 - 12 on the mirror oposite side of the bucket.

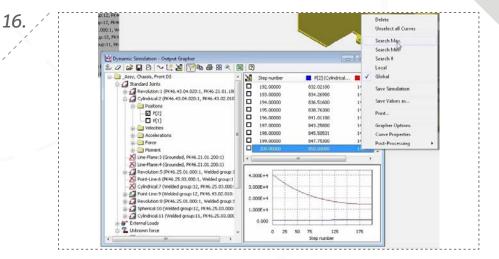
Force Location 1 Torque Jack Kinematics Jant: Cytindinusic (HK46.43.04.020.1), PK46.43.02.010.1) Totsi poston: Mobili poston:	Unknown Force		
Jack Kimematics Jork: Cysimakica/12 (MK46.43.04.020.11, MK46.43.02.010.11) Intelvid position: Intelvid position: Final position: B50.000 mm B50.000 mm Degree of freedom: # of steps:	© Force	Location 1	
Kinemakis Joint: Crylindrical:2 (PK46:45.04.020.1, pK46:45.02.010:1) • Initial position: 400.000 mm 850.000 mm Degree of freedom: # of steps:	Torque	Location 2	
Joint: Crylindricaliz (PK46-43.04.02011, PK46-43.02.01011) Initial position: 400.000 mm 850.000 mm Degree of freedom: # of steps:	Jack		
Cylindriceliz (PK66.43.04.020:1), PK66.43.02.010:1) Initial position: Initial positio: Initial position: Initial positi			
400.000 mm 850.000 mm Degree of freedom: # of steps:		1, PK46.43.02.010:1) -	
Degree of freedom: # of steps:			
		850.000 mm	
		Cancel <<	
(D) OK () Cancel <<	Scale		
	Display 0.001		

On the 'Unknown Force' dialog box, uncheck 'Display' then click 'OK'.

n 🕺 - 🗽 - 🐂 Inspect Simplify Design 3D Model Sketch Tools Manage ₩ Dynamic Motion 😤 Publish Movie Har Unknown Force Dublish to Studio itus Force Torque Output Grapher V Trace Results Load Animate x 2 (Dynamic Simu 23 Simulation Player 2 14 100

14.

Select 'Unknown Force Dynamic Simulation > Results > Unknown Force'



Once the simulation completes in the 'Output Grapher', right click on 'Step 200', select 'Search Max'.

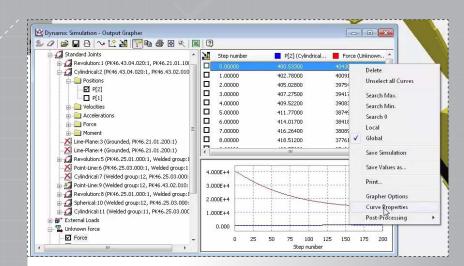




18

17.

19



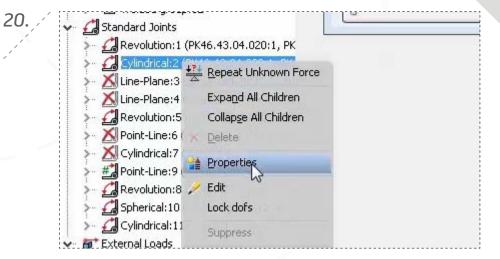
Right click the hightlighted step (Step 0), select 'Curve Properties'.

nic Simu	Simulation Pla	yer		
			P	2
	1, Construc	tion Mode	1	*
	0.00 s	0%	00:00	00
	0			
0:1, PK				

Exit the animation by clicking 'Construction Mode' on the 'Simulation Player'. Save your work.

Dynamic Simulation - Properties × **Curve Properties** Values On whole curve. On displayed area Minimum : Average Maximum 21682.27612 14296.20000 Median : Standard deviation : Amplitude 18935.00000 7519.68521 26134.20000 Multiplier Active Value : Color : 1 Center 2 Cancel OK Apply

Note the Maximum force of 40430.40 Close the 'Properties' dialog box. Close the 'Output Grapher'.



In the browser, under 'Standard Joints', right click 'Cylindrical:2' and select 'Properties' from the short cut menu.



21.

3	1			Ś	
	<u></u>			9	
	Enable joint force		Damping:		
	1	F	Constant val	ue	
	Spring Free position:		Input graphe	r	
	0.000 mm	•	0.000 N/mm	F.	
	Dry friction				
	Coefficient:				
	0.000	•			

In the 'Joint Properties' dialog box, on the 'dof 2 (T)' tab, select 'Edit Joint Force', check on 'Enble Joint Force', select 'Constant Value'.

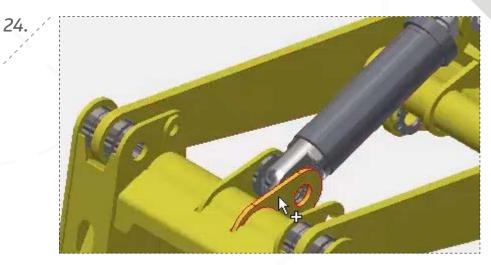


Select 'Export to FEA Dynamic Simulation > Stress Analysis > Export to FEA'.

1			~	1	5
0~		-			
I Er	able joint force		Damping:		
	41500	۲	0.000 N s/mm	+	
	Spring Free position:		Elastic stiffness:		í I
	0.000 mm	F	0.000 N/mm	•	
	Dry friction Coefficient:				
	0.000	•			

Enter a 'Constant Value' of 41500 N, Click 'OK'.

22.

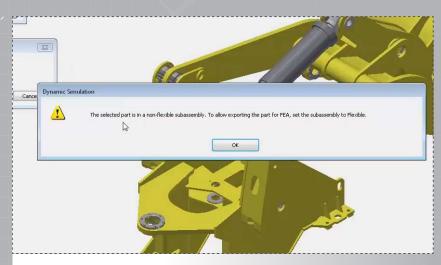


In the graphics window, select the part shown.

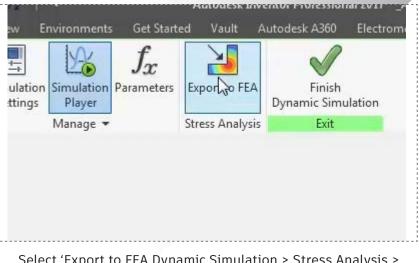




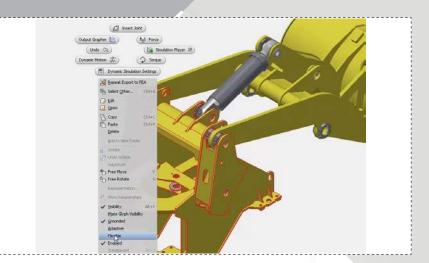
25.



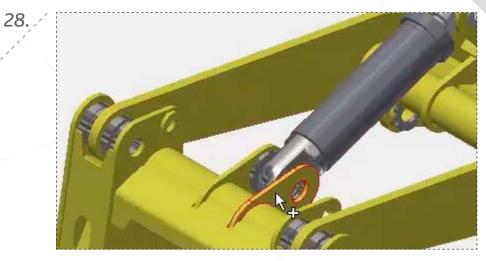
Select 'OK' on the warning message dialog box.



Select 'Export to FEA Dynamic Simulation > Stress Analysis > Export to FEA'.



In the graphics window, select the sub-assembly shown, right click and select 'Flexible' from the short cut menu.



In the graphics gindow, select the part shown. On the 'Export to FEA' dialog box, click 'OK'

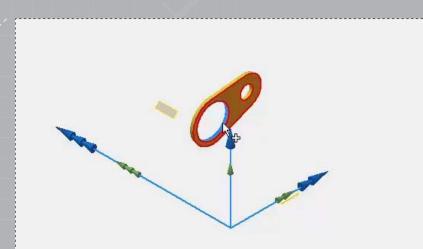


29.

31.

bup:14, PK4

PK46.21.01

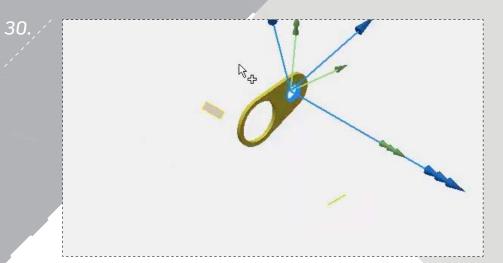


On the 'FEA Load-Bearing Faces Selection', under 'Joints to complete': 'Load Bearing', select 'Welding:13'. Select the large inside circular face shown.

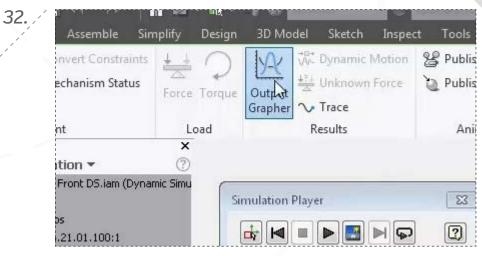
ion

namic Simu	Simulation Player	Σ
		in or replay the simula
	0.00 s 0%	00:00:00

Run the simualtion, click 'Run' on the 'Simulation Player'.



On the 'FEA Load-Bearing Faces Selection', under 'Joints to complete': 'Load Bearing', select 'Revolution:14'. Select the small inside circular face shown. Click 'OK'.

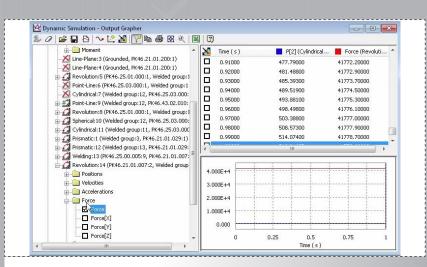


Open the 'Output Grapher'.

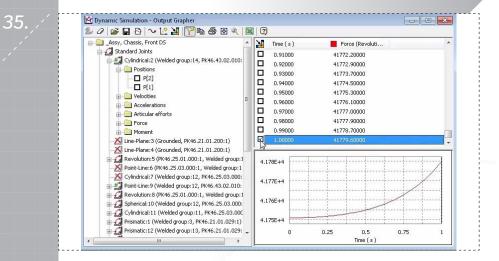
'Dynamic Simulation > Results > Output Grapher'.



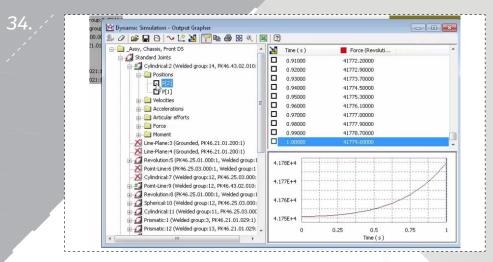
33.



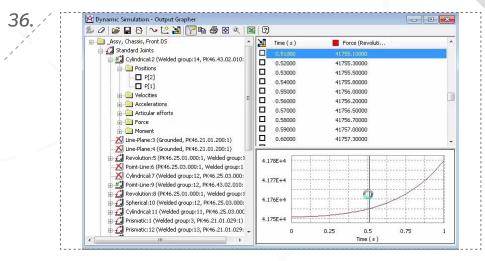
On the 'Output Grapher', expand 'Joint Revolution:14', then expand the Force folder, then check 'On Force'.



On the 'Output Grapher', check 'Time Step 1.00000' to export it.



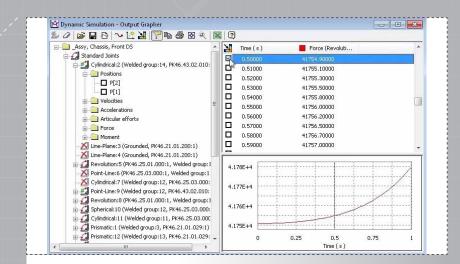
On the 'Output Grapher', under 'Joint Cylindrical:2', uncheck 'P[2]'.



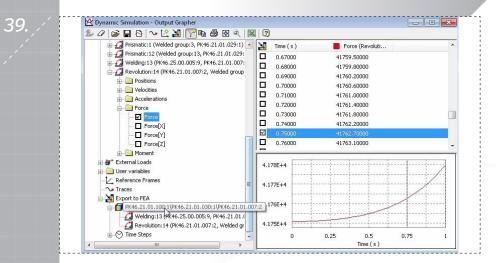
On the 'Output Grapher', double click on the graph at the '0.5 time mark'.



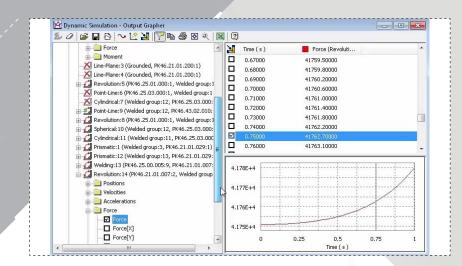
37.



On the 'Output Grapher', check 'Time Step 0.50000' to export it.

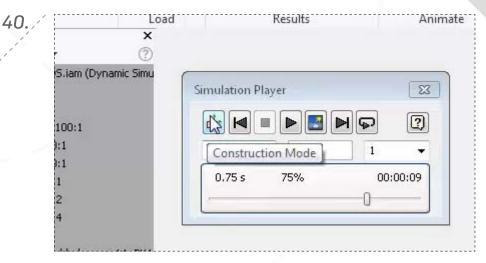


Notice that the time steps have been added to the 'Export to FEA' folder in the 'Output Grapher'. Close the 'Output Grapher'.



38.

Repeat previous steps for the 0.75 time mark in the graph.



Exit the animation by clicking 'Construction Mode' on the 'Simulation Player'. Save your work.



page: 12

		*	2	×°
Dynamic Simulatio	Sthiss Analysis	Frame Analysis	Inventor Studio	Cable and T Harness
	_			Begin
Dynami	c Simulat	ion 💌	0	(
-			(Dynamic Sim	

Start the 'Stress Analysis Environment, Environments > Beign > Stress Analysis'.

galanii zecki Androna Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sodd State Sudy Trynii Sudy Try	Create New Study Name:	Static Analysis:1		
State Analyse Part The State Part The State Part			-	
State Analyss State Analyst State Analyst State Analyst State Analyst State A	Study Type Model St	tate		
Spanista Dresses Across Contact Surfaces Across Across Across Contact Surfaces Part Prote 21:01:100:10/Post 21:01:00 Vacial Analysis Vacial	@ Static Analysis			
Part Tene Step Part U.S. Particular Participad Holds 0.000 Contracts Trapped Accentor Contracts Targenizal Stiffness	Detect and E	liminate Rigid Body Modes		
Part Term Step Pres 21.01.109.10945.21.01.00 • Et3.5 • Modal Audors • Modal Audors • Moder & B • mountry Storge • constant Friended Modes • phranced Aucuracy Contacts Tolerance Name Step • Contacts • Conta	I Separate Stre	esses Across Contact Surf-	aces	
Pede 21.01.100:19/Pede 21.01.00 • 10.0.5 • 10.0.5 • 10.0.5 • 10.0.5 • 10.0.5 • 10.0.5 • 10.0.5 • 10.0.5 • 10.0.0 • 10.00	P Motion Loads	Analysis		
Hudd Andyse Hudd Andyse Marker of Hodes Marker of Hodes Marker of Hodes Granuts Presided Hodes Defancer Acceses Contacts Telerance Acceses Talerance Named Strives Targetid Strives	Part		Time Step	
And Andrés Martine et Modes Martine et Modes Compute Preloadel Modes Compute Preloadel Modes De Nanced Accuracy Contacts Tolerance Tope 0.100 mm Straffeds Turgerstal Softmess	PK46.21.01.10	0:1\PK46.21.01.05 +	1:0.5	
Contacts Training Contacts Type Contacts Typ	C Modal Analysis			
Compute Prelixadid Modes Contacts Tolerance 0.100 mm Standad Variation Varia	☑ Number of Mo	odes	8	
Contacts Tolerance 0.00 m Introde Accus acy Internal Int	El Frequency Ra	ange	0.000 • 0.000	
Contacts Tolerance Type 0.100 mm Banded • Normá Saffress Targential Saffress				
Tolerance Type 0.100 mm Bondled • • Normal Soffness Tangential Soffness	E thanced Acc	08.90		
0.100 mm Bisnited • Normal Stiffness Tangential Stiffness	Contacts			=/
Normal Stiffness Tangential Stiffness	Tolerance	Туре		
				1
(and days) (manufacture)	0.000 N/nm	0.000 /	Avres .	
Shell Connector Tolerance 1.750 (as a multiple of shell thickness)			1.750	12

On the 'Create New Study' dialog box, check on, 'Detect and Elminate Rigid Body Modes', 'Separte Stresses Across Contact Surfaces', 'Motion Loads Analysis'.

Create Study	Parametric Table	Assign	Fixed Frictionless	Force Pressu
Ma	inage	Material	Constraints	Lo
			×	
Study	*		0	
1000		Front DS.iar	(7) m (Dynamic Simu	
1000		Front DS.iar	⑦ n (Dynamic Simu	

42.

Create a 'New Study Analysis > Manage > Create Study'.

1	Create New Study		
1	Name:	Static Analysis:1	
1	Design <u>O</u> bjective:	[Single Point +]	
	Study Type Model St	ate	
	Static Analysis		
	Detect and D	ininate Rigid Body Modes	
	😰 Separate Stre	esses Across Contact Surfaces	
	2 Motion Loads	Analysis	
	Part	Time Step	
		0:199096.21.01.05 - 102	
	Modal Analysis	10.75	
	📝 Number of Mo	des a 👔	
	Prequency Ra		00
	Compute Prei		
	Effhanced Acc	tur eCy	
	Contacts Tolerance	Туре	
	0.100 mm	Bonded	
	Normal Stiffness	Tangential Stiffness	
	0.000 N/mm	0.000 N/mm	
	Shell Connector T	olerance 1.750	

From the 'Time Step' drop down list, select the 't:1' time step Click 'OK'.



45.

47.

odesk A360
Electromechanical
Analysis

Animate
Same Scale
Smooth Shading *

Probe
Color Bar
Adjusted x1

Convergence
Probe Labels

Result
Display

Turn Off the display of the load glyphs by unchecking 'Boundary Conditions Analysis > Display > Boundary Conditions'.

Electromech Get Started Vault Autodesk A360 😤 Animate Midsurface I Same Ľ A Probe Color I Offset Simulate Mesh View 间 Convergence Probe Prepare Mesh Solve Result

Click 'Mesh Settings Analysis > Mesh > Mesh Settings'.

In the browser right click on (Materials)

- Cont 💩 Assign Materials

🗛 _Assy, Chassis, Front DS.iam (Dynamic Simu

🛱 _Assy, Chassis, Front DS.iam (Dynai

Show All Materials

Repeat Boundary Conditions

STUDY

🗸 🕌 Static Analysis:1

Scons

🌄 Mest

.

46.

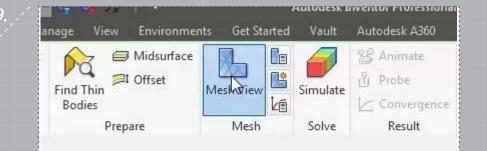
48.

In the browser, right click on 'Materials', select 'Show All Materials' from the short cut menu.

Common Settings		
Average Element Size	0.250	
(as a fraction of bounding box leng	th)	
Minimum Element Size	0.375	
(as a fraction of average size)		
Grading Factor	1.500	
Maximum Turn Angle	60.00 deg	
Create Curved Mesh Elements		
Assembly Option		
☑ Use part based measure for Ass	embly mesh	

On the Mesh Setting dialog box, enter the values: Average Element Size: 0.250, Minimum Element Size: 0.375, Check 'ON', 'Create Curved Mesh Elements'. Click 'OK'.





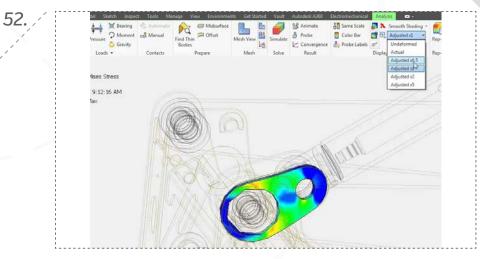
Click 'Mesh View Analysis > Mesh > Mesh View'. Turn off Mesh View by clicking 'Mesh View' again.

Model: _Assy, Chassis, Front DS.ia	am (Dynamic Simulation)
1 study, 1 configuration will be run	
Current configuration only	
Ready to run study.	

Click 'Run', on the 'Simulate' dialog box.

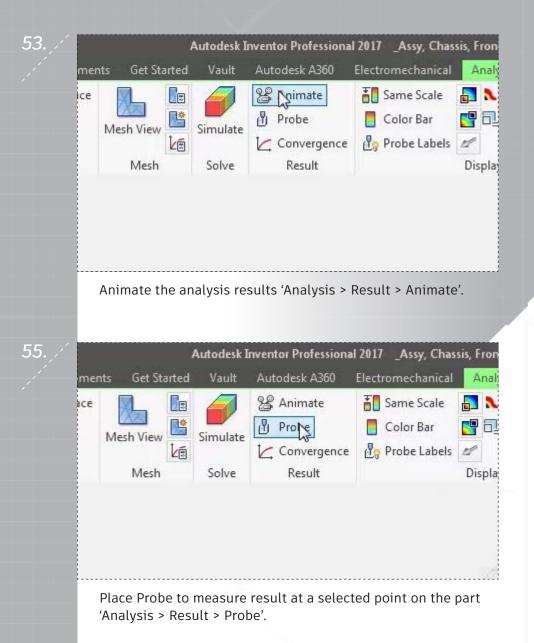
		nts Get Started	Vault	Autodesk A360	Electromec
/	Midsurface			🔓 Animate	Same
	n ^{∭‡} Offset	Mesh View	Simulate	Probe	Color
		iviesii view	Singher	Convergence	Probe
	Prepare	Mesh	Solve	Result	

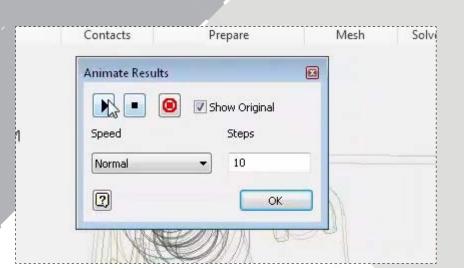
Run simulation, click 'Simulate Analysis > Solve > Simulate'.



Adjust the displacement display, Select 'Adjusted x0.5 Analysis > Display > Displacement Adjustment'.

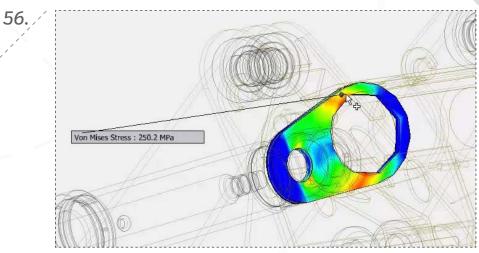






54.

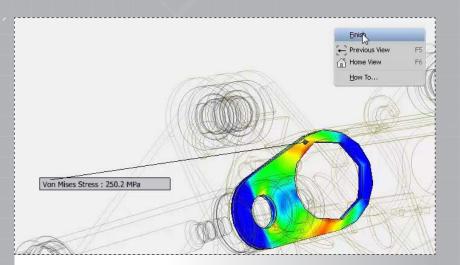
Click the 'Play' button on the 'Animate Results' dialog box. View the results animation. Areas of red are areas of concern. Click 'OK' to close the animation.



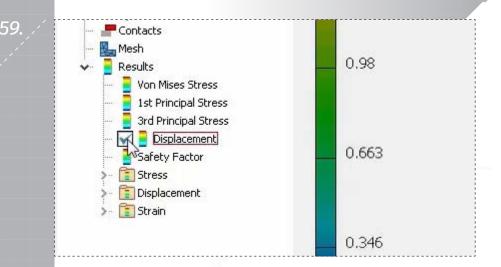
Click on the part in the graphics window to place a Probe.



57.



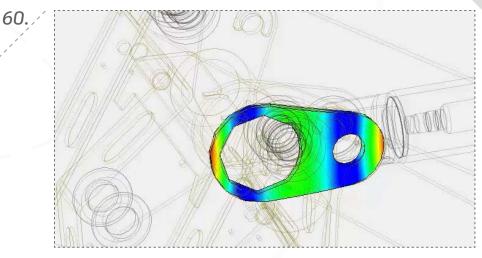
Right click in the graphics gindow, select 'Finish' from the short cut menu.



In the browser, double click 'Displacement' under the 'Results'.

58. atodesk Inventor Professional 2017 _Assy, Chassis, Front DS.iam (Dynamic Simi Vault Autodesk A360 Analysis C3 -😤 Animate Same Scale 🛐 👠 Smooth Shading 🔹 Probe Adjusted x0.5 🝷 Color Bar Simulate 🖧 Probe Lybels 💋 Convergence Solve Result Display

> Hide the Probe display by clicking Off the Probe Lables 'Analysis > Display > Probe Lables'.



View the Displacement results in the graphics window. Areas of red are areas of concern. To finsh the analysis, Click 'Finish Analysis'.





Autodesk, Autodesk Inventor, and the Autodesk logo are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product offerings and specifications at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document.

© 2017 Autodesk, Inc. All rights reserved.