AUTODESK INVENTOR Trial Projects

Shape Generator Design a stiff lightweight part



In Inventor, click the 'Projects' icon in the ribbon. Navigate to where you saved the project files and select Assembly Conveyor.ipj. Then open Assembly Conveyer.iam.



Next select the blue part, right click on it and select 'Open' to open the part. This will allow you to work on this part only.

	👗 🗈 - 🖙 🔚	← • →	· 🔺 🖻 •		📲 🛞 Mananat	- 0	Appearance		Q	$f_x + =$	
	File Assemble Simplify Design		Design 3	Select Component Priority		Too	s Manage	View E	ew Envi	nvironments	
	Distance	Material Materi	Appearance		Select Part Priority Select Part Priority Select Feature Priority Select Faces and Edges	ent Is	Exchange App Manager Options		Customize .inks Add-Ins	🎯 Macros	
	Model ▼ ▼ ⁴ # Assembly View ▼ ∰ Assembly Conveyor.iam >- ∰ 3rd Party		l.	90 77	Select Sketch Features Select Annotation Priority						
			Select Visible Only								
	Relationships Fig Representations Origin				Invert Selection Previous Selection						
	Assembly-1_1 Relationsh Grigin Grigin	ips ations	TEM 1		Select All Occurences Select Constrained To Select Component Size	-					

Next you will separate the blue part to lighten it. Begin by separating it from the assembly using the 'Select Part Priority' from the 'Selection Priority' drop down menu.



4.

On the 3D Model tab select 'Shape Generator' in the 'Explore' panel.



Steel, High Strength, Low Alloy	
Steel, MAN-TEN	
Steel, Mild	
Steel, Mild, Welded	
Steel, Non-Alloy	
Steel, Paint Finish, Blue	
Steel, Paint Finish, Dark Gray, Matt	
Steel, Paint Finish, Ivory, Glossy	
Steel, Paint Finish, Ivory, Matte	
Steel, RQC100	
Steel, Wrought	
Stone	
Stucco	
Teak	
Terrazzo	
Thermoplastic Resin	J
Document Materials	
🗸 Autodesk Material Library	
InventorMaterialLibrary	

You will next need to set some variables for Inventor to make calculations. Go to the Material drop down menu and select 'Steel, Mild' in the 'Autodesk Material Library' to set the material for this part.

Fixed Constraint
Location
Cancel Apply <<
Use Vector Components
□ × 0.000 mm
□ y 0.000 mm
z 0.000 mm
☑ Display Glyph
Name
Fixed Constraint:2

The 'Fixed Constraint' dialog appears.



Next create constraints by going to the 'Constraints' panel and select 'Fixed'.



8.

Select the two faces of the mounting pads as shown in the image below. Select 'OK'.





Next apply a load. Go to the 'Loads' panel and select 'Force'.

11.



Next select the inside face edge on all for all four mounting slots as shown in the image to the right.

k I	ocation
Magnit	
riagrina	
2	OK Cancel Apply <<
	Vector Components
_ Ose	vector components
Fx	0.000 N
Fy	0.000 N
Fz	0.000 N
V Disp	lay Glyph
Disp Scale	lay Glyph
V Disp Scale	lay Glyph

The 'Force' dialog appears, place a value of 100 in the 'Magnitude box'.



Select 'OK'.

10.





Next select the top face of the part as shown and then use the arrows on the box to expand it around the top portion of the part as shown in the images below.

Location Region Orientation Global XYZ Resize Options			
Region Box Orientation Global XYZ Resize Options Image: Move	Location		
Orientation Global XYZ Resize Options Image: Move	Region	Вох	T
Resize Options	Orientation	Global XYZ	*
	Resize Options	[♪ Move]	Size

The 'Preserve Region' dialog appears.



Select 'OK'.



17.

19.



Next define symmetry planes by selecting 'Symmetry Plane' in the 'Goals and Criteria' panel. The 'Symmetry Pane' dialog appears. Under 'Active Planes' select 'Horizontal' and 'Vertical' for directions.

View BIM Get Started Autodesk A360 Analys @ Bearing E E D Moment Preserve Symmetry Shape Generator Mesh View Gene 🎽 Gravity Region Plane Settings Sha Goals and Criteria Mesh Ru x 2

Next select 'Shape Generator Settings' in the 'Goals and Criteria' panel. The 'Shape Generator' Settings dialog appears.



Select 'OK'.

18.



Under 'Mass Target' select 'Reduce original by (%)' and place a value of 60 in it's box.



21.



Next select 'Mesh Settings' button in the 'Mesh' panel. The 'Mesh Settings' dialog appears.



 M
 Get Started
 Autodesk A360
 Analysis
 Image: Construction of the started sta

Select 'Mesh View' in the 'Mesh' panel. The part will begin to calculate.

Average Element Size	0.020
(as a fraction of bounding box length)	
Minimum Element Size	0.200
(as a fraction of average size)	
Grading Factor	1.500
Maximum Turn Angle	1.05 rad

Under 'Common Settings' in the 'Average Element Size' box set the value to 0.02.



The final mesh appears after it's calculation, as shown in the image below.



page: /





Finally select 'Generate Shape' in the 'Run' panel.

The final shape study is completed and a final generative shape is created as shown in the image below.

Senerate Shape	
Model: IONIZER_HOLDER_BRACKE	T.1_1.ipt
1 study, 1 configuration will be run	197 - 205 • 01
Current configuration only	*
Ready to run study.	
9	
	Pun Cancel

The 'Generate Shape' dialog appears. Click 'Run'.





Open the shape generated part you created in the previous project.

Promote Shape		
Promote Shape To		
Current Part File		
🔘 STL File		
a		C
	OK	Cance

30

The 'Promote Shape' dialog appears, select 'Current Part File' and select 'OK'.



29.

31.

Next you will promote this part into your part file. Select 'Promote Shape' from the 'Export' panel.

romote Shap)e	
Shap The Recc - Cl - Us	e promoted success shape was added to ommended next step nange the Visual Styl se the shape as a gu	fully! the part browser s le of the part to wireframe ide to modify your part
(ОК	Prompts >>

You will receive a prompt explaining to you that the shape has been successfully promoted select 'OK'.





32.



Next go to the 'Sketch tab' and start a new 2D sketch on the front face of the part.



Select 4 edges.



33.

In the 'Sketch' tab select 'Project Geometry' and project the edges existing part into the sketch.



You will next select the 'Line' command and place a line between the midpoint of the upper projected line and the midpoint of the lower projected line.



page: 10

36.



Select the line and in the 'Format' panel select 'Centerline' and change the line to a center line, as shown in the images below.



Using the mesh as a guide use the 'Line' tool to draw in line segments to cut away material from the existing design. Begin by using the 'Line 'tool in the 'Create' panel and follow the images above.



Line changes to centerline.



Next select the 'Mirror' command in the 'Pattern' panel.



page: 11



The 'Mirror' dialog box appears, click on 'Select' and expand a box over the line segment drawing you just created.

42.



Next select 'Finish Sketch' in the 'Exit' panel to exit out of the sketch.



41

43.

Next select the centerline created earlier and click 'Mirror Line' in the 'Mirror' dialog box. Then select 'Apply' to mirror the lines you just created.

D/		Pauska	Sweep	Emboss	Decal
2D Sketch	Extrude	Revolve	S Coil	🖒 Rib	
Sketch	-	.	Crea	te	
					×
Model •					(?)
7 #4					
		BDACKET :	1 1 int		

Next click the '3D Model' tab and select the 'Extrude' command in the 'Create' panel.







Then select all the closed profiles on the part as shown in the image below. Select 'Cut' from the drop down menu. Then select 'Through All'. Then select 'OK'.



Your model should look similar to the image above.



Go to the browser click on the mesh item and then right click to select 'Visibility' to turn the mesh off.



Next select the 'Fillet' button in the 'Modify' panel. The 'Fillet' toolbar appears, click 'Select Feature'.



48

50.

page: 14



Then select the extrude feature previously created and add a 5 mm radius to any corner in that feature. Select 'OK'.



Next add another group of fillets, two at the top and two at the lower portion of the part as shown in the images above. Place a value of 5 mm radius for all four fillets. Select 'OK' to create the fillets.



Create two more fillets on the upper corners of the part, make them 20 mm radius, as shown in the image above.



Finally, go to the browser, click on the mesh item and then right click to select 'Visibility' to turn the mesh on. Make sure to save the part file before moving on to the next project.



page: 15



Create Study	Assign	Fixed O Pin	Force Pressure
Manage	Material	Constraints	Loads 💌
Study TON	▼ IZER_HOLD hape Genera IONIZER_H Material Constraints Loads	X (?) IER_BRACKI ator:1 OLDER_BR4	

Next select the 'Create Study' in the 'Manage' panel under the 'Analysis tab'.



Select the 'Stress Analysis 'tool in the 'Simulation' panel under the '3D Model tab'.

55	Create New Study	
00.	Same:	Static Analysis:1
X	Design gbjective:	Single Point 🗸
	Study Type Model St	ate
	Static Analysis	
	Detect and Ele	iminate Rigid Body Modes
	Separate Stre	esses Across Contact Surfaces
	Motion Loads	Aralysis
	Part	Time Step
	O Nodal Analysis	
	E Number of Mo	ides 8
	Frequency Ra	MQN - 0.000 - 0.000
	Compute Prek	oaded Modes
	Enhanced Acc	STRY
	C Shape Generator	
1	Contacts	
	Tolerance	Type
	0.100 mm	Sonderd v
	Normal Softness	Tangental Softness
	S, Soc Hynan	10.000 (1998)
	Shell Connector To (as a multiple of sh	bell hidness)
	0	Reset OK Cancel

The 'Create New Study' dialog box apppears, select 'Static Analysis' under the 'Study Type' tab. Select 'OK'.



page: 16



Next set your constraints by going to the 'Constraints' panel and select 'Fixed'. The 'Fixed Constraint' dialog appears. Set the fixed constraint to the two bottom mounting faces on the part.



Next select the inside face edge on all for all four mounting slots as shown in the image above. Select 'OK'.



57.

Next you will next need to apply a force. Go to the 'Force' tool in the 'Loads' panel and in the 'Force' dialog box place a value of 100 N in the 'Magnitude' box, as shown in the images above.



Next select 'Mesh Settings' button in the 'Mesh' panel. The 'Mesh Settings' dialog appear. Under 'Common Settings' in the 'Average Element Size' box set the value to 0.02 and select 'OK".



60.

62.

page: 17



Select 'Mesh View' in the 'Mesh' panel. The part will begin to calculate. The final mesh appears after it's calculation, as shown in the image above.



Notice in the analysis that based on the shape generation that there is little-to-no stress at all.

Simulate	
Model: IONIZER_HOLDER_BRACKET.1_1.ipt	
Processing study 1 of 1.	
Current configuration only	N.
Study "Static Analysis: 1" in progress	
Computing results	

61.

Finally select 'Simulate' in the 'Solve' panel to run the study. The 'Simulate' dialog box appears, select 'Run'.



Next exit the analysis by selecting 'Finish Analysis' in the 'Exit' panel. Next go back to the **Conveyer Assembly** and notice that you now have a lighter part that is as stiff as the original intended design. Make sure to save all your files before closing.



AUTODESK[®] INVENTOR[®]



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.