

MFG463370-L

Stressing Out: Simulation Workspace in Fusion 360

Elizabeth Bishop University of Warwick, UK

Learning Objectives

- Learn about the basics of setting up a simulation in Fusion 360.
- Learn how to understand which simulation study type to use.
- Explore simulation outputs and how to use them.
- Explore iterative design methods.

Description

Fusion 360 software could be your most powerful tool as a designer, maker, or engineer. Fusion 360 is an all-encompassing piece of software that helps users with processes from start to end, design to manufacture, utilizing CAD, CAM, Simulation, and Rendering to envisage a design. This class will focus on the middle of that process—Simulation for Fusion 360. Simulation lets designers test (simulate) their designs before reaching the manufacturing process. This enables iterative changes without the time and money (and potential waste) that can go into making a product only to find it doesn't meet the standards needed. The Simulation workspace in Fusion 360 can feel a little intimidating at first glance—with so many options to choose from. This class will take you through the steps, showing you which study type of simulation can be used depending on what your aims are. You will leave this class feeling more confident and ready to improve your whole design process!

Speaker



Elizabeth Bishop is а Postgraduate Researcher at the University of Warwick researching Large-Scale Additive Manufacturing (3D Printing). She has been interested in 3D printing for several years now, following a successful project surrounding designing and making a humanitarian rescue UAV. Elizabeth also volunteers as a Maker in Residence in the Engineering Build Space at Warwick University where she explores making, CAD and CAM alongside 3D printing.

@LizBish94



Table of Contents

_earning Objectives	. 1
Description	. 1
Speaker	. 1
ntroduction	. 3
Static Stress	. 4
Exercise 1A+B: Static Stress, Spanner	. 4
Exercise 1C+D: Static Stress, Spice Shelf	. 9
Nodal Frequencies	14
Exercise 2A	14
Thermal	19
Exercise 3A	19
Thermal Stress	28
Exercise 4A	28
Structural Buckling	33
Exercise 5A	33
Nonlinear Static Stress	40
Exercise 6A	40
Event Simulation	48
Exercise 7A	48
Shape Optimization	58
Exercise 8A	58
Conclusions	69



Introduction

In this class we are going to go through the entire Simulation workspace in Fusion 360. We will go through the basics of setting up a simulation; how to choose the study type; how to review the results; and how these can be used to influence future design decisions. We will cover the following study types in the simulation workspace:

- 1. Static Stress
- 2. Modal Frequencies
- 3. Thermal
- 4. Thermal Stress
- 5. Structural Buckling
- 6. Nonlinear Static Stress
- 7. Event Simulation
- 8. Shape Optimization

There is a supplied data set for this class which you will need to upload into your data panel. For each section follow the example through the step-by-step instructions to solve the simulation study.



Static Stress

Static stress analyses can be used when a known load is applied in a static (non-dynamic) way to an object. The resultant stress, strain, and deformation results analysed to determine the likelihood of failure of the design. For a Linear Static Stress analysis, it is assumed that the model will behave in an elastic way i.e. will return to the original shape when the load is removed.

Exercise 1A+B: Static Stress, Spanner In this exercise we will perform a static stress simulation on a spanner.

1. Open the supplied file 1A_Spanner_No_Fillet





	Simplify geometry for use	in Simulation			
Static Stress	Addel Frequencies	Electronics Cooling (Preview)	Themal		1
Thermal Stress	Structural Buckling	Norlinear Static Stress	Event Simulation	Simplify Choose "simplify" to clean up your CAD model for or select a study type from the left and "Create Study can also choose to smithy your model after you've c	simulation, dy". You created a
				suury.	



4. Set your material to Steel (Same as Model)

Category	Name	Component	Study Materials	Safety Factor
/letal	Steel	Simulation Model 1:1	(Same as Model)	Yield Strength

5. Apply the constraints to the two surfaces as below, use a fixed constraint, and click ok



6. Apply the loads. Select the split face. Using a force load, change the direction type to angle and a magnitude of 500 N.





7. Check that the pre-check is ready

Ready to Solve	×
$\sqrt[3]{5}$ The study setup has all the inform	nation required.
	OK

8. Click Solve and choose either on cloud simulation (requires 5 cloud credits) or solve locally.

		Man Orfers
	Statue	Cloud Credite
Simulation Model 1 - Study 1 - Static Stress Static Stress	Solved	cioud creats
Simulation Model 1 - Study 2 - Static Stress Static Stress	Ready	5
lving simulation studies uses cloud credits Learn More		



9. Review your results. You can change the result type using the drop-down list



10. You can also animate your results, change the deformation scale, and share your results by generating a report.





11. Repeat steps 1 to 10 using the supplied file 1B_Spanner_With_Fillet and compare the results

A Advance have not dependent conver-					R Automati I prov. WE (Manifest property)				
R BI R ALLE	B 10, Same, Solder		The Association of Con-	-+ = 0 = 0 6	B B B B B B B B B B B B B B B B B B B	Talana June 1	1.1	The Assessment of the Assessme	
						I I. E. I			
+)						•			
A II make					A III manta				14.3
A = 170 more than a				-	A - Distances and a				100
D BE Interface				-	6 W. Interface				
D (1) BE (mail)					In 11 Million (
D C M Comments					P C M Deserve				
A COMPANY OF THE OWNER.					A COMPANY OF A DESCRIPTION OF				
0 to () to be taken					5 to 45 million				
 Interaction and an 					 Investigation and an 				
in terms					in local				
8 mm					P				
A all hour martines.									
D D mutanes									
and the second									
Be man									
B. Smith									
				and instants					and the second second
				And in a					And A Designed
				Jackson V					Dentes 1
		1		-					inter T
		A COMPANY OF A DOMESTIC					A		
		Second Second Second					ACCURATE ACCURATE		



Exercise 1C+D: Static Stress, Spice Shelf

1. Open the supplied file 1C_Spice Shelf Bracket_No Cross Brace



2. Move from the Design Workspace into the Simulation Workspace



3. Choose a static stress simulation from the study type menu





4. Change the material to ABS Plastic

Category	Name	Component	Study Materials	Safety Factor
astic	Steel	Bracket:1	ABS Plastic	Yield Strength

5. Apply a fixed constraint to the back face of the bracket, then click ok





6. Apply a load of 60 N to the top surface, acting downwards. Change the type to Force. Direction type to Normal (acting downwards) and a magnitude of 60 N.



8. Click Pre-Check to ensure that everything is set up for the Simulation





9. Select Solve and either solve on the cloud (using 5 cloud credits) or solve the simulation locally. Then click Solve 1 Study

	On Cloud	O Locally		
TUDIES OF THE ACTIVE DOCU!	IENT			View Options
Study			Status	Cloud Credits
Simulation Model 1 - Study Static Stress	1 - Static Stress		Solved	
 Simulation Model 1 - Study Static Stress 	2 - Static Stress		Ready	5
olving simulation studies uses o	loud credits Learn N	Aore		

10. Review the results. Observe that the displacement is quite high.





11. Repeat steps 1 to 10 using the supplied file 1D_Spice Shelf Bracket_Wtih Cross brace and compare the results. Note the reduction in displacement.





Modal Frequencies

A modal frequency study can be used to determine a part's natural frequency. This is important as you do not want a design to naturally vibrate when in use and to shake itself apart. Structures exhibit multiple natural frequencies, and this study type can be used to determine the fundamental mode (lowest frequency) and its harmonic modes. Engineers need to design components so that the natural frequency is not close to the frequency at which the component operates.

Exercise 2A

1. Open the supplied file 2A_Fan_Mount. We are going to perform a modal frequency study on a mount for a fan, the fan operates at 50 Hz so this is the frequency we want to avoid.

2. Move from the Design Workspace into the Simulation Workspace



- <text><complex-block><complex-block><complex-block><complex-block><complex-block><complex-block><complex-block>
- 3. Choose a Modal Frequencies simulation from the study type menu

4. Select the material, choose ABS Plastic



STUDY MAT	ERIALS				
View All Mate	erials	✓ Search			
Category	Name	Component	Study Materials	Safety F	actor
Plastic	Steel	Quick Mount:1	ABS Plastic	Yield Strength	
4aterial Librai	y All Libraries	~			<< Properties
					OK Cance

5. Set the contraints. Choose a pin constraint and apply to the two cylindrical pins as shown below



6. Apply the Loads. Apply a load across the two mounting holes for the fan. This will be a load of 1 N and an angle pointing directly downwards.



F Autodesk Fusion 360 (Education License)		- 6 ×
田 聖 · 田 ち · ル · stur	2A_Fan Mount v2*	× + = 0 @ 0 9
SIMULATION * SAPLEY * MATERALS * CONSTANTS * LOADS * CONTACTS *		
• BROWSER • I		۵
Units: Custom		un ann
Simulation Model1 Named Views		
D IIIII Origin		I . STRUCTURAL LOADS
Construction Model Components		Type A Force •
A 📀 🛃 Study 1 - Modal Frequencies	a	Farce Per Entity
Study waterans Image: Study waterans Image: Study waterans		Direction Type 🔍 🔂 🕹
Contacts		Reference D Select
Results		XAngle 0.0 deg •
Study 2 - Modal Frequencies Study 2 - Modal Frequencies Study Materials		Z Angle 0.0 deg •
D E Load Caset O		File Direction 🙀
San Contacts		Magnitude 1 N *
Results		OK Cannel
(e		
POINTE		5 Fores 186 - Distance - 53 Ab aver
	○○○ (4, 4, 4) ■.	a recent (the distance - second the
7 In the pro-sheek we can as	a thora is the following arror	
7. In the pre-check we can se	e mere is me following enor.	
톤 Ready to Solve with Warnings	×	
×-		
🛛 🔨 🚃 The missing input may impact	your results.	
× <u>*</u>		
Loads are not considered for th	ie analysis. To include them,	
🔰 🍈 📥 check Compute Preloaded Mo	des in the Settings dialog.	
Click here to select 'Setting	s', to change the option	
Compute Preloaded Mode	5.	
	Close	

8. Go to Settings and click 'Apply pre-loads'





F Settings		×
Study Type Modal Frequencies		~
General	Gen	eral
Mesh Adaptive Mesh Refinement	Name	Study 2 - Modal Frequencies
	Number of Modes	
		8
	Frequency Range [Hz]	
	Compute Preloaded Modes	
		If checked, structural loads will be included and the f
		OK Cancel

9. Recheck the pre-check

톤 Ready to Solve	×
The study setup has all the inform	nation required.
	OK

10. Select solve and Solve on the cloud (requires 5 cloud credits) or solve locally



OLVE • FAQ On Cloud Cocally		
TUDIES OF THE ACTIVE DOCUMENT		View Options
Study	Status	Cloud Credits
Simulation Model 1 - Study 1 - Modal Frequencies Modal Frequencies	Solved	
Simulation Model 1 - Study 2 - Modal Frequencies Modal Frequencies	Ready	5
olving simulation studies uses cloud credits Learn More		

11. Review the results. Look at the different modes using the drop-down list. You can animate your result, edit the deformation scale and share your results by creating a report.





Thermal

Thermal analysis can be used to modelling thermal problems such as cooling fins for heat distribution or to look a\at multiple possible materials for an insulation. In the tutorial we will look at a pipe and different insulation materials and thicknesses.

Exercise 3A

- 1. Open the supplied file 3A_MultiLayer_Pipe_1
- 2. Move from the Design Workspace into the Simulation Workspace



3. Choose a Thermal simulation from the study type menu



- 4. Set up the materials. We will need to set three different materials
- a. Inner Piper set as Copper
- b. Middle Pipe set as Polyurethane Foam
- c. Outer Pipe set as ABS Plastic.



STUDY MATERIALS						
View All Materials View Search						
Category	Name	Component	Study Materials	Safety Factor		
Metal	Steel	Inner Pipe:1	Copper	Yield Strength		
Plastic	Steel	Middle Pipe:1	Polyurethane Foam	Yield Strength		
Plastic	Steel	Outer Pipe:1	ABS Plastic	Yield Strength		
Material Libra	ry All Libraries	\checkmark		<< Properties		
				OK Cancel		

This will also update the appearance of the three pipe sections in the display window



5. Hide the Middle and Outer pipe in the browser tree by clicking the eye next to them. Apply a Load to the inner pipe.

- a. Select all three sections of the Inner Pipe.
- b. Choose type to Radiation
- c. Set the value to 100°C





6. Show the Middle and Outer Pipe and set a thermal load to the outer surface of the pipe. This is going to represent room temperature. Choose the type of Radiation and apply the thermal load of 24°C. The click ok.



7. In this simulation we have multiple components (or bodies) so we need to apply contacts between the surfaces so the model knows how to behave.





8. Select a Contact Detection Tolerance of 0.1 mm and click Generate

	IATIC CONTACTS	•
▼ Contact	Detection Tolerance	
Solids	0.10 mm	•
		Generate Cancel

9. We can review and manage these under Manage Contacts





○ List by Bodies					다. Create Contact Set	
	Contact Set		Contact Type	Penetration Type	Bodies	Entities
Y	Bonded1	Ø	Bonded 🔹 👻	Symmetric 🔹		
					Middle Pipe:1/Body1	Face 38
				1	Outer Pipe:1/Body1	Face 35
Y	Bonded2	Ø	Bonded 🔹	Symmetric 🔹 👻		
					Middle Pipe:1/Body1	Face 37
				1	Outer Pipe:1/Body1	Face 34
Y	Bonded3	Ø	Bonded 🔹	Symmetric 🔹 👻		
					Middle Pipe:1/Body1	Face 36
					Outer Pipe:1/Body1	Face 33
>	Bonded4	1	Bonded 🚽	Symmetric 🔹		
>	Bonded5	/	Bonded -	Symmetric 🔹		
>	Bonded6	0	Bonded 🔹	Symmetric 🔹		

10. Review the Mesh Settings by right clicking on the mesh in the browser tree and Generate the Mesh – we can see that the mesh is quite large and may not be suitable for the simulation.



🗧 Autodesk Fusion 360 (Education License)	
	T 3A_MultiLayer_Pipe_1 v4*
SIMULATION • SETUP SIMULATION • STUDy • SIMPLEY • MATERIALS • LOADS •	DISPLAY* MANAGE* SOLVE* DISPLAY* NEPECT*
← BROWSER ●	
 Simulations Units: Custom Named Views Named Views Notel Components Model Components M	



Modify the mesh by clicking on Settings in the Tab at the top and selecting the following:a. In Mesh - Change the Model-based Size to 3%

E Settings		\times
Study Type Thermal		\sim
General	Mesh	
Mesh		
Adaptive Mesh Refinement	Average Element Size	
	Model-based Size 🔘	
	1% 10%	
	Scale mesh size per part	
	Absolute Size 🔘	
	Advanced Settings	
	OK Cano	el

b. In Adaptive Mesh Refinement – Change Refinement Control from None to High and change the Results for Baseline Accuracy from Heat Flux to Temperature



E Settings				×		
Study Type Thermal				~		
General	Adaptive Mes	h Refinement)		
Mesh Adaptive Mesh Refinement	Refinement Control					
				•		
	None Low	Medium	High Custom	1		
	Results Convergence Tolerance (%)	5				
	Portion of Elements to Define (%)	40				
	Results for Baseline Accuracy	Temperature	~			
			ОК	Cancel		
c. Click ok and regen	erate the Mesh for review			- σ ×		
田 動 * 田 ち * み * SETUP	3A_Mu8Layer_Pipe_1 v4*			× + 및 O © 0 9		
SIMULATION * STUDY * SIMPLEY * MATERIALS * LOADS * CONTACTS *	B B S S S S S S S S S S S S S S S S S S					
BROWSER Binulations				2 24		
Const Custom Generalization Model 5 O Imm Named Views				a contraction of the second		
Crigin Model Components Model Components						
Immer Vpe 1 Immer Vpe 1						
Study 1 - Thermal Study 2 - Thermal Study 2 - Thermal Study 1 - Thermal						
Study 4-Thermal Study 5-Thermal Study 5-Thermal						
Dentacts						
Results						
COMMENTS	今・郎 今 4 頁・ 11-11-11-11-11-11-11-11-11-11-11-11-11-					
12. Check the pre-chec	ck					
🗜 Ready to Solve	×					
$\sqrt[3]{5}$ The study setup has al	I the information required.					
	UK					

13. Solve the study on the cloud (using 5 cloud credits) or using a local solve



OLVE • FAQ OLVE • FAQ OLVE • FAQ OLVE • FAQ	/	
TUDIES OF THE ACTIVE DOCUMENT		View Options
Study	Status	Cloud Credits
Simulation Model 1 - Study 1 - Thermal Thermal	Ready	
Simulation Model 1 - Study 2 - Thermal Thermal	Solved	
Simulation Model 1 - Study 3 - Thermal Thermal	Solved	
Simulation Model 1 - Study 4 - Thermal Thermal	Ready	5
olving simulation studies uses cloud credits Learn More		
	The document is modified.	A new version will be created before the solv

14. Review the results. In an thermal simulation we can review the Temperature Heat Flux and Thermal Gradient.





COMMENT

01

15. Use the inspection tools such as surface probes, point probes and slice planes in the Inspection tab to further analyse the results



16. Investigate the pipe by selecting different materials for the Middle Pipe and see how the surface temperature changes. You can also change the thickness of the insulation. Compare your results. The supplied file 3B_MultiLayer_Pipe_2 has a thicker insulation layer.



Thermal Stress

Thermal Stress analysis combines both the Static Stress Simulation as well as the Thermal Simulation Types. This exercise will consider the thermal stress on a high temperature steam pipe hanger.



Choose a Thermal Stress simulation from the study type menu 3.

New Stu	dy				×
	Simplify geometry for use	n Simulation			
Ļ	44		l	Total.	
Static Stress	Modal Frequencies	Electronics Cooling (Preview)	Thermal		
Thermal Stress	Structural Bucking	Noninear State	Event Simulation	Simplify Choose "Smoth," to clean up your CAD model for simulation or select a study type from the left and "Create Study". You can also choose to armothy your model after you've created a	n,
				study.	
Shape Optimization					



4. Select Simplify from the top bar to enter the Simplify Workspace. Remove the Nuts from the Model Component list to remove them from the simulation. This removes them from the simulation only, they will still exist in the Design Workspace. When you have deleted them click FINISH SIMPLIFY.

Ⅲ ■・ 目 ら	• ? •	4A_ThermalStress_PipeHanger v3*	× + 4 0 0 0 9
	SIMPLIFY SOLID SIMPLIFY SOLID CREATE • NOOFY •	TOOLS CONSTRUCT* NOPECT* SELECT* FINGH SMALFY*	
- BROWSER	•)		
Binvation Mo Binvation Mo Binvation Mo Unix. D Binvation Mo Bi	edah minimum kooreali @ minimum strongenetika strongenetika strongenetika morgan morga		Z [*]
	MoveCopy M M Mev Component Greats Selection Set m beect Al Occurrences Select Al Occurrences Select Smilar Occurrences		
	Rigit Group Appearance A C Texture Map Controls Properties Export Save Copy As		
	Copy Ctri- Cut Ctri-		
	Remove Remove All Except Selected		
	Show/Hide V		
	Show All Components		
	Show All Bodies		
	Selectable/Unselectable		
COMMENTS	Upacity Control	◆・茴 壱 ℃ Q、 ■・■・	
4 4 b b	Find in Window Find in Timeline	_	0

5. Set the materials for the study using the MATERIALS, change all the components to Steel Alloy.

STUDY MATERIALS

Category	Name	Component	Study Materials	Safety Factor	
letal	Steel	/_B3102-12_body:1	Steel, Alloy	Yield Strength	
letal	Steel	/2:1	Steel, Alloy	Vield Strength	
letal	Steel	/_B3102-12_top:1	Steel, Alloy	Yield Strength	
Material Library All Libraries V					

6. Change the Mesh settings using the Settings button in the top bar. Change the Modelbased Size to 3%. Tick Scale mesh size per part to ON, and generate the mesh to review.



F Settings		×
Study Type Thermal Stress		~
General .	Mesh	
Adaptive Mesh Refinement	Average Element Size	
	Model-based Size 🔘	<u> </u>
	1%	10%
	Scale mesh size per part 🗹	
	Absolute Size 🔿	
	Advanced Settings	
		OK Cancel

7. Apply the Loads, start with the Structural Load. Apply a Load of 500 N to the curved inside face as shown below. This represents the weight of the pipe inside the hanger.



8. Apply a Thermal Load to the same inside face. Change the type to Applied Temperature and set a Magnitude of 500°C.





9. Add an additional Thermal Load. Select Load > Thermal Load. Change the Type to Convection. Toggle on Select all Faces and choose all the faces of all the parts (except the face we have already applied a load to). Set the convection value to 30 W / (m^2 K) and the Ambient Temperature to 20°C.

F Autodesk Fusion 360 (Education License)		- a ×
田 書 * 田 ち * み *	44_ThermalStress_PipeHangerv3*	× + 4 0 0 9
	4 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2	
STUDY* SURJEY* MATERIALS* CONSTRAINTS* Co STUDY* SURJEY* MATERIALS* CONSTRAINTS* Co Total code Total code		Constantiane Constantiane
computs • 9	今・節 ひ ぺ 身・圓・	Multiple selections

10. Check the pre-check



Ready to Solve	×
🗧 The study setup has all the informa	tion required.
	OK

11. Solve the simulation on the cloud (requires 5 cloud credits) or solve locally.

TUDIES OF THE ACTIVE DOCUMENT	[View Options
Study	Status	Cloud Credits
Simulation Model 1 - Study 1 - Thermal Stress Thermal Stress	Solved	
 Simulation Model 1 - Study 3 - Thermal Stress Thermal Stress 	Ready	5

12. Review the results. With a Thermal Stress Analysis you can review both the results that are available in a Static Stress simulation as well as those available when performing a Thermal analysis.





Structural Buckling

Buckling is when there is a sudden change in the shape of an object when subjected to compressive forces. This usually happens when the length is long compared to the cross-sectional area of the object. You may have experienced this when pushing on the end of a long tube, it suddenly pushes out to the side.

There are two ways to approach buckling in Fusion 360, the results given are in the form of a multiplier based on the inputted force. The first approach is to apply a known load, and the result will be a multiplier of that load at which the object will buckle. Alternatively, we can apply a load of 1 N and the output will be the load at which the part will buckle.

Exercise 5A

1. Open the supplied file 5A_Bar_Stool. We are going to investigate some different materials for the long tube to see what might be suitable for the design. Our target load is that the stool must be able to take a weight of 110 kg.



3. Choose a Structural Buckling simulation from the study type menu



New Study	×
Simplify geometry for use in Simulation	
Static Stress Model Frequencies Electronics Cooling (Preview)	
Themai Stress	Simplify Choose "Smplify" to clean up your CAD model for smulition, or select a study type from the left and "Create Study". You can also choose to smplify your model after you've created a study.
Shee Optimation	
Help me choose a study type.	Simplify Model Cancel

4. In the Simplify Workspace remove the Components: Seat; Upper Strut; and Lower Strut. You can do this by right clicking on them in the browser tree and selecting remove. When you have finished click FINISH SIMPLIFY to return to the Simulation Workspace.



5. Set the Materials for the first study to Steel (Same as Model)



STUDY MATE	RIALS	✓ Search		
Category	Name	Component	Study Materials	Safety Factor
Metal	Steel	Base:1	(Same as Model)	Yield Strength
Metal	Steel	/Single Strut:1	(Same as Model)	Yield Strength
Aaterial Library	All Libraries	~		<< Properties
				OK Cance

6. Apply a Constraint to the bottom face of the base



7. Apply a load to the top of the tube – use a Force load of 1 N. This will give our results so we can find the load at which the chair will buckle.





※·尚 む c* Q · 見·目·

9. Check the Pre-Check

COMMENTS 0



F Ready to Solve	×
Solution The study setup has all the information	n required.
	ОК

10. Solve the Simulation on the Cloud (requires 15 cloud credits) this study type cannot be solved locally

SOLVE • FAQ On Cloud Locally		
STUDIES OF THE ACTIVE DOCUMENT		View Options ~
Study	Status	Cloud Credits
Simulation Model 1 - Study 1 - Structural Buckling Structural Buckling	Solved	
Simulation Model 1 - Study 2 - Structural Buckling Structural Buckling	Solved	
Simulation Model 1 - Study 3 - Structural Buckling Structural Buckling	Solved	
Simulation Model 1 - Study 6 - Structural Buckling Structural Buckling	Ready	15
Solving simulation studies uses cloud credits Learn More		
	The document is modified. A	A new version will be created before the solve
		Solve 1 Study Close

11. Review the Results





12. Clone the Study by right clicking on the study and selecting clone study, do this twice so you have 3 studies in total. Change the material in the 2 new studies to:

- a. Aluminium for the tube
- b. ABS Plastic for the tube

Leave the base as Steel as it is not part of the study.

Autobesk Fusion Jou (Education License)				
III III STUP				
SIMULATION - STUDY - SIMPLI	Fy. MATERIALS.	CONSTRAINTS *	LOADS*	
# BROWSER	•)			
Simulations Units: Custom Units: Custom Simulation Model1 O Market Views D Model Components	Ð			
🔬 🔯 🛃 Study 1 - Structural	Clone Study			
Sildy Materials Load Case1 Contacts	Settings	E Del		
Results	Study Materials			
Study 2 - Structural Bi Study 3 - Structural Bi	uci Report Properties			
Study Materials Eacl Case1	G Auto Setup - Robo	DK		
Contacta Mesh Results				

13. Compare the results of the different materials to determine which of the three materials is most suitable. Remember that the chair needs to take a load of 110 kg. Use the Compare option in the top bar to view multiple results at the same time.







Nonlinear Static Stress

Many materials do not behave in a fully elastic way. i.e. when a load is applied, they do not return to their original shape. Most materials follow a elastic-plastic behaviour and have the following characteristic stress strain curve.



Exercise 6A

1. Open the supplied file 6A_Thin_Beam





3. Choose a static stress simulation from the study type menu



New Study	×
Simplify geometry for use in Simulation	
Static Stress Model Prequencies Cooling (Preview)	
Thermal Stress Structural Buckling Nonlinear Static Structural Buckling Nonlinear Static	Simplify Choose "Simplify." to clean up your CAD model for smultiton, or select a study type from the left and "Greate Study". You can also choose to simplify your model after you've created a study.
Shee Optimization	
Help me choose a study type.	Simplify Model Cancel

4. Set up the materials. Choose from the Fusion 360 in built library of Nonlinear Materials. This leaves far fewer materials to choose from. Set the material to Aluminium – Pure (low Strength).

E Autodesk Fusion 360 (Education License)				- 0 ×
⊞ ∎ * ⊟ • • > ·		🗊 6A_Thin Beam v3		× + 🖛 🤈 🕲 🤪
SIMULATION *			NSPECT • SELECT •	
+ BROWSER		•		
🔺 🔚 Simulations				1200
Units: Custom				FOCHER DIGHT
🖉 💿 🚼 Simulation Model 1 💿				
Named Views				X
Cingin Digin				
Model Components	STUDY MATERIALS			
Study Materials	View All Materials	V Search		
🕨 💷 Load Case1 🧿	Category Name	Component Study Materials	Safety Factor	
Contacts	Metal Steel	Simulation Model 1:1 Aluminum - Pure (Low-Strength)	Yield Strength	
😂 🏪 Mesh				
Results				
	Material Library Fusion 360 Nonlinear Mi All Libraries	sterial Library 💛	<< Properties	
	Fusion 360 Additive Mat Fusion 360 Nonlinear Mi	terial Ubrary sterial Ubrary	OK Cancel	
	Fusion 360 Material Libr	ary		
		42		
		•		
COMMENTS O				

5. Select Properties and then click view advanced material properties.



•	STU	DY MATER	RIALS				
	View	All Mater	ials	✓ Search		Material	Aluminum - Pure (Low-Strength)
	Ca	tegory	Name	Component	Study Materia		
	Met	al	Steel	Simulation Model 1:1	Aluminum - Pure (Low-Strength)		
						Your material view it.	data is using advanced properties, dick below to
							View advanced material properties
	<		-		>		
	Mater	rial Library	Fusion 360 Nonlinear Materia	l Library 🗸	Properties >>		
							OK Cancel

6. This takes you to the Materials Properties Browser. Click on the double arrow to expand the window then click on the pencil next the Aluminium to edit the material properties.

P Material Browser								×
Physical								
earch							ntity Appearance	Physical
Document Materials	Jocument Materials						Name Alumi	inum - Pure (Low-Strength)
Name Category	Name Category						escriptive Informatio	n
AluminumStrength) Metal						Description		
Steel Metal							Туре	Metal -
							Comments	Stress-Strain curve following ASME B&PV Code 2015, S
							Keywords	
						_		
♠ ▼ Fusion 360 Nonlinear Material Library ▼	$\langle \rangle$					Pr	oduct Information	
Gas 🔨		Name	▲ Category		^		Manufacturer	
Glass	۲	Aluminum - High-Strength Alloy	Metal				Model	
Te Metal	۵	Aluminum - Moderate-Strength Alloy	Metal				Cost	
Misc .	12	Aluminum - Pure (Low-Strength)	Metal				URL	
 Plastic Stone 	۲	Copper - High-Strength Alloy	Metal					
T Wood	٢	Copper - Pure (Low-Strength)	Metal					
Fusion 360 Additive Material Library		Steel - Carbon Steel for Pressure Vessels	Metal					
Fusion 360 Appearance Library	٥	Steel - High-Strength Structural	Metal					
v	26	Steel - High-Strength Low-Allow	Metal		~			
🔤 • 🚳 • 🗏					~			Add to Favorites & Edit

7. Click on the tab called Physical and the Advanced Properties. Here the Material Model, Behaviour and Type can be changed. Change the type to Elasto-plastic(Bi-Linear) this matched the curve shown above. Click Ok, then close the panel.



E	E Material Browser X										
Ph	Physical										
Sea	Search Q Identity Appearance 🕮 Mysical 🖽 ×										
Do	Document Materials >					Basic Properties	A V	dvanced Propert	ties		
	Name 🔺	Category						Marcal Madal	Instantia		Advar
0	AluminumStrength)	Metal						Materiai Model	isotropic		
0	Steel	Metal						Behavior	Nonlinear		-
						Туре	Elasto-plastic(Bi-lin	ear)	•		
								▼ Plastic Parameters			
n	Fusion 360 Nonlin	ear Material Library 🔻	\rangle			8	I≣ •	Tangent Modulus	6.895 GPa		4. V
	🖕 Gas	^		Name	 Category 		^	Hardening Rule	Isotropic		-
	Glass		۲	Aluminum - High-Strength Alloy	Metal			▼ Yield Function			
	Metal		۵	Aluminum - Moderate-Strength Alloy	Metal			Yield Criterion	von Mises		-
	Misc 🖌		0	Aluminum - Pure (Low-Strength)	Metal			Initial Vield Stress	14 809 MPa		•
Plastic		0	Copper - High-Strength Alloy	Metal			indui ricio scicia	141003 1111 2		•	
Wood			Copper - Pure (Low-Strength)	Metal		10					
Fusion 360 Additive Material Library		D	Steel - Carbon Steel for Pressure Vessels	Metal							
Fusion 360 Nonlinear Material Library			Steel - High-Strength Structural	Metal							
		· · · · · · · · · · · · · · · · · · ·	26	Steel - High-Strength Low-Allow	Metal		~				
E7	ä∗ Q+ = <					~<		ОК	Cancel	Apply	

8. Apply the constraints. Apply a fixed constraint to the bottom face.



9. Apply a Load to the top face. Apply a Force load of 350 N at an angle of 20° then click ok.



E Autodesk Fusion 360 (Education License)		– a ×
Ⅲ ■ × 目 ち · ル ·	🗊 6A_Thin Beam v3*	× + 🖛 🧿 🙆 🤧
	ar. ≶5 22 = 0	
+ BROWSER •)	0	
STUDY SARLYY MATERIAS CONSTRAINTS LOADS CONTACTS BEFLAY MAAN DEDICATE ON THE STANDARD CONTACTS BEFLAY MATERIAS CONSTRAINTS LOADS CONTACTS BEFLAY MAAN DEDICATE ON THE STANDARD CONTACTS BEFLAY AND	ос. 2004. БОЛА. КАКСА. 2008. К. К. С. 20 МЛ. Д С	COT STRUCTURAL LOAD Force Type Type
сомилта • 1	令·翁 ⑦ ♀ Q、 ■・■・	1 Face Area : 100.00 mm*2

10. Change the Mesh Settings in Settings > Mesh to 3%

E Settings		×
Study Type Nonlinear Static	Stress	~
General	Mesh	
Mesh	⊿ Average Element Size Model-based Size	
	1% 10% Scale mesh size per part Absolute Size O	
	Advanced Settings	
	OK Cance	:I

11. Finally, we set up the number of steps in the Simulation. This is how many discrete points in time the simulation will perform over. Set this to 10 Steps in Settings.



E Settings	×	
Study Type Nonlinea	r Static Stress ~	
General	General	J
Mesh	Name Study 1 - Nonlinear Static Stress Number of Steps 10	
	OK Cancel	

12. Solve the simulation on the cloud (this requires 15 cloud credits).

Solve		×
SOLVE · FAQ On Cloud Locally		
STUDIES OF THE ACTIVE DOCUMENT		View Options 🗸
Study	Status	Cloud Credits
Simulation Model 1 - Study 1 - Nonlinear Static Stress Nonlinear Static Stress	Solved	
Simulation Model 1 - Study 2 - Nonlinear Static Stress Nonlinear Static Stress	Ready	15
Solving simulation studies uses cloud credits Learn More		
	The document is modified.	A new version will be created before the solve.
		Solve 1 Study Close

13. Once the Simulation has solved the results can be reviewed. The results can be reviewed at each of the steps in the simulation.





A 2D chart can also be reviewed over the steps.





Event Simulation

Event Simulation in Fusion 360 can be used to explore dynamic problems. Event simulation is a fully dynamic tool – it can take into account, the mass; the acceleration; the velocity; the inertia; throughout the simulation. This is really important for simulating impact events, for example a bird strike, or a crash test. We will briefly look at 4 different examples to give an idea of the study type is capable of.

Exercise 7A



3. Choose Event Simulation from the study type menu



New Study	×
Simplify geometry for use in Simulation	R
State Stress Model Frequences	ŧ
Thermal Stress Structural Bucking Nonlinear State: Thermal Stress Structural Bucking Nonlinear State: Thermal Stress Structural Bucking Nonlinear State: Stress Structural Stress Structural Bucking Nonlinear State: Stress Structural Stress Structural Structure St	del for simulation, ite Study". You ou've created a
Shape Optimization	
Help me choose a study type. Simplify Model	Cancel

4. Usually we work from left to right along the top bar, but this time we will start by setting up the simulation settings. Click Manage > Settings to open up the settings menu. Chage the following:

- Movement to Dynamic (considers inertia) Total Event Duration to 0.006 s a.
- b.
- Number of Result Intervals to 12 c.

E Settings	×
Study Type Event Simulation	~
General	General
Mesh	
Damping	Name Study 1 - Event Simulation
Result Output	Movement Dynamic (considers inertia) V
	Total Event Duration 0.006 s
	Number of Result Intervals 12
	Solve Status Information Interval (Heartbeat) 50
	OK Cancel

For the Mesh settings:

Change the mesh to absolute size of 3 mm and then click ok. a.



톤 Settings	×
Study Type Event Simulation	~ _
General	Mesh
Mesh Damping Result Output	▲ Average Element Size Model-based Size ○ Absolute Size ● 3 mm Advanced Settings
	OK Cancel

5. Define the stationary and moving bodies. Set Body 1 to be rigid and Body 2 will be moving. We will set Body 1 to be rigid as we are not concerned with its movement or stress in the simulation. In the browser tree select Rigid Bodies, click the pencil to edit, and select Body 1 as the target body. Then click ok.

F Autodesk Fusion 360 (Education License)		- 0 X
	7A_ShupFitv3	× + = 0 @ 0 9
SIMULATION -		
+ BROWSER •		1
Branner District State(C) Dist		Canon
COMMENTS • F	今・豊 今 ら の・ 御・圖・	Bodyt

6. Set up the materials for the study. Change the materials for both parts to be Nylon 6/6. Then click ok.



STUDY MATE	RIALS ials	✓ Search.		
Category	Name	Component	Study Materials	Safety Factor
Plastic	ABS Plastic	Simulation Model 1:1/Bo	Nylon 6/6	Yield Strength
Plastic	ABS Plastic	Simulation Model 1:1/Bo	Nylon 6/6	Yield Strength
aterial Library	Fusion 360 Nonlinear Mate	erial Library 🗸		<< Propertie
				OK Canc

7. Apply the constraints. Start with a constructural constrint. Choose Body 1 and set this to be fixed.



8. Apply a constraint of a prescribed translation. This is to allow the Body 2 to move into Body 1 in a single direction and axis. Choose Constraints > Prescribed Translation





9. Select the face on Body 2, and enter a value of -10 mm in the Uz magnitude box.



10. Click on the Multiplier Curve and enter the following values. This is where we define how the motion will be over time during the simulation. We will set a movement which will decelerate into the second part over the course of the simulation. Click ok to save the curve, then ok to save the constraint.



	RANSLATION	٠
Selection	Ŋ 1 Face 🗙	
Object Type		
Constraint Type	Displacement •	
Direction Type	🕹 Components (x, y, 🕶	
Components	Ux Uy Uz	
Magnitude Ux	0.00 mm -	
Magnitude Uy	0.00 mm 👻	
Magnitude Uz	-10.00 mm 💌	
Change Units	🕒 mm 🔹	
Multiplier curve		
0	OK Cancel	



11. Set up the Loads. Start with Load > Initial Linear Velocity. Use Vector for the Direction Type and set a value of -3000 mm/s in the Z component.





12. Define the contact between the two bodies. If this was a complex problem you may need to enable to check for self contact. In this case we know that the parts will not intersect so we can disable this option which means that the simulation will compute more quickly. Set this in Contacts > Global Contacts.

- a. Ensure that Global Separation Contact is enabled
- b. Proximity Bonded Contact is disabled
- c. Under Advanced Settings, Enable Allow Self Contact
- d. Click Generate

I SLOBAL CONTACTS	44
Global Separation Contact	
Default Friction Coefficient	•
Proximity Bonded Contact	•
▼ Advanced Settings	
Allow Self Contact	
0	Generate Cancel

13. Apply Mesh refinement. This allows us to put a finer mesh in an area of interest, without wasting computing power on areas that are not of interest. In Manage select Local Mesh Control.





Select the central area (using a left to right selection) which will detect 18 faces/edges. Then change the mesh size in this area to 1 mm.



14. Check the pre-check. There may be a warning that a nonlinear material has been selected. In this case this is fine so click ok.



E Ready to Solve with Warnings	×
The missing input may impact your results.	
 Warning: One or more of the physical materials do not contain nonlinear properties. The material may not behave as expected in the current analysis type. Nonlinear properties are recommended. 	
Click here to select 'Study Materials' to set a material.	
Close Close	

15. Solve the simulation in the Cloud (requires 15 cloud credits). Again, it shows the warning about the nonlinear material. Click Solve 1 Study. This may take some time.

SOLVE FAQ On Cloud Locally STUDIES OF THE ACTIVE DOCUMENT View Options Study Status Cloud Credits Simulation Model 1 - Study 1 - Event Simulation Solved Solved Solved Simulation Model 1 - Study 2 - Event Simulation Owarning • Repair Solved Simulation Warning • Repair Solved Study Solved Study Solved Study The document is modified. A new version will be created before the solved Solve 1 Study Close	Solve		×
STUDES OF THE ACTIVE DOCUMENT velocities	SOLVE · FAQ		
Study Status Cloud Credits Image: Study 1 - Event Simulation Solved Image: Solved <	STUDIES OF THE ACTIVE DOCUMENT	View	/ Options ~
Simulation Model 1 - Study 1 - Event Simulation Solved Simulation Model 1 - Study 2 - Event Simulation Warning • Repair Event Simulation Warning • Repair Solving simulation studies uses cloud credits Learn More Solving simulation studies uses cloud credits Learn More The document is modified. A new version will be created before the solve. Solve 1 Study Close Close	Study	Status	Cloud Credits
Simulation Model 1 - Study 2 - Event Simulation Warning • Repair 15 Event Simulation Warning • Repair 15 Solving simulation studies uses cloud credits Learn More The document is modified. A new version will be created before the solve. Solve 1 Study Close	Simulation Model 1 - Study 1 - Event Simulation Event Simulation	Solved	
Solving simulation studies uses cloud credits Learn More The document is modified. A new version will be created before the solve. Solve 1 Study Close	Simulation Model 1 - Study 2 - Event Simulation Event Simulation	🔴 Warning • Repair	15
Solving simulation studies uses cloud credits Learn More The document is modified. A new version will be created before the solve. Solve 1 Study Close			
Solving simulation studies uses cloud credits Learn More The document is modified. A new version will be created before the solve. Solve 1 Study Close			
Solving simulation studies uses cloud credits Learn More The document is modified. A new version will be created before the solve. Solve 1 Study Close			
Solving simulation studies uses cloud credits Learn More The document is modified. A new version will be created before the solve. Solve 1 Study Close			
Solving simulation studies uses cloud credits Learn More The document is modified. A new version will be created before the solve. Solve 1 Study Close			
The document is modified. A new version will be created before the solve. Solve 1 Study Close	Solving simulation studies uses cloud credits Learn More		
Solve 1 Study Close		The document is modified. A new	version will be created before the solve.
			Solve 1 Study Close

16. Review the results.



F Autodesk Fusion 360 (Education License)		- a ×
Ⅲ ■ · 目 ち · ∂ ·	TA_SnapFitv3*	× + 🛱 Ø 🖲 🚱
RESULTS		
+ BROWSER •)		
🖌 🔚 Simulations		1-1-1
Units: Metric (SI)		Z- neur
🖉 🐼 🛐 Simulation Model 1 💿		have a second
Named Views		X
D 😳 🛗 Origin		
D 🐵 📷 Construction		
D iiii Model Components		
⊿ 😒 📕 Study 1 - Event Simulation		
👍 💭 Rigid Bodies		
Simulation Model 1:1(Body1		
D 🐵 🛞 Study Materials		
🔏 💷 🔜 Load Case1 🧿		
A @ 24 Loads		
Gravity		
📝 🐵 🖽 Initial Linear Ve		
D 🐵 🖂 Constraints	and the second se	
Contacts		
😂 🏪 Mesh		
D 🏪 Local Mesh Controls		
Results		
A 🛃 Study 2 - Event Simulation		136 Max.
D 💭 Rigid Bodies		Stress • 120
D Study Materials		Von Maes *
D I Load Case1		MPa+
Contacts		45 A
and Mesh		3,4 24
D Local Mesh Controls		Step 3.714 :
D 🗧 Results		
		- 30
		- 0 Min.

COMMENTS O



Shape Optimization

Shape Optimization is a simulation tool which can be used to remove mass in areas not needed based on a given load. This is for a single material and does not take manufacturing methods into account. We will consider a robot gripper arm and use a mass target for the simulation target to remove material along non-critical load paths. We will then look at using the output to influence future designs.

Exercise 8A

1. Open the supplied file 8A_Gripper_Arm. It is important that our new design will comply with the original design specification so we will aim for a safety factor of 2.





3. Choose a Shape Optimization from the study type menu



New Study	×
Singlify geometry for use in Sinulation	-
Static Stress Model Frequencies Electronics Cooling (Review)	Ħ
Themal Stress	your CAD model for smulation, left and "Create Study". You model after you've created a
Shape Optimization	
Help me choose a study type.	Implify Model Cancel

4. • Set up the materials. Leave as Steel (Same as Model)

TUDY MATERIALS

Category	Name	Component	Study Materials	Safety Factor
letal	Steel	Simulation Model 1:1	(Same as Model)	Yield Strength

5. Apply the Constraints, change the Type to Pin and select the cylindrical surfaces as below.





6. Apply a Load to the gripper arm end. Use a Force Load of 500 N to the face as below.



7. Next, we want to set the Preserve Regions – these are the areas which will be kept when the simulation is performed. From the top bar select Shape Optimization > Preserve Region. Select the First Cylindrical surface and set a radius of 8 mm.





Do the same for the second cylinder surface and set a radius of 5.5 mm.



8. Under shape optimisation select a symmetry plane. This is so that when material is removed it is symmetric across this plane.





COMMENTS

∲•≝ ở 약 Q• ≣•∭•

9. Now set the criteria for the study. Select Shape Optimization > Shape Optimization Criteria. Set the Target Mass to be 40%. Click OK.

1 Face | Area : 3387.524 mm*2



Parameter Expression Value Units Global Objectives Iarget Mass < OR = 40 % Stiffness Maximize Global Constraints Preserve Boundary 1 N/A Preserve Boundary 2 N/A Symmetry Plane 1 N/A Preserve Entities with Loads and Constraints Image: Mass in the state of the state o	SHAPE OPTIMIZATION CRI	TERIA			44
Global Objectives Target Mass < OR = 40 Stiffness Maximize 40 V Global Constraints Preserve Boundary 1 Preserve Boundary 2 N/A 9 Symmetry Plane 1 N/A 10 Preserve Entities with Loads and Constraints V OK Cancel		Parameter	Expression	Value	Units
Target Mass < < OR = < 40	🗸 Global Objectives 🕈				
Stiffness Maximize Global Constraints Preserve Boundary 1 N/A Preserve Boundary 2 N/A Symmetry Plane 1 N/A Preserve Entities with Loads and Constraints Preserve Entities with Loads and Constraints OK Cancel 		Target Mass 👻	< OR = 👻	40	%
Global Constraints Preserve Boundary 1 N/A Preserve Boundary 2 N/A Symmetry Plane 1 N/A N/A Preserve Entities with Loads and Constraints OK		Stiffness 🔹	Maximize 🚽		
Preserve Boundary 1 N/A Image: Constraints of Constraints Preserve Entities with Loads and Constraints Image: Constraints of Constraints Image: Constraints of Constraints of Constraints	🗸 Global Constraints 🕈				
Preserve Boundary 2 N/A Image: Constraints of Cons		Preserve Boundary 1	N/A		
Symmetry Plane 1 N/A Image: Symmetry Plane 1 Image: Symmetry Plane 1 Image		Preserve Boundary 2	N/A		
Preserve Entities with Loads and Constraints 🖉		Symmetry Plane 1	N/A		
Preserve Entities with Loads and Constraints 🖉					
Preserve Entities with Loads and Constraints OK Cancel					
Preserve Entities with Loads and Constraints OK Cancel					
Preserve Entities with Loads and Constraints 🖉 OK Cancel					
Preserve Entities with Loads and Constraints OK Cancel					
Preserve Entities with Loads and Constraints OK Cancel					
Preserve Entities with Loads and Constraints 🖉					
Preserve Entities with Loads and Constraints OK Cancel					
Preserve Entities with Loads and Constraints OK Cancel					
Preserve Entities with Loads and Constraints OK Cancel					
Preserve Entities with Loads and Constraints OK Cancel					
Preserve Entities with Loads and Constraints OK Cancel					
Preserve Entities with Loads and Constraints OK Cancel					
OK Cancel	Preserve Entities with Loads	and Constraints 🗹			
				Ok	Cancel

10. Set the Mesh Settings under Manage > Settings. Choose a absolute mesh size of 1 mm then click ok.

E Settings	>
Study Type Shape	Optimization V
General	Mesh
Mesn	▲ Average Element Size Model-based Size ○ Absolute Size ● 1 mm Advanced Settings
	OK Cancel

11. Check the Pre-Check.



F Ready to Solve	×
The study setup has all the information requi	red.
ОК	

12. Solve the study on the cloud (requires 5 cloud credits) this type of study cannot be solved locally. Click Solve 1 Study.

Solve		×
SOLVE · FAQ On Cloud	Locally	
STUDIES OF THE ACTIVE DOCUMENT		View Options 🗸
Study	Status	Cloud Credits
Simulation Model 1 - Study 1 - Shape Optimization Shape Optimization	Ready	5
Solving simulation studies uses cloud credits Learn More		
	The document is modified	I. A new version will be created before the solve
		Solve 1 Study Close

13. Once the simulation is complete, we can review the results. The results are shown as Load Path Criticality. You can drag the arrow to view how the material was removed over the study, you can also go beyond what the mass request was in the criteria.









We are going to add this mesh to our design workspace

I O PROMOTE		**
Add mesh object to:	Design Workspace	•
0	ОК	Cancel



15. Click Finish Results to move back into the Design Workspace. You can see we now have the original body and the new mesh body.



Choose the operation type as New Body and select OK.



MESH TO E	BREP	**
Mesh Body	▶ 1 selected	×
Operation	New Body	•
0	OK	Cancel

This gives you a new TSpline body that can be manipulated as you would a design in the Design Workspace. This can be re-simulated using a Satic Stress Simulation (see Exercise 1 for how to do this). Check that no areas exceed the safety factor of 2.

17. Option 2 – Use the Mesh as a guide to edit the original Body and simulate using a Static Stress Simulation to check the safety factor does not fall below 2.

- a. Copy Body 1 and paste so we have a new body to work with.
- b. Create a sketch on the top surface and using the mesh as a guide create areas which will be removed from the body.



c. Extrude cut the sketch through the part and fillet any sharp corners where needed.





d. Move back into the Simulation workspace and set up a new study as Static Stress type. Perform a static stress simulation to check that the safety factor does not drop below 2. If the results indicate it does, then go back and edit the design and repeat until happy with the design.





Conclusions

In this class we have covered the entire Simulation workspace in Fusion 360. Including the basics of setting up a simulation; how to choose the study type; how to review the results; and how these can be used to influence future design decisions. The following study types in the simulation workspace were covered:

- 1. Static Stress A Spanner and Spice Shelf Bracket
- 2. Modal Frequencies A Fan Bracket
- 3. Thermal A Pipe and its insulation options
- 4. Thermal Stress A Pipe hanger and how it reacts to a hot steam pipe
- 5. Structural Buckling A bar stool style chair
- 6. Nonlinear Static Stress Simple beam example
- 7. Event Simulation Dynamic examples
- 8. Shape Optimization Removing mass on a robot gripper arm