ME345: Modeling and Simulation

Analysis of I-cantilever beam by using SimulationXpress

Stevens Institute of Technology

Professor Frank Fisher

Last updated: July 5th, 2011

Written by Lin Dong

Software versions used in the tutorial: SolidWorks 2011

1. Start SolidWorks

Click on the New tab on the top left corner of the screen and then create a new Part.

SolidWorks → □ 2 - 10 - 10 - 10 - 10 - 10 - 10 - 10 -	? • _ = = ×
New SolidWorks Document Pert Part Part Part New SolidWorks Document Part Part Part	(11) (12) (12) (12) (12) (12) (12) (12)
a 3D arrangement of parts and/or other assemblies	
a 2D engineering drawing, typically of a part or assembly Drawing	
Advanced OK Cancel Help	8
SoldWorks Education Edition - Instructional Use Only	

2. Create a sketch

Click on the Sketch command and Sketch tab to start.



Select the Front Plane as the sketch plane and then it automatically goes to orient a model

normal to the nearest global XYZ coordinates. If not, right click to selcet Normal to command.



To display a sketch grid in an active sketch and set options for the grid display and snap

functionality, go to the **Tools** > **Options** > **Document Properties** > **Grid/Snap** > Check **Display Grid** > change **grid spacing** to 10mm

ument Properties - Gr	id/Snap	
system Opt ons Document F	Properties	
Azeran Sandard Antonio Anton	Go To System Snaps	

If the units are not in MMGS automatically, go to Units > check MMGS

Drafting Standard						
Annotations	MKS (meter, kilogram	m. second)				
Dimensions	OCCS (contimeter, gr	am, second)				
- Virtual Sharps	MMGS (millimete), gr	am, second)				
tel Tables	O IPS (inch, pound, se	cond)				
Section of the sectio	Custom					
units D				1		
Colors	Туре	Unit	Decimals	Fractions	More	
Material Properties	Basic Units					
Plane Display	Length	millimeters	.12			
DimXpert	Dual Dimension Length	inches	.12			
- Size Dimension	Angle	degrees	.12			
- Location Dimension Chain Dimension	Mass/Section Propert	ties				
- Geometric Tolerance	Length	milimeters	.12		1	
- Chamfer Controls	Mass	grams				
- Display Options	Per Unit Volume	milimeters^3				
	Motion Units					
	Time	second	.12			
	Force	newton	.12			
	Power	watt	.12			
	Energy	joule.	.12			
	- MISSING CO	and the second s	10 me			

Complete the following sketch by clicking on the **Line** command and starting at the **Origin**. Do not worry about the lengths of the structure, they can be defined later, but make sure the lines are horizontal or vertical. It can be implied by a green square with the line inside.



In order to make the I-cantilever beam symmetric, add relation of the lines. Click Shift and



selected two lines > Add relations of **Equal** > Click \checkmark to close



Repeat the same procedure to define the **equal** relation of the other three groups of lines.

Solid Work	(s 🕨 🗋 - 🔗 - 🖬 - 🗞 - 🍤 - 🕞	• 8 🗷 •	Sketch1 of Part1 *	?•= = ×
Exit Sketch Features Ske	No No No No PropertyHanager Image: Im	A Mirror Entities Linear Sketch Pattern Wove Entities Display/Delett Relations Display/Delett Relations	* ************************************	<u>3</u> -=×
Part1 (Default O Sensors Annotation Haterial <r td=""></r>	Selected Entities Relation Rel	H s H		
Front Plane Top Plane Right Plane Corigin Corigin Corigin	Existing Relations			· · · · · · · · · · · · · · · · · · ·
	Add Relations A Horizontal Image: Collinear Collinear Image: Collinear Parallel Equal Fix Fix			
C Mode SolidWorks Education	Options Per construction For construction Front Edition - Instructional Use Only		Normal Distance: 159.51mmTotal Length: 62.68mm Under Dr	efred Eding Stetchi 🔮 💈 🧭

Then go to smart dimensions to define the length of the structure. Click on Smart Dimension >

Select the sketch of the line > edit the value



3. Extrude the sketch to create a solid in 3D dimension

Click on the **Features** Tab > **Extruded Boss/ Base**. In the **PropertyManager** window, change the dimension of **D1** to 6000.00mm.

🚳 Solid Works 🕴 🗋 - 🔌 - 🙀 - 😓 - 🦃 - 💽 - 🛢 🗉	Part1 *	?•- #×
Extruded Revolved & Lofted Boss/Base Boss/Base Boss/Base to Cut Waard Cut	Swept Cut Cafed Cut Paltern & Draft 😑 Done Reference Curves Pattern & Draft 🧐 Done Reference Curves Instant3D	
Features Sketch Evaluate DimXpert Office Products Flow Simula	on 🧠 २२, ४४ 🖬 🕮 - 🗊 - ६४ - 🚱 👷 - 😡 -	- 8 ×
PropertyManager		
Boss-Extrude1 ?		
* 🕍 🖌 🏹 6°C		
From A		/
Sketch Plane		·
Direction 1 8		
Bind		
> 5000.00mm		
VD1		
Draft outward		
Direction 2 ¥	7//	
Selected Contours 🛛 🕹	42	
< > *Trimetric		
SoldWorks Education Edition - Instructional Lise Only		Edition Part 2

4. Start SimulationXpress

Go to Tools, add-ins and remove Simulation from the checked list. Then go to Evaluate Tab and

SolidWorks File Edt Vew 2n	sert Tools ANSYS 13.0 Window	140 2	· 💽 • 8 🗇 🖾 •	Cartlever_J_bean *	😵 Search Soldworks Help 🛛 🔎 🕴 🖉 🖉
Stady Design Study Measure Properties Properties	Statistics	 Deviation Analysis Zebra Stripes Curvidure Parting Line Analysis 	Symmetry Check Symmetry Check Thickness Analysis Sig Compare Documents	Chel Chel Sinulatoritoress Atoress DPHOpress DrivelVoratoress Sust Analysis Wated Analysis Maard Waard	anotr 35
Transa So Cruster Upon Carlos Lamo Orbito Co Carlos Lamo Orbito Co	Office Products [19.00	R III (1) (1) (1) (1) (1) (1) (1) (1) (1) (1)	
3					
			//		What would you like to do?
1					Vers the part is different control from the second Control from the second Control and demonstrain the second Control and demonstrain the second Control and assembly from the second Control and second Control and Secon
*isomet	x ric				How do I close this window?

click on **SimulationXpress** Command.

Click on the **Options** to change the **units** to SI.

SolidWorks File Edit Vew	Insert Tools ANSYS 13.0 Windo	Help 2	9 · 📳 8 🗇 🗐 ·	Cantle	ver_L_beam *	🎯 Search SolidWorks Help 👂 ? - 🖶 2
esign Measure Mass Section Se Properties Properties	Check ensor Statistics	Deviation Analysis Deviation Analysis Zebra Stripes Dudercut Analysi	A Symmetry Check Q is thickness Analysis On Activ	A Simulator/Upress Plotpress DPH0 Analysis Wizard Analysis Anal	press DriveWorksXpress Sustanabilit; ysis Wizard	
ratures Sketch Evaluate DimXpe	ert Office Products	Constraine Strandine vie	l 😵 🖬 🕼 - 🗊 - dr - 🔮 🎗	Wizard Wiz	*d -	→ B × SolidWorks SimulationXpress
Caroliners J. berm (Default < D Sensor The Sensor The Sensor T					1	Welcome to SolidVorks Similatoria/press such for the part To deter first day and start span, door than the part To deter first shart and start span, door than the part To deter first shart and start span, door than the part To deter first shart shart span and start span. The options Start Over
A SimulationXpress Study (-Def. Parameters Cantilever, Learn (-Alloy Ste Finances External Loads A Results	SimulationXpress Of	otions				for the
Stress (-vonMiser-) Displacement (-Res disp-) Deformation (-Displacem Factor of Safety (-Max von Factor of Safety (-Max von	Results location:	C:\Users\\dong\App[Data\Local\Temp			
			Cancel			What woold you like to do?
20	i,					Create a drawing from this part. Create an assembly from this part. Create an assembly from this part. Create an assembly form this part. Create an asse
*Iso Model Motion Study 1	metric SimulationXpress Study (Fr Des	gnXpress Study				

5. Run simulations

First, click on **Next** to **Add a fixture**. Select the I-shaped face to restrain the face from moving then you will see a green fixture there.

Second, click **Next** to define the **Loads**. In this tutorial, we will define a force on the top face of the beam. Click on **Add a force**, then select the top face, the pink force will appear on the solid. Then add the value of 30000N to the beam, this distributed load is applied normal to the face and over the whole surface of the beam.

Third, click **Next** to apply the **Material**. Choose the material of Alloy Steel from the library. Click on Apply then close this window.

SolidWorks Materials	Properties	Favorites			
🕂 🔠 Steel	Material	properties			
→ 1023 Carbon Steel Sheet (SS)	Materia	Is in the default lib	rary can not be	edited. You must first co	py the material to
201 Annealed Stainless Steel (SS)	a custo	m library to edit it.			
A286 Iron Base Superalloy	Model 1	Type: Linea	r Elastic Isotropic	· •	
AISI 1010 Steel, hot rolled bar					
AISI 1015 Steel, Cold Drawn (SS)	Units:	SI - N	I/m^2 (Pa)	•	
\$∃ AISI 1020	Catego	ry: Stee			
-3∃ AISI 1020 Steel, Cold Rolled					
AISI 1035 Steel (SS)	Name:	Alloy	Steel		
3∃ AISI 304					
3∃ AISI 316 Annealed Stainless Steel Bar (SS	Descrip	tion:			
Steel Sheet (SS)	Courses.	. —			
∃ AISI 321 Annealed Stainless Steel (SS)	Source				
∃ AISI 347 Annealed Stainless Steel (SS)	Sustain	ability: Defin	ed		
3∃ AISI 4130 Steel, annealed at 865C					
∃ AISI 4130 Steel, normalized at 870C	Dranaster		Makia	Unite	
3∃ AISI 4340 Steel, annealed	Flootic Mo	dulua	210:011	M/mA2	
3∃ AISI 4340 Steel, normalized	Paissons	Patin	0.28	N/A	
3∃ AISI Type 316L stainless steel	Shear Mo	dulus	7.9e+010	N/m^2	
3∃ AISI Type A2 Tool Steel	Density		7700	kg/m^3	
-SE Alloy Steel	Tensile St	trength	723825600	N/m^2	
Alloy Steel (SS)	Compress	sive Strength in X		N/m^2	
-3∃ ASTM A36 Steel	Yield Stre	ingth	620422000	N/m^2	
Cast Alloy Steel	Thermal E	xpansion Coeffici	ent 1.3e-005	/K	
Cast Carbon Steel	Thermal C	Conductivity	50	W/(m·K)	
Cast Stainless Steel	Specific H	teat	460	J/(Kg·K)	
Chrome Stainless Steel	materiality	amping Ratio		I N/A	
1 1 9					

After the **Fixture**, **Loads**, **Material** are fully defined; click **Next** to **Run the simulation**.

Then you will see the screen below when it is running, solid mesh is in progress.

Mesh Progress	1
Meshing: Boss-Extrude1	
Memory usage: 426 524K	
Elapsed time:2s	
Solid mesh in progress	
Cancel	

Once the simulation is completed you will see the **Results** icon on the left. To see how this simulation works, click **Play animation**. After then, click **Yes, continue.**

6. Results Analysis

By default, SimulationXpress starts the analysis in terms of FOS (factor of safety).

(It is a term describing the structural capacity of a system beyond the applied loads or actual loads. Essentially, how much stronger the system is than it needs to be for a given load. So FOS tells you that the part will fail at locations where FOS <1. To account for uncertainties in values, flaws in materials, etc., values for FOS are typically greater than 2. If a FOS is "too large", then the designer may consider changing the geometry of the part as the design may be over-conservative. However, there may be other reasons, for example, other loading conditions that put further restrictions on the geometry.)

To see the results for the entire analysis, here enter a value greater than the default FOS=1, for example enter 20, then you will see the results that **RED** <= **FOS=20**<=**BLUE**.

Click on the other results, for example, **Show displacement**, and then it gives you an intuitionistic view of the displacement and as it is shown that the maximum displacement occurs at the end of the beam in red.

Repeat the same procedure to look at the stress distribution. Click Show von Mises stress

Note that all the variables of the **Fixture, Loads, and Material** can be redefined by simply going back to each step to change. Then run the simulation again to get new results.

Congratulations! You have completed this tutorial.

Now that you are done, can you try to...

- 1) What happens if you fix both ends of the beam (rather than just one end)? How does this effect the stresses and displacements? Does this make sense?
- 2) What happens if you change the material? How do the stresses in the structure change? How does the factor of safety (FOS) change? (Hint: do a Google search and compare the values of the Yield Stress for different materials.)
- 3) For the given initial geometry, calculate by hand the theoretical maximum force that can be applied given a Factor of Safety of 3.0. Apply this load within the software are verify that the stresses are the same.
- 4) What happens if you load the structure such that it acts like an 'H beam' rather than an I beam?
- 5) Change the cross-section of the beam so that you minimize weight but maintain a FOS of greater than 3 (keeping the load the same and maintaining a width of 250 mm and keeping the symmetry of the beam). Also allow the maximum deflection not to exceed 1.5 mm.