Welcome to

Solidworks Simulation Tutorial

of

Torin BIG RED Aluminum Jack



In this tutorial, we will use Solidworks Simulation to perform stress analysis on an aluminum floor jack built by Torin. We will apply a maximum load that this floor jack can handle as claimed by the manufacturer and see if this floor jack can actually support that maximum load without failure on any of its component.

SolidWorks Simulation Tutorial

1) Open Big Red Aluminum Racing Jack.SLDASM.



2) First of all, we need to suppress the front wheel, rear wheel assemblies and handle. To suppress all of those parts simultaneously, left-click **Front Wheel** and ctrl+left-click **Rear Wheel Assembly** <1>, <2> and **Handle**.



3) Now, right-click and select **Suppress.**

Ð



4) This window will pop up. Click close.

🕫 Wha	at's Wro	ng			
Туре	Feature	Preview	Help	Description	
Marning	2 Equations			Big Red Aluminum Racing Jack: The following equations could not be evaluated because of deleted/invalid dimensions: Equation 14: DD1@Sketch1@Front Wheel.Part Equation 55: DD1@Sketch1@Handle.Part Equation 56: DD2@Extrude4@Handle.Part Equation 14: D1@Sketch1@Front Wheel.Part Equation 57: D1@Extrude4@Handle.Part Equation 14: D1@Sketch1@Front Wheel.Part Equation 57: D1@Sketch1@Front Wheel.Part Equation 56: DD2@Extrude4@Handle.Part Equation 56: D1@Sketch1@Front Wheel.Part Equation 56: D2@Extrude4@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 57: D2@Extrude4@Handle.Part Equation 57:	
Show errors Show warnings Display What's Wrong during rebuil					

5) The assembly should now look like this.



6) Now, we will define a distance between the plate, which supports the load from a car, and the ground. For this

case, we assume the bottom surface of the frame as the ground. Click on <u>Mate</u> Mate. The Mate window will appear on the left. Under Mate Selections, we can first select top surface of the plate as shown in the picture below.



7) Second, select the bottom face of the frame as shown in the screen shot below. Under **Standard Mates**, click on



8) The top face of the plate and the bottom face of the frame are now 4" from each other. Now we will use SolidWorks to analyze the stress on the floor jack assembly when the load is applied. Click on the Simulation tab



9) Click on 'New Study'.



10) Under Type Select **Static**



12) Now, we need to assign material for the floor jack assembly. Under Study1, right-click Parts and select Apply Material to All.



13) The material window pops up. From the material list click the + sign next to Aluminum Alloys.

Material									×
	*	Properties	Tables & Cu	rves A	ppearance	CrossHatch	Custom	Application Da	ita F 🔹 🥬
		 Material n 	roperties —						
		Material	s in the defa	ult librar	y can not be	edited. You	must first (copy the materi	al to
		a custor	n library to e	dit it.					
		Model T	vne:	inear El	lastic Teotror	vic .			
→ 3 201.0-T43 Insulated Mold Casting (SS)		1 Hodor I	7pc.	Linear Li	1850C 1500 Of	//L			
→ 3 = 201.0-T6 Insulated Mold Casting (SS)		Units:		SI - N/m	^2 (Pa)		•		
→ 📑 201.0-T7 Insulated Mold Casting (SS)		Categor	nve E	Aluminiu	im Allovs		7		
	Ξ	catogo	· ·	/ Inclining					
} Ξ 2014-O		Name:		2024 All	loy				
		Default	failure 🚺	Maxwaa	Missa Stree				
3 Ξ 2014-T6		criterion	10	Max von	I Mises Stres	s ,			
3 2018 Alloy		Descript	tion:						
2024 Alloy			ſ						
3 2024 Alloy (SN)		Source:	l						
§ ≣ 2024-O		Sustaina	ability:	Defined					
≣ 2024-T3									
3 Ξ 2024-T361		Descent			Maha	11-3-			
§ ⊒ 2024-⊤4		Property Electic Mo	dulua	_	Value	Units			_^î
		Poissons I	ouius Ratio		0.33	N/A			
3 Ξ 2219-T31		Shear Mod	dulus		2.8e+010	N/m^2			
∃ Ξ 2219-T37		Density			2800	kg/m^3			
∃ Ξ 2219-T62		Tensile St	rength		18612600	0 N/m^2			E
2219-T81		Compress	ive Strength	in X		N/m^2			
		Yield Stree	ngth		75829100	N/m^2			
2618-T61 (SS)		Thermal E	xpansion Co	efficient	2.3e-005	/K			
3003 Alloy		Thermal C	onductivity		140	W/(m·K)			
3003-H12		Specific H Material D	eat amping Datia		800	J/(Kg·K)			
3003-H12, Rod (SS)		, waterial D	amoino katio			IWA			
3003-H14	-	ſ	Apply	Cloco	6-14	e Confu		Help	
		l	Арріу	Close	Sav	Connig		heip	

14) Under Aluminium Alloys, select 2024 Alloy. Click Apply, then Close.

15) Now, we will apply force to the assembly. Right-click External Load select Force.



16) Select the top face of the jacking surface



17) Change the Units to English (IPS). Under Force, select normal to direction and set 3000 as the magnitude of force. Check that force is pointing downward.



18) Now, we will define constraints to the assembly. Right-click **Fixtures** and select **Fixed Geometry**



19) For Faces, Edges, Vertices for Constraint, select two upper half of the cylinder where the shaft for the front wheel is mounted. Refer to the picture below.

		IIIdWorks rie Edit vew Snert Tools Toolson Simulation Wordow Help 2 🗋 - 🗗 - 🔄 - 🖓 - 🧐 - 🖁 👘 🔤 -	Big Red Aluminum Racing Jack	Search SoldWorks Help P • 7 • - @
		「「四白北市江戸町」なりらりたちに今回人 第19月日のため回転日本 「ロット」の日本の日本		
		H H H H H H H H H H H		
	Image:	Apply Indultis Extensis Carrier Main Action Deformed Compare (22 Pol Tools - Advisor Result Result Result Result Result Result Result		
		by Layout Sketch Evaluate Office Products Flow Simulation Simulation	A. 19.	
		Police Po	A	
		andard (Pract Generative) &	Z	
		Posed Hinge		
		Face(1) (frame-1 Face(2) (frame-1)		
			at Geometry	
		hanned V		
		and fatting V		
		moor settings 0		
Inder Defred Löberg Assembly Diversion of the State of th				
Inder Defred Edeng Assembly 10 Education State (Labora Assembly 10) Education State (Labora Assembly 10)	Video Under Dialogo 11 ki Skedy 1 Ki Education (doon - IndiagCood Muc Orly Video Defined Editing Assembly			
Inder Defred Edeng Assembly 10 Education States 1 to State 1 to State 1 to State 1 to State 2 to S	A B C al			
Inder Defend Löting Assembly 10				
Index Defined Editing Assembly II	toder Tedors Ted Start 1 + Stedy 1 to Education Editors 1 + H Stedy 1 to Education Educa			
Index Defined Editing Assembly II I Education Interfactors Interfactors Under Defined Editing Assembly II OPTION Control Interfactors	All Defending Assembly II All School To Letters Defending Assembly II Control California Californ			
I focus (difere Infractional UL Off) I focus (difere Infractional UL Off) Inder Definet Editing Assembly II I focus (differe Infractional UL Off) I focus (differe Infract	La sa la deficie de la deficie			
I force for the start of the st	I deter Defined Edition Assembly I Control Edition Assemb			
I Gozzini Gildon - Indiacita Gildon - Jul Steely 1	A B C A	20-Lex		
International Index Index International Up Only Under Defined Editing Assembly Up Only 2017	Model Model Russet 1 Mc Steely 1 Sectory Control Russet 1 Mc Steely 1 Sectory Control Russet 1 Sectory Assembly 1 Sectory Russet 2 Sectory Rus			
Linder Defined Editing Assembly III 2017	Inder Defined Editing Assembly	Model Model Study 1 1 Study 1		x - 1
		As Education Edition - Instructional Use Only		Under Defined Editing Assembly

- 20) Now, we will define the second restraint to the assembly. Right-click Fixtures and select **Fixed Geometry**. (see step 18)
- 21) Under the rear wheel mounts, click on the two circular faces as shown in the picture for the **Faces, Edges, Vertices for Restraint**. These two faces are in contact with the bearings from the rear wheels. Therefore, they are restrained.



22) On the part lists, right-click **Bottle Jack** and select **Bible**.



- 23) The Bottle Jack is now hidden and we can go back to the SolidWorks Study.
- 24) Right-click Fixtures and select Advanced Fixtures



25) For Faces, Edges, Vertices for Restraint, click on the end face of the piston as shown in the picture.

26) Select the Front Plane from the assembly tree as the **direction for restraint**.(next to the pink box)

27) Under translation, click **Normal to plane** and set the value to 0.



28) We have completed applying restraints to the assembly. Now, we can show the Bottle Jack back to the assembly.



29) Go to FeatureManager design tree, right-click Bottle Jack and select Show.



30) The Bottle Jack is now shown back to the assembly.



31) Now, we will define a contact type to the assembly. Expand the tree item labeled **Connections** then **Component Contacts** and right click on **Global Contact** and click Edit Definition

32) Select Bonded. Click OK.



33) Now, it is time to create mesh. Right-click Mesh and select Create Mesh.



- 34) The Mesh window appears on the left. Click **OK** to accept the default values for meshing.
- 35) The program is now creating the mesh for the floor jack assembly.
- 36) Meshing is now complete and it is time to run the Solidworks analysis. Right-click **Study 1** and select **Properties**.



37) Select **Direct sparse** as a solver and click **OK**. Using Direct sparse as a solver will minimize the time to run the analysis and it also gives more accurate results. However, the downside of using this solver is that it uses too much RAM space which will cause the computer to run slow.

Static	<
Options Adaptive Flow/Thermal Effects Remark	-
Gap/Contact Include global friction Friction coefficient: Ignore clearance for surface contact	
Large displacement	
Solver	
O Direct sparse Use inplane effect	
OFFE Use soft spring to stabilize model	
OFFEPlus Use inertial relief	
Results folder D:\Documents and Settings\Class2007\Desktop	
OK Cancel Apply Help	

38) Now Click **Run** at the top of the screen.



- 39) SolidWorks is now running the FEA analysis.
- 40) The analysis is now complete.



41) The von Mises stress plot, is shown by default



42) To better understand how the parts deform under loading, we can use an animation function to show us the stress and deformation of the assembly over a period of time. Go to **Simulation** menu on top, then go to **Result Tools** and select **Animate**.



43) Hit the **play** button to run the animation.



44) Click OK to end animation

45) Note that some components actually intersect each other. This is not desirable because in real life, parts interference does not occur. Therefore, accurate results will not be obtained if the interference is present. The ninture below points out the interferences.



46) Interference can also be seen here.



47) To fix the interference problem, we need to further define the contact or gaps for the assembly. First, put the mouse pointer anywhere on right frame surface and right-click it. Then, select Change Transparency.



48) The Right Frame is now transparent.



49) Right-click Connections and select Contact Set.



50) For Faces, Edges, Vertices for Source, select the face of Parallel Linkage as shown in picture below. (Note: The Parallel Linkage surface that you need to select is highlighted in green.)



51) For Faces for Target, select the surface of the **Right Frame** as shown in the picture below. Under **Type**, select **No Penetration**. Click **OK**. (Note: The **Right Frame** surface that you need to select is highlighted in pink.)





52) Right-click Connections and select Contact Set.



53) For Faces, Edges, Vertices for Source, select the face of Spacer Pin2<2> as shown in picture below. (Note: The Spacer Pin2<2> surface that you need to select is highlighted in green.)



54) For Faces for Target, select the surface of the **Right Frame** as shown in the picture below. Under **Options**, select **Node to Surface**. Click **OK**. (Note: The **Right Frame** surface that you need to select is highlighted in pink.)



55) Right-click Connections and select Contact Set.



56) For **Faces, Edges, Vertices for Source,** select the face of **Bottle Jack** as shown in picture below. (Note: The **Bottle Jack** surface that you need to select is highlighted in green.)







- 58) We have completed defining gaps to avoid interference of the parts with the right frame. We can now change the transparency of the right frame back to normal. Put the mouse pointer anywhere on right frame surface and
 - right-click it. Then, select Change Transparency.



59) The Right Frame now is no longer transparent.



60) Now, let's fix the interference problem of the parts with the Left Frame. Put the mouse pointer anywhere on the

Change Transparency. left frame surface and right-click it. Then, select SolidWorks _ re 2 D · 2 · 1 · 2 · 9 · 1 · 1 · 1 P . 7 . - 8 × 雪式目目前非 Shidy Assembly Larout Sketch Evaluate Office Products Flow Simulation Simulation QQ800.0.0.0.0.0. «.g - = × Gett 10 m 0 66 Li 🗞 🌬 🖆 8 P Onlin Del Pr e -99 Parent/Ch Y Insert Ean t 03 😢 (1) - B + a

61) The left frame is now transparent Right-click Connections and select Contact Set.



Faces, Edges, Vertices for Source, select the face of **Parallel Linkage** as shown in picture below. 62) For (Note: The Parallel Linkage surface that you need to select is highlighted in green.)





64) Right-click Connections and select Contact Set.



65) For Faces, Edges, Vertices for Source, select the face of Spacer Pin2<1> as shown in picture below. (Note: The Spacer Pin2<1> surface that you need to select is highlighted in green.)











68) For Faces, Edges, Vertices for Source, select the face of Bottle Jack as shown in picture below. (Note: The Bottle Jack surface that you need to select is highlighted in green.)



69) For Faces for Target, select the surface of the Left Frame as shown in the picture below. Under Options, select Node to Surface. Click OK. (Note: The Left Frame surface that you need to select is highlighted in pink.)



70) We have completed defining gaps to avoid interference of the parts with the left frame. We can now change the transparency of the left frame back to normal. Put the mouse pointer anywhere on left frame surface and right-

click it. Then, select Change Transparency.



71) The left frame now is no longer transparent.



72) Right-click Mesh and select Create Mesh.



73) Click OK when this message prompts.

<u>.</u>	Reme	shing will delete	e the results for	study: Study 1.
		ОК	Cancel]

74) Click **OK** to accept the default values.

75) Re-meshing is now complete.

76) Click Run Study



77) SolidWorks is now running the FEA analysis.

78) SolidWorks has completed the analysis.



79) The von Mises Stress is shown by default.







81) Hit the **play** button to run the animation.



82) Note that parts interference no longer occurs.



83) No more interference.



84) Click on Simulation > Design Check wizard



85) Click OK to use default Settings



86) Now, let us find out the safety factor of individual component. Since the **Main Arm** is one of the most important parts that determine how much load the floor jack can support, let us plot the safety factor of the **Main Arm**. Right-click **Factory of Saftey** and **Edit Definition**.







88) The safety factor of the **Main Arm** is plotted



89) This floor jack assembly was modeled with a parameterized function that allows the user to modify the geometry of certain component and having the SolidWorks to automatically change the geometry of the other components associated with the component which its geometry was manually modified. For instance, if you change the diameter of a pin, the hole in which the pin is inserted into is automatically modified with respect to the change in the pin diameter.

90) Now, let us change a radius of **Pin2** to utilize the parameterized function. Right-click **Pin2** and select **Open Part**.



91) Pin2 now appears on a new window.



92) Under Revolve1, right-click Sketch1 and select Show. .



93) Double-click Sketch1 to view the dimension of Sketch1.



94) Place the mouse pointer on the dimension noted as **D1@Sketch1**. Then, double-click that dimension to modify its value.



95) The **Modify** box appears. Change the dimension from 13/32" to 1/2".





97) Right-click Sketch1 and select Hide.



98) Go to Window and select Big Red Aluminum Racing Jack to go back to the assembly.



99) Click Yes when this window pops up.

SolidWorks 2006 (Automatically dismis
Models contained within the assembly have changed. Would you like to rebuild the assembly now?
Yes No
Don't tell me again in this session

100) Select Close when this message prompts.

🛙 What's Wrong							
Туре	Feature	Preview	Help	Description			
A Warning	Σ Equations			Big Red Aluminum Racing Jack: The following equations could not be evaluated because of deleted/invalid dimensions: Equation 14: D1@Sketch1@Front Wheel.Part Equation 55: D1@Sketch1@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 57: D1@Extrude4@Handle.Part Equation 14: D1@Sketch1@Front Wheel.Part Equation 55: D1@Sketch1@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 56: D2@Extrude4@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 57: D1@Sketch1@Handle.Part Equation 57: D1@Sketch1@Sketch1@Handle.Part Equation 57: D1@Sketch1@Sketch1@Handle.Part Equation 57: D1@Sketch			
Show er	rors 🔽 Show w	varnings 🔽] Displ	ay What's Wrong during rebuil Close Help			

- 101) Click Study1 tab at bottom to return to study
- 102) Since the dimension of the **Main Arm** was modified, we need to re-create the mesh to update the SolidWorks results. Right-click **Mesh** and select **CreateMesh**.



⚠	Remeshing will delete the results for study	: Study 1.
	OK Cancel	

- 103) Click **OK** to proceed with the default mesh parameters values.
- 104) **Run** the study
- 105) When SolidWorks has completed re-running the analysis, double-click **Factory of Saftey** to find out how the change in the **Pin2** radius affects the safety factor of **Main Arm**.



106) According to the manufacturer of this floor jack, this specific floor jack has a maximum lift of 17 ¹/₂" (distance from the **Top Plate** to the ground). In this tutorial, we analyzed the stress on the floor jack when the **Top Plate** is supporting a 3000 lb load at floor jack's minimum lift which is 4" from the ground. At its maximum lift (17 ¹/₂"), the stress throughout the floor jack will be different in comparison to its stress at its minimum lift (4"). To find out how the lift height affects the stress throughout the floor jack, you can perform the stress analysis on your own by simply changing the distance between the **Top Plate** and the **Top Plane** as discussed in **Step 7** of this tutorial. You will then need to re-create the mesh and re-run the analysis to update the SolidWorks results.