

SOLIDWORKS
WORLD 2015
FEBRUARY 8-11
PHOENIX CONVENTION CENTER | PHOENIX, AZ

SOLIDWORKS, Duct Tape
and ~~Bailing~~ Wire

*Baling*⁷

Creative solutions to everyday challenges



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Technical Services Director, CATI

Agenda

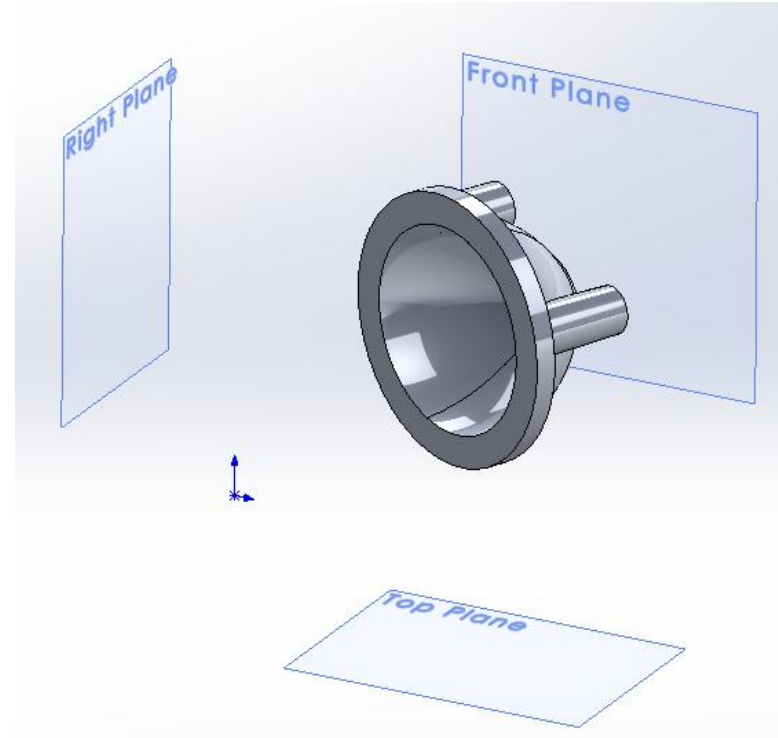
- Show you a problem
- Show you a solution to the problem
- Repeat until we run out of time



Imported part orientation

Problem:

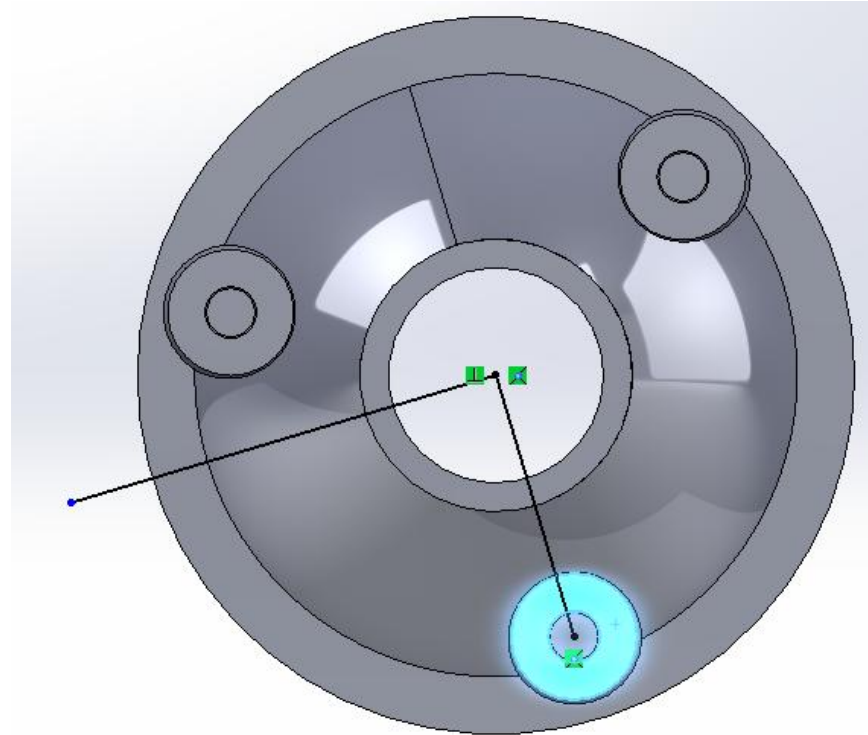
Your part that you imported has the wrong orientation and the location of the Origin is not where you want it.



Imported part orientation

Solution Step1:

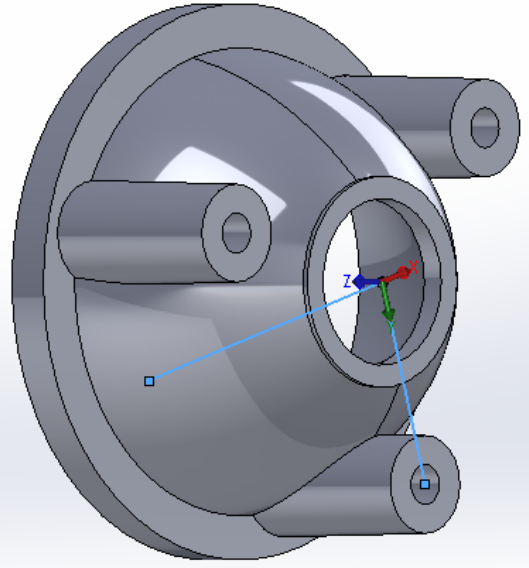
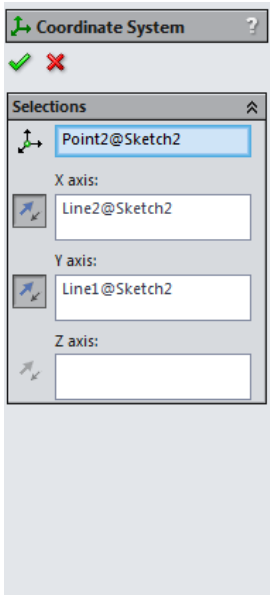
Create a Sketch to dictate our new orientation



Imported part orientation

Solution Step2:

Insert a Coordinate System (Reference Geometry) setting your Origin and Axis.



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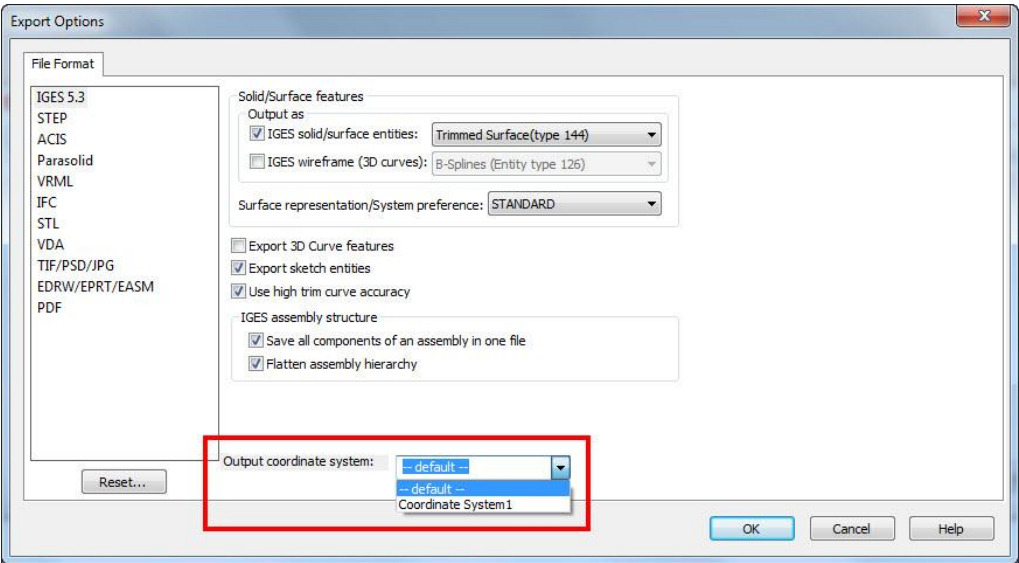
Imported part orientation

Solution Step3:

Export the file, setting the Option for Output coordinate system.

Solution Step4:

Import the file you just saved.



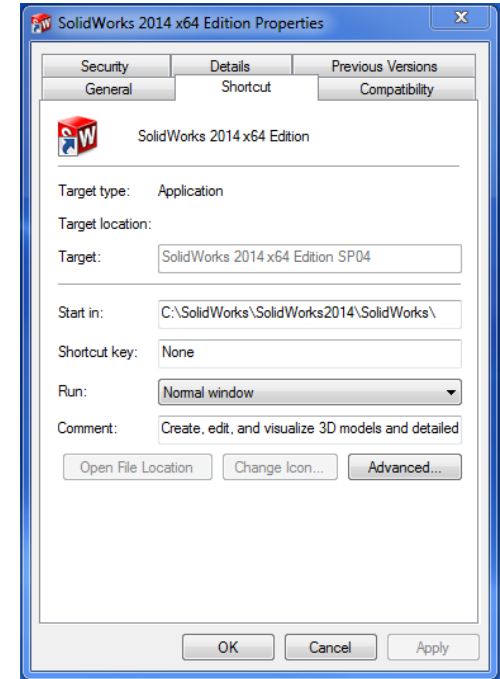
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How to change you SolidWorks Desktop Icon

Problem:

"I'd like to change the desktop icon to a Pac-Man icon so I can put it next to the AutoCAD icon (I've already changed this to a blue ghost) and make it look like SolidWorks is about to eat AutoCAD. However, I cannot change the icon using the normal method. Can this be done?"

Unfortunately SolidWorks does not allow you to Right Click on their icons and go to Properties and select the Change Icon



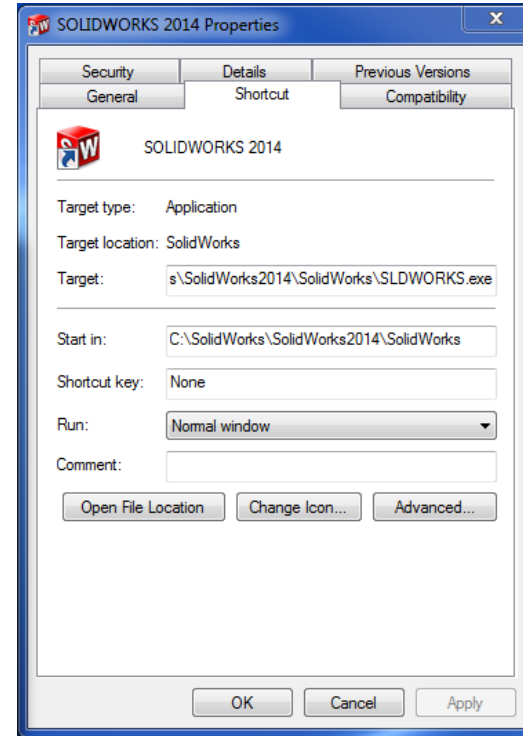
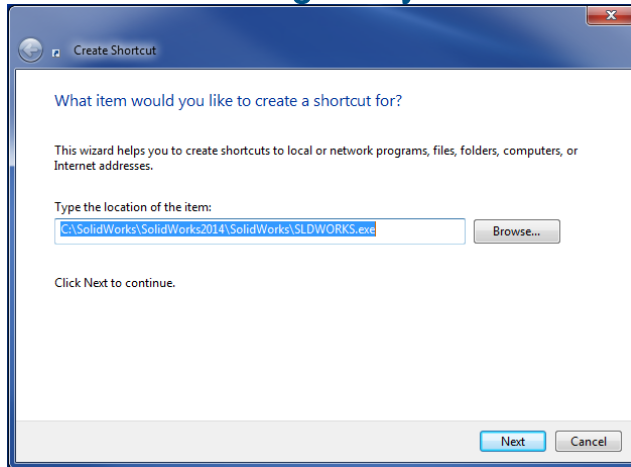
How to change you SolidWorks Desktop Icon

Solution Step1:

Create a new shortcut to SLDWORKS.exe.

Solution Step2:

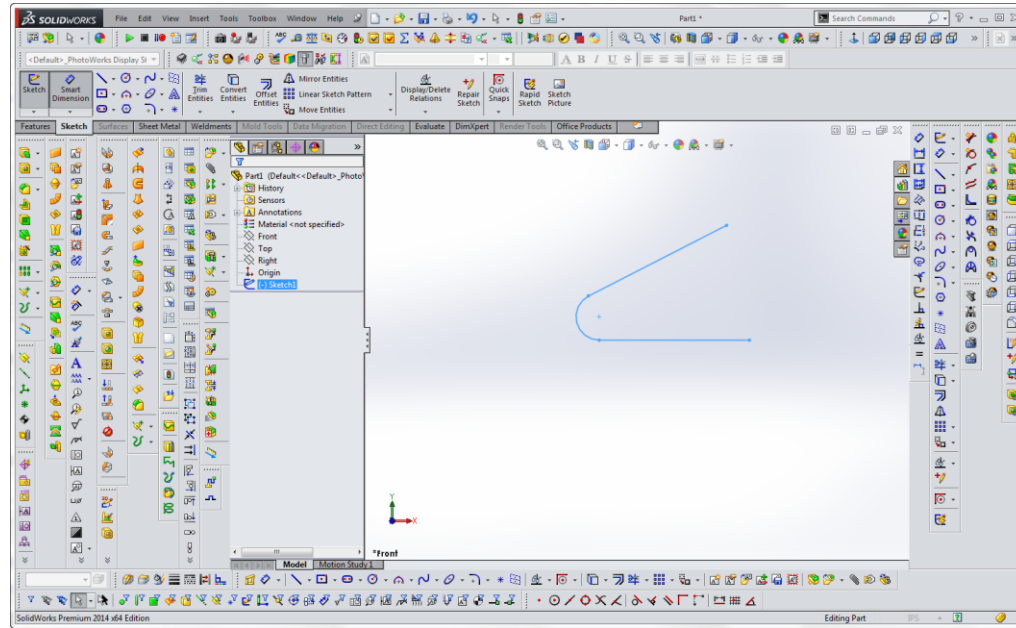
Change the icon to the image of your choice.



Toolbar Modification the Easy Way

Problem:

You are looking for the easiest way to modify your SOLIDWORKS toolbars.

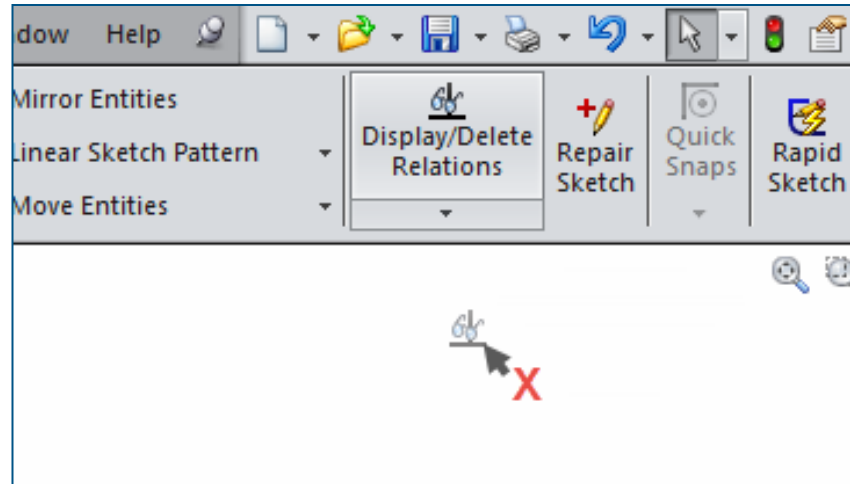


Toolbar Modification the Easy Way

Solution:

Use the **Alt** key

Option (1): To remove icons hold down **Alt** and drag your unwanted icon off the toolbar.

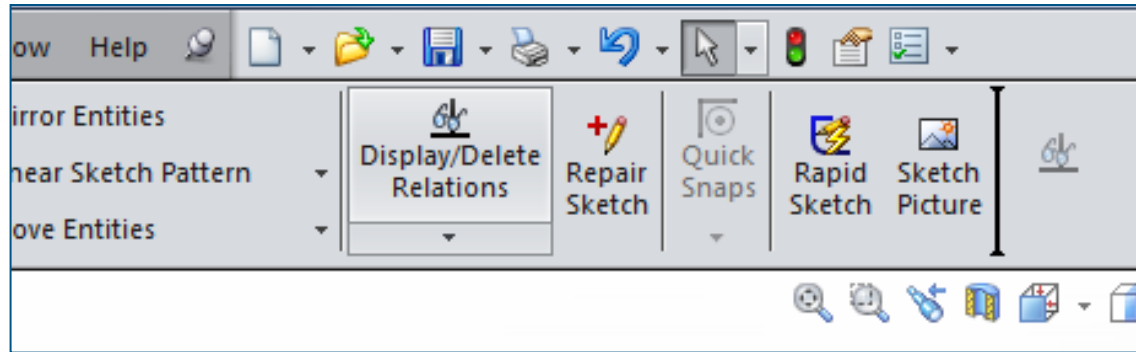


Toolbar Modification the Easy Way

Solution:

Use the **Alt** key

Option (2): To move icons hold down **Alt** and drag you icon to its new location.



Problem 1

Problem:

You want one of those fancy big flat screen TVs that all your friends have but are low on cash.



Solution 1

Solution:

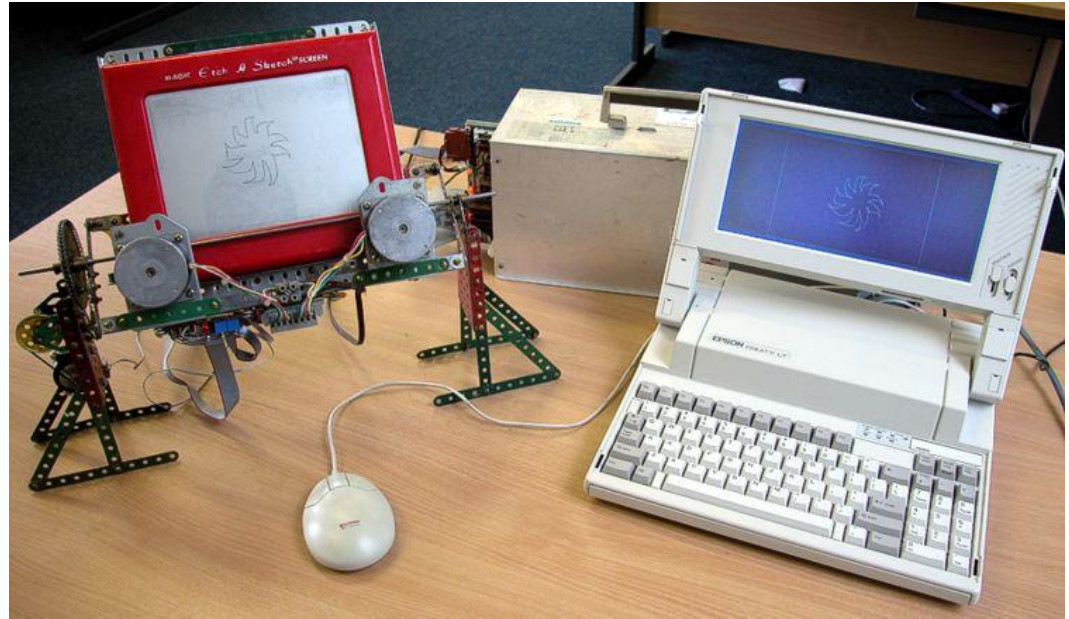
Get a used big screen from Craigslist and make it look like a flat screen.



Efficient Sketching Tips

Problem:

You want to make sketches as quickly and efficiently as possible.

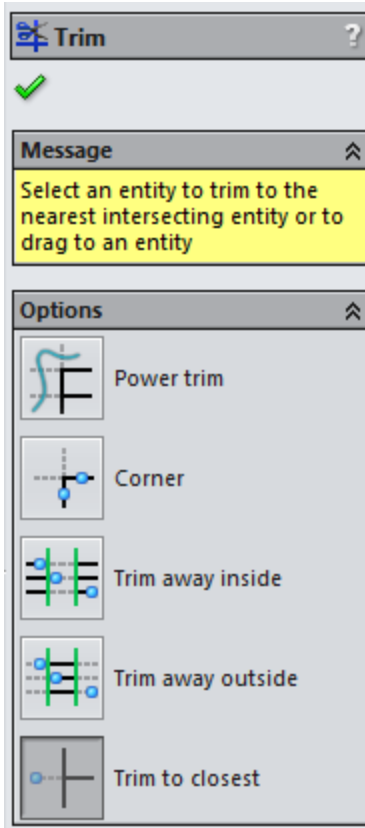


Efficient Sketching Tips

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Solution 1:

Use Trim to Extend. With ***Trim to closest*** option on you can click and drag the end of a line/arc to the next entity in a sketch.

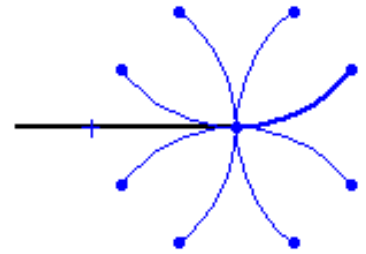


Efficient Sketching Tips

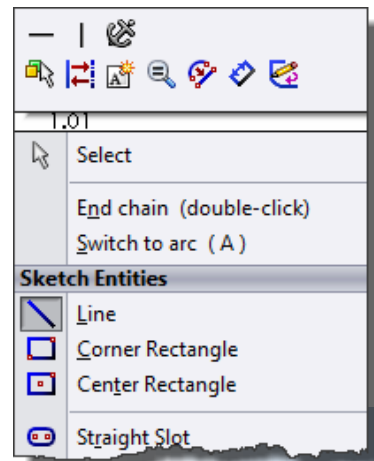
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Solution 2:

Use the Line tool to add Arcs. While in the Line tool sketch your first line Click/Click method when creating the next line move back to the end point and it will switch to a Tangent Arc, results vary depending on where you start and end your arc.



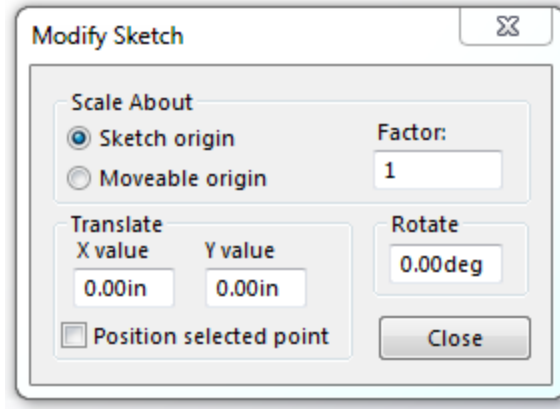
Note: Hot Key is A.



Efficient Sketching Tips

Solution 3:

Modify Sketch, found in Tools, Sketch Tools this can be used to move, rotate, scale or mirror you sketch.

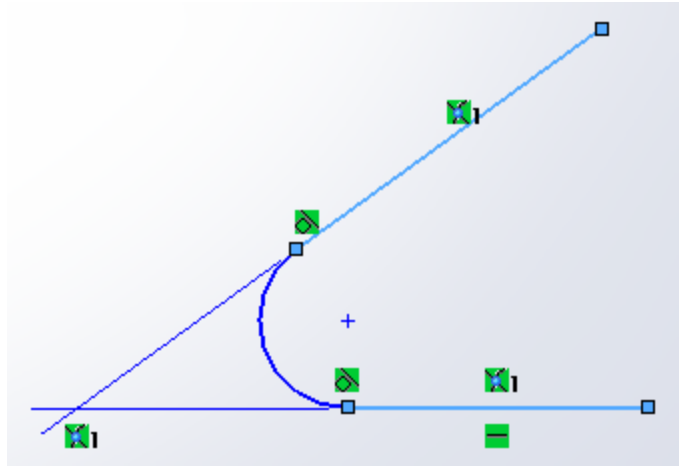
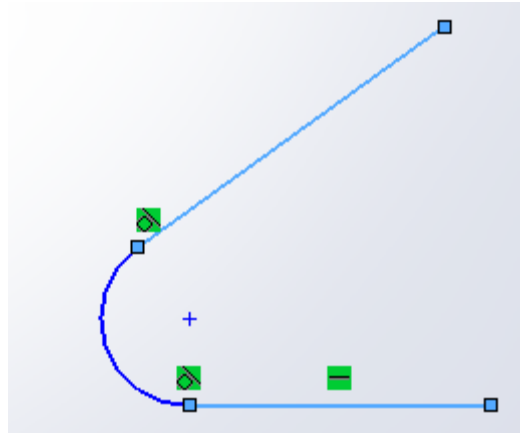


Efficient Sketching Tips

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Solution 4a:

Virtual Sharp, CTRL select two entities and select the Point tool from the Sketch toolbar.

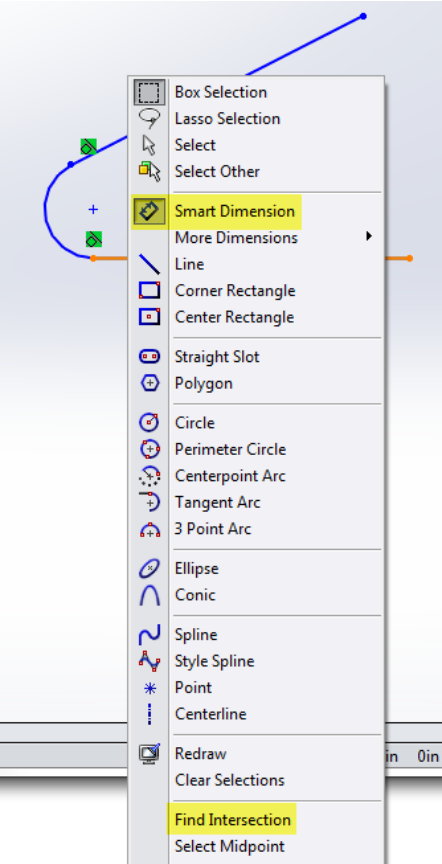


Efficient Sketching Tips

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Solution 4b:

Virtual Sharp, while in Smart Dimension Right Click on an entity and select Find Intersection.

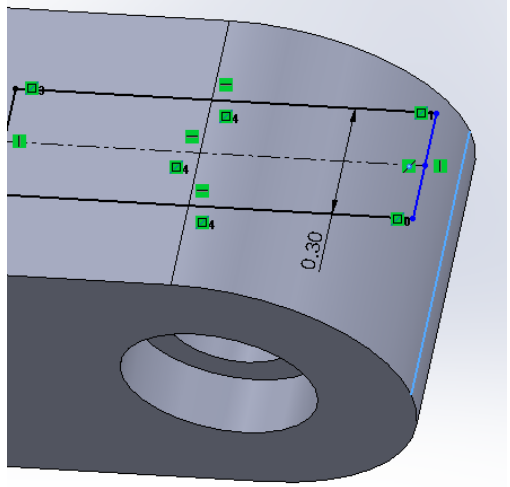
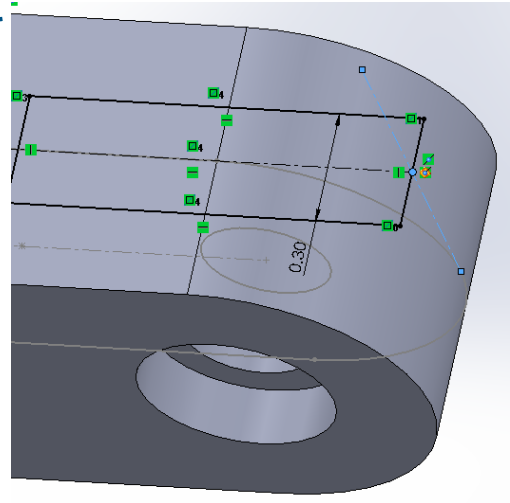


Efficient Sketching Tips

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Solution 5:

Can not select a Silhouette Edge, add construction geometry to your existing sketches and make the relation to that geometry.

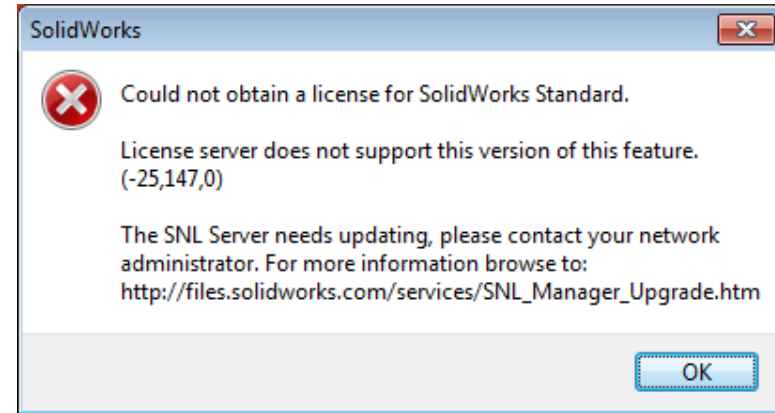


Moving a SOLIDWORKS Network License server

Problem:

Your SOLIDWORKS clients can not get licenses because the SNL server name has changed or has been replaced.

You now have to update 20 machines to look at the new server.



Moving a SOLIDWORKS Network License server

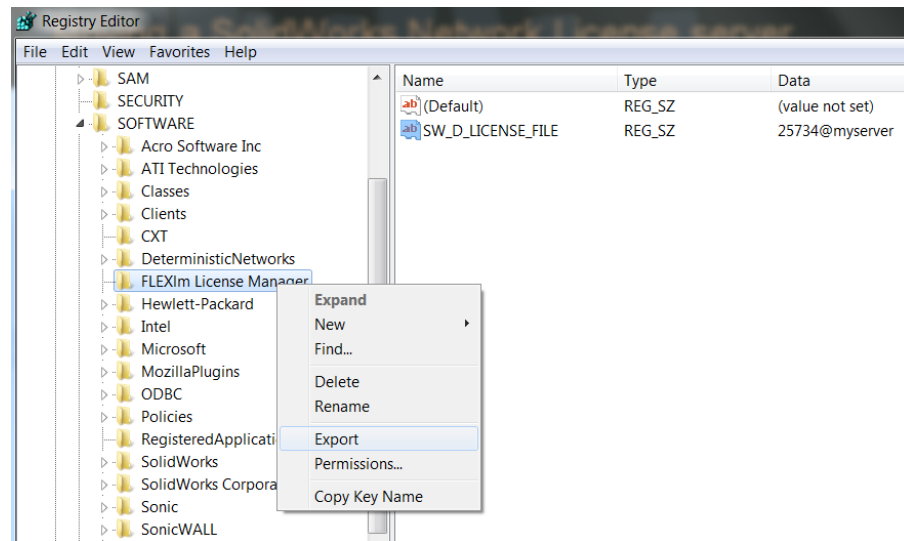
Solution Step1:

Export the registry key “HKLM\Software\FLEXIm License Manager\”

Windows Registry Editor Version 5.00

[HKEY_LOCAL_MACHINE\SOFTWARE\FLEXIm License Manager]

"SW_D_LICENSE_FILE"="25734@myserver"



Moving a SOLIDWORKS Network License server

Solution Step2:

Change the key to reference the new server and save the file

Windows Registry Editor Version 5.00

[HKEY_LOCAL_MACHINE\SOFTWARE\FLEXIm License Manager]

"SW_D_LICENSE_FILE"="25734@newserver"

Moving a SOLIDWORKS Network License server

Solution Step3:

Build a batch file that will import the reg file

```
Regedit /S "\\<path>\<reg file name>.reg"
```


Moving a SOLIDWORKS Network License server

Solution Step4:

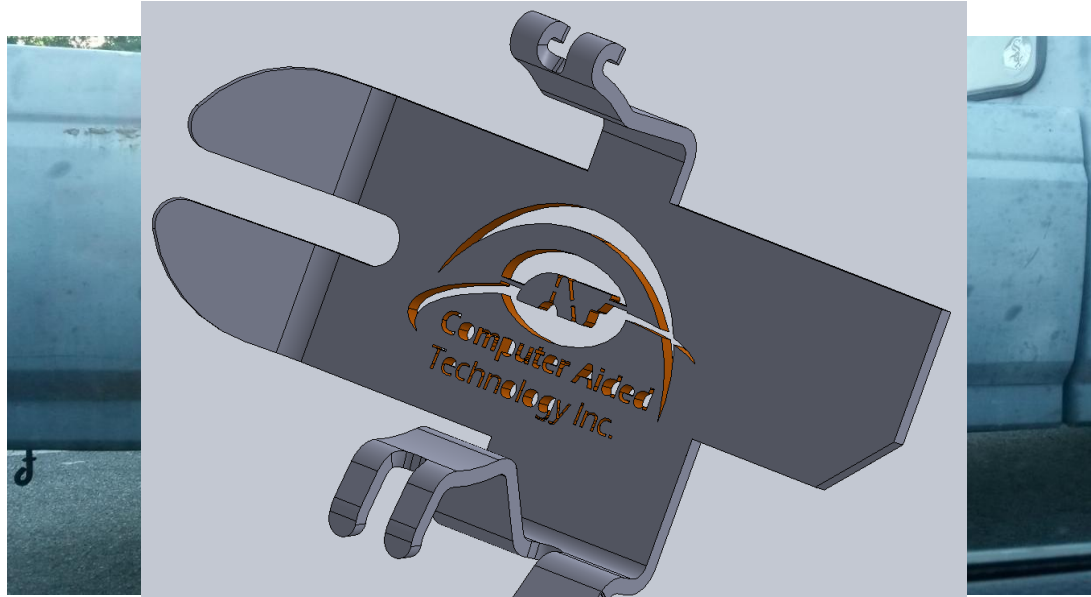
Deploy the command as you prefer:

- Send via email
- Offer it on an internal webpage
- Add the command to the login script for a couple of weeks

Incremental Sheet Metal Bending

Problem:

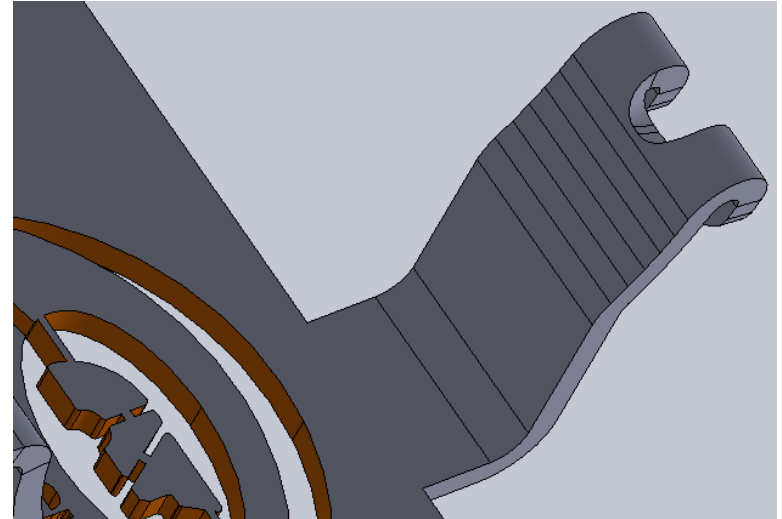
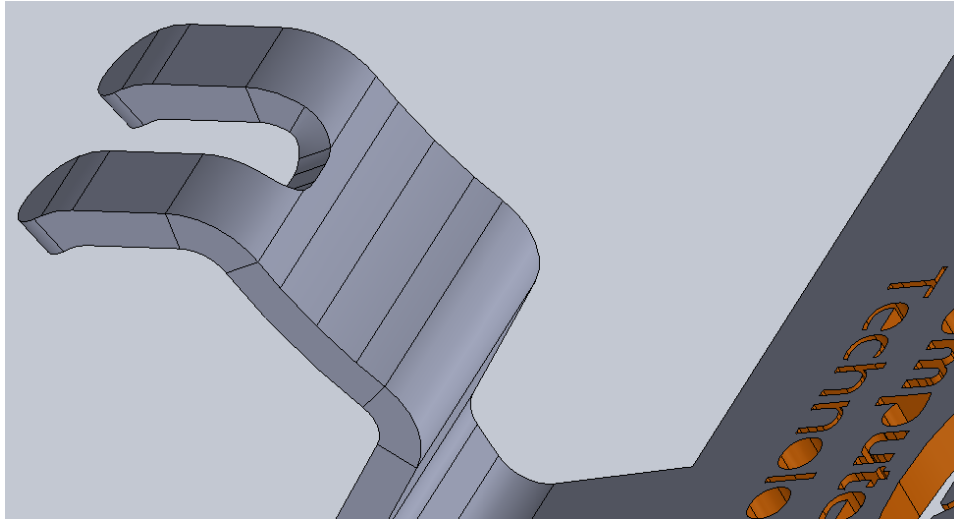
You have a complex sheet metal part that you need to form incrementally.



Incremental Sheet Metal Bending

Solution Step 1:

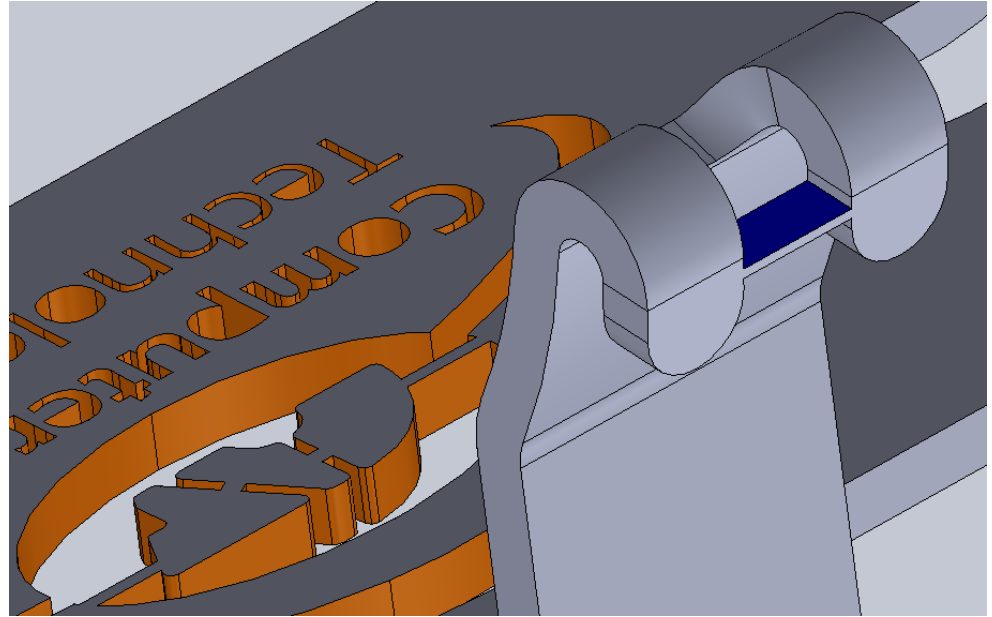
- Using Sketches and the Split Line Tool we were able to break the faces where we wanted our bends.



Incremental Sheet Metal Bending

Solution Step 2:

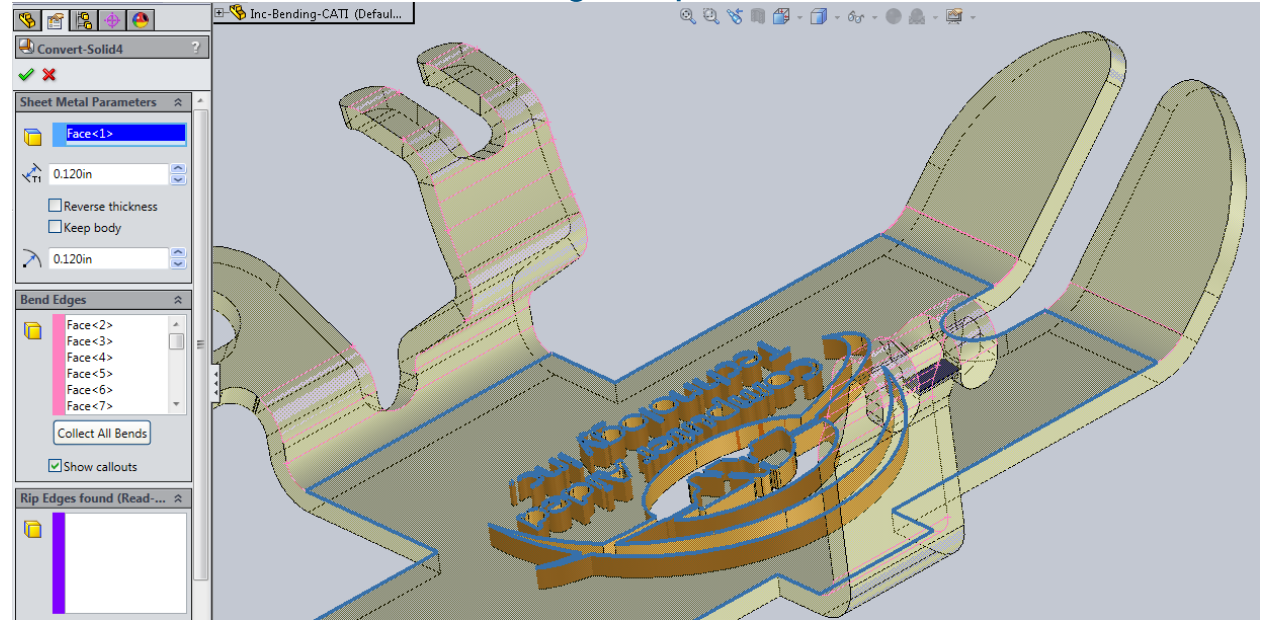
- Adding a bridge of material between two of the ears (shown in blue) allows them to flatten together.



Incremental Sheet Metal Bending

Solution Step 3:

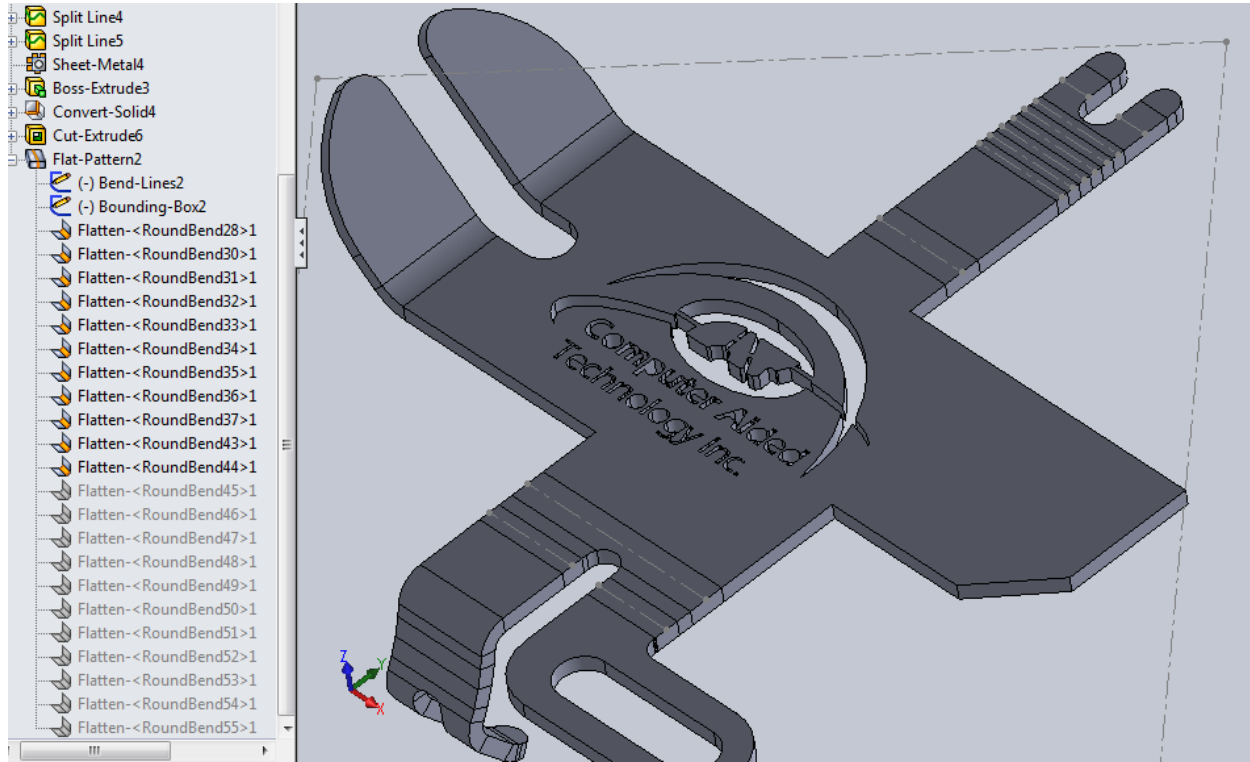
- Use Convert to Sheet Metal and Collect All Bends to get a part that has incremental bends.



Incremental Sheet Metal Bending

Solution Step 4:

- Flatten.

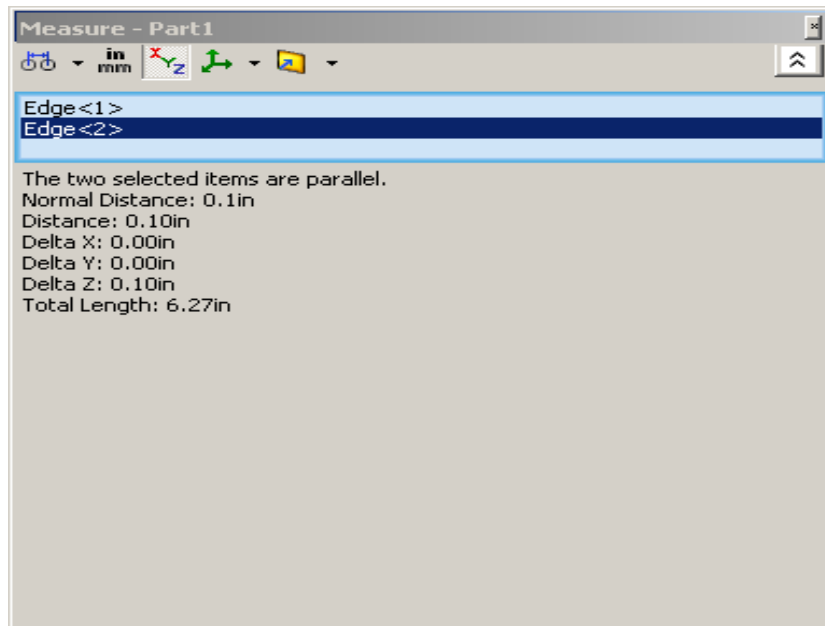


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Disappearing Dialog Box

Problem:

My Measure dialog box is no longer visible on the screen.



Disappearing Dialog Box

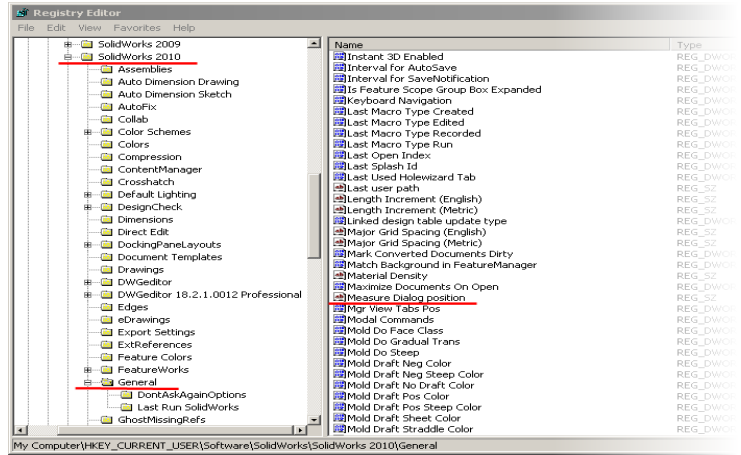
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Solution:

Option (1): Change the resolution of your screen to the maximum allowed.

Option (2): Edit the Registry key that is the position holder for that dialog box.

- HKCU\Software\SolidWorks\SolidWorks201X\General
- Find the “Measure Dialog Position” entry.
- Edit this key, delete any/all values, OK.
- Run SolidWorks, try Measure



Problem 2

Problem:

You got a new hunting trailer and need to move it to your hunting property.



Solution 2



Home Use License Upgrade

Problem:

You have a Home Use License of SOLIDWORKS and it will not allow you to upgrade it.



Home Use License Upgrade

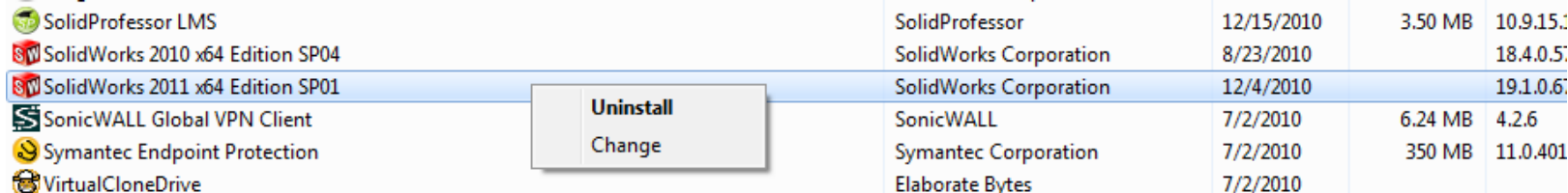
Solution:







Step1 – Go to Add/Remove Programs and find SOLIDWORKS and select Change.

Step2 – In the SW Installation manager enter in a valid SW Serial Number (network SN from work). Note: You do not need to go through the install process.

Step3 – Update to the latest Service Pack.

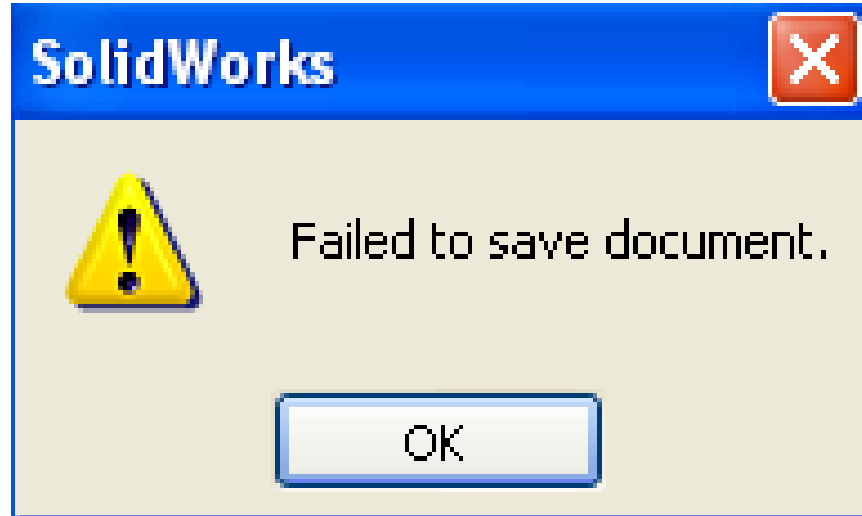
Step4 – Switch back to your HUL Serial Number to run SOLIDWORKS with the updated Service Pack.



 SolidProfessor LMS	SolidProfessor	12/15/2010	3.50 MB	10.9.15.1
 SolidWorks 2010 x64 Edition SP04	SolidWorks Corporation	8/23/2010		18.4.0.5
 SolidWorks 2011 x64 Edition SP01	SolidWorks Corporation	12/4/2010		19.1.0.6
 SonicWALL Global VPN Client	SonicWALL	7/2/2010	6.24 MB	4.2.6
 Symantec Endpoint Protection	Symantec Corporation	7/2/2010	350 MB	11.0.401
 VirtualCloneDrive	Elaborate Bytes	7/2/2010		

The Dreaded Failed to Save Document Error

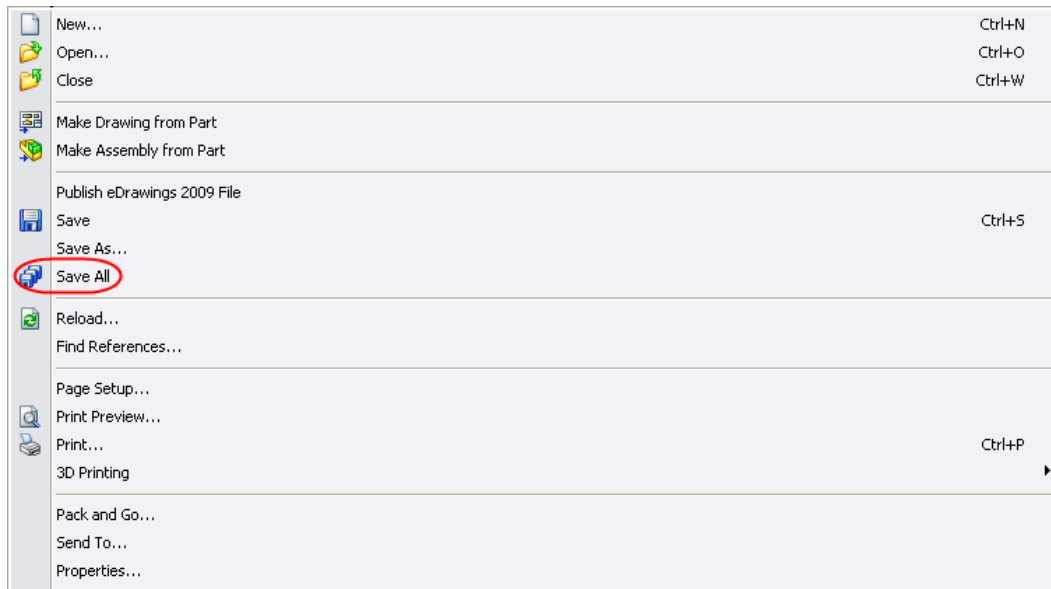
Problem:



The Dreaded Failed to Save Document Error

Solution1:

Does “Save all” work?



The Dreaded Failed to Save Document Error

Solution2:

Try to use Ctrl+Q to perform a forced regeneration.



The Dreaded Failed to Save Document Error

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Solution3:

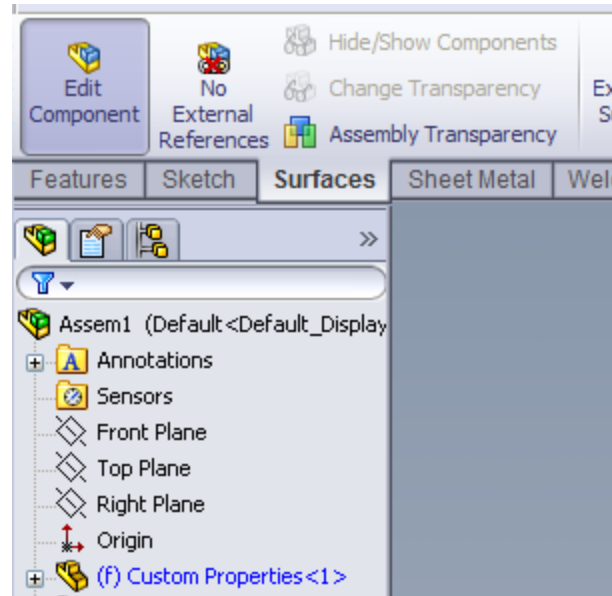
Do you have any configurations?



The Dreaded Failed to Save Document Error

Solution4:

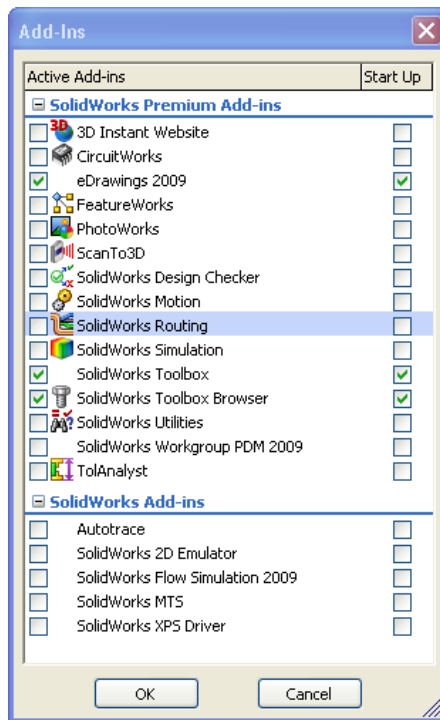
Did you just edit a part?



The Dreaded Failed to Save Document Error

Solution5:

Turn off all Add-ins



The Dreaded Failed to Save Document Error

Solution6:

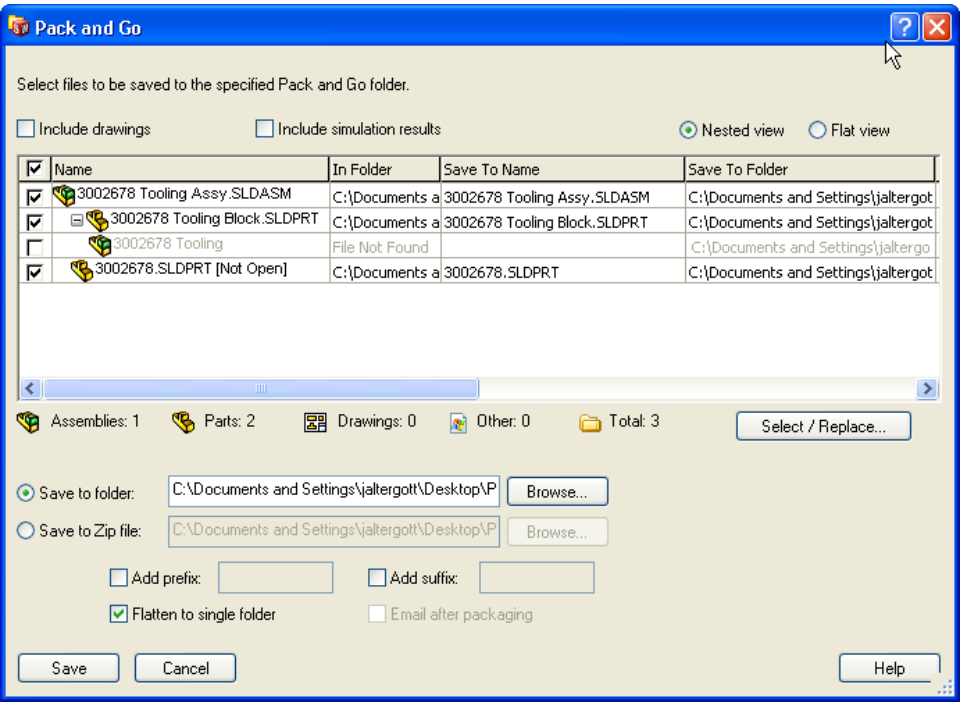
Resave the assembly with a new name
to a new location



The Dreaded Failed to Save Document Error

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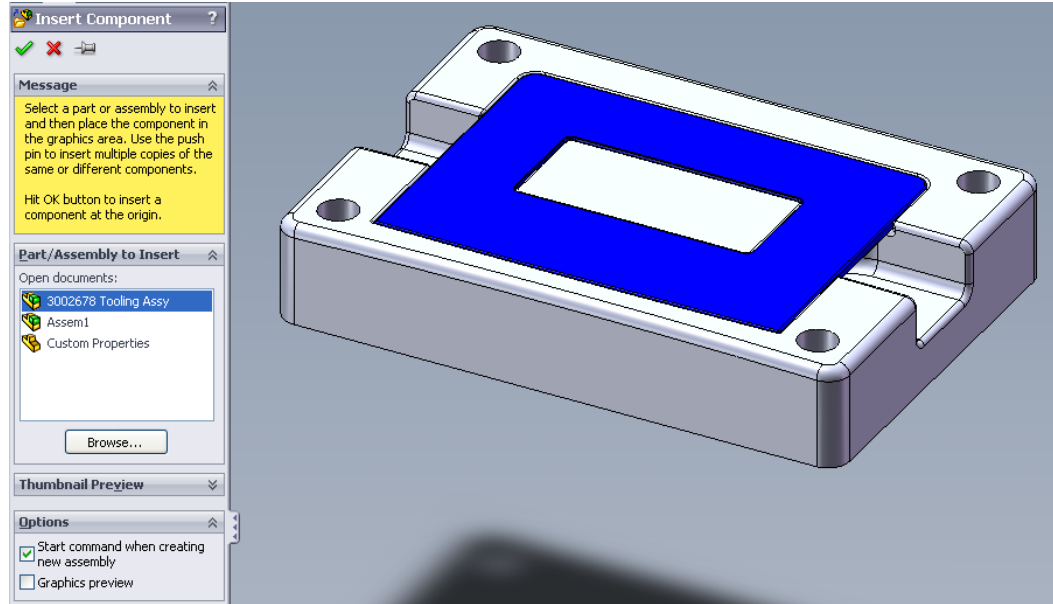
Solution7: Pack and Go



The Dreaded Failed to Save Document Error

Solution8:

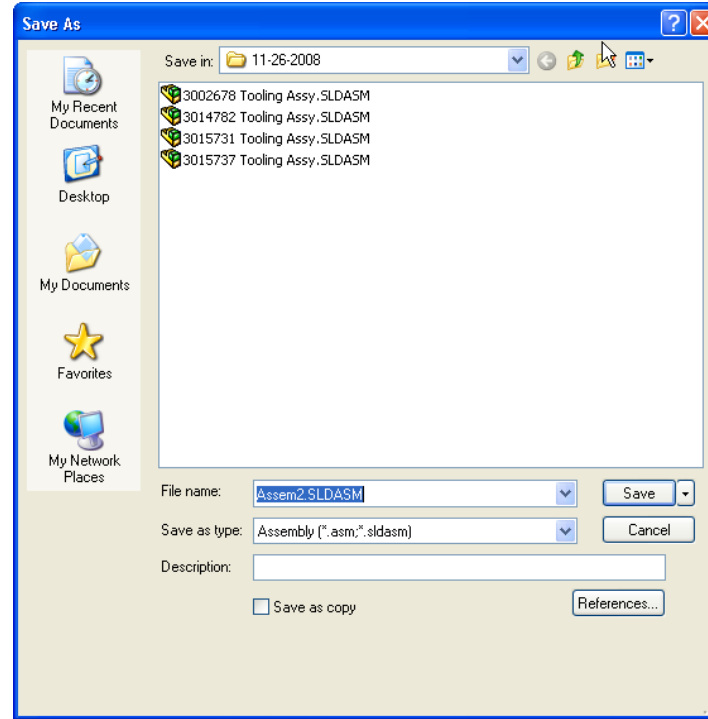
Insert the part or assembly into a new assembly



The Dreaded Failed to Save Document Error

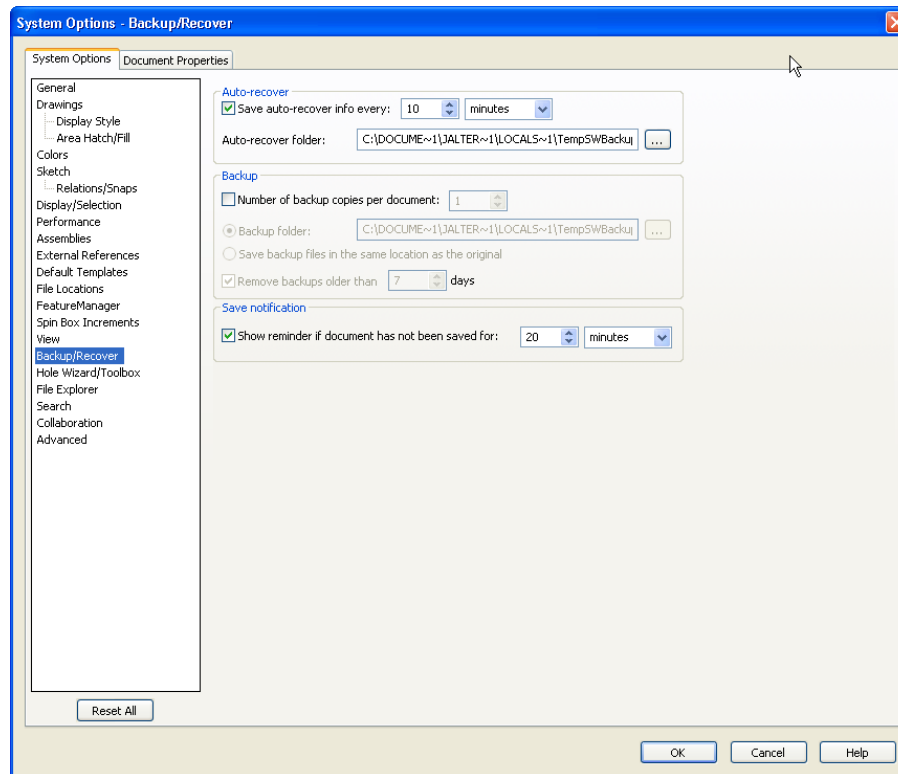
Solution9:

Can you save any of the sub parts?



The Dreaded Failed to Save Document Error

Solution10: SOLIDWORKS Backup?

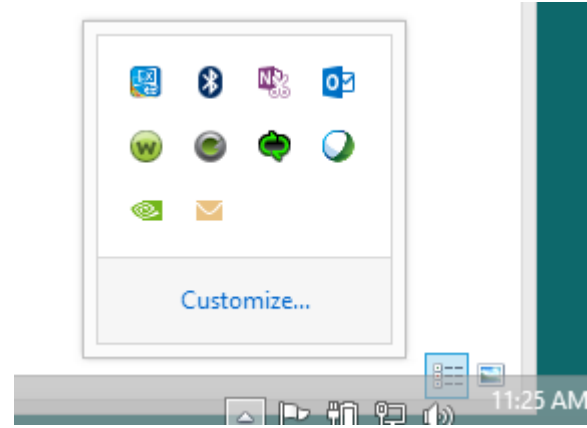
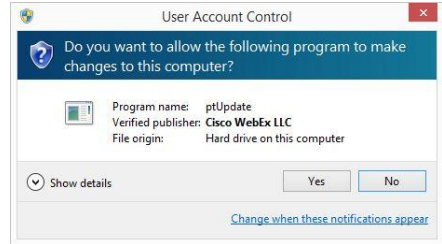
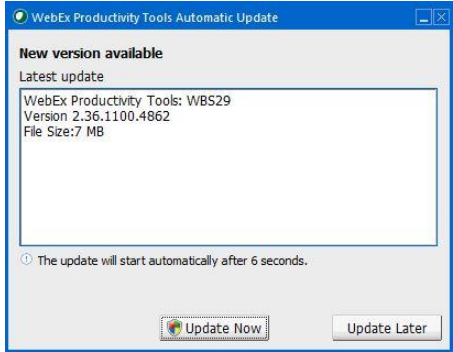


Junk is randomly starting on my Workstation

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Problem:

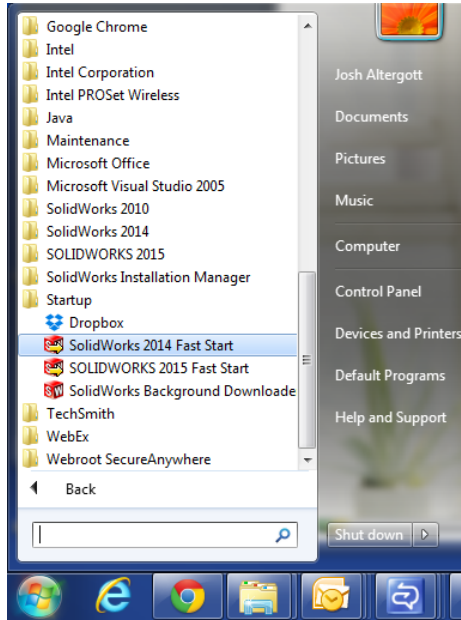
How can I turn this stuff off?



Junk is randomly starting on my Workstation

Solution1:

Startup Folder



C:\Users\

Shortcut for any application WIN7 or WIN8

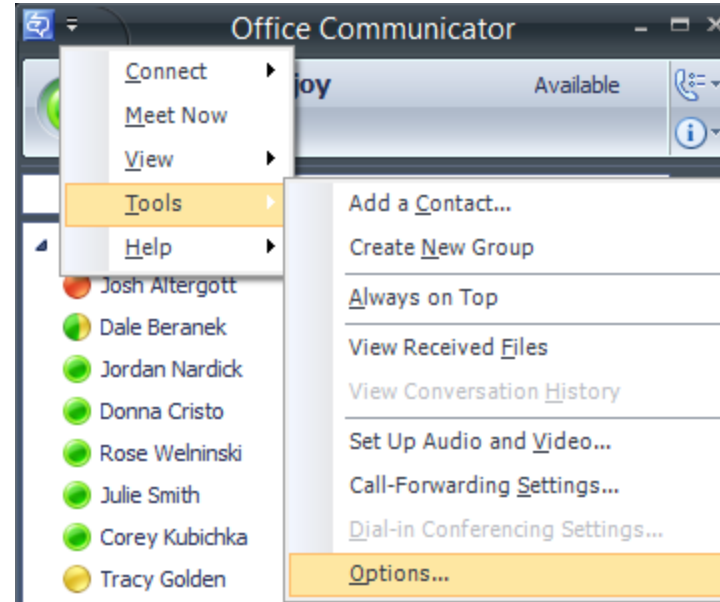
WIN7 Only

Junk is randomly starting on my Workstation

Solution2:

Startup Folder

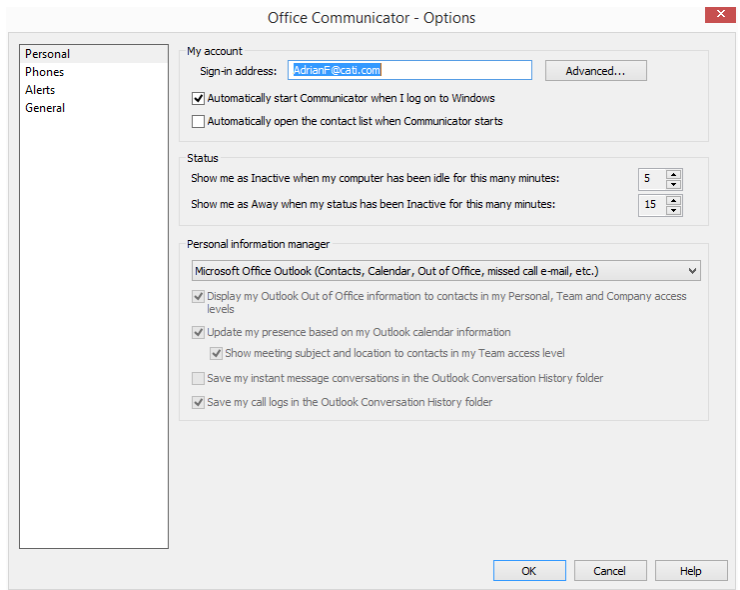
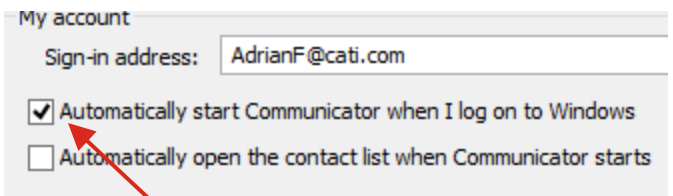
Application



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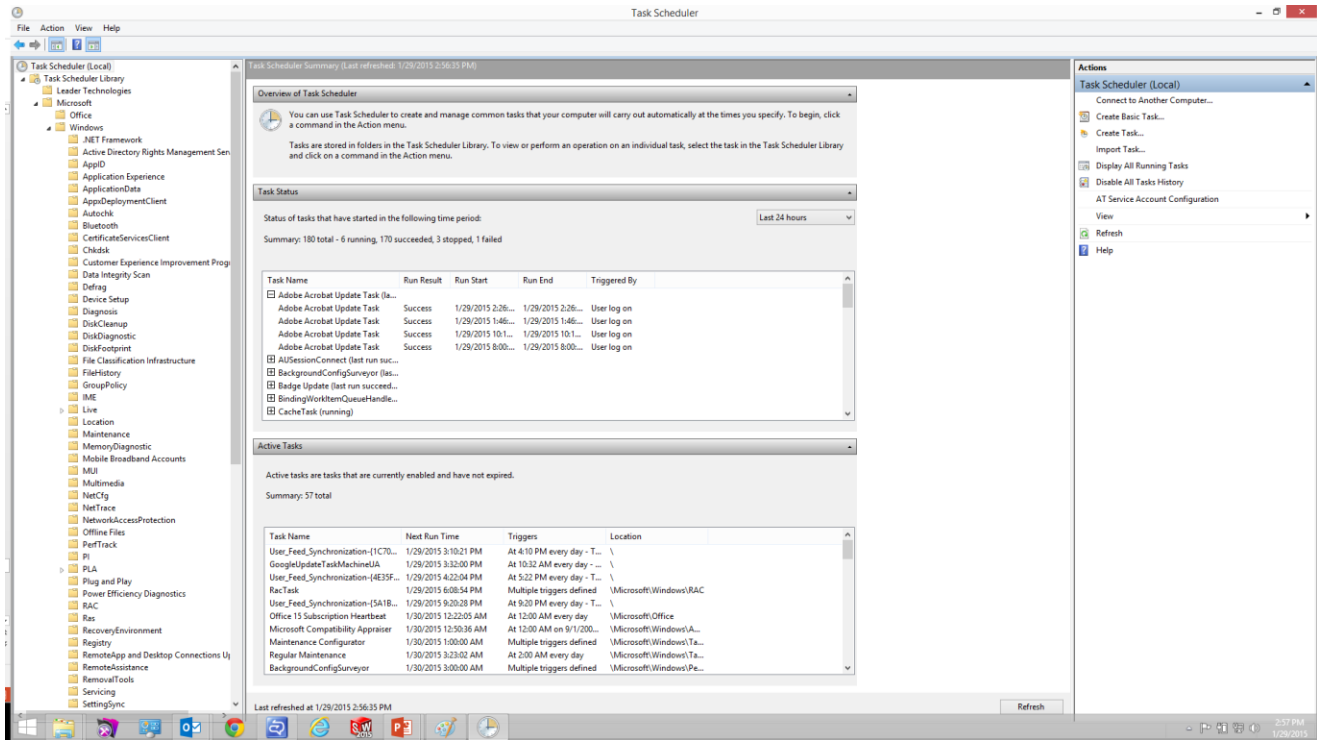
Solution2:
Startup Folder
Application



Junk is randomly starting on my Workstation

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Solution3:
Startup Folder
Application
Task Scheduler



Junk is randomly starting on my Workstation

Solution4:

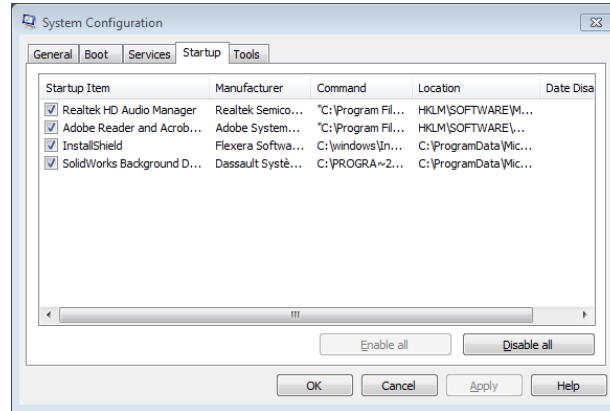
Startup Folder

Application

Task Scheduler

Mscconfig

TaskManager



WIN7

Junk is randomly starting on my Workstation

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Solution4:

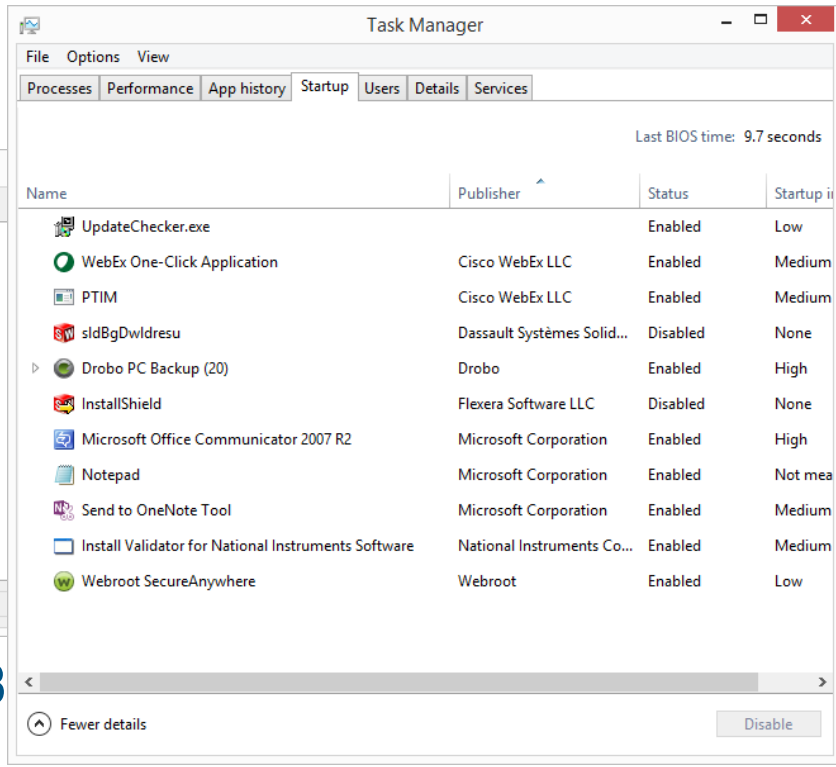
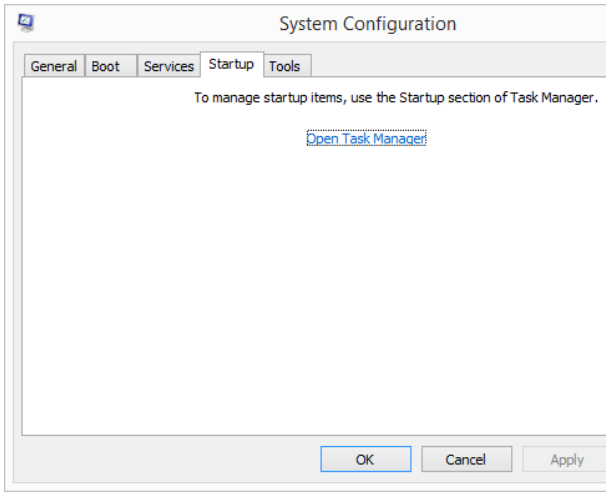
Startup Folder

Application

Task Scheduler

Mscconfig

Task Manager



WIN8

Junk is randomly starting on my Workstation

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Solution5:

Startup Folder

Application

Task Scheduler





Msconfig

Task Manager

Registry

HKLM\SOFTWARE\Microsoft\Windows\CurrentVersion\Run

HKLM\SOFTWARE\Wow6432Node\Microsoft\Windows\CurrentVersion\Run

Name	Type	Data
 (Default)	REG_SZ	(value not set)
 Communicator	REG_SZ	"C:\Program Files (x86)\Microsoft Office Communicator\communicator.exe" /fromrunkey
 InstallValidator.exe.F...	REG_SZ	C:\Program Files (x86)\National Instruments\Shared\NIUninstaller\InstallValidator.exe -s
 WRSVC	REG_SZ	"C:\Program Files (x86)\Webroot\WRSa.exe" -ul

Junk is randomly starting on my Workstation

Solution5:

Startup Folder

Application

Task Scheduler

Msconfig

Task Manager

Registry

HKLM\SOFTWARE\Microsoft\Windows\CurrentVersion\Run

HKLM\SOFTWARE\Wow6432Node\Microsoft\Windows\CurrentVersion\Run

HKCU\SOFTWARE\Microsoft\Windows\CurrentVersion\Run

Name	Type	Data
(Default)	REG_SZ	(value not set)
Drobo PC Backup	REG_SZ	C:\Program Files (x86)\Drobo\Drobo PC Backup\DroboPCBackup.exe AUTOSTART
PTIM.exe	REG_SZ	C:\Program Files (x86)\WebEx\Productivity Tools\PTIM.exe
PTOneClick	REG_SZ	C:\Program Files (x86)\WebEx\Productivity Tools\ptoneclk.exe /AutoRunning="2"

Junk is randomly starting on my Workstation

Solution 666:

Using this for EVIL

HKLM\SOFTWARE\Microsoft\Windows\CurrentVersion\RunOnce

HKLM\SOFTWARE\Wow6432Node\Microsoft\Windows\CurrentVersion\RunOnce

HKCU\SOFTWARE\Microsoft\Windows\CurrentVersion\RunOnce

- You can Auto Launch any program or file that has a default program linked to it.
- It will launch on next login
- Will automatically remove itself and never launch again
- Use your imagination

Problem 3

Problem:

You have a laundry list of crap in the garage.

- PVC
- 2x4s
- Frisbee
- Wagon wheels
- Plastic Chair
- Door hinges
- Misc hardware
- String
- Cordless drill
- Zip ties

BEER

Solution 3

Solution:

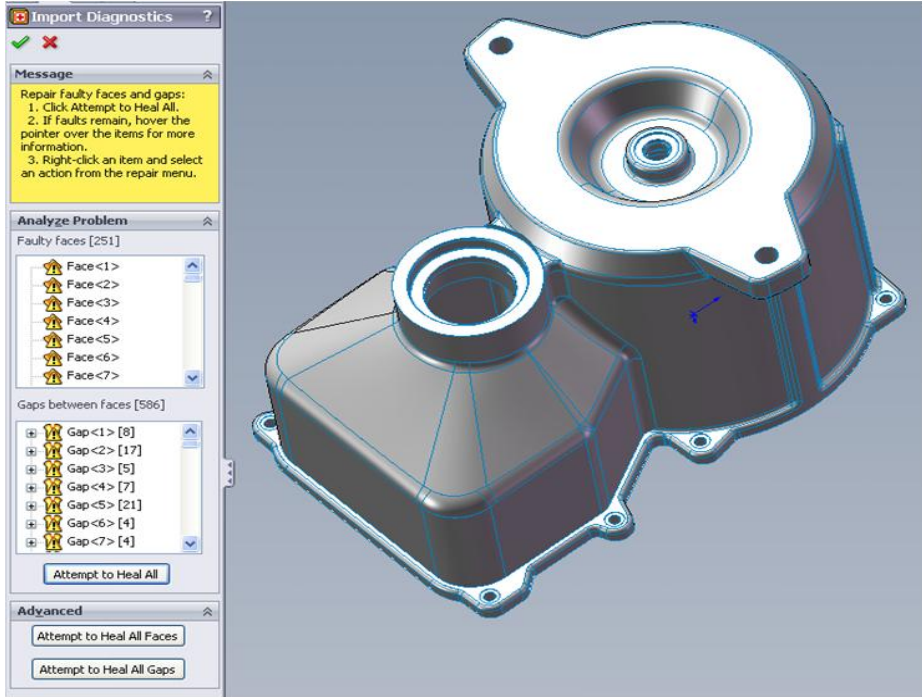


Surfaces, surfaces and more surfaces

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Problem:

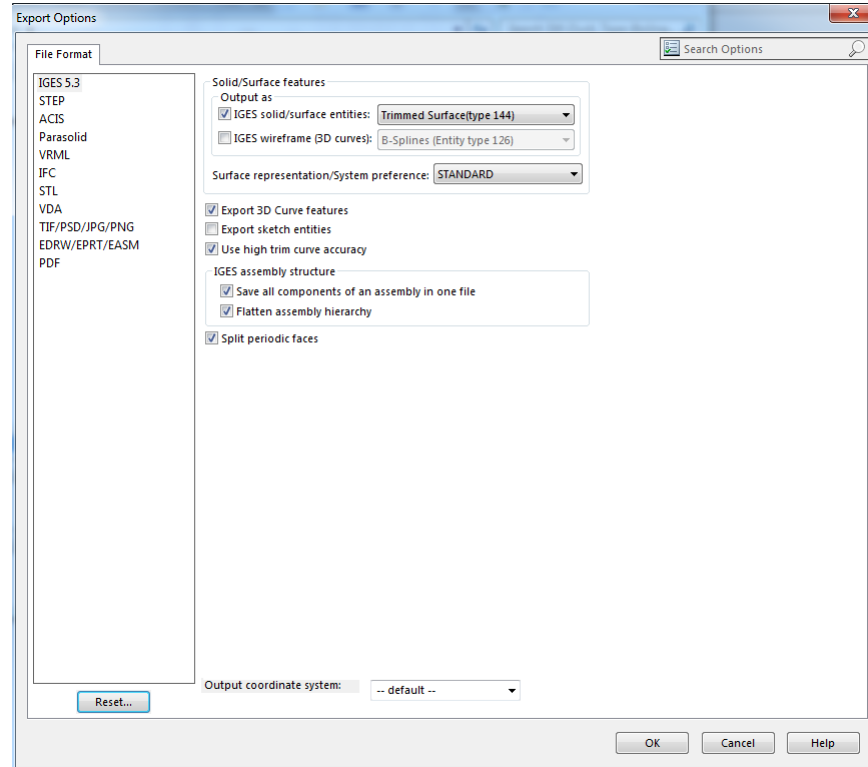
You import a customer's file(s) and they are made up of 100's or 1,000's of surfaces.



Surfaces, surfaces and more surfaces

Solution Step 1:

Export the file as another neutral file format, I have had the best success with .IGES.

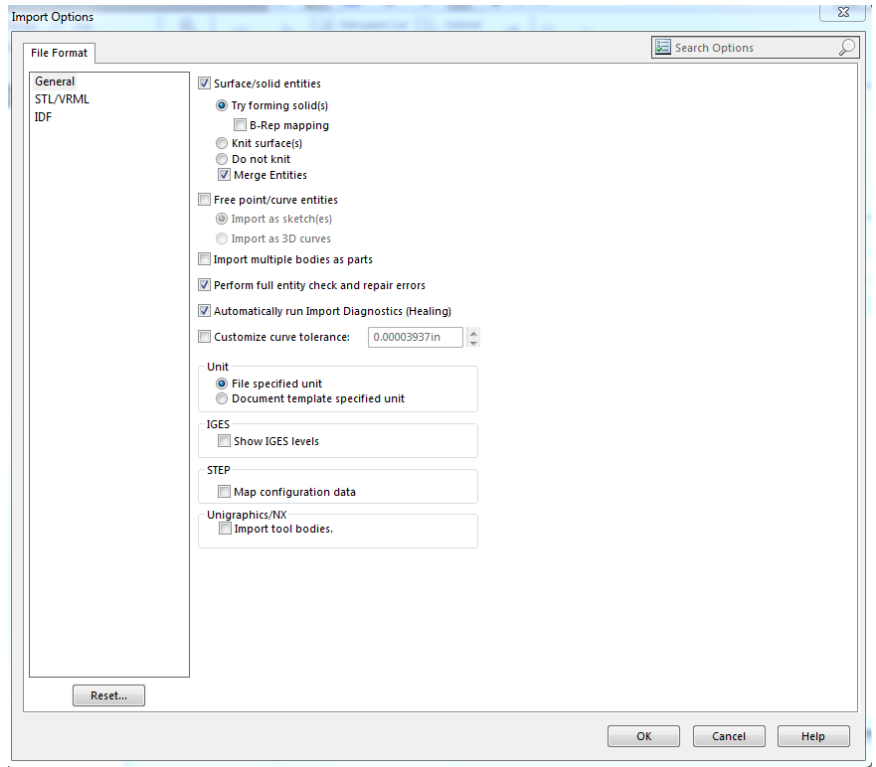


Surfaces, surfaces and more surfaces

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Solution Step 2:

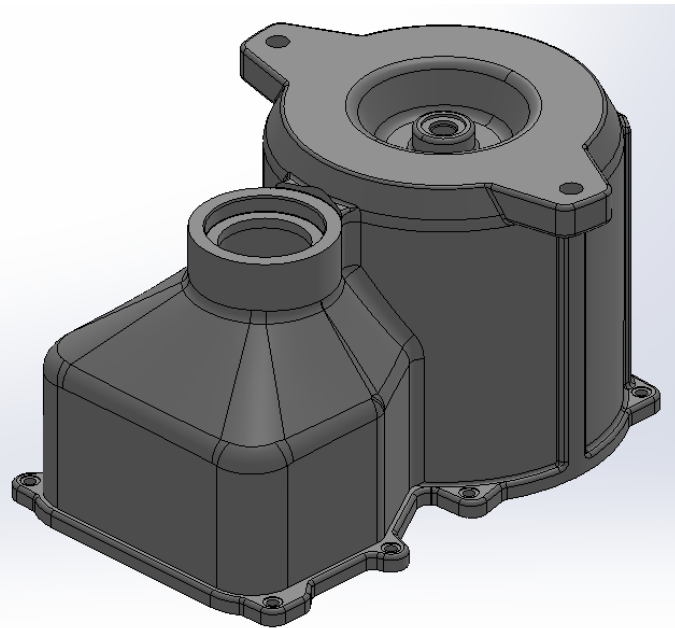
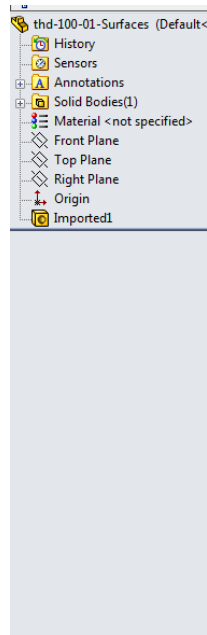
Import the file you just exported, making sure to check the Import Options.



Surfaces, surfaces and more surfaces

Solution Step 3:

The end result is a file that has far fewer surfaces and possibly just one solid body.



Our file had 541 surface and imported as a single solid body with no errors.

SW Options that satisfy “IT”

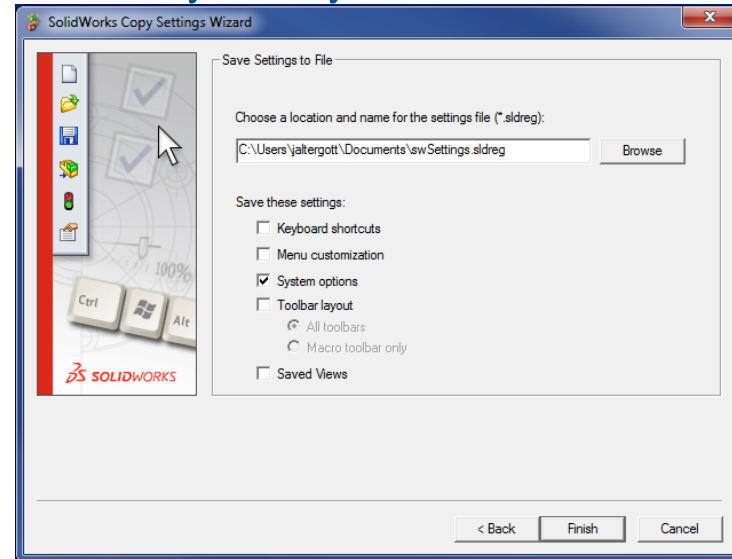
Problem:

Users have the ability to change their System Options at will. Which can cause problems with consistency in modeling standards and practices. Also users may use improper templates, dim standards, etc...

SW Options that satisfy “IT”

Solution Step 1:

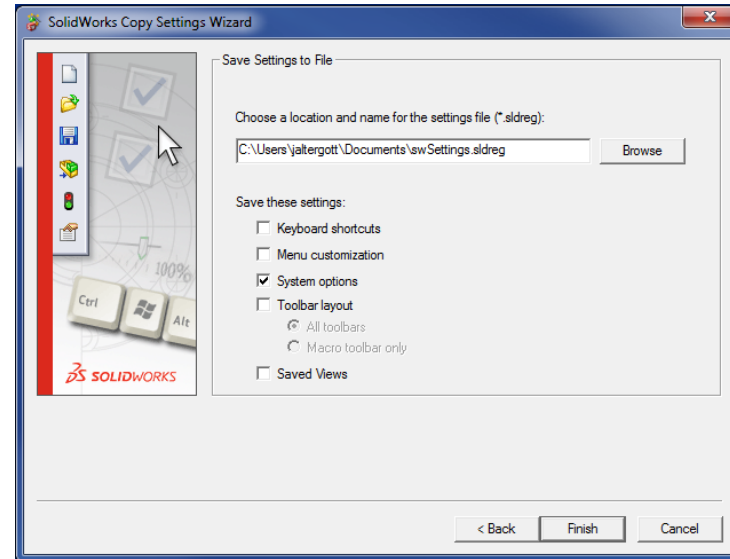
Using the Copy Settings Wizard create a swSettings.sldreg file that has the system options set the proper way. Make sure that you only save the system options.



SW Options that satisfy “IT”

Solution Step 2:

Change the file name to swSettings.reg and place it in a common location where users can read but not write.



SW Options that satisfy “IT”

Solution Step 3:

Deploy the new file in one of two ways:

Option1: Add the following line to the Login in script of your SOLIDWORKS users:

```
Regedit /S "\\<server name>\<share name>\<Path>\swSettings.reg"
```

Option2: Replace the users SOLIDWORKS launch icons with a batch file that has these commands in it:

```
Regedit /S "\\<server name>\<share name>\<Path>\swSettings.reg"
```

```
"C:\<install location>\SolidWorks\swspmanager.exe"
```

How big are my parts

Problem:

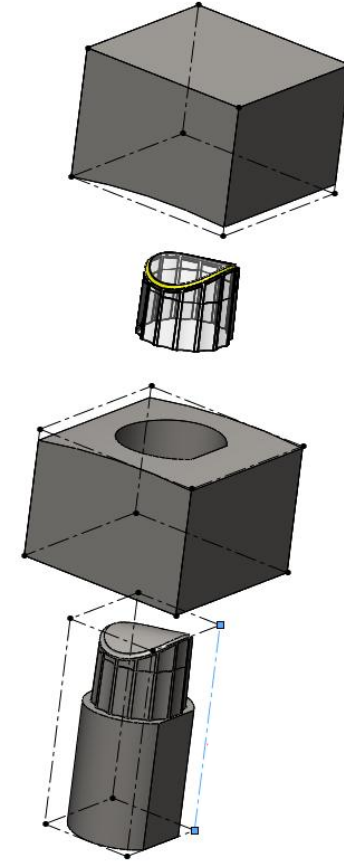
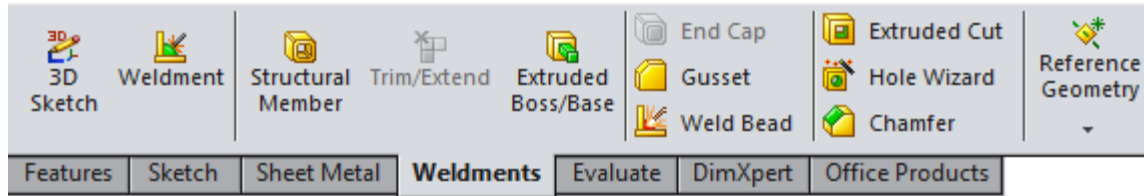
You have a mold or other multi-body part and want to know the sizes for purchasing and manufacturing.



How big are my parts

Solution Step 1:

- Make your part into a Weldment.

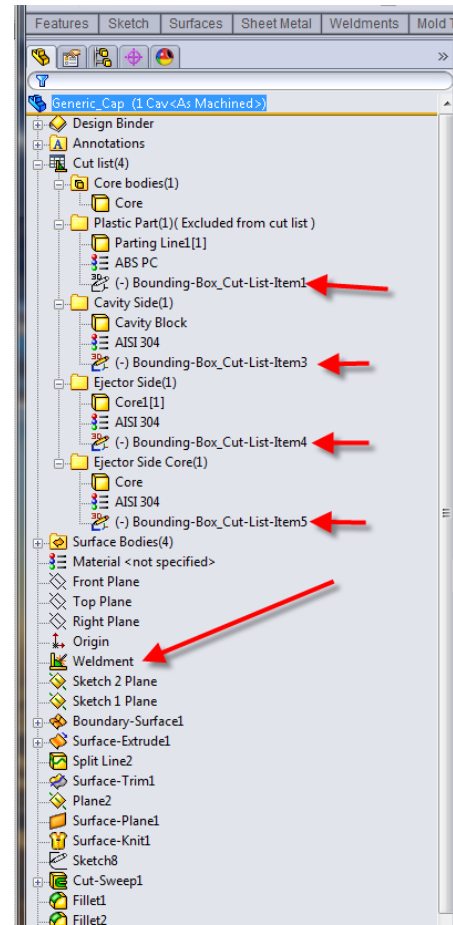
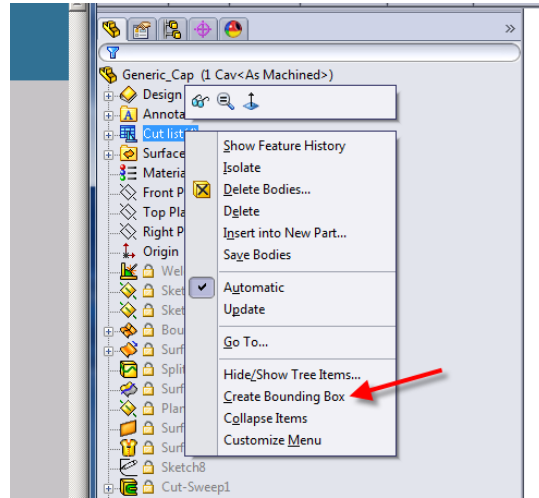


How big are my parts

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Solution Step 2:

Right Click on each Cut List item and use Create Bounding Box.



How big are my parts

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Solution Step 3:

The Properties of each item show their overall size as well as volume and material.

Properties Summary | Cut List Table

Delete Exclude from cut list BOM quantity: None Edit List

Property Name	Type	Value / Text Expression	Evaluated Value
1 MATERIAL	Text	"SW-Material@@@Cavity Side@Generic_Cap.SLDPRT"	AISI 304
2 DESCRIPTION	Text	AISI 304, SW-Thickness x SW-Width x SW-Length	AISI 304, 4.358 x 5.75 x 6
3 3D-Bounding Box Thickness	Text	"SW-3D-Bounding Box Thickness@@@Cavity Side@Generic_Cap.SLDPRT"	4.358
4 3D-Bounding Box Width	Text	"SW-3D-Bounding Box Width@@@Cavity Side@Generic_Cap.SLDPRT"	5.75
5 3D-Bounding Box Length	Text	"SW-3D-Bounding Box Length@@@Cavity Side@Generic_Cap.SLDPRT"	6
6 3D-Bounding Box Volume	Text	"SW-3D-Bounding Box Volume@@@Cavity Side@Generic_Cap.SLDPRT"	150.335
7 <Type a new property>			

OK Cancel Help

Problem 4

Problem:

You need to move out of a third floor apartment.

You're fairly lazy.

Fairly resourceful.

You really love your sofa.

Solution 4

Solution:

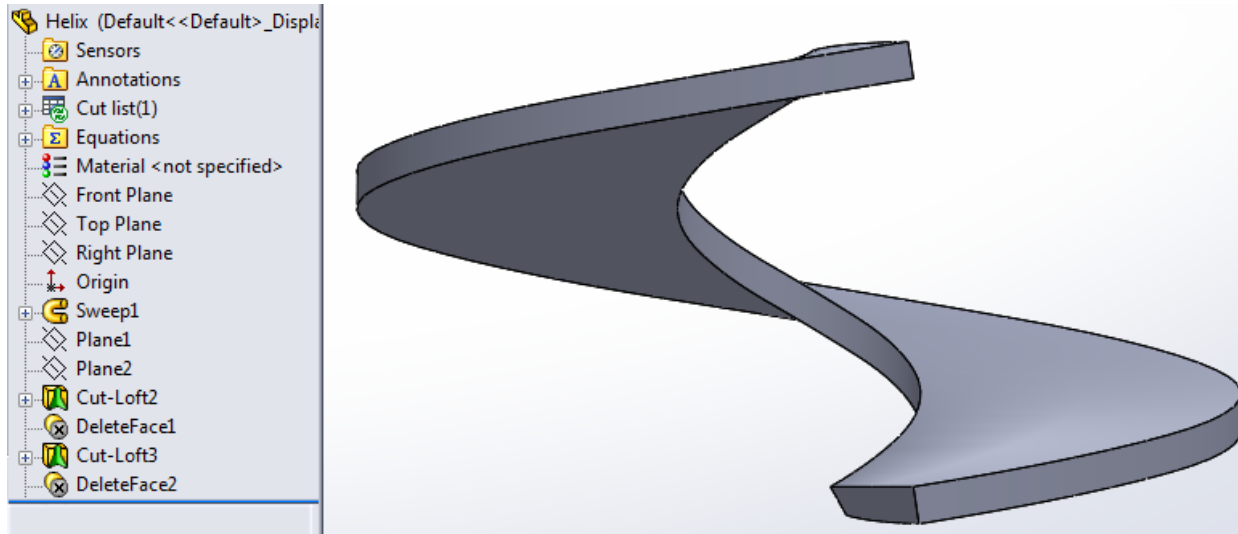
3 long ropes and a couple of stout tie downs.



Flattening Complex Sheet Metal Parts

Problem:

You have a complex sheet metal part that you need a flat pattern of for manufacturing.

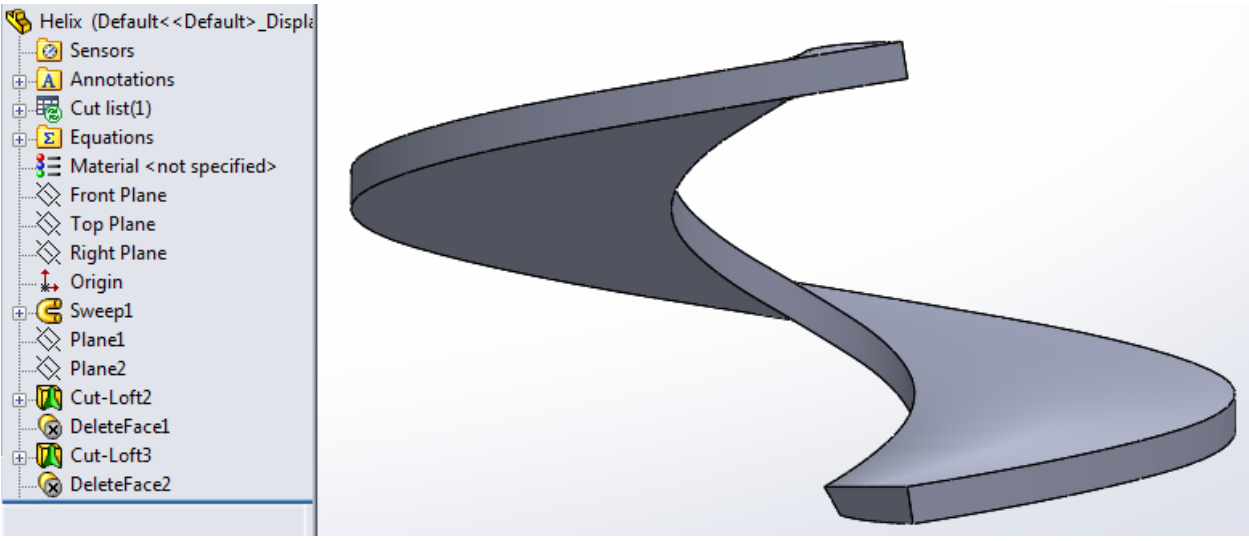


Flattening Complex Sheet Metal Parts

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Solution Step1:

Use a sweep, loft, or other features to create the end result shape you are looking for.



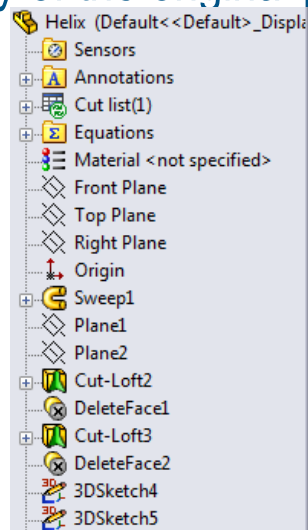
Flattening Complex Sheet Metal Parts

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Solution Step2:

Create a 3D Sketch of each profile.

Delete the body of the original part.

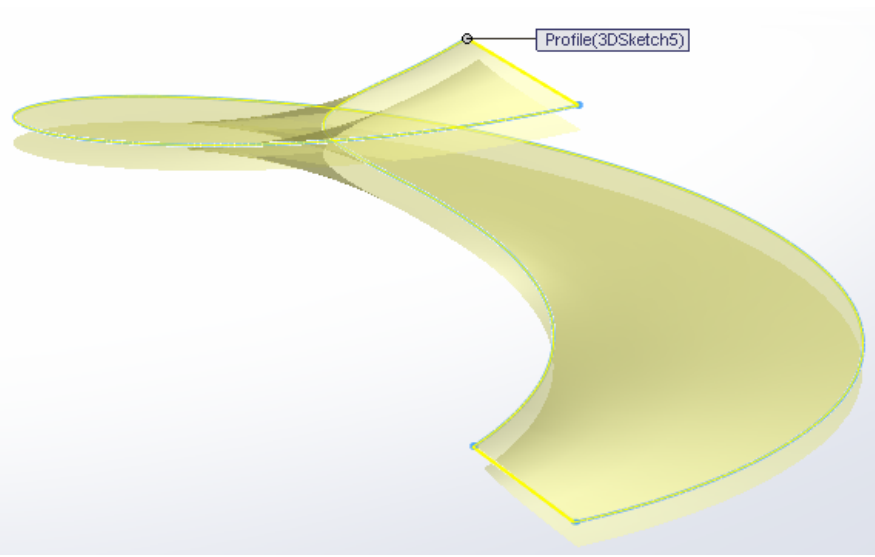
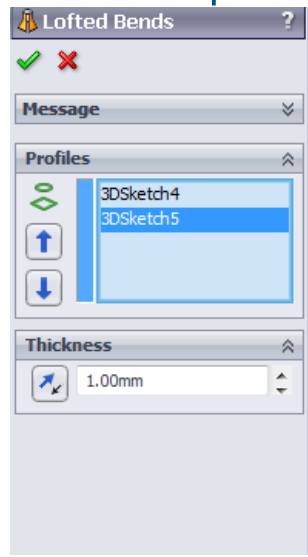


Flattening Complex Sheet Metal Parts

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Solution Step3:

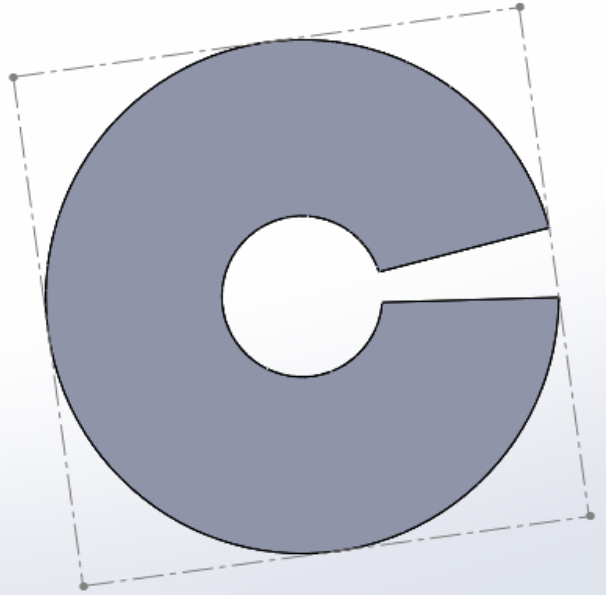
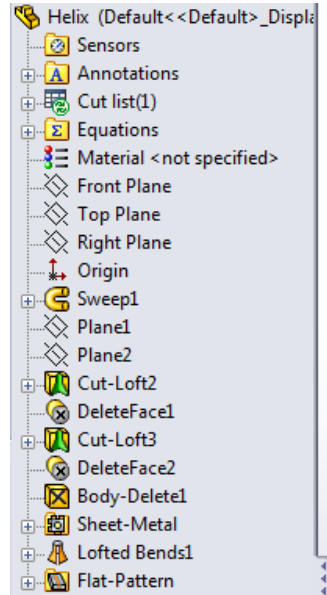
Use Lofted Bends and the new 3D Sketches to create the sheet metal part.



Flattening Complex Sheet Metal Parts

Solution Step4:

Test your new part to make sure it flattens properly.



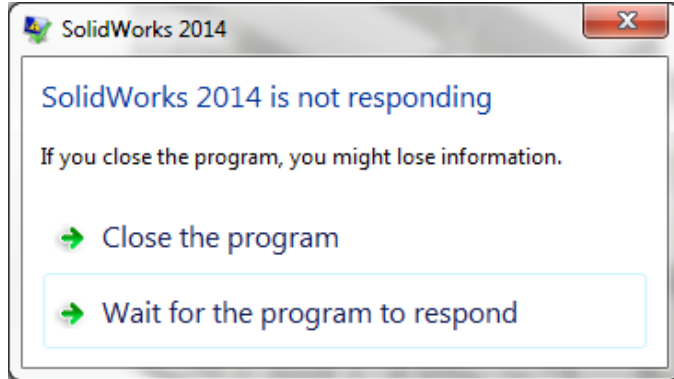
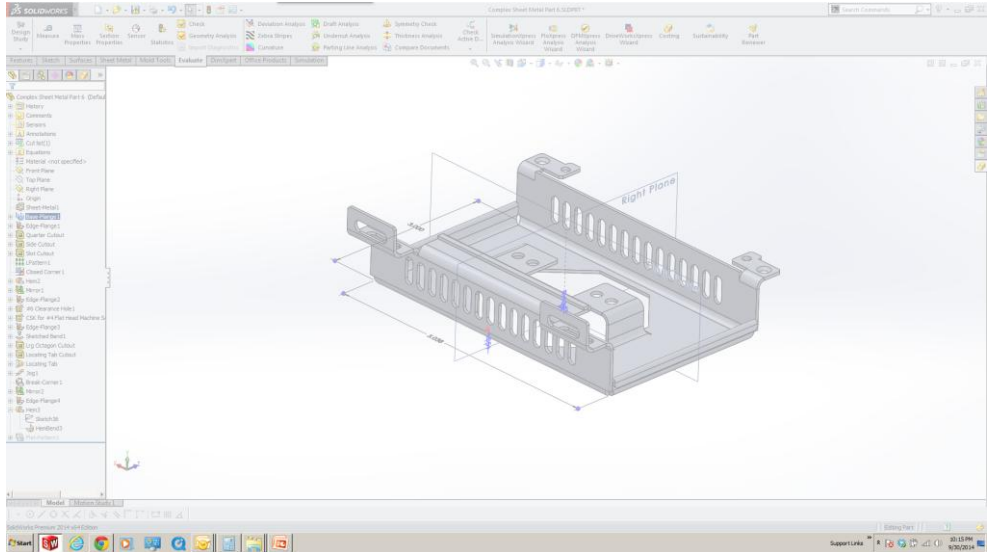
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SOLIDWORKS White Out

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Problem:

You are opening, rebuilding or doing any process that takes an extended period of time to complete and you get a White Out screen.

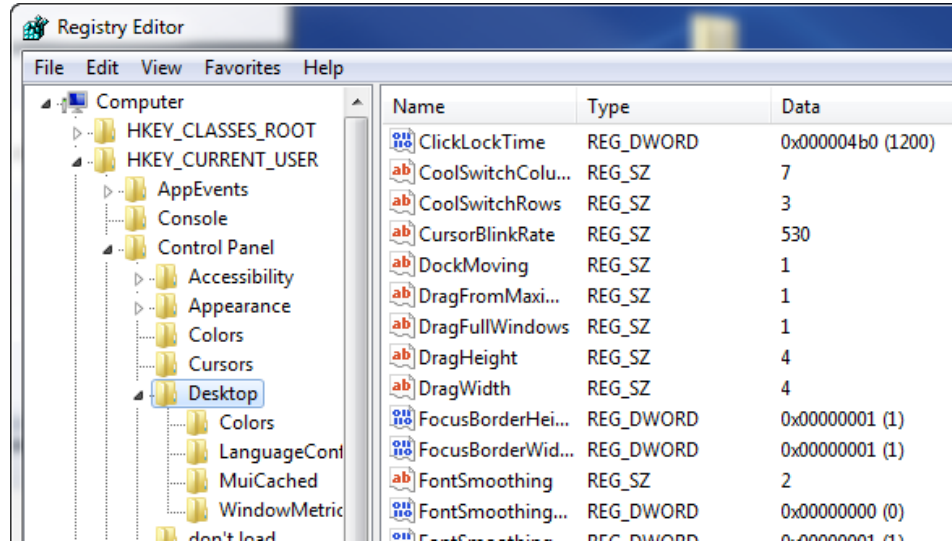
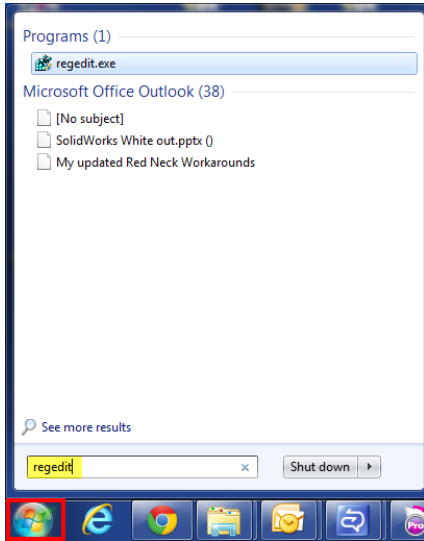


SOLIDWORKS White Out

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Solution Step 1:

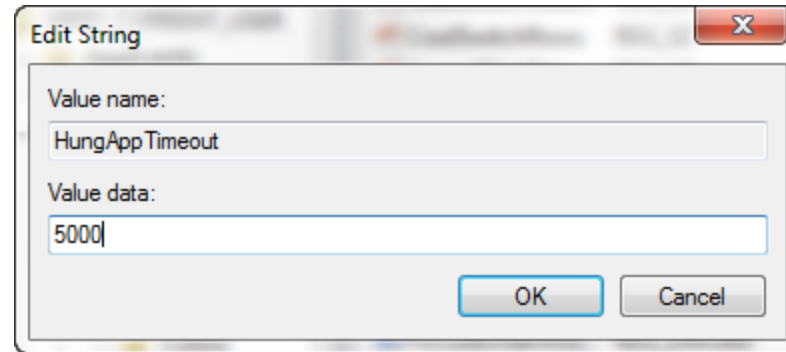
- Start the Windows registry editor and navigate to HKEY_CURRENT_USER\Control Panel\Desktop



SOLIDWORKS White Out

Solution Step 2:

- In the right pane of Desktop, double/click tap on the HungAppTimeout string value to modify it. The default value is **5000** milliseconds (5 seconds).

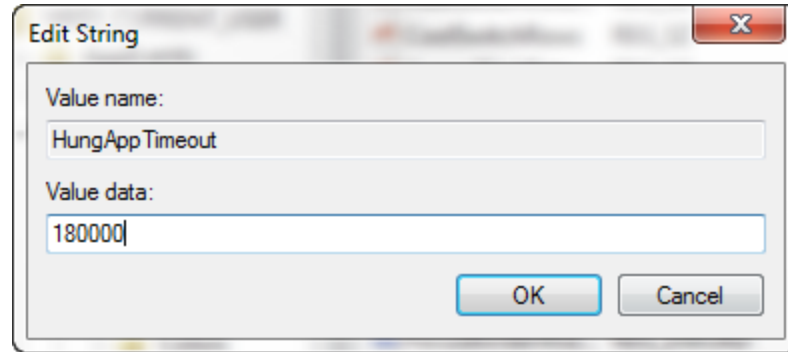


NOTE: If you do not have a **HungAppTimeout** string value, then right click or press and hold on an empty space in the right pane of **Desktop**, click/tap on **New** and **String Value**, type **HungAppTimeout** for the name, and press Enter.

SOLIDWORKS White Out

Solution Step 3:

- Type in a number for how many milliseconds you want to have for the HungAppTimeout value, and click/tap on OK.



NOTE: It is recommended to not set a value below **1000** milliseconds.

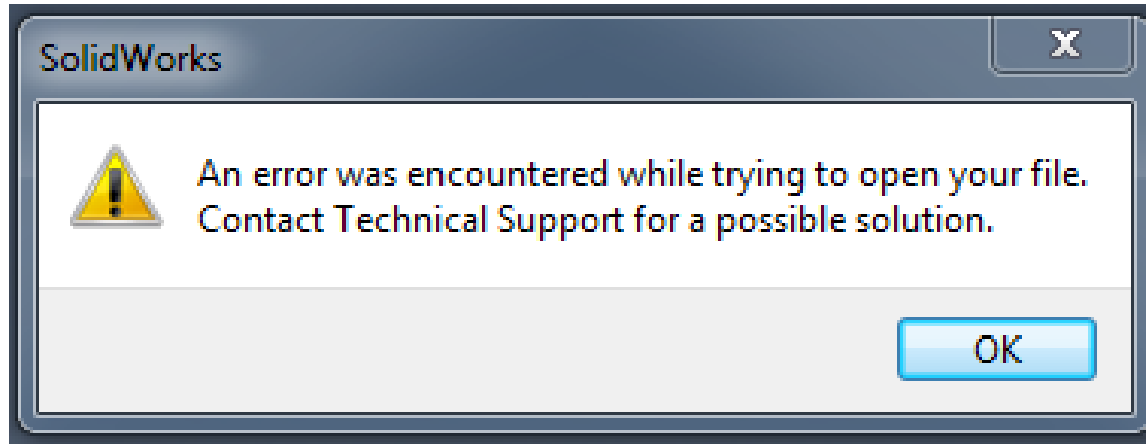
1000 milliseconds = 1 second

180000 milliseconds = 180 seconds (3 minute).

Open Assembly = Fail

Problem:

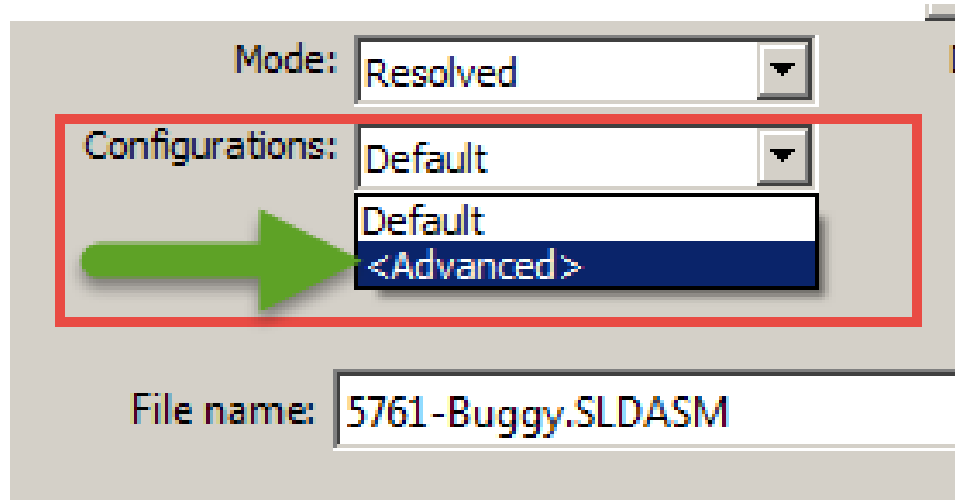
You are trying to open a SOLIDWORKS Assembly file and you get the following message:



Open Assembly = Fail

Solution Step 1:

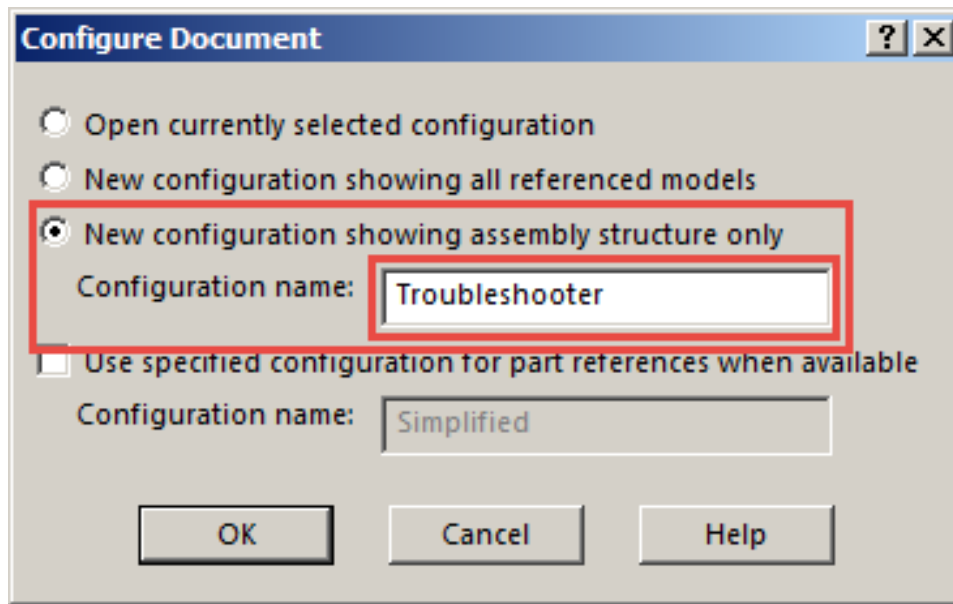
- From the Open Dialog, and the dropdown menu for Configurations select the <Advanced> configuration.



Open Assembly = Fail

Solution Step 2:

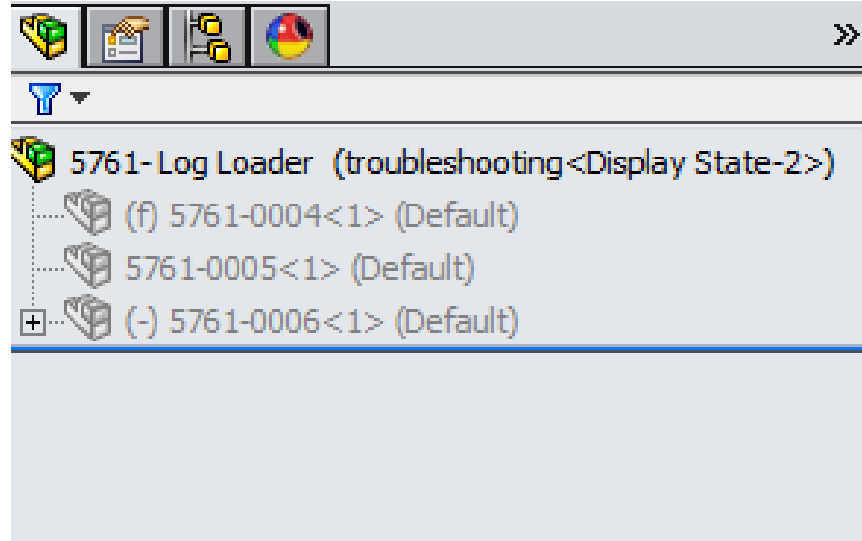
- Select the option that opens the assembly with everything suppressed.



Open Assembly = Fail

Solution Step 3:

- If the assembly opens then it is not corrupt and you can unsuppress the parts and assemblies until we find the bad file.

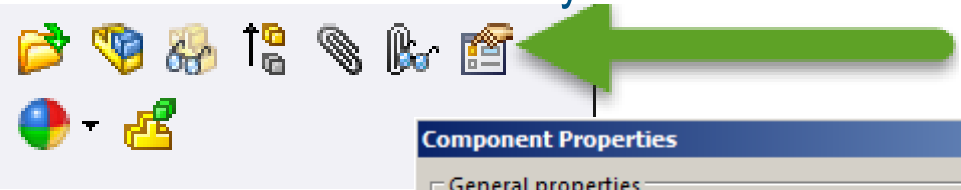


Open Assembly = Fail

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Solution Step 4:

- Use component properties to find the location of the bad file and open that file on its own and troubleshoot any errors.



Component Properties [X]

General properties

Component Name: Instance Id: Full Name:

Component Reference:

Spool Reference:

Component Description:

Model Document Path:

(Please use File/Replace command to replace model of the component(s))

CATI and InFlow Presentation Schedule

Monday February, 9th

- **Discovering and Understanding SOLIDWORKS Dismissed Messages** - 1:30pm-2:30pm - Room TBD, Bryan Pawlak and Phil Whitaker
- **SOLIDWORKS Composer: As Easy as Building with Legos** - 1:30pm-2:30pm – Room North-225B, Brian Reel
- **Achieving Extreme SOLIDWORKS Performance: Hardware/Configuration** - 2:45-3:45pm - Room North-121A-C, Adrian Fanjoy and Josh Altergott
- **Configuring SOLIDWORKS Toolbox** - 2:45-3:45pm - Room North-124A/B, John Van Engen
- **Successfully Prepare for Your EPDM File/Data Migration** - 2:45-3:45pm - Room North-120D, Jeff Barker

Tuesday February, 10th

- **SOLIDWORKS Duct Tape & Bailing Wire: Creative Solutions to Everyday Problems-** 1:30-2:30pm - Room North 128A/B, Josh Altergott and Adrian Fanjoy
- **Drawings: Setup of Templates and Sheet Formats-** 1:30-2:30pm - Room North-221, Jim Krivoshein
- **EPDM Web: Implementing and Configuring** - 1:30pm-2:30pm - Room TBD, Justin Webster
- **How to Build and Maintain Effective Design Tables** - 2:45-3:45pm - Room North-226B, Josh Altergott
- **DriveWorksXpress Essentials Hands-On Lab** – 4:30pm-6:00pm - Room North-227C, Ryan Dally

Wednesday February, 11th

- **SOLIDWORKS Composer: As Easy as Building with Legos** - 1:30pm-2:30pm – Room North-225B, Brian Reel
- **Achieving Extreme SOLIDWORKS Performance: Modeling Methodology** - 2:45-3:45pm - Room North-120D, Josh Altergott, and Adrian Fanjoy
- **Sharing Assemblies—Without Giving It All Away-** 2:45-3:45pm - Room North-221, Jim Krivoshein

For a copy of all CATI and InFlow presentations please go to <http://www.cati.com/sww2015>

THANK YOU

http://blog.cati.com/redneck_workarounds/

<http://www.cati.com/>

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