Inventor ~~Jeopardy~~ Jeffardy:

Things I think I should know, but don’t, and am too embarrassed to ask

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Class summary

- Attend this session for a new way to learn and possibly win a prize.
- We will be running a game of Inventor Jeopardy, Where the points in fact do matter.
- Ever wonder why something you have done before no longer works? Ever try to get something done easily but don't know where to click to make it happen? How about a simple work flow that should happen in two clicks when you are doing it in 20?
- Here is your chance to learn in a competitive environment and have a great time doing it. This will be more than a clicks and tips session, it will be a knowledge war where we all win something.

Key learning objectives

At the end of this class, you will be able to:

- Leave with at least 5 buttons you have never thought about using!
- Realize some of your work flows can be simplified by choosing the correct tool or hidden button.
- Show off your new found hidden stuff to those that did not come to AU.
- Comprehend the value of paying attention to the dialog boxes, they are not wallpaper

Topic 1	Topic 2	Topic 3	Topic 4	Topic 5	Topic 6
10	10	10	10	10	10
20	20	20	20	20	20
30	30	30	30	30	30
40	40	40	40	40	40
50	50	50	50	50	50
60	60	60	60	60	60
70	70	70	70	70	70
80	80	80	80	80	80
90	90	90	90	90	90
100	100	100	100	100	100

I have two sheet metal vendors. One vendor's results always match my designs. The other's folds are always off by a bit.

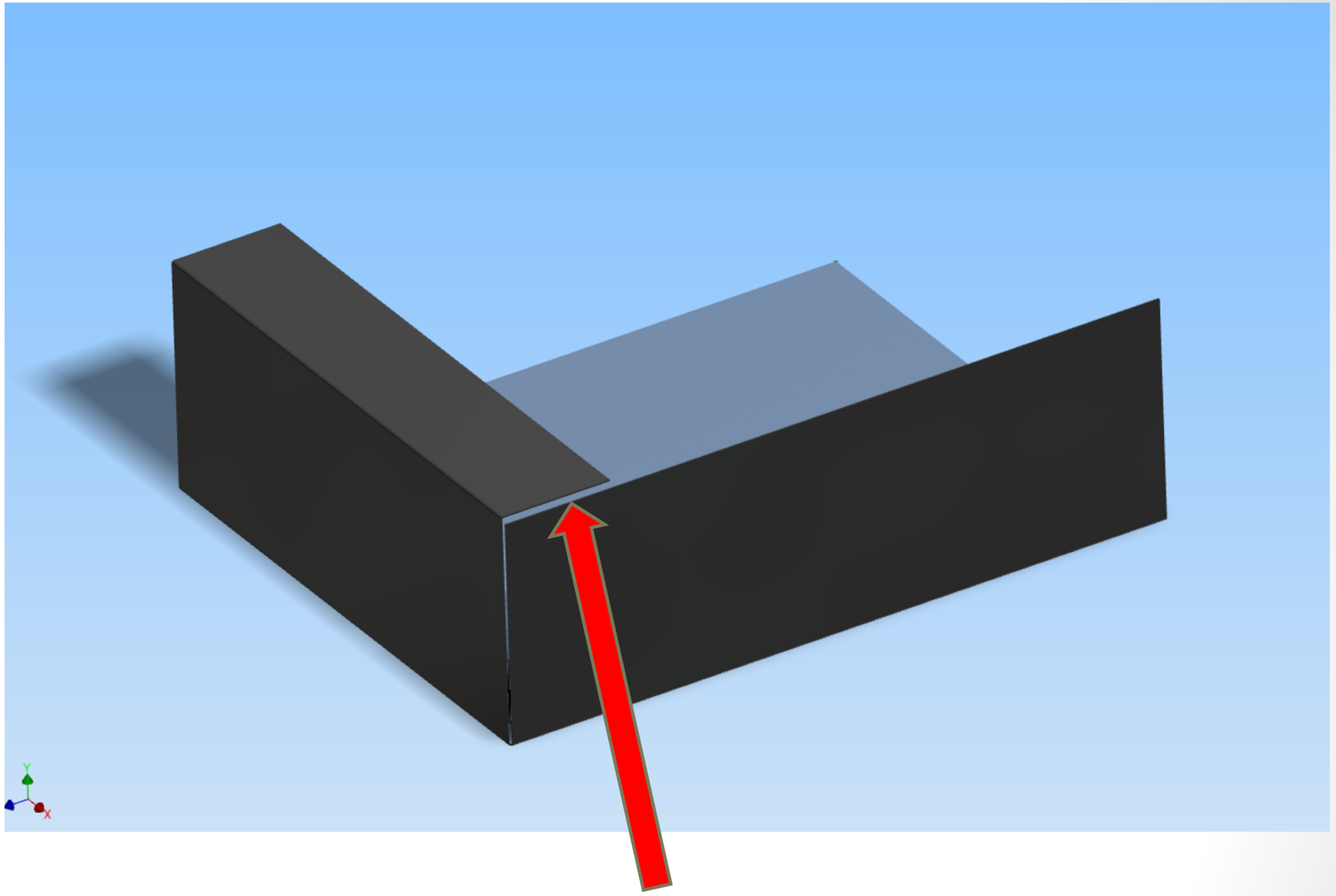
What should I do?

A: The vendor is messed up, find a different one.

B: Create a new unfold rule for the wayward vendor.

C: This is always a problem with sheet metal, work around it.

D: Go to application options and modify sheet metal settings.

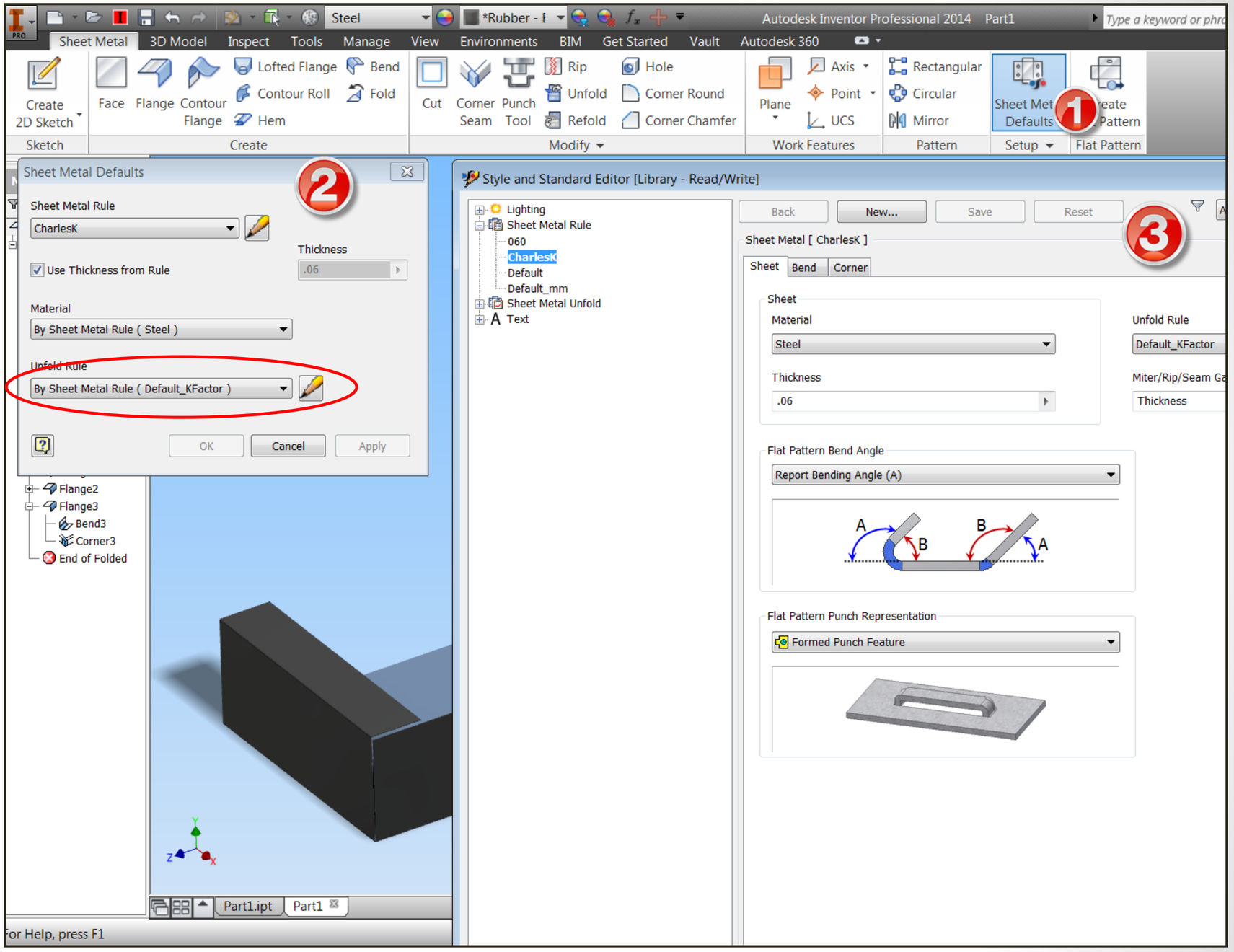


B: Create a new unfold rule for the wayward vendor.

The sheet metal rule is used to create the flat pattern based on the fabricator's tooling. Work with the vendor to create a sheet metal rule that yields what you need.

Go to the sheet metal tab

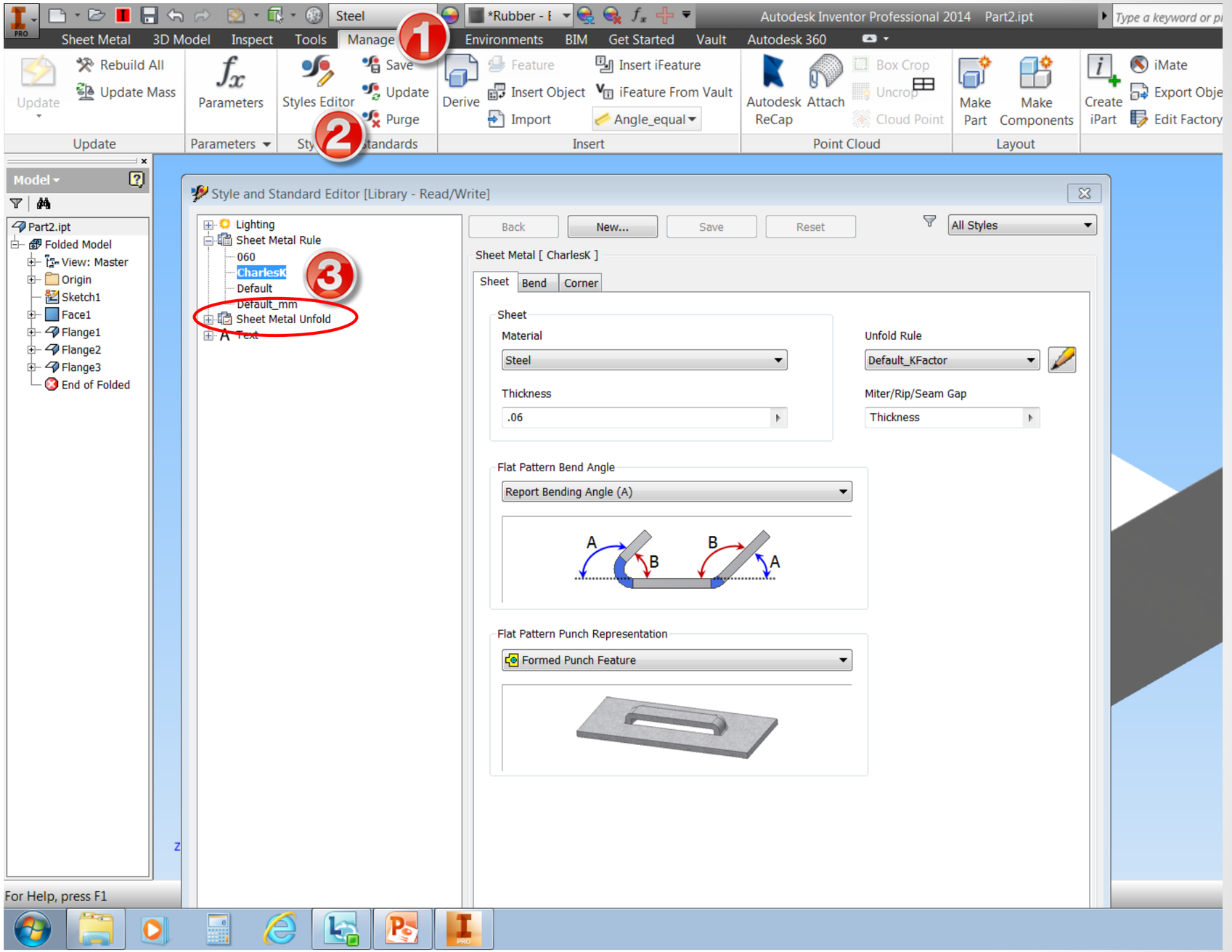
1. Sheet metal defaults
2. Pencil next to rule
3. Create a new rule or modify an existing rule



B: Create a new unfold rule for the wayward vendor.

Another way to get to the sheet metal rules is under the manage tab.

1. Select manage tab
2. Select styles editor
3. Modify or create a new rule



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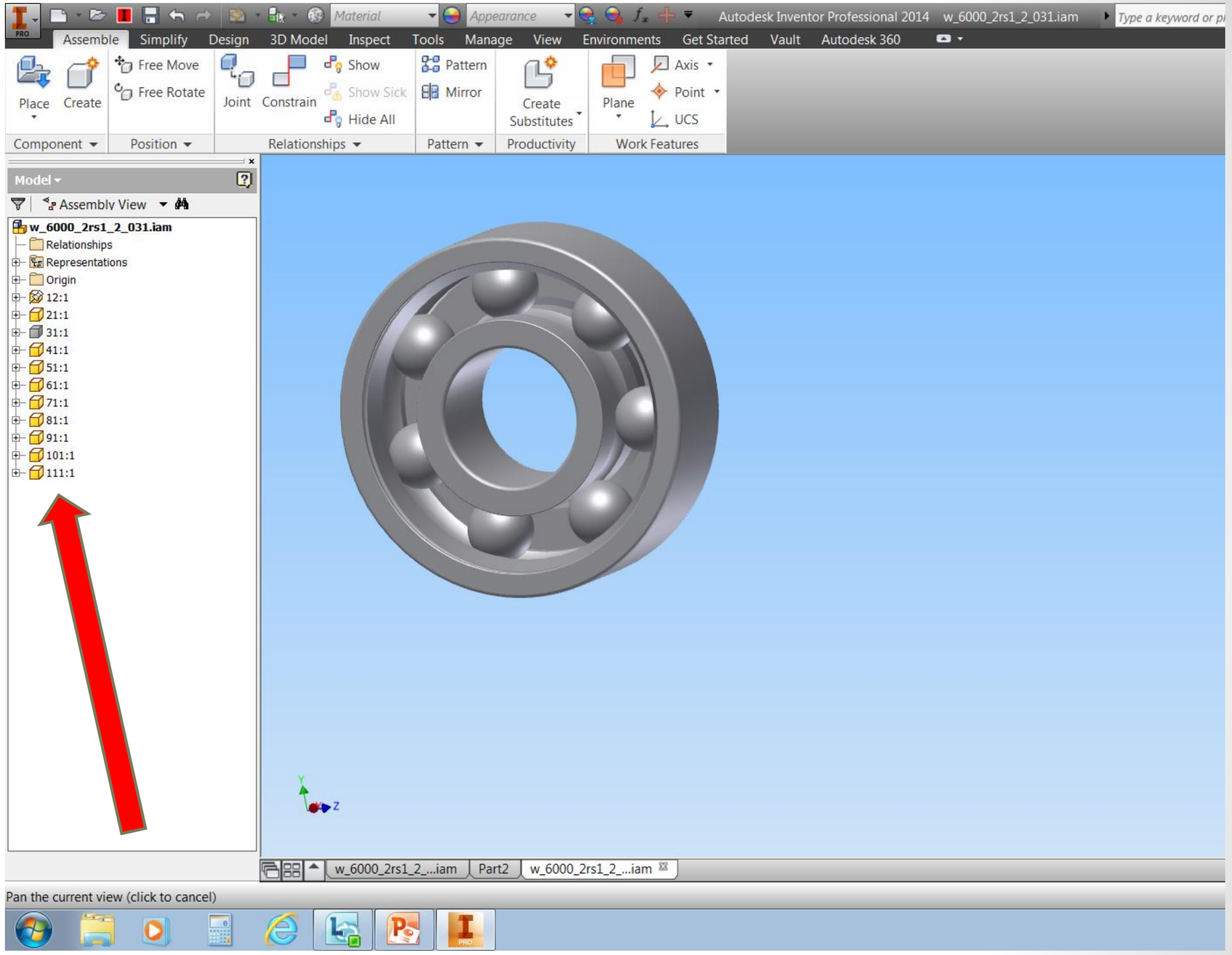
I have an assembly that I downloaded from a vendor. Great getting content, but not so great since the parts are named Part1, Part2, Part3, …
How do I get out of this mess?

A: Delete all the other parts in my workspace with the same names.

B: Rename each part with Design Assistant.

C: Derive a new part from the assembly..

D: Ask the vendor for a better model.



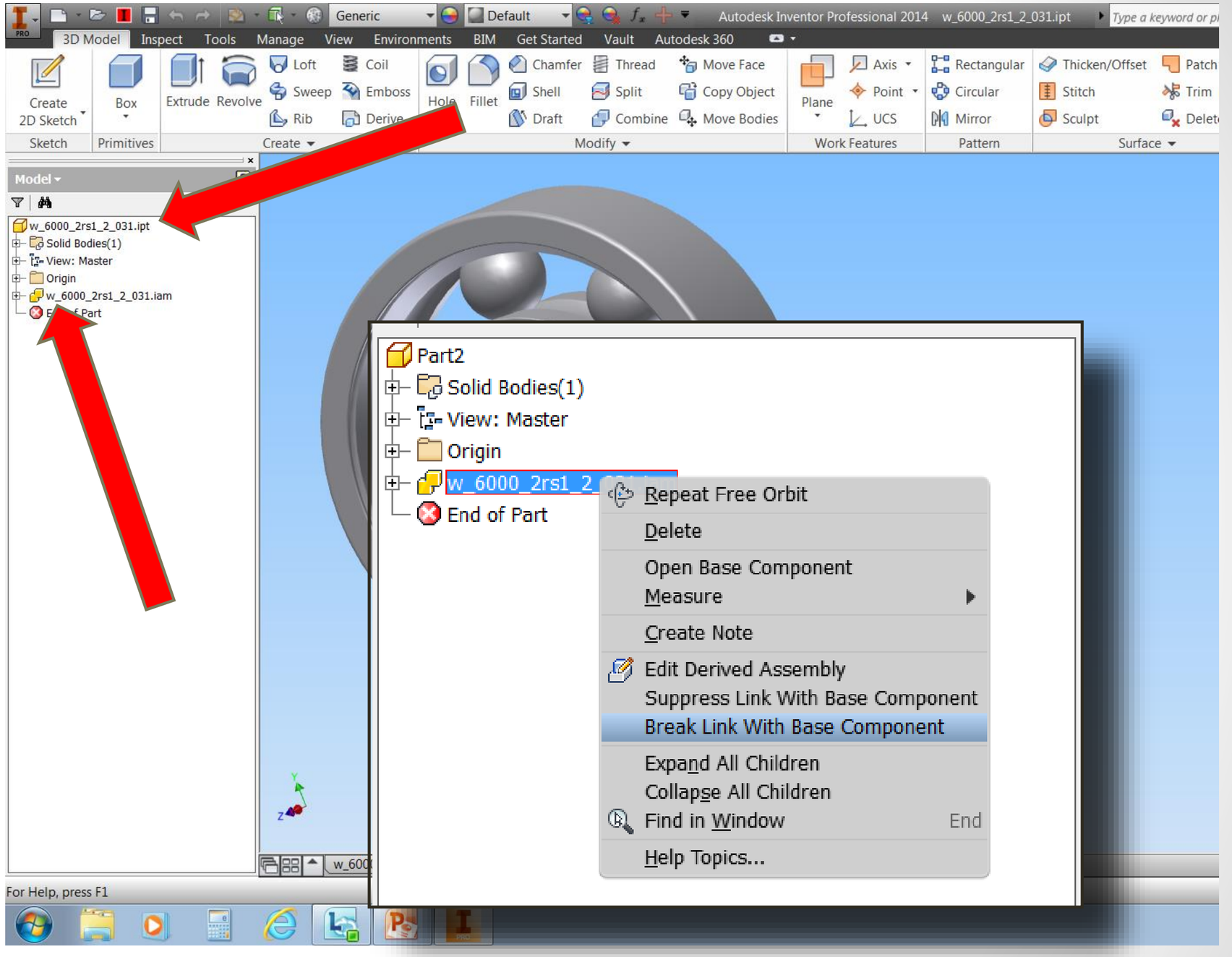
C: Derive a new part from the assembly.

You could have renamed each part in the assembly. This would have kept the individual parts usable as parts.

However if you have downloaded the assembly to use as purchased, you do not need to spend the time “playing” with the offending assembly.

Derive a new part from the assembly. This will have the same shape without naming conflicts.

Then break the link with the base component.



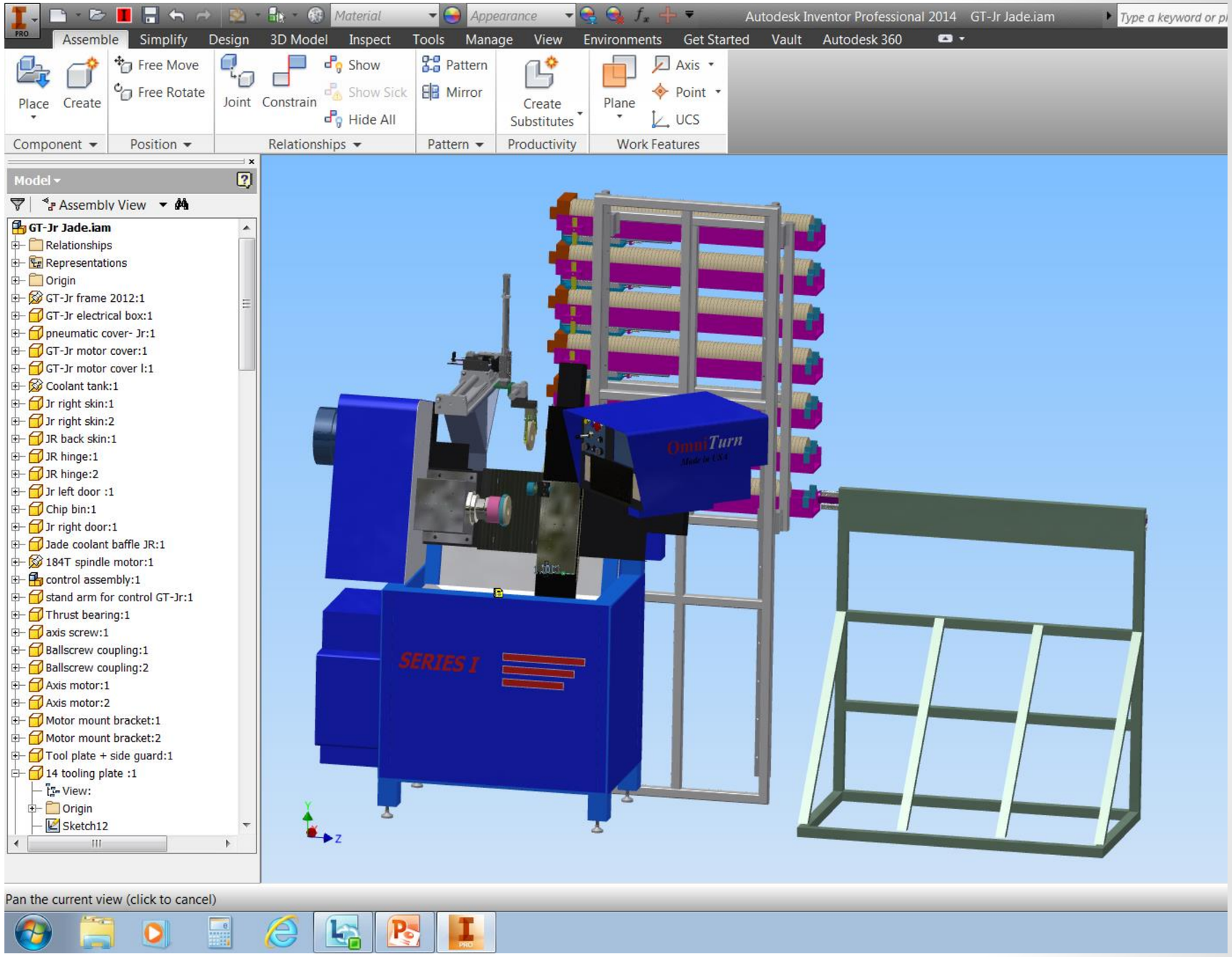
I want to send my customer a model that contains details that he needs, but not to much so that he gets confused or can rip off all my work.
Oh by the way, he also uses dialup. What tool do I use?

A: Derived parts

B: Part suppression and representations

C: Visibility and enable

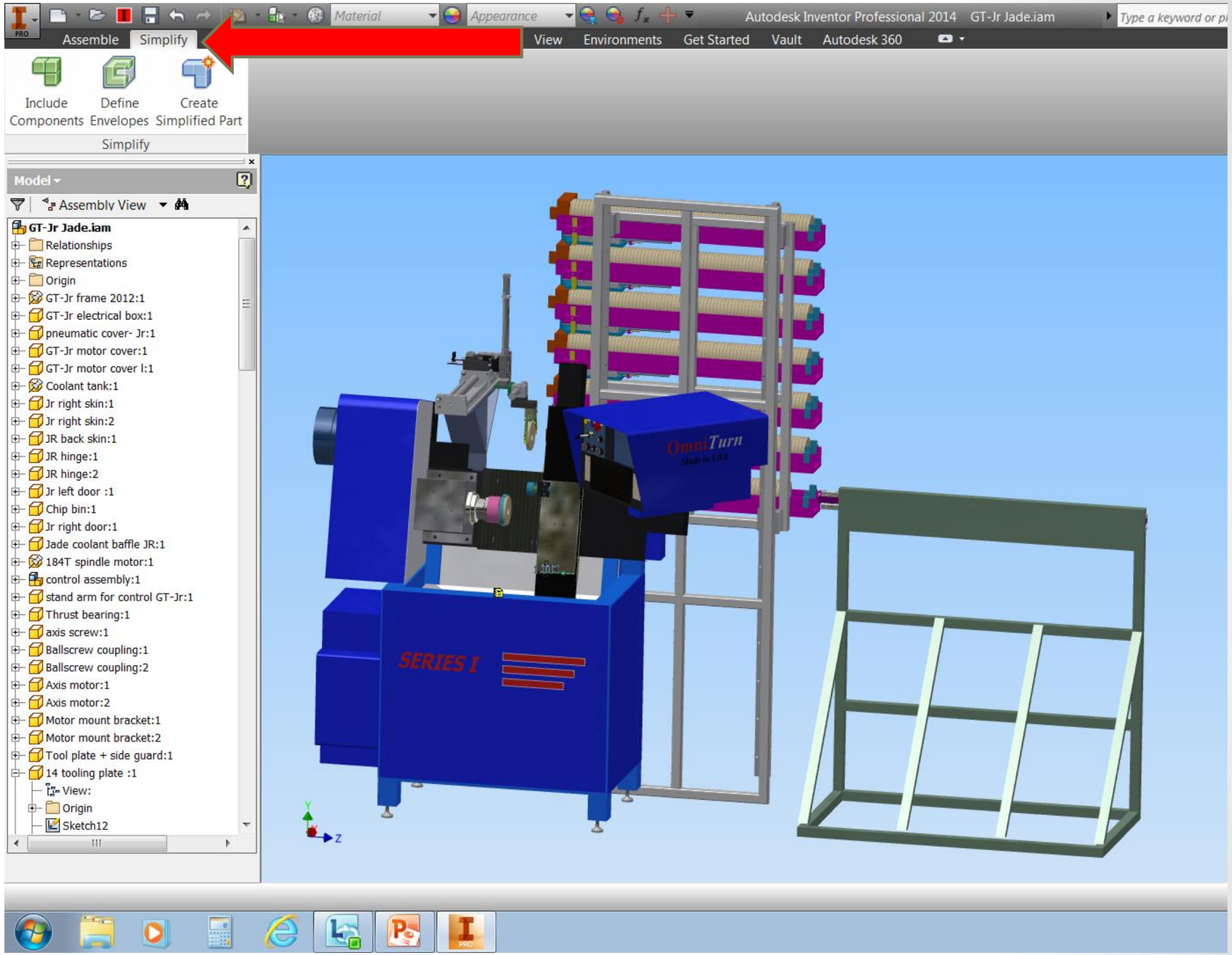
D: Shrink wrap



D: Shrink wrap

Shrink wrap with simplicity will get it done. Small file size, Hide the details, Get the idea across.

With the assembly open, go to the Simplify tab. Select the operation that best suits your need.



I need to send my customer a 3D model so he can see what I am talking about.

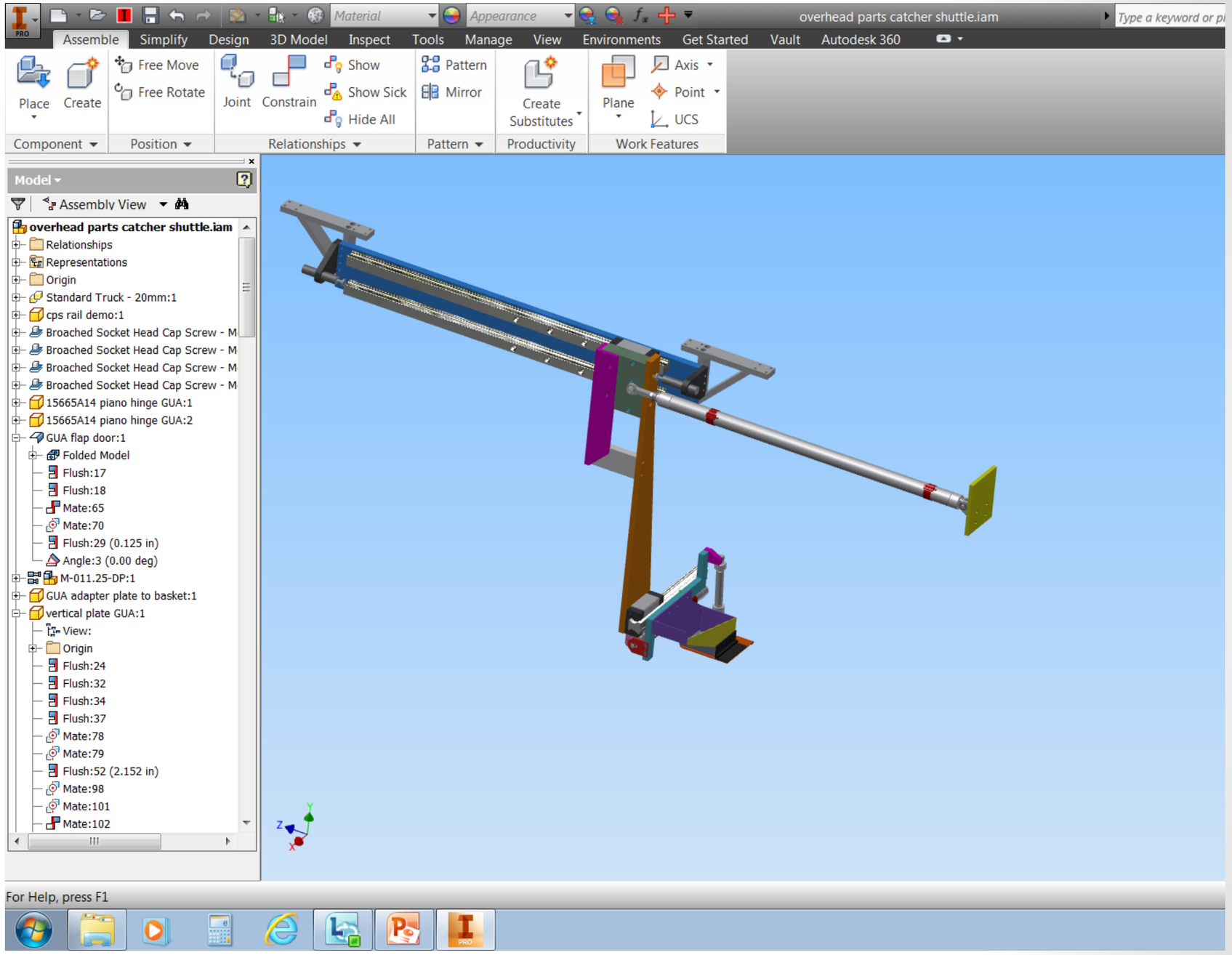
He is at a facility that will not allow him to install Design Review. So I want to send him a 3D PDF. What tool do I use?

A: 3DS Max

B: Showcase

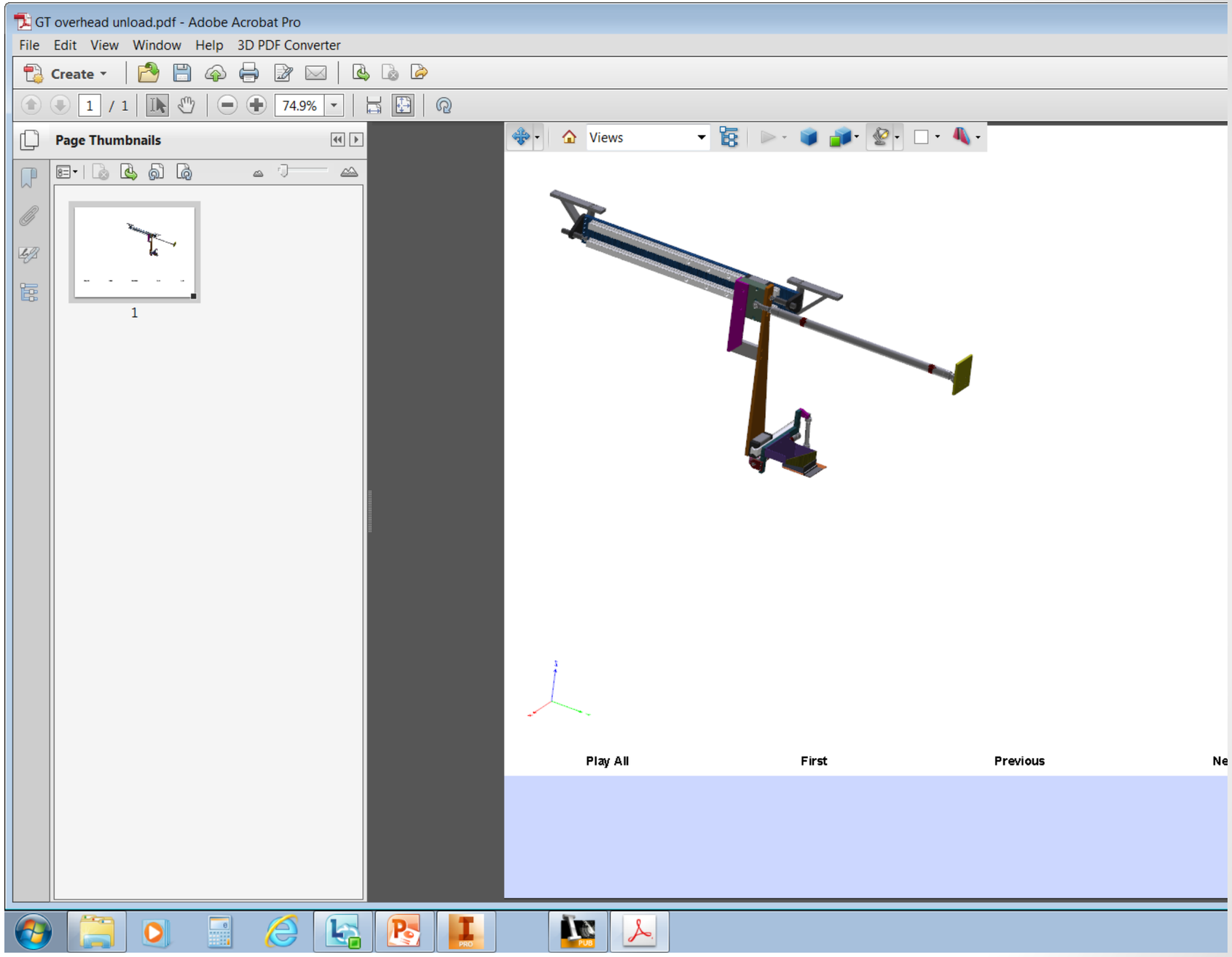
C: Inventor Studio

D: Inventor Publisher



D: Inventor Publisher

Not as useful as a DWF, but in a pinch it gets it done.



What tool do I use to get real time, high quality renderings?

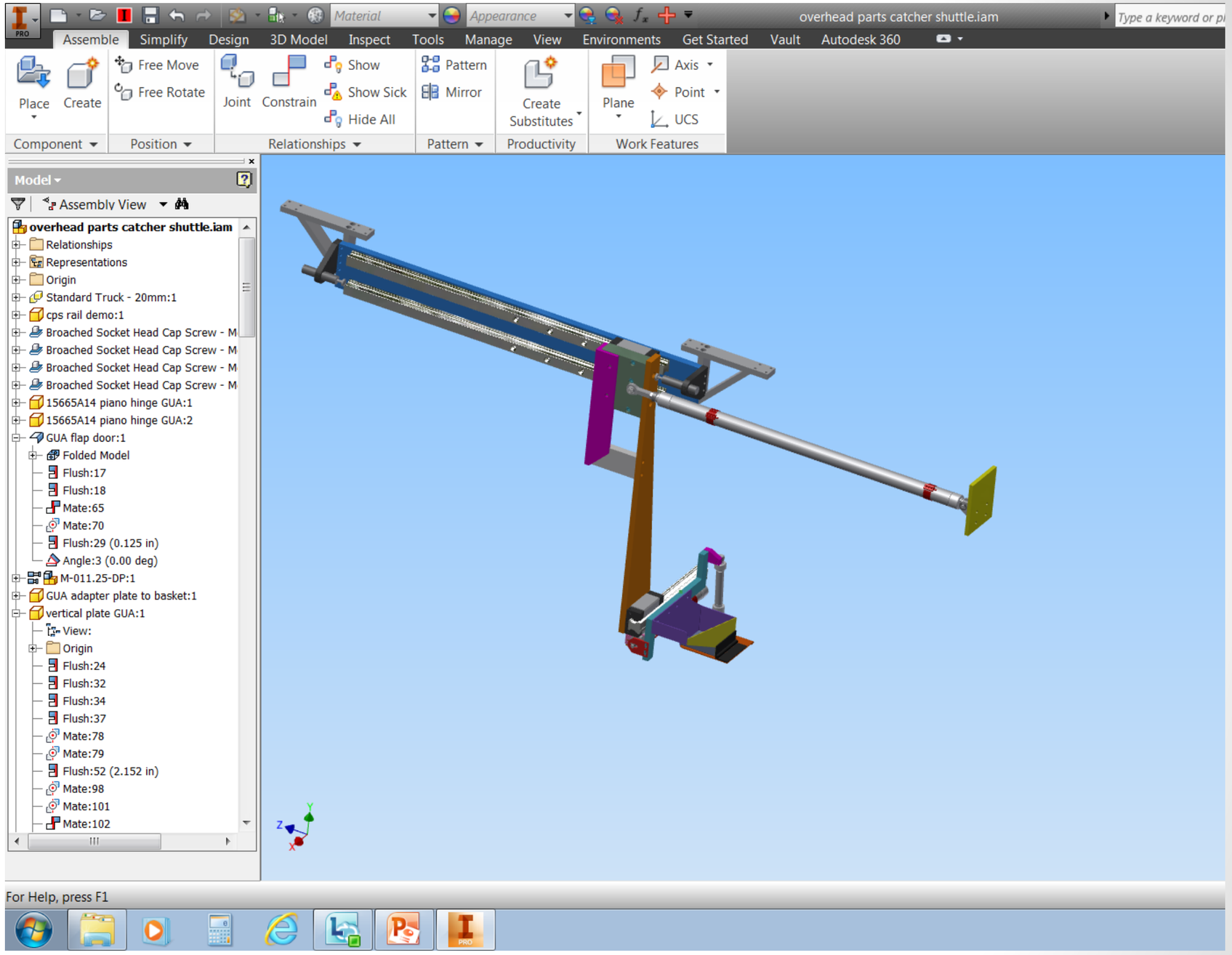
Also quickly experiment with visual changes to material, textures, colors and more.

A: Showcase

B: Photoshop

C: Inventor Studio

D: Inventor Publisher



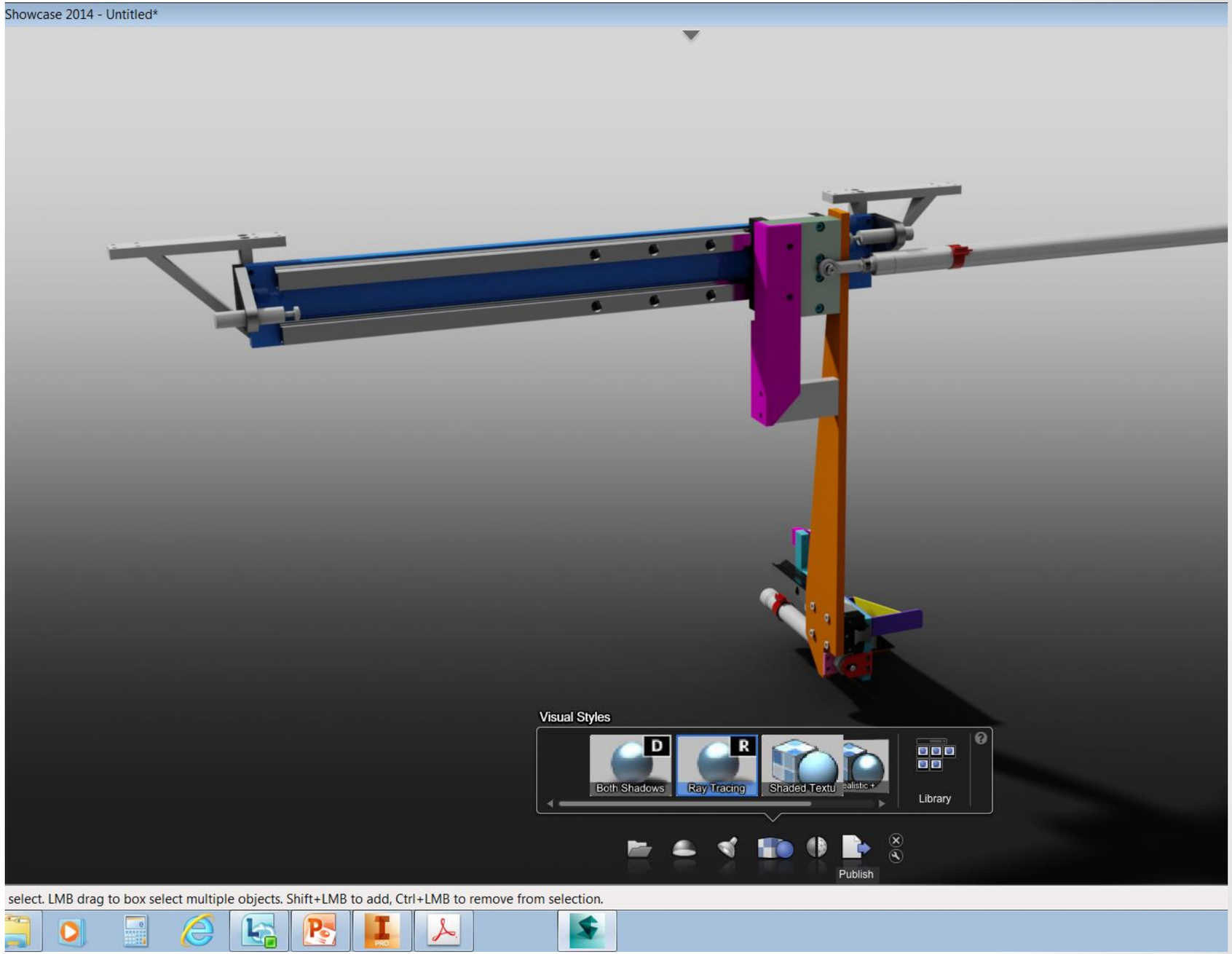
A: Showcase

Simple to use, fast for quick beauty shots and simple videos.

Ray Tracing is also simple.

Once the assembly is imported you can change the appearance in a moment.

Create great looking presentations in a fraction of the time of other imaging tools.



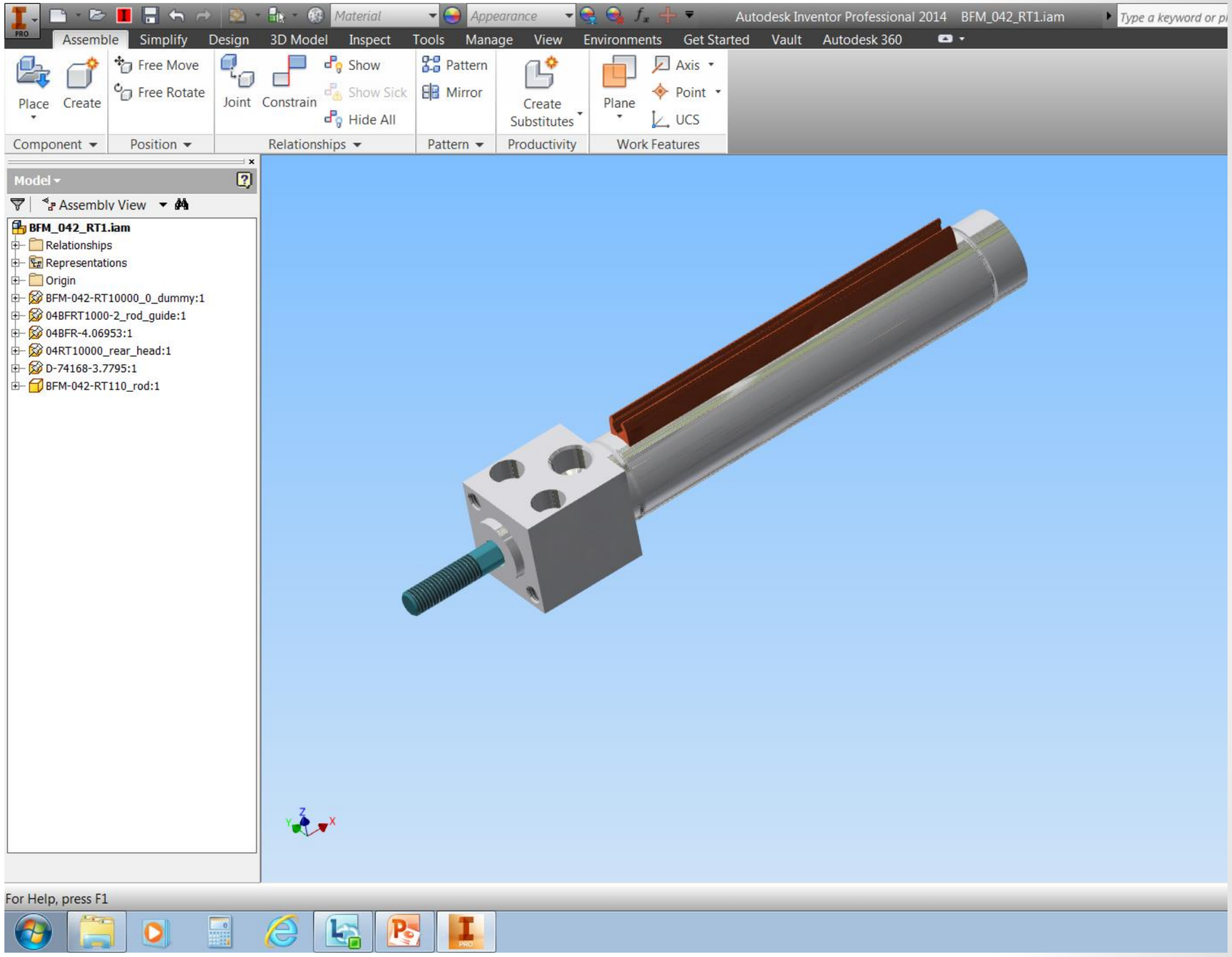
We are used to being able to constrain the shaft of this piston to the cylinder. Also limiting the motion of the shaft to match the real assembly. Now with 2014 it can be done in one dialog box.
What is it?

A: iAssembly

B: iLogic

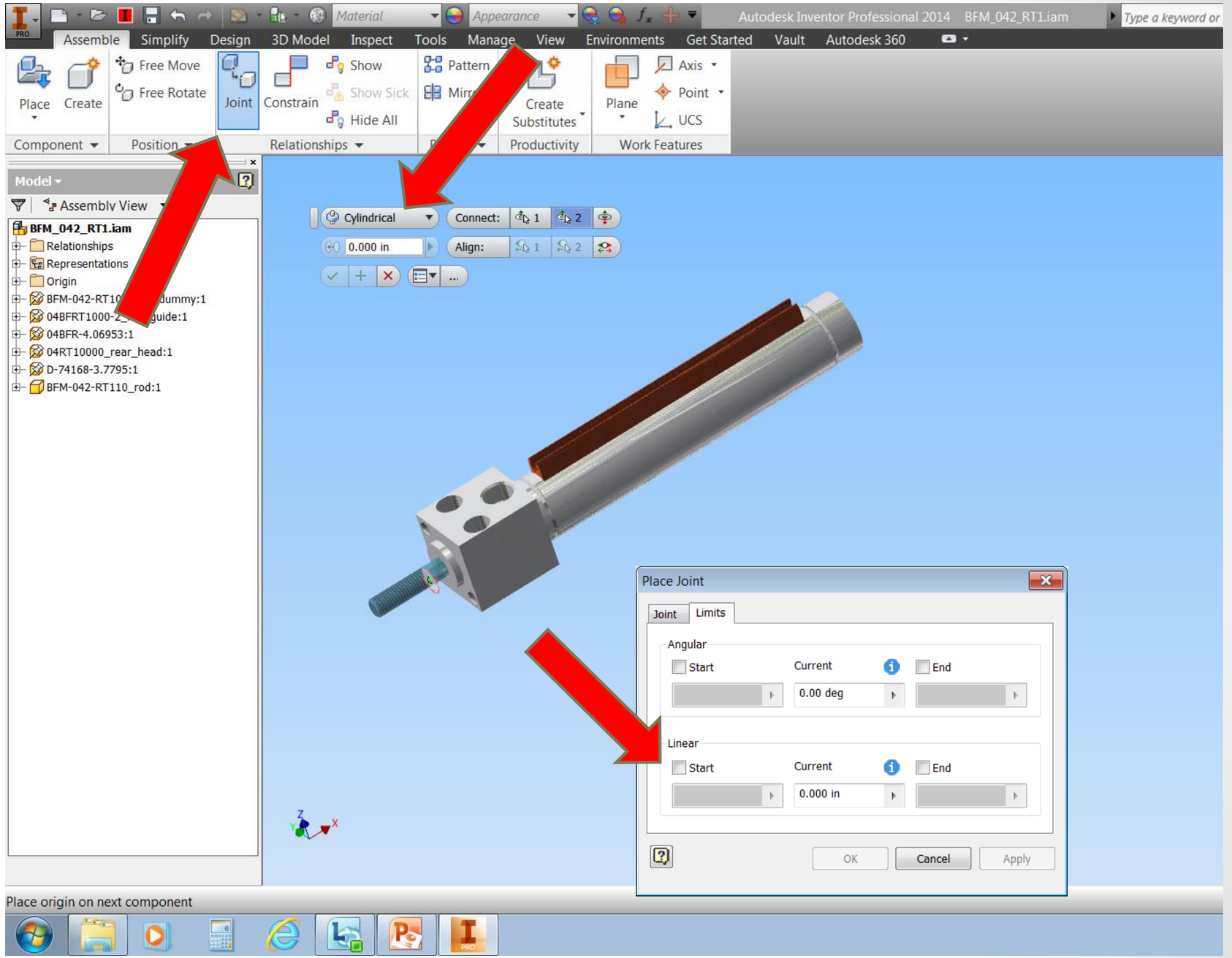
C: iDon't know

D: Joints



A: Joints

Joint not only enables new constraint types, but it will also pass along design intent to analysis tools.



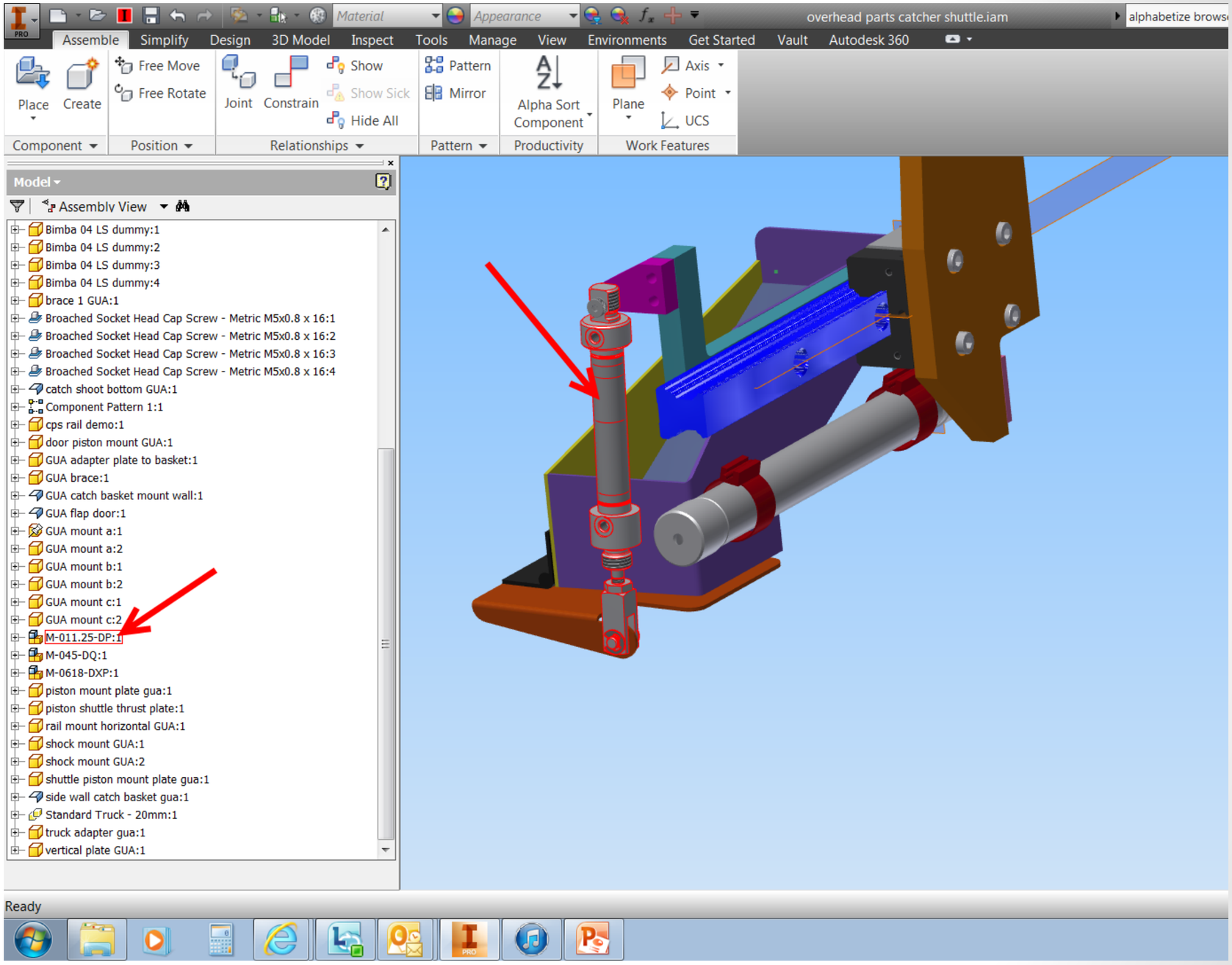
In this assembly I would like to be able to open and close the trap door. This means that the piston with the arrow needs to be able to move within the constrains of its subassembly. What do I do to make this happen

A: Locked

B: It can't be done easily. Another AUGI wish list item.

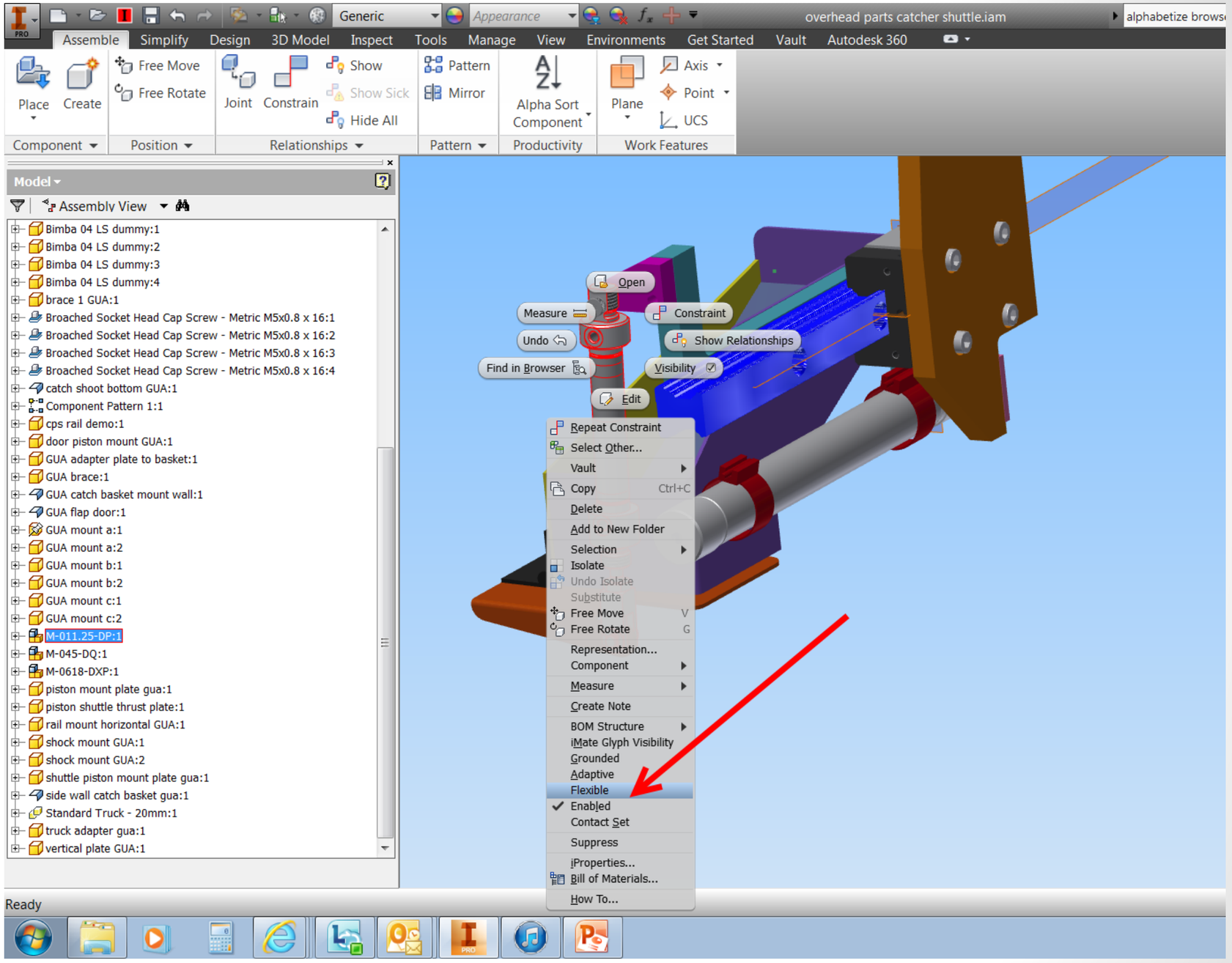
C: Bring the Sub-assembly components into the assembly and constrain them as needed.

D: Flexible



D: Flexible

This is another great tool. Turn on Flexible for sub assemblies. They can then move as they would if they were the top level assembly. Constraints and grounding work as setup in their assembly.



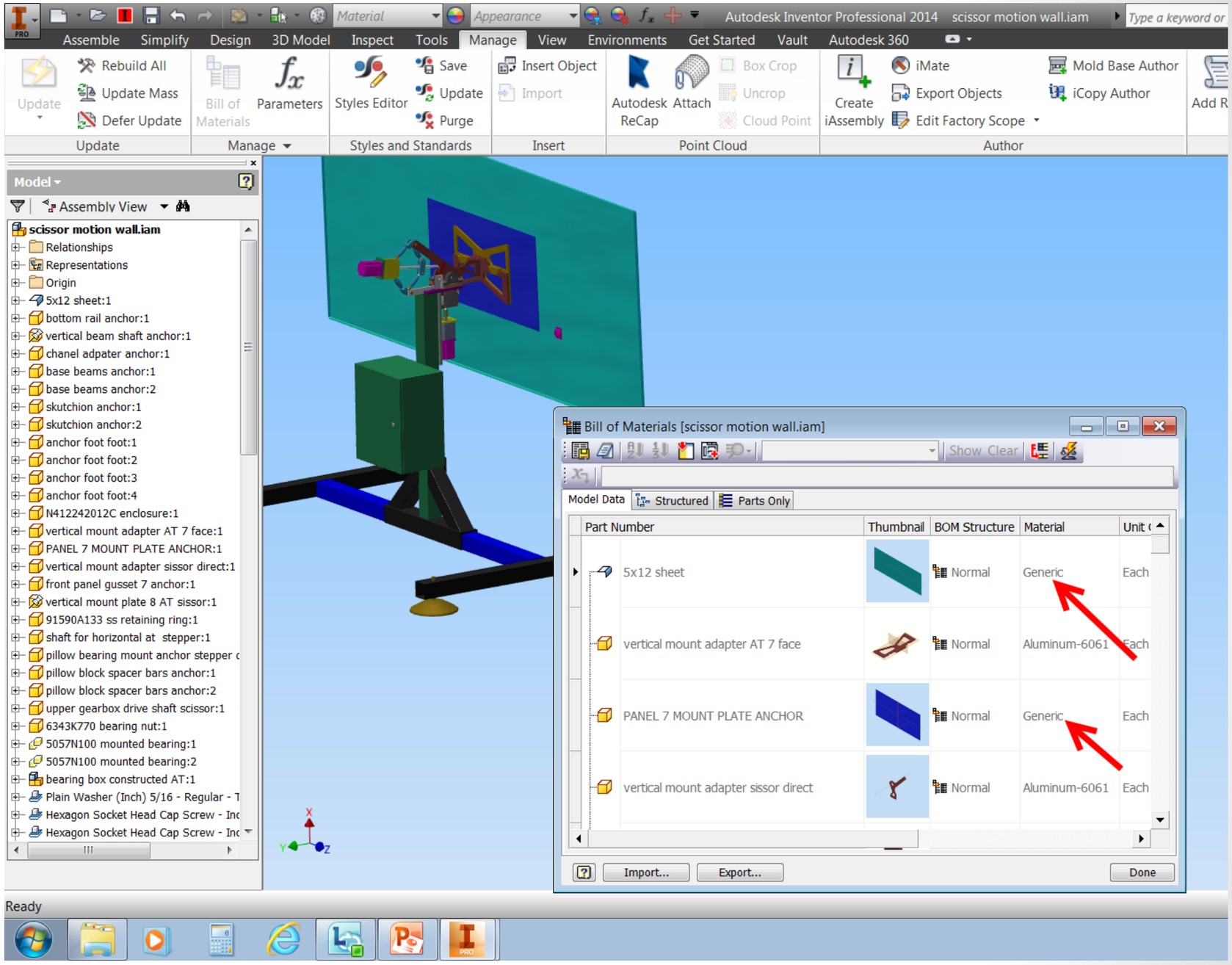
I am working in the BOM to change many part's materials. This is neat since you can change many parts quickly without having to open and close each one individually. However the parts are grayed out and can not be changed. Why?

A: Locked

B: Vaulted and not checked out

C: iProperties

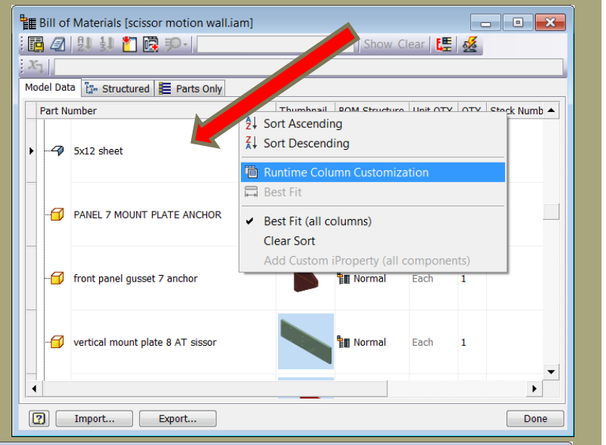
D: Constrained

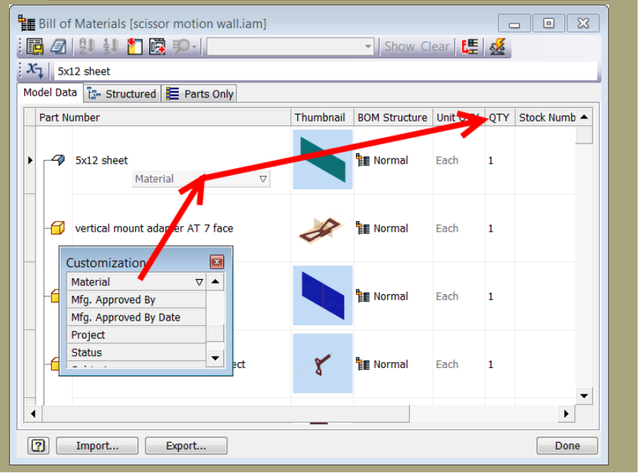


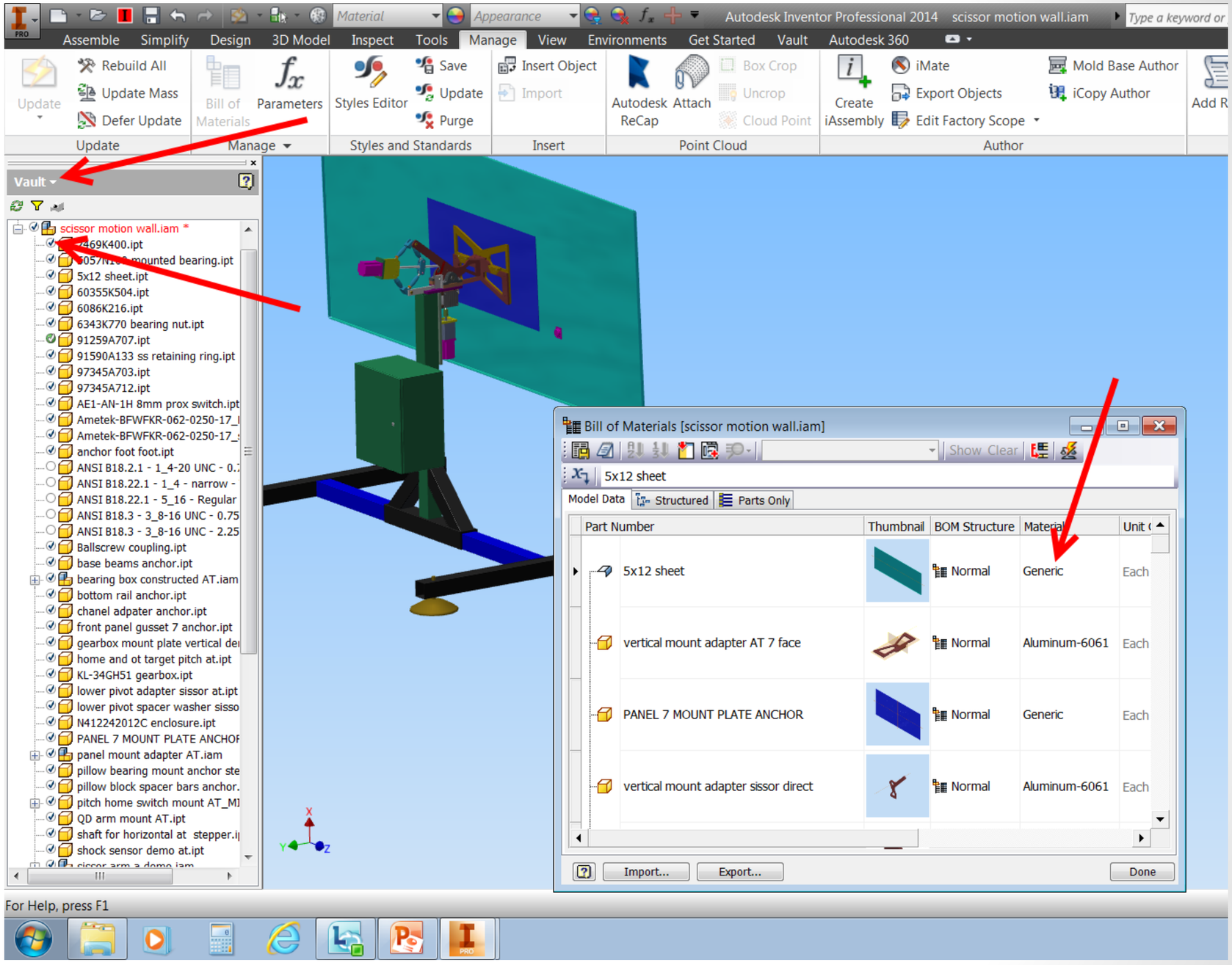
B: Vaulted and not checked out.

Items that are not checked out can not be changed.

Bonus note: to change materials, open the BOM, RMC title bar to be able to select Runtime Column Customization, then drag and drop the Material into the heading bar.







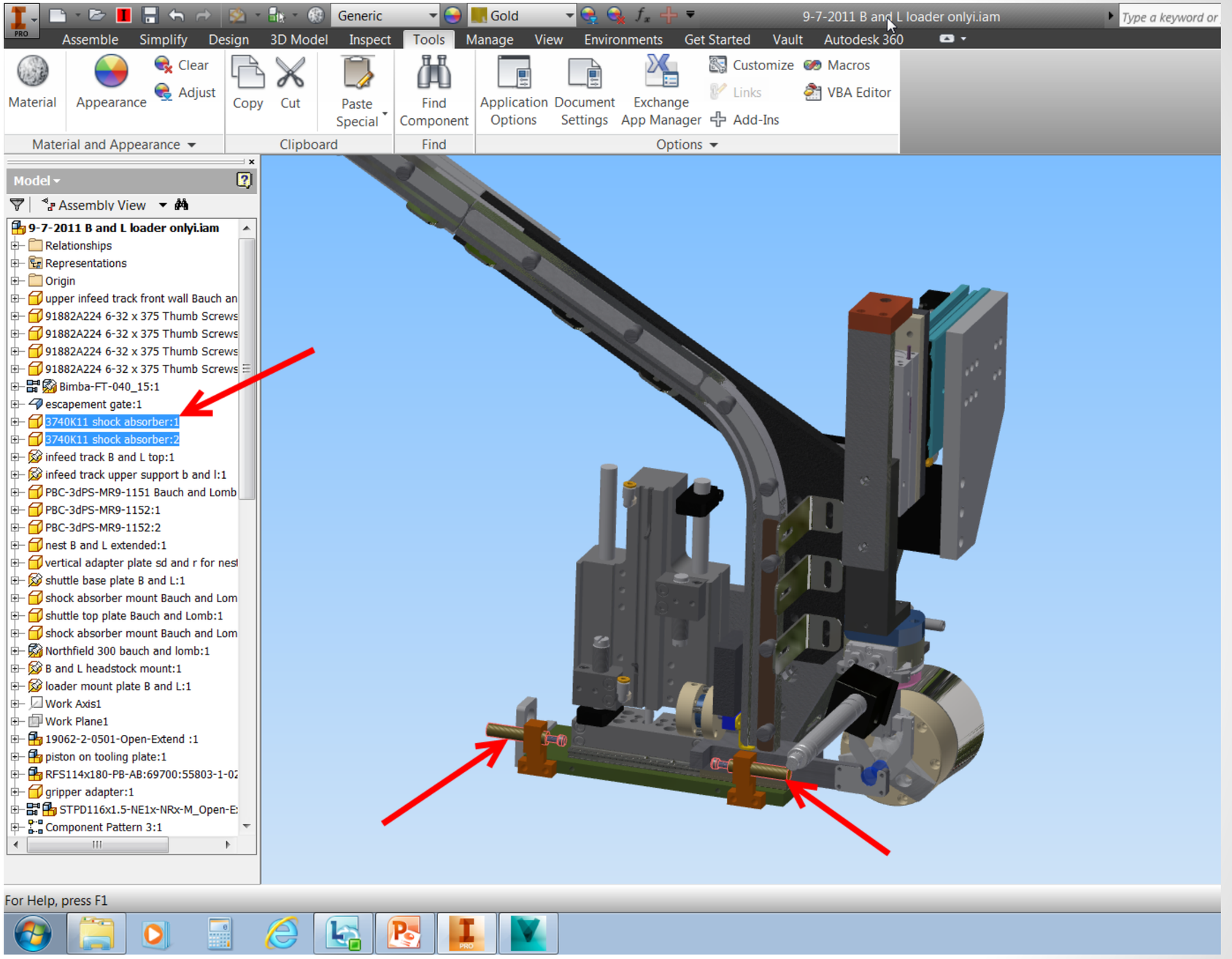
A part that is used in many assemblies has just been discontinued by our supplier. I now need to find where it is used and replace it with the new part. How can it be done in just a few minutes?

A: Vault

B: Windows Explorer

C: It can't be done easily. So sit back, let your wife know you won't be home for dinner, *again*, then use the Find tool.

D: Design Accelerator

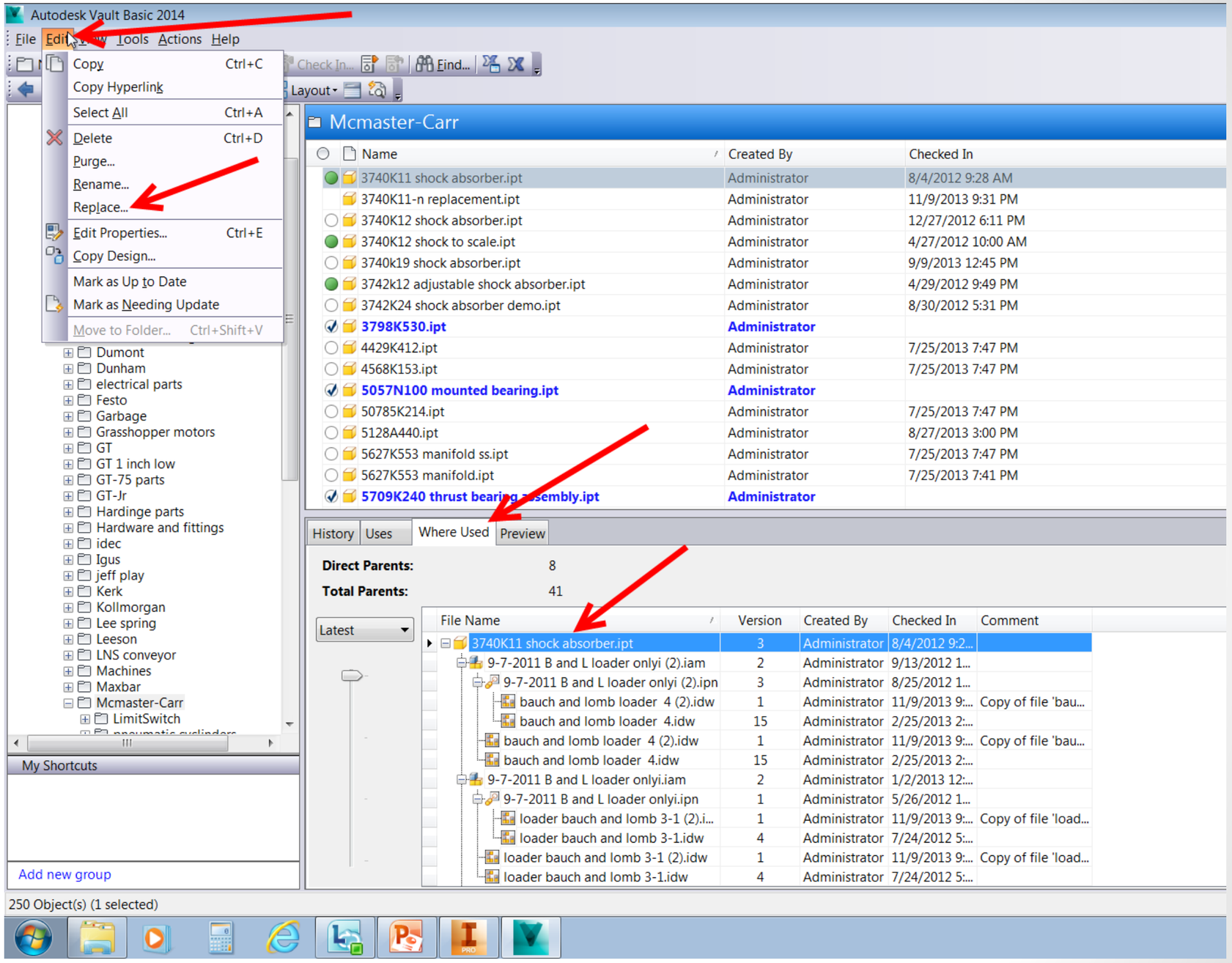


A: Vault

Vault is such a cool tool. And it is free. Even if you are a single seat and don't share material with others you should be using Vault.

In this example you could find and replace all the discontinued occurrences in just a few minutes. This is one of MANY cool Vault tools.

Process:
Find the part in the Vault, go to the "Where Used" tab, drop the Edit tab, then select "replace". Then follow the instructions.



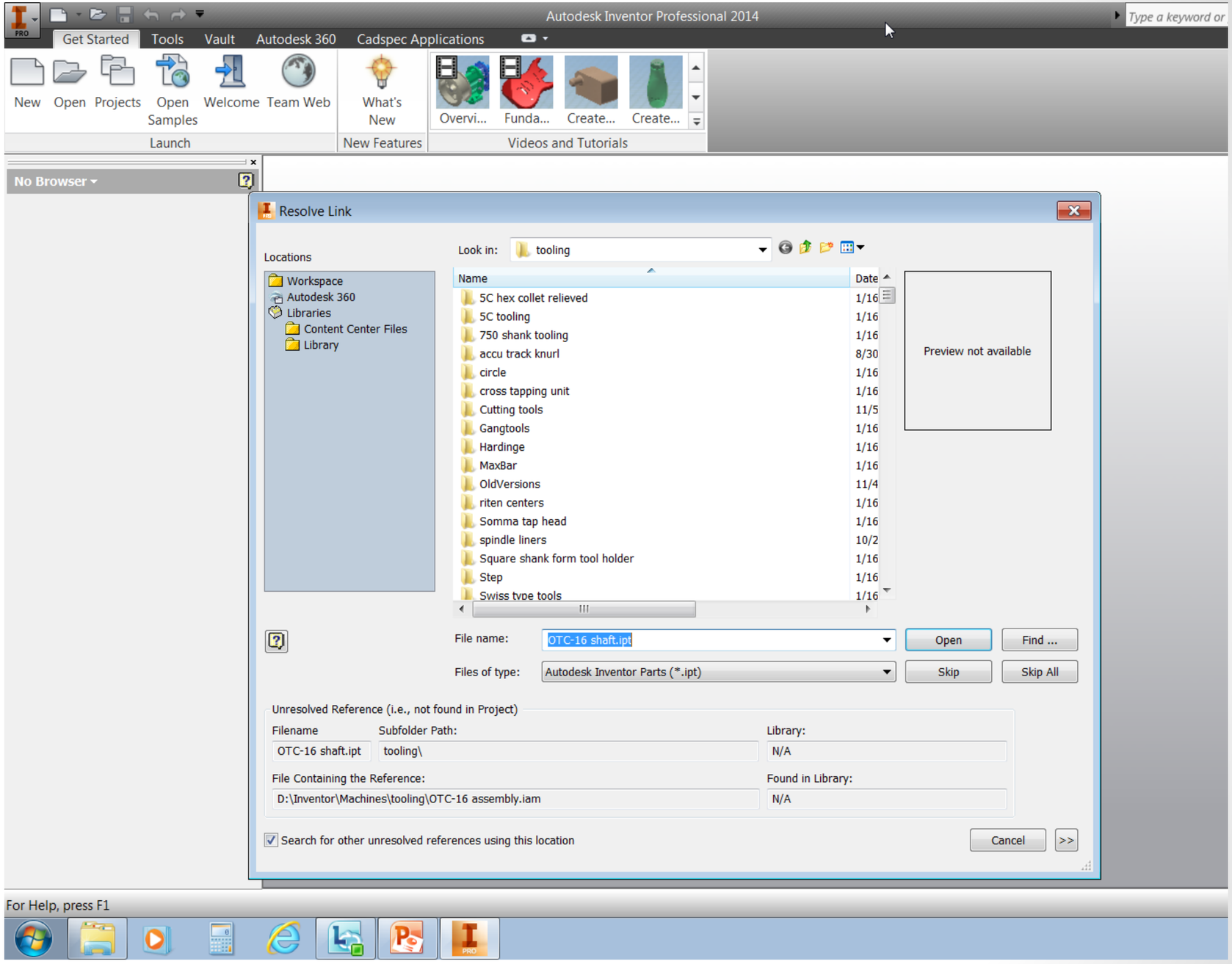
You get a warning when you open an assembly that you have unresolved links. Any idea what might have happened? **One** possibility is:

A: Application settings are not configured correctly

B: Project selection is messed up.

C: You need more RAM. You are running out of memory before the files can be loaded.

D: The new intern has been experimenting after hours again.

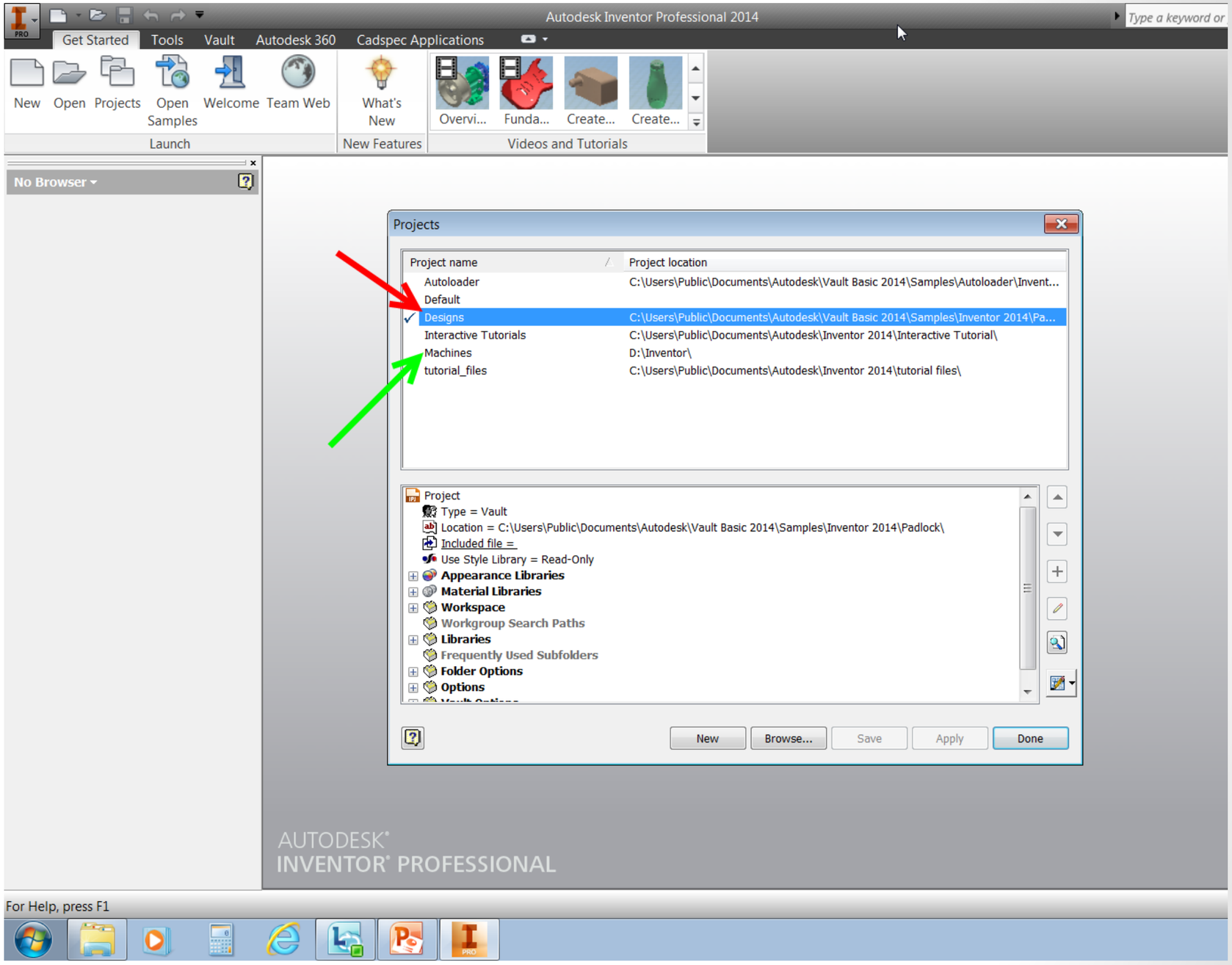


B: The Project is not set to your workspace

When opening an assembly, Inventor looks to the workspace for the required parts and sub assemblies. If the project selection has gotten messed up these errors will happen. Also if you don't save your parts and assemblies in the workspace this will also happen.

Go to the "Get Started" tab, select Projects, make sure your project is the one checked.

Note: Another result of this will be if you don't see your templates when creating a new file. The Project points to where your templates are stored.



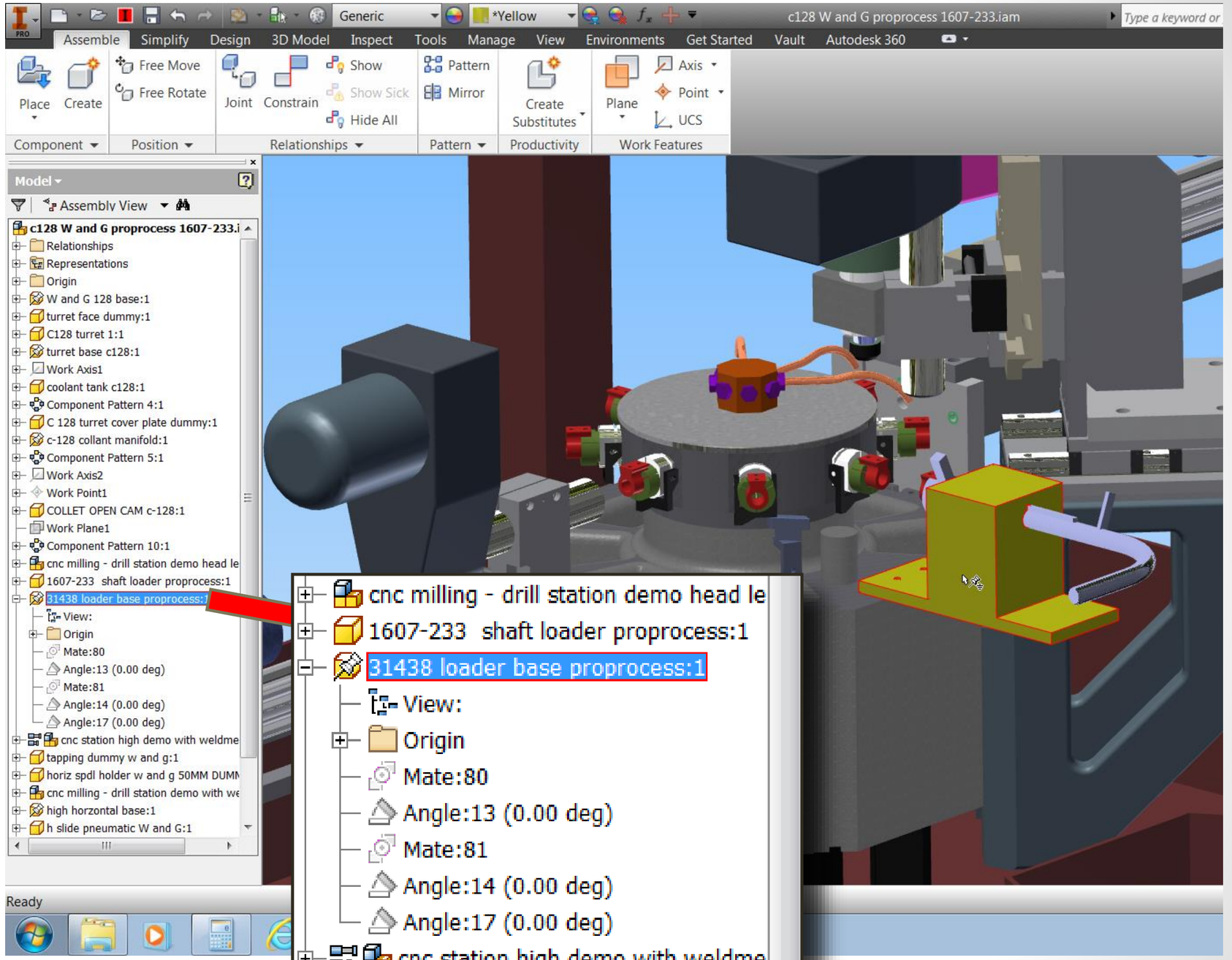
You would like to move the yellow part "***31238 loader base proprocess***". But it won't move. Why?

A: It is grounded

B: It is constrained

C: It is not flexible

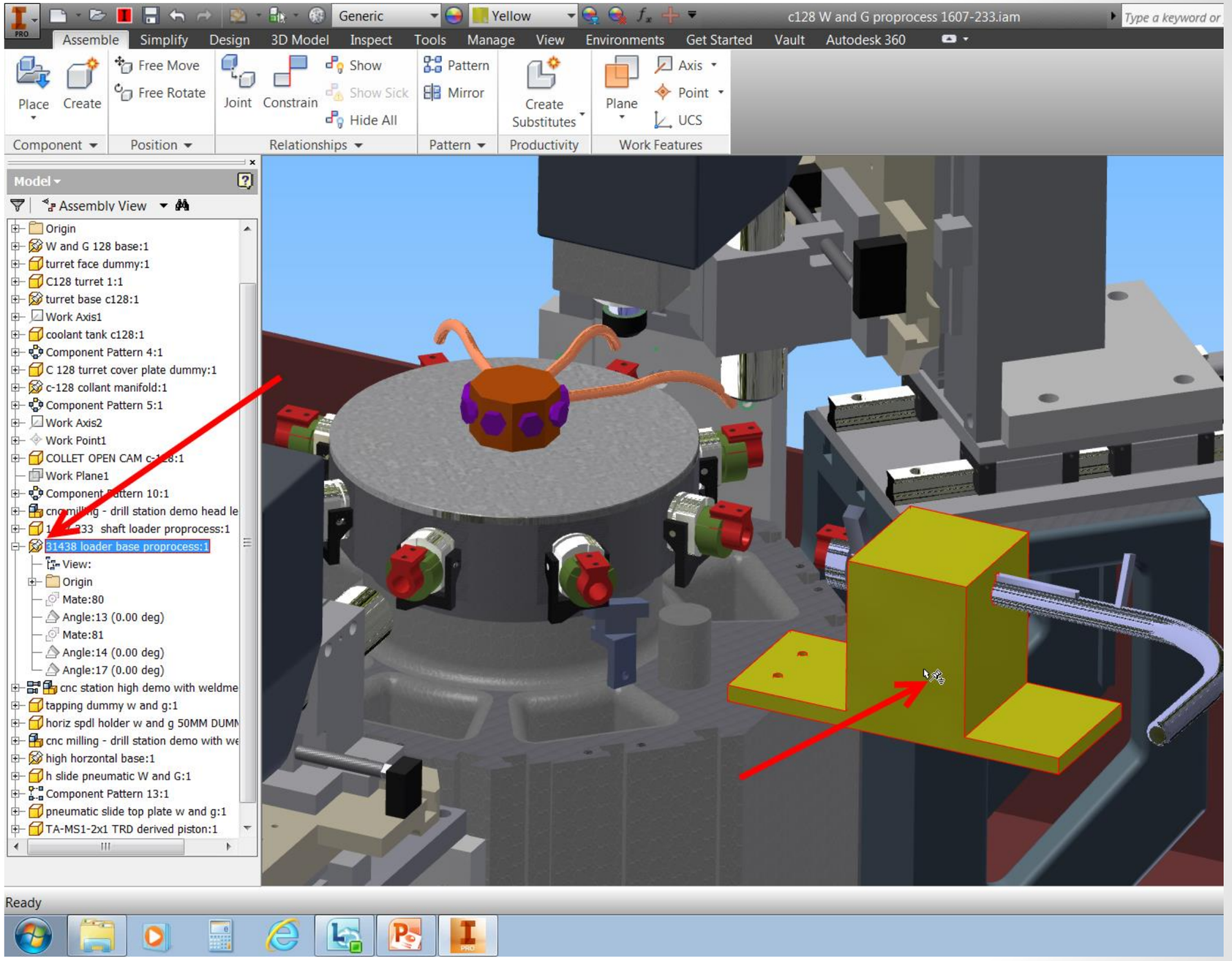
D: Adaptivity



A: It is grounded.

You can see in the browser there is a thumbtack on the type icon. This shows that the part is grounded. Also if you hover the mouse over the part, the cursor turns into a thumbtack. Also indicating its being grounded.

If you selected constrained, you were not necessarily wrong. If another part in the assembly was adaptive and linked to the yellow base it would not move. You would need to turn off the adaptivity.



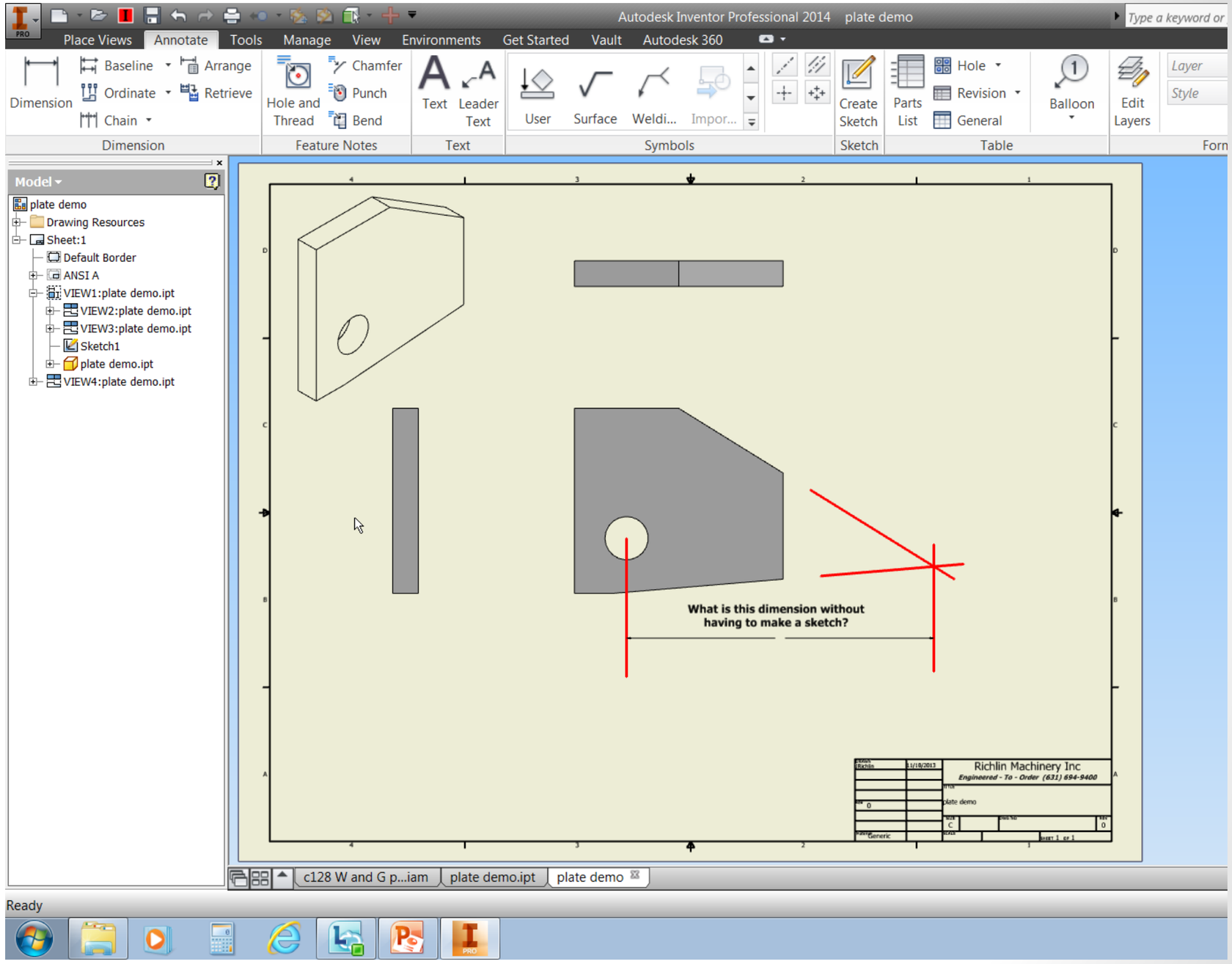
You need to place a dimension from the hole center to the intersection of the two angled sides. There is a tool that can do this quickly without needing to create a sketch. What is it?

A: This is a trick question. You need to draw a sketch

B: Make a new part including the intersection.

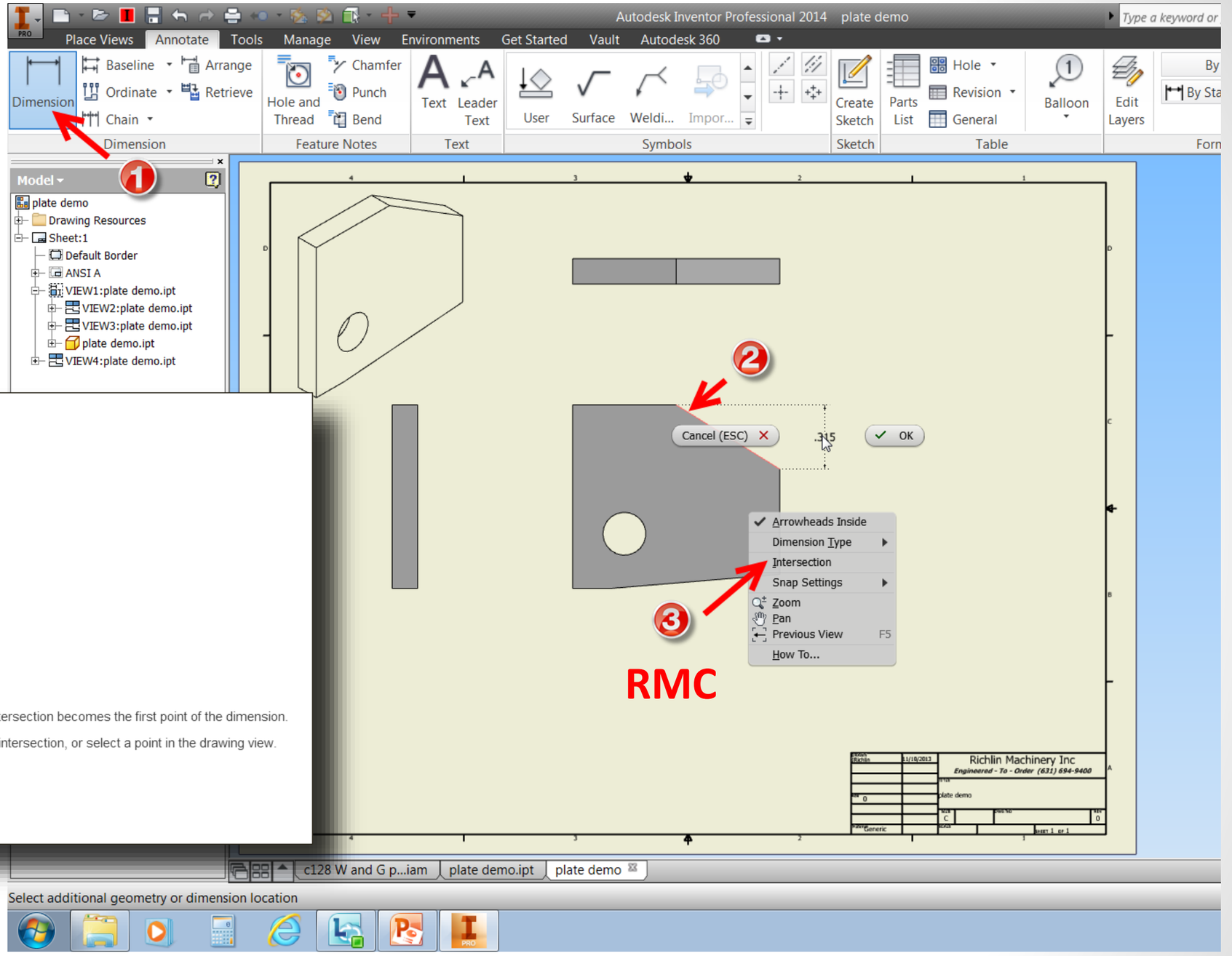
C: Intersection tool

D: Call for pizza, it will be a long night

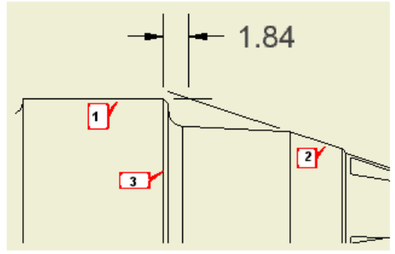


C: Intersection tool

The Intersection tool is found with a RMC after selecting one leg of the intersecting lines.



Add dimensions to intersections of extension lines



1. On the ribbon, click Annotate tab ➤ Dimension panel ➤ Dimension .
2. Select the first line (1) to define the intersection.
3. Right-click and select Intersection from the menu.
4. Select the second line (2) to define the intersection. The lines are extended as needed, and their intersection becomes the first point of the dimension.
5. Select a model edge (3) to define the dimension. Alternatively, perform steps 2-4 to define another intersection, or select a point in the drawing view.
6. Drag the dimension to the desired position.
7. Click to place the dimension in the drawing.

Note: All geometry that specifies the dimension must belong to the same drawing view.

You have a right handed assembly that that needs to be mirrored and modified.

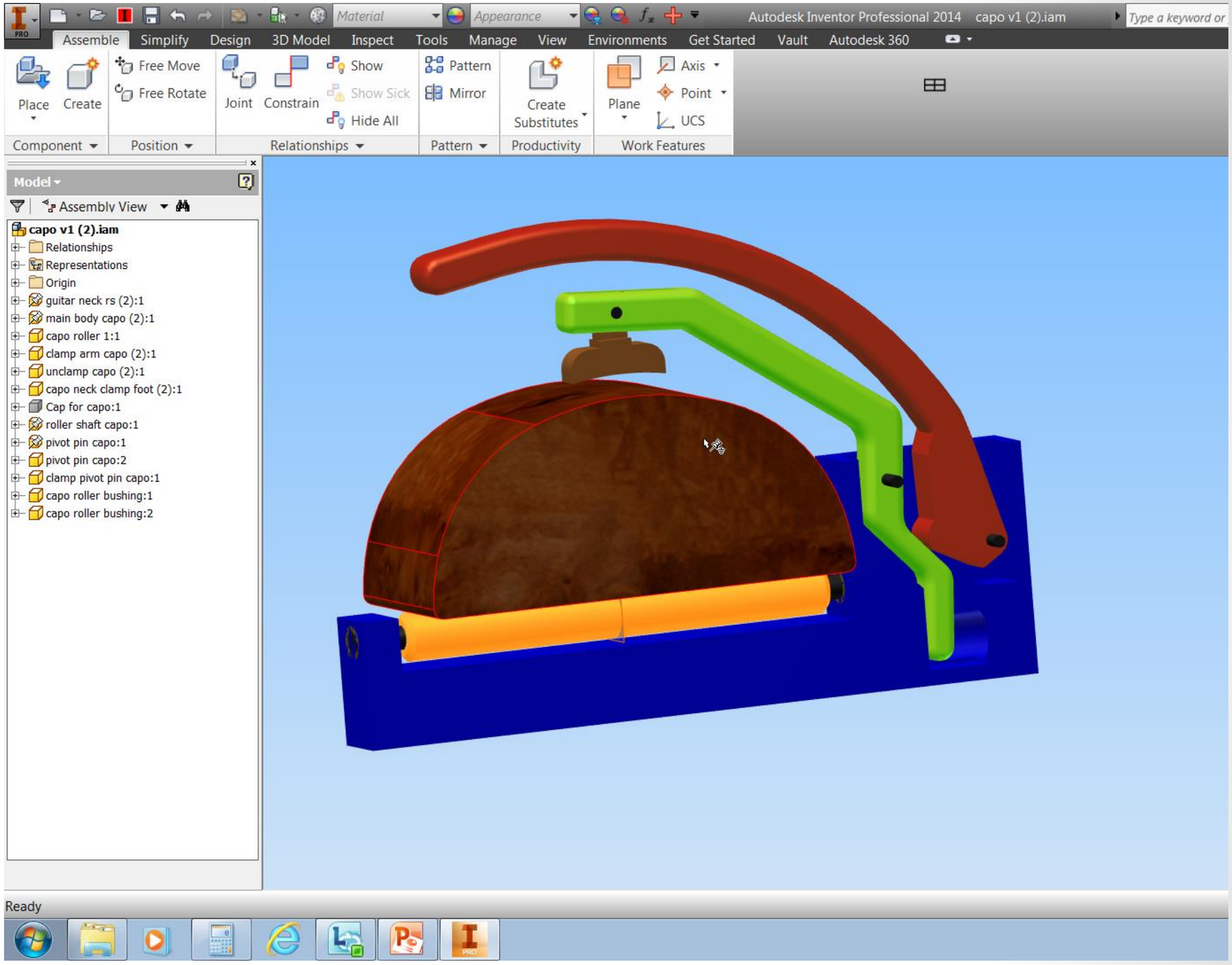
To modify it successfully, it will need to be constrained as before. Can you do this quickly?

A: No, you will need to re constrain the like the original assembly

B: Use Vault to mirror constrained

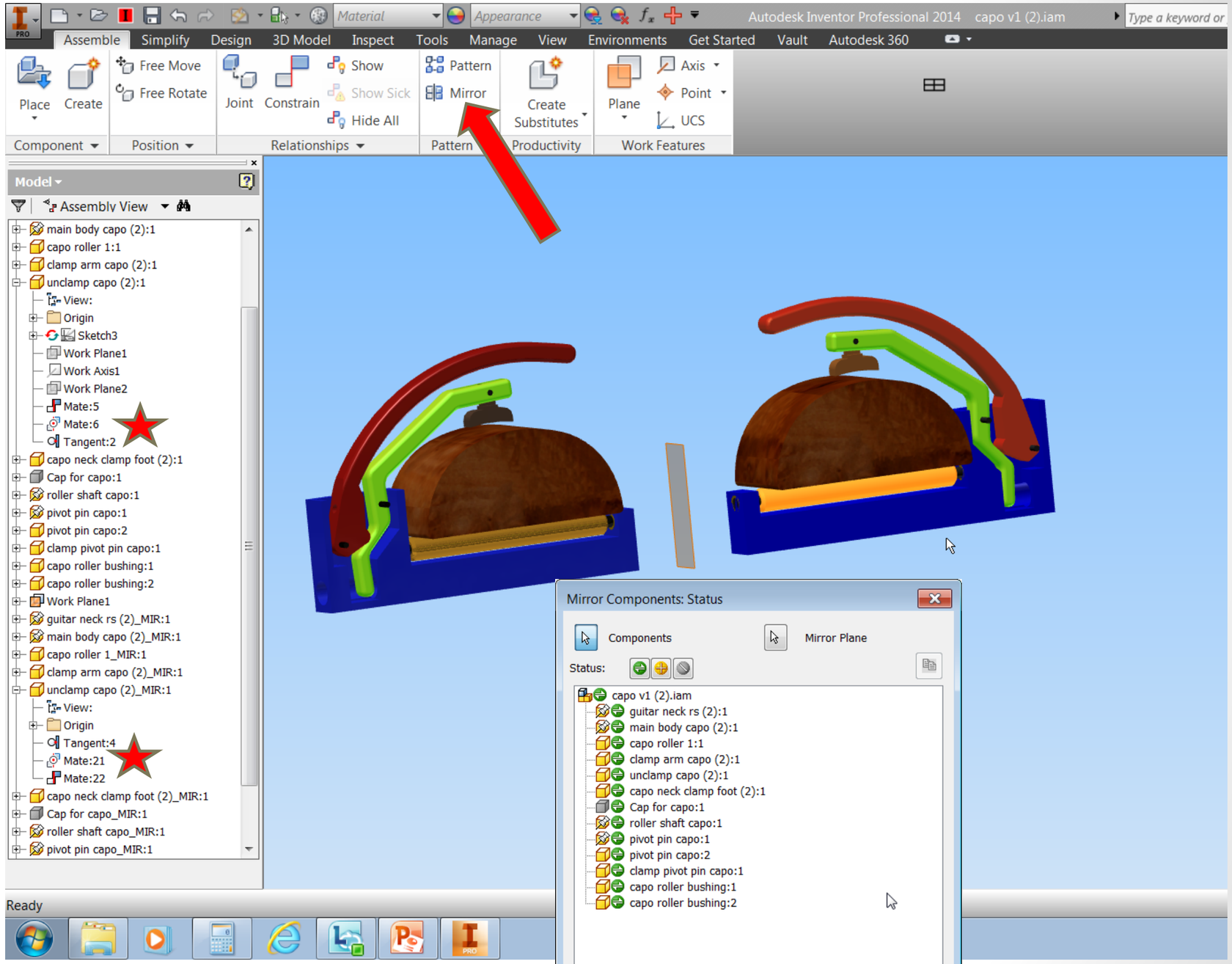
C: Use Derive to create a new assembly

D: Get Inventor 2014



D: Get 2014

With 2014 one of the great new features is the ability to mirror and keep the new assembly constrained. Note in this mirrored demo the new parts are constrained like the originals.



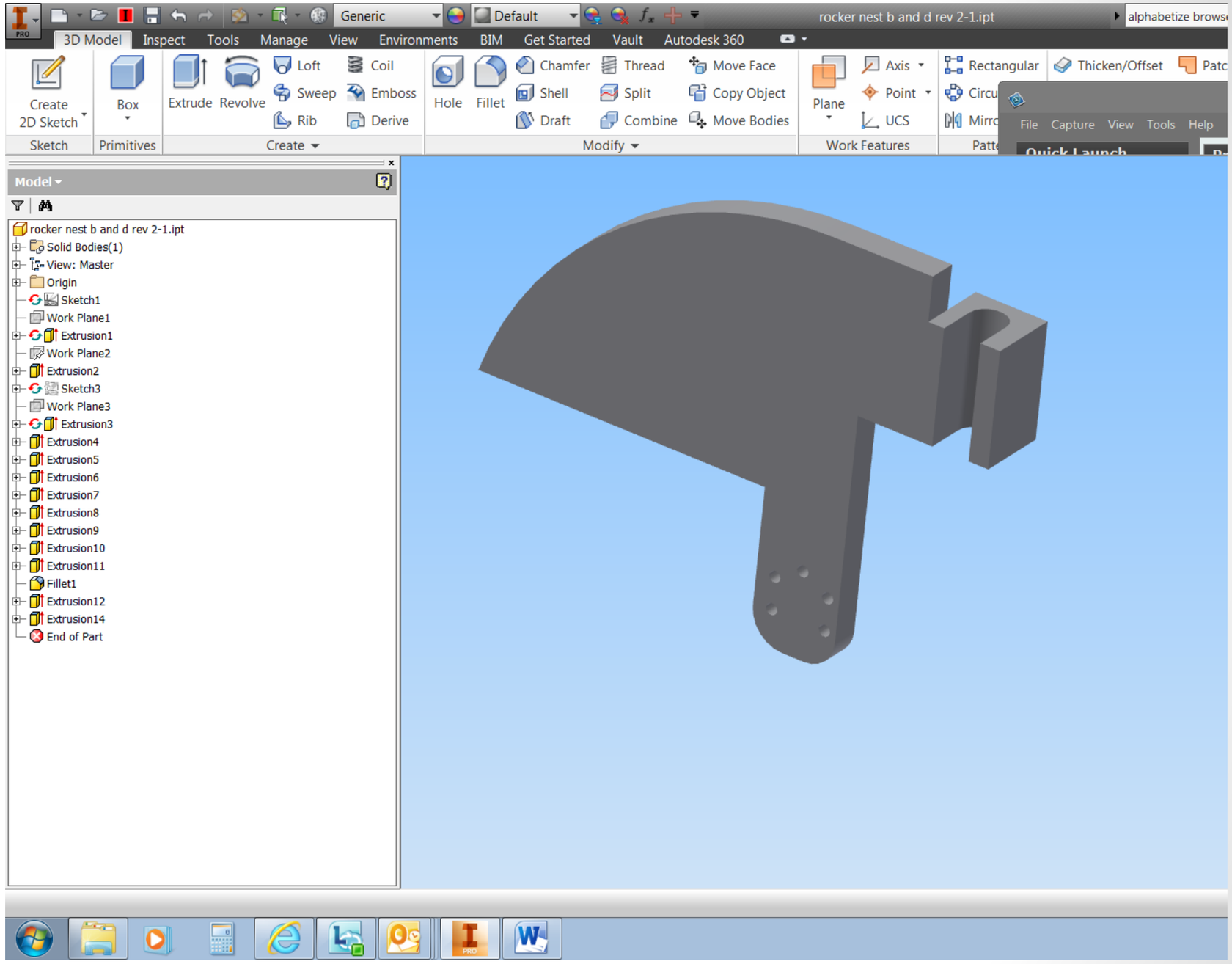
We just got some files in from our sister company. As you can tell by the browser details, there are none. Those guys are a freaking mess. What tool can I use to try and figure out what their intent was with this part? Or at least see what they did?

A: Vault

B: Design Review

C: Psycotherapy

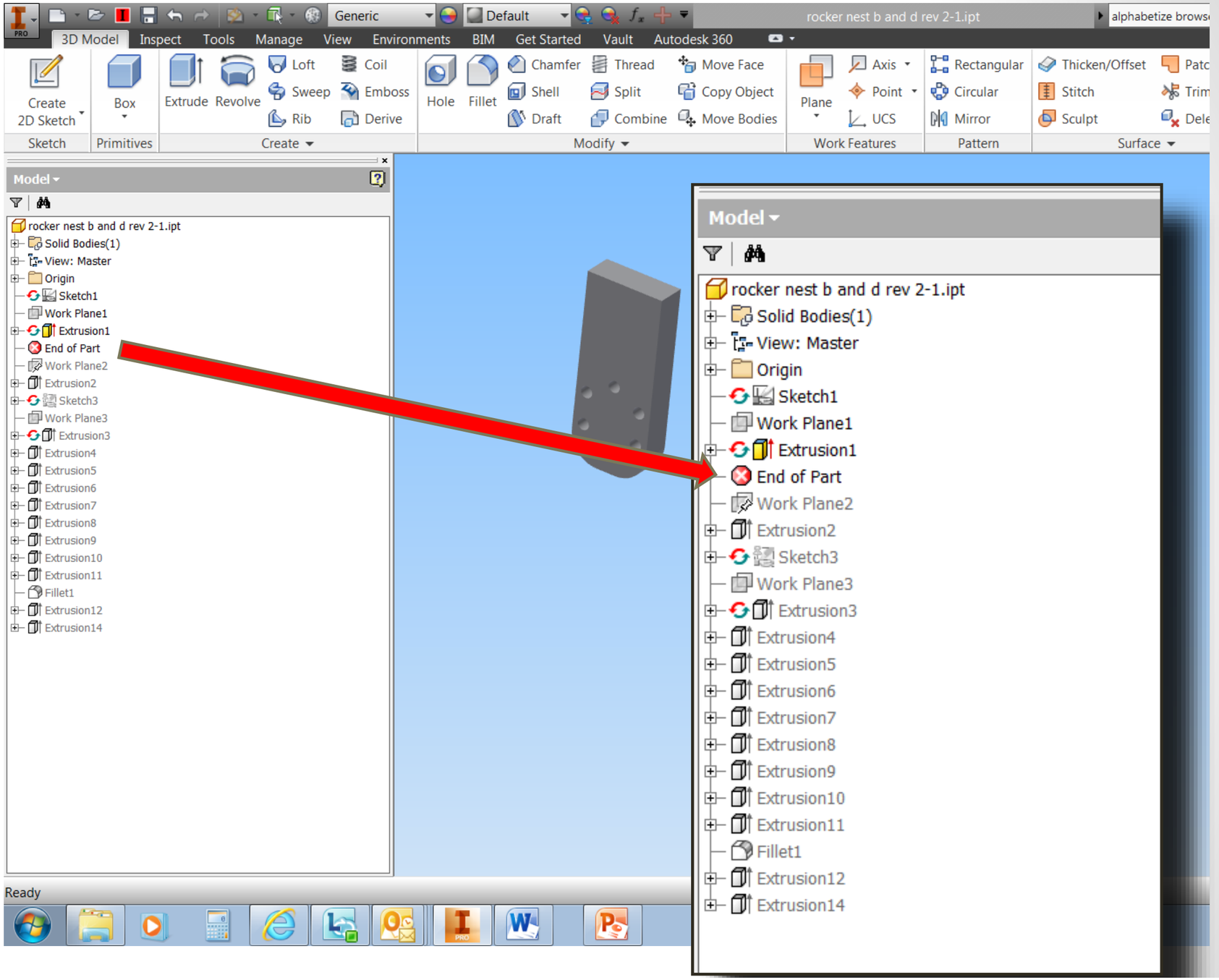
D: EOP (end of part)



D: End of Part - EOP

If you drag the “End of Part” up the browser tree features will suppressed. I find working my way through the part, step by step, I can get an idea of what the other team was thinking, if they were.

When done, just drag the EOB back to the end.



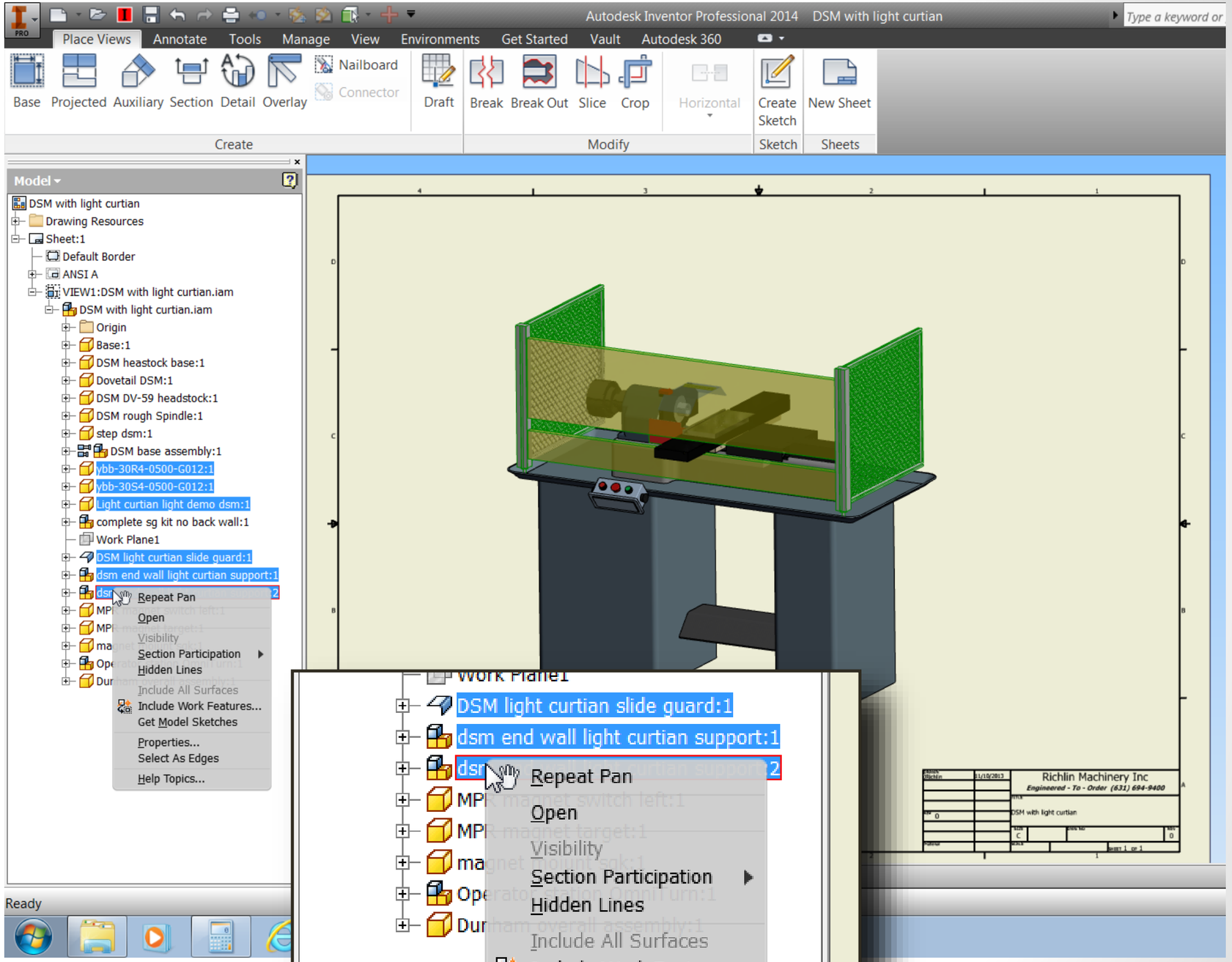
In this drawing I would like to show the model without the light curtain. Instead of creating new assembly states I want to just turn the visibility off for some of the components. Notice in the RMC on the selected items that the visibility command is grayed out. Why?

A: Edit View needs changes

B: Can't be done. Need to create new assembly details

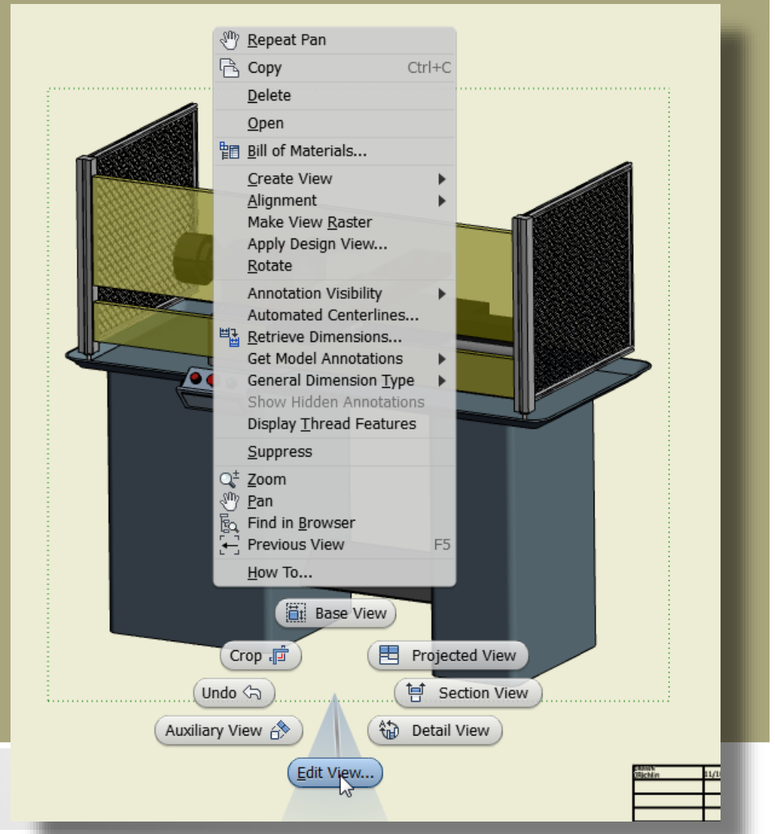
C: Go to the view tab, use the isolate command

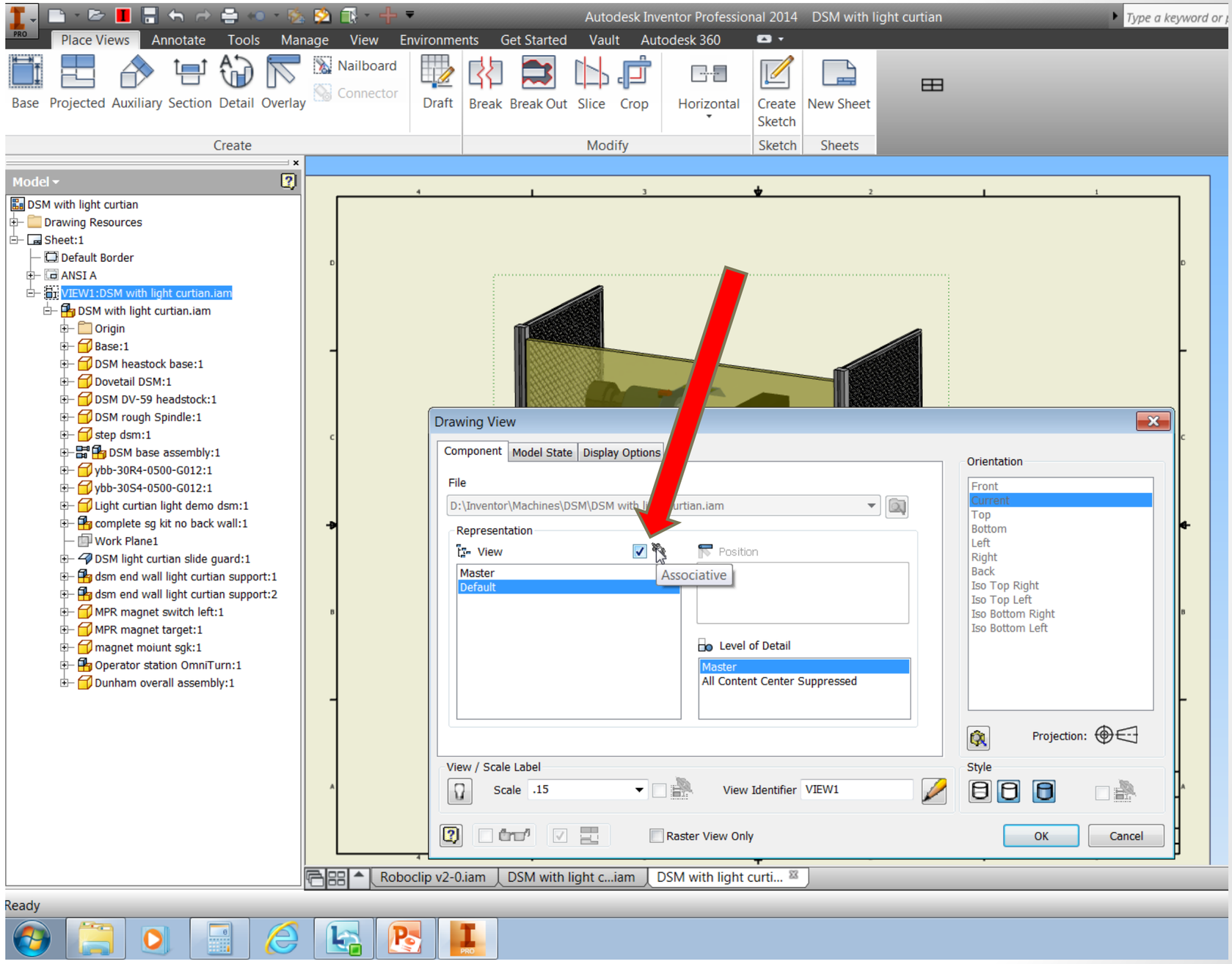
D: Use BOM to make the changes needed.



A: Edit View

Edit the view command (see below). In the Drawing View dialog box uncheck the Associative box. This will enable you to control the visibility of individual items.





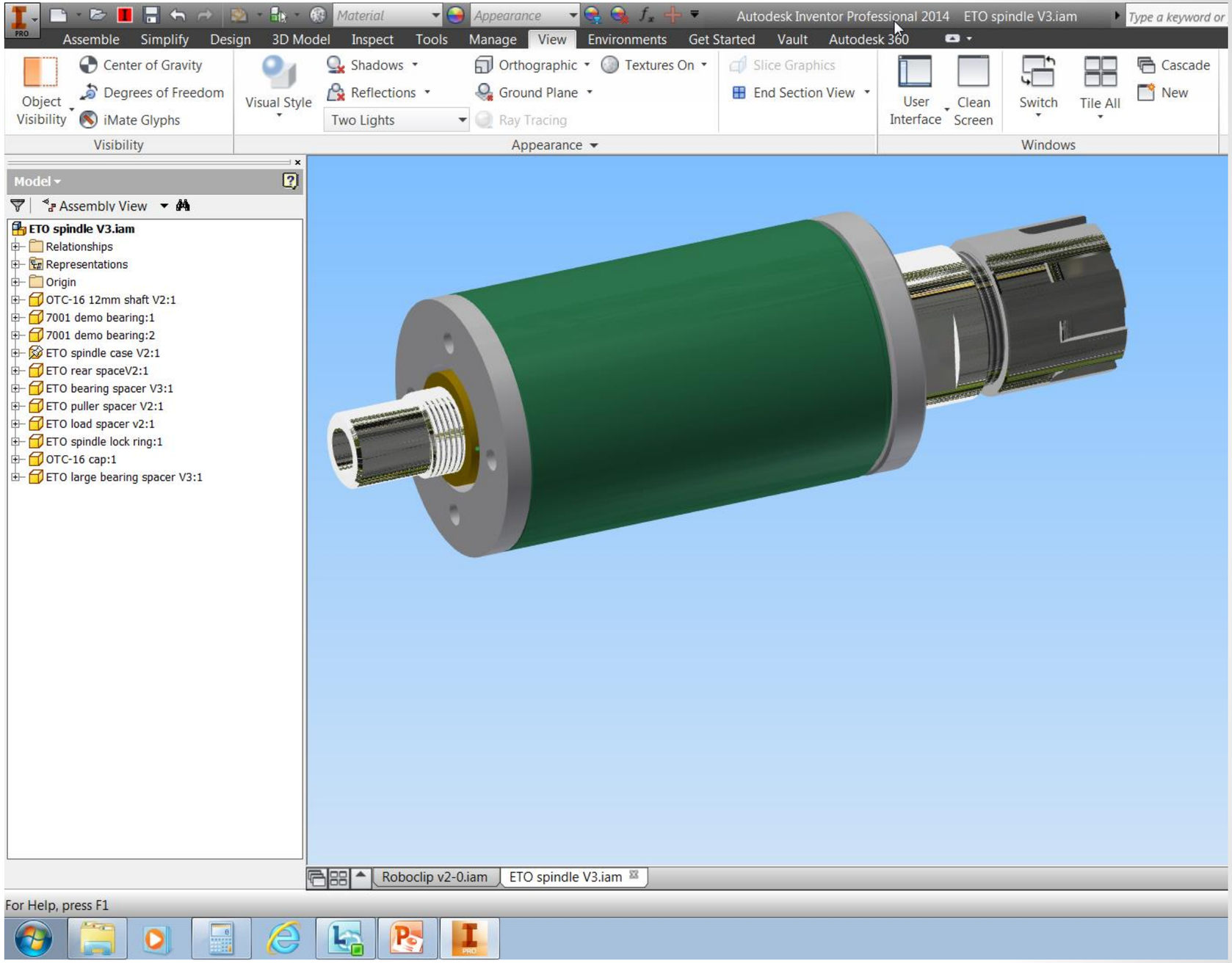
While working on this assembly you need to look at the bearings and spacer inside the spindle. What tool can you use to move constrained parts out of the way?

A: Edit View

B: Can't be done. Need to suppress constraints to get this done

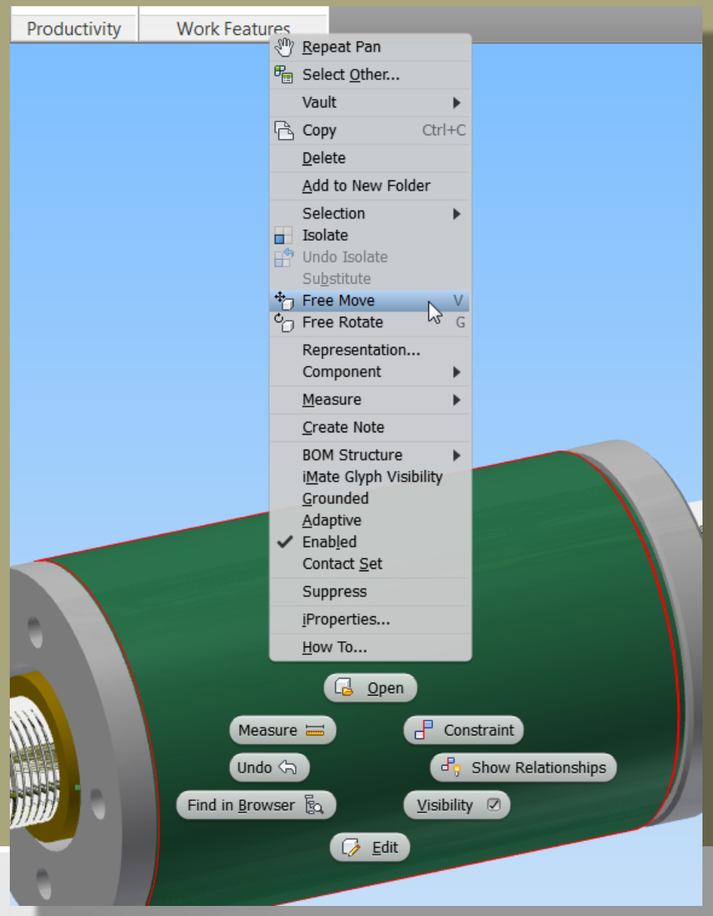
C: Go to the view tab, use the isolate command

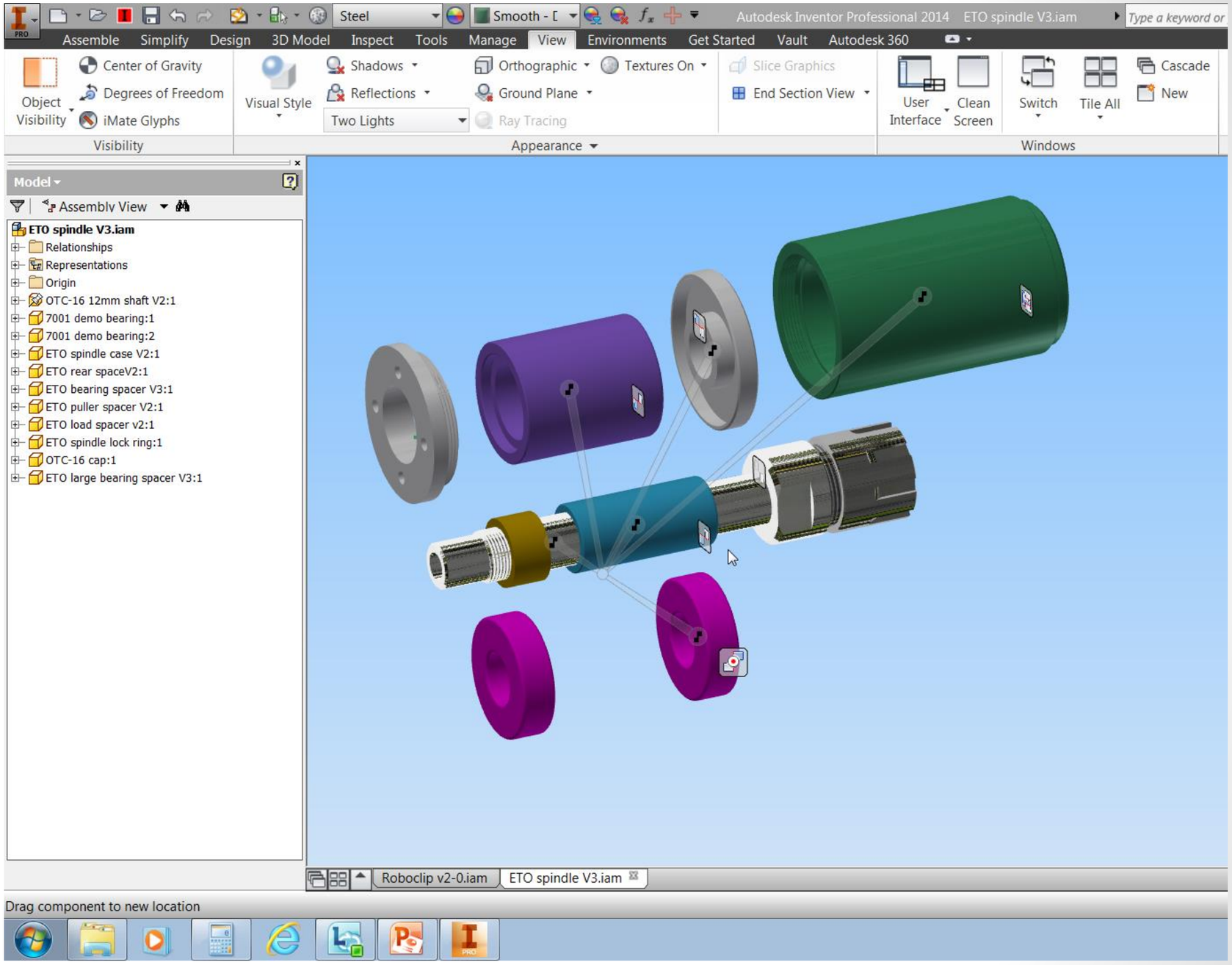
D: Free Move tool



D: Free Move Tool

RMC any part, select the “Free Move” tool. Now you can move parts out of the way. Constraints are not lost, just temporarily suppressed. When you turn the tool off and move any part, they all snap back in place.





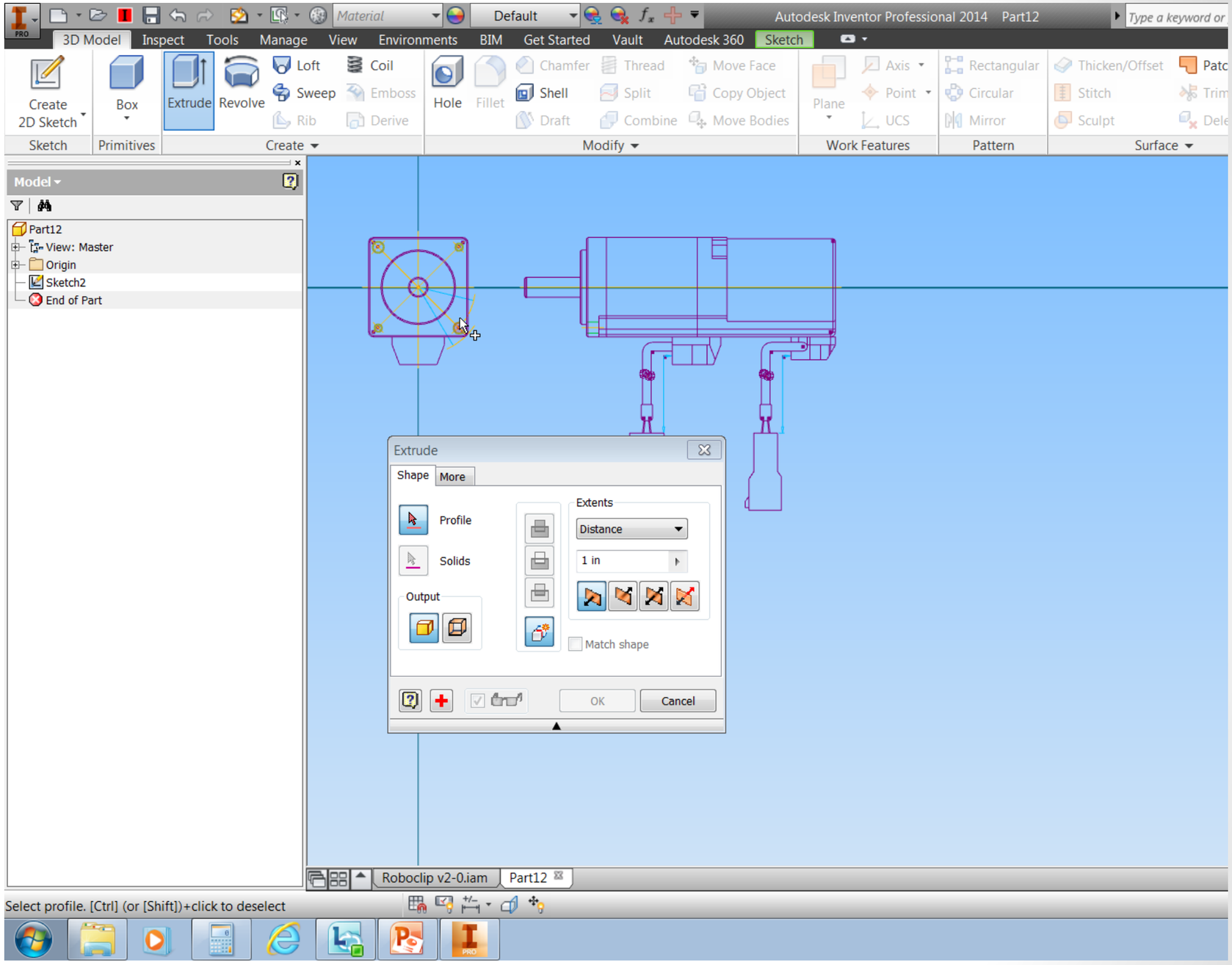
You have brought a DWG into a part sketch. When you go to extrude features you find that what appears to be closed loops are not. There are no constraints at all. Every line and arc is on its own. What went wrong?

A: Get an intern to spend hours closing loops and constraining. Productivity takes a hit, but your company is helping with job creation.

B: Import settings need to be configured.

C: Export from Autocad all on one layer, they will automatically constrain when placed in the sketch.

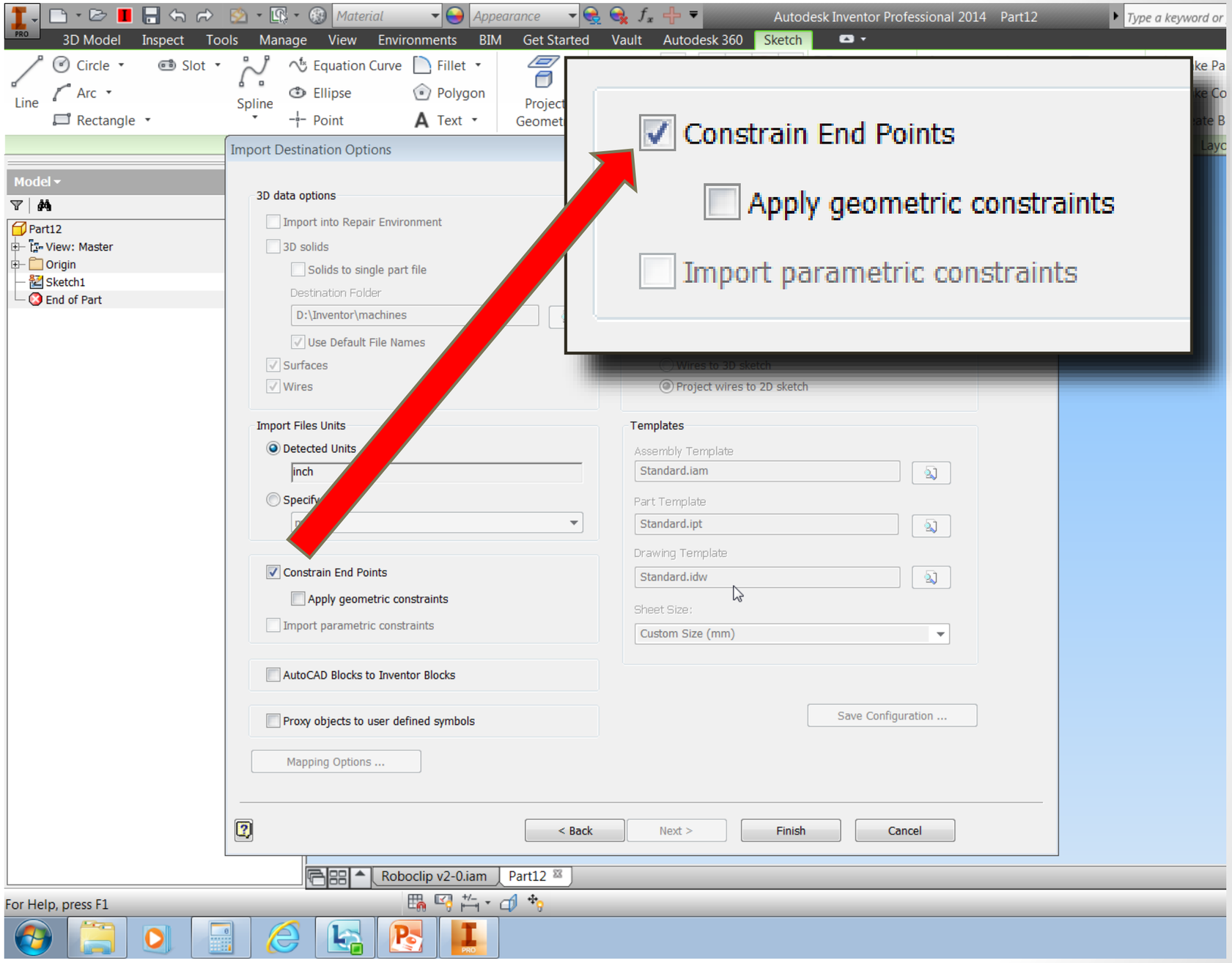
D: Document settings. Turn on the Auto connect with the parasitic auto processing doohicky.



B: Import settings need to be configured.

After you select the DWG file for import, the dialog box shown on the right comes up.

There are a number of check boxes that will make your imported data more usable. In particular, the check box shown will solve most of the closed area issues.



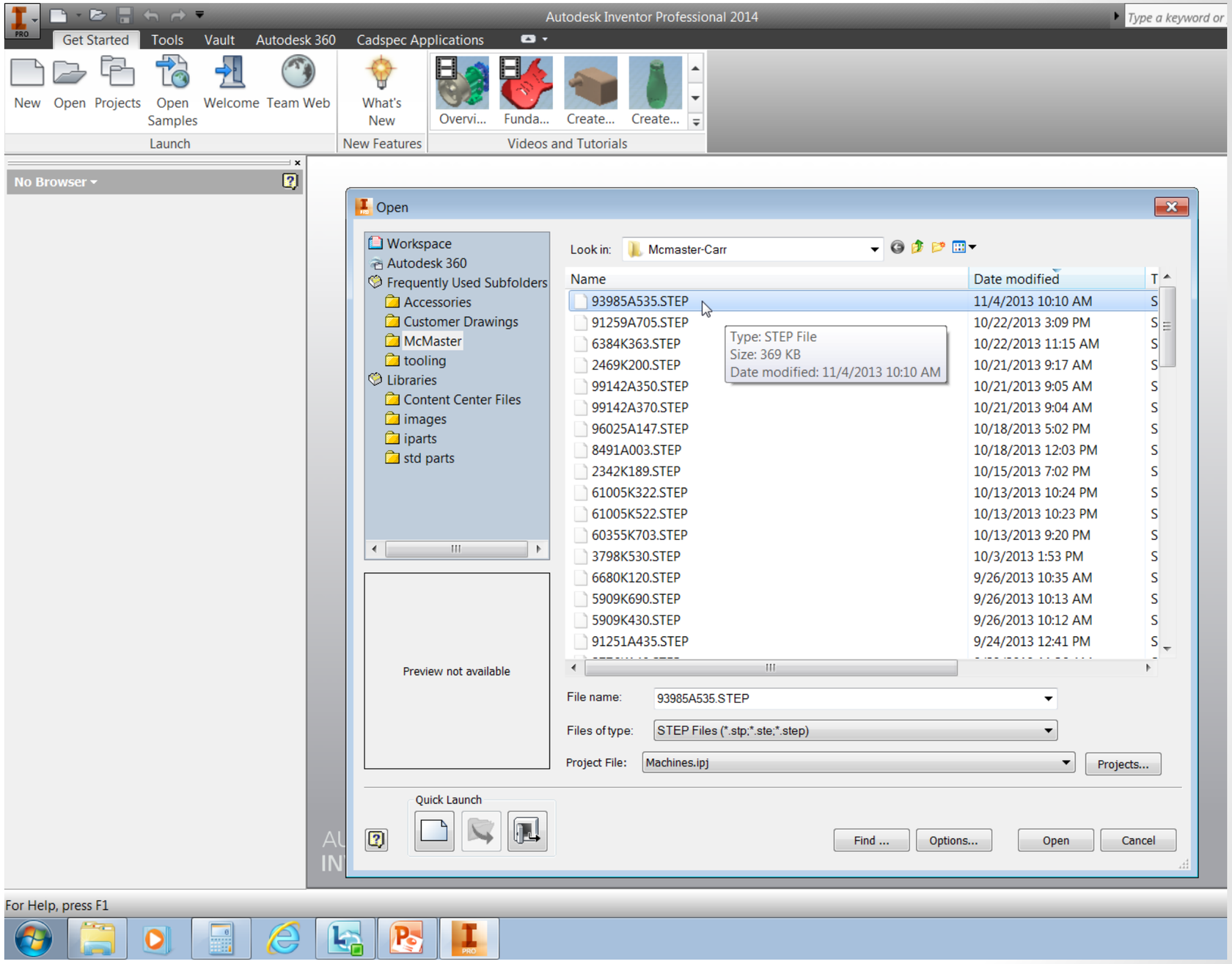
I have a .step file from a vendor. When I open it and Inventor creates an .ipt or .iam, where do they end up?

A: Where the .step file was.

B: Location that I choose.

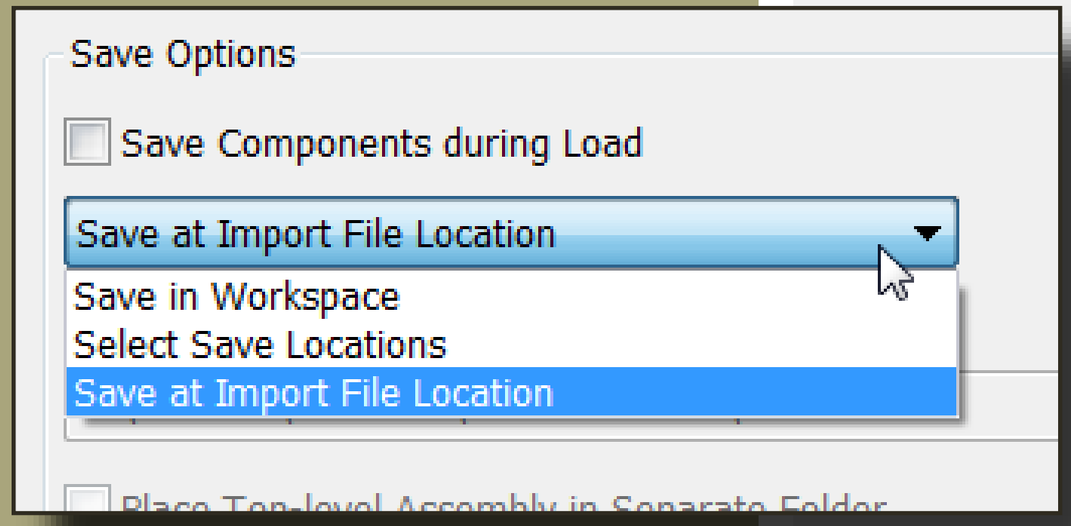
C: In the Workspace.

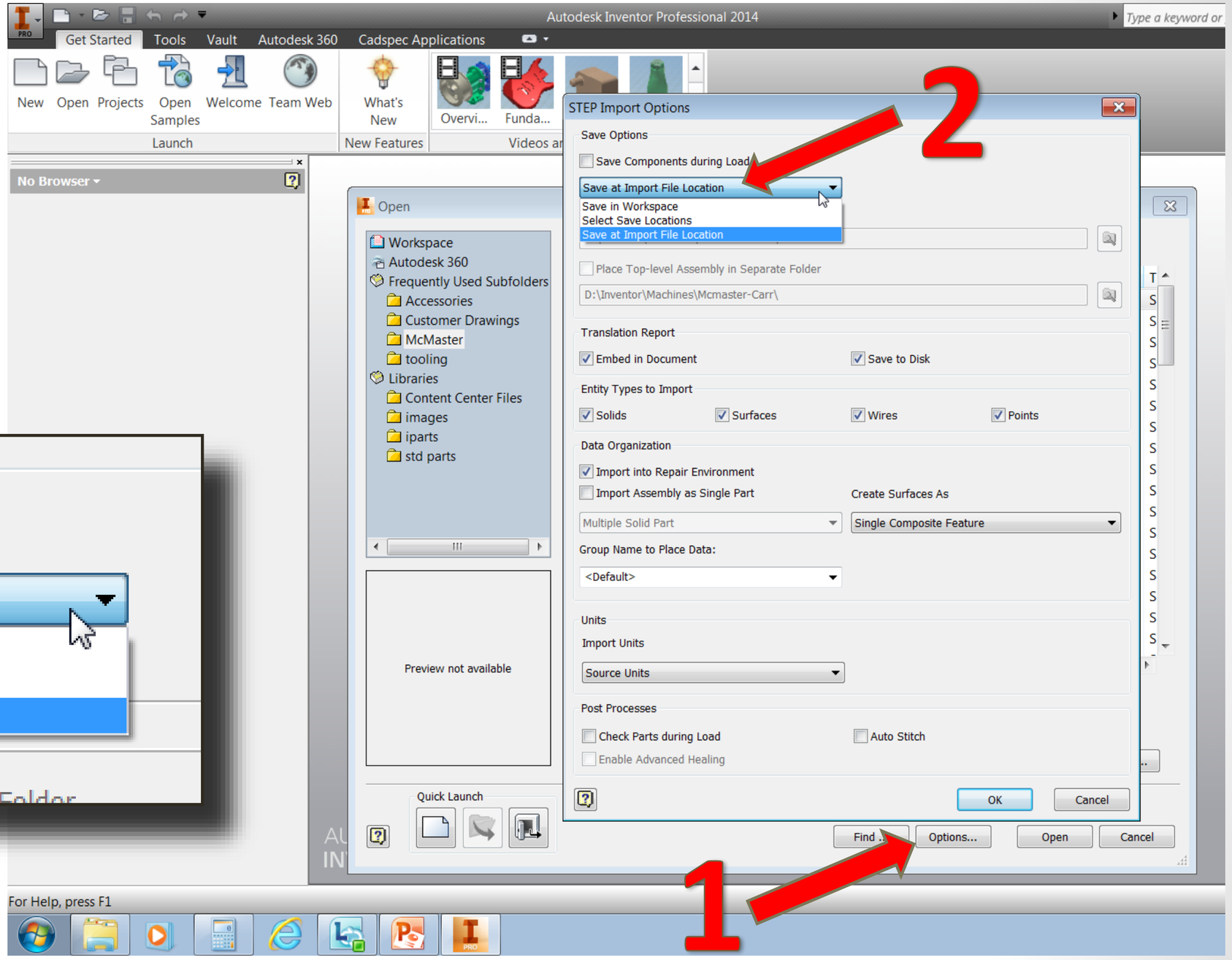
D: This was a trick question, it can be any of the above.



D: This was a trick question, it can be any of the above.

After you select the file to be opened, the STEP Import Options dialog box opens. Select the options. Now you have a choice where the files will go.





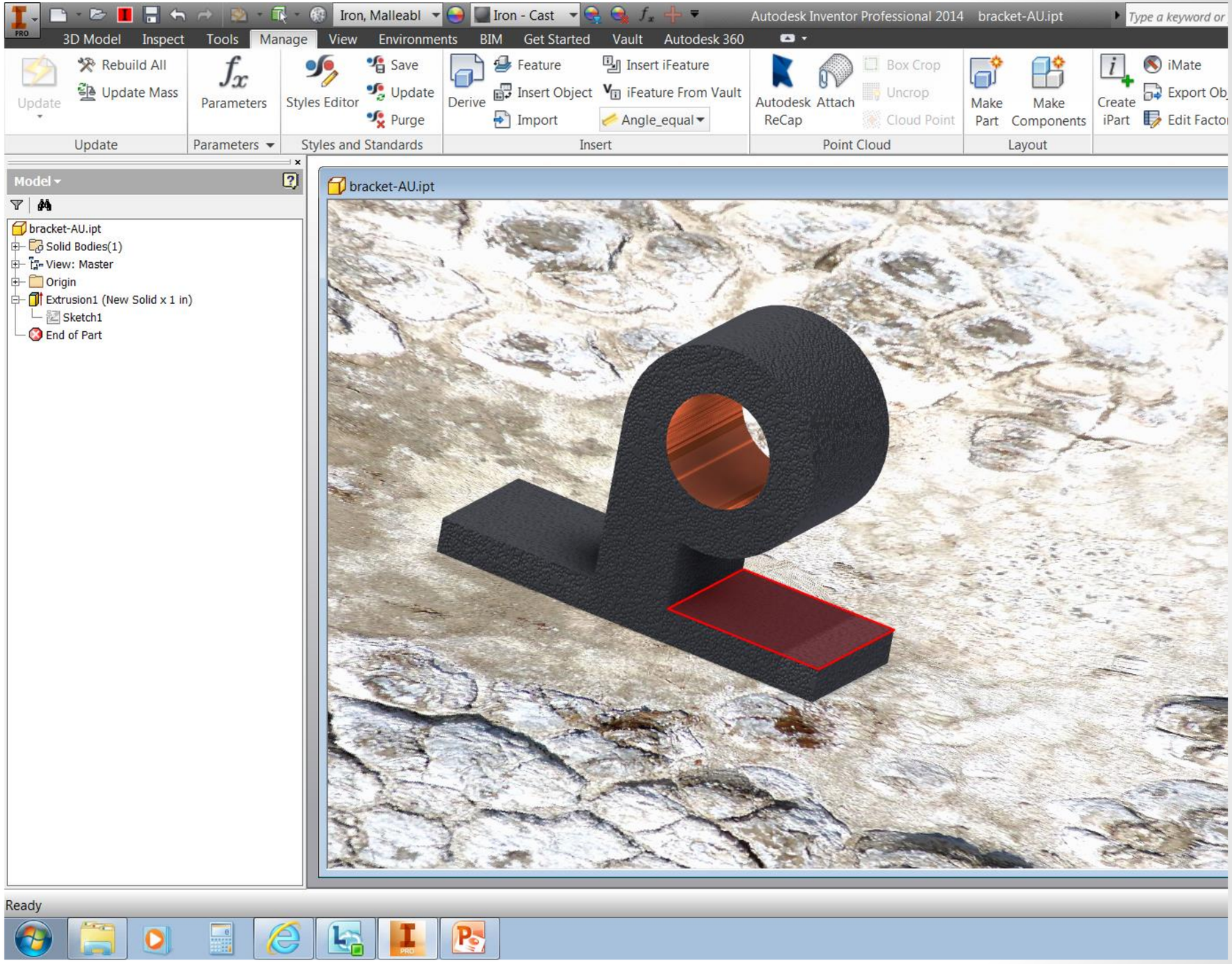
You have a part that has the a limited number of variations. Like this support bracket. Instead of drawing the part over and over for different cases, what could you use to get it done quickly?

A: iParts

B: iMagic

C: I Don't know

D: iDesign

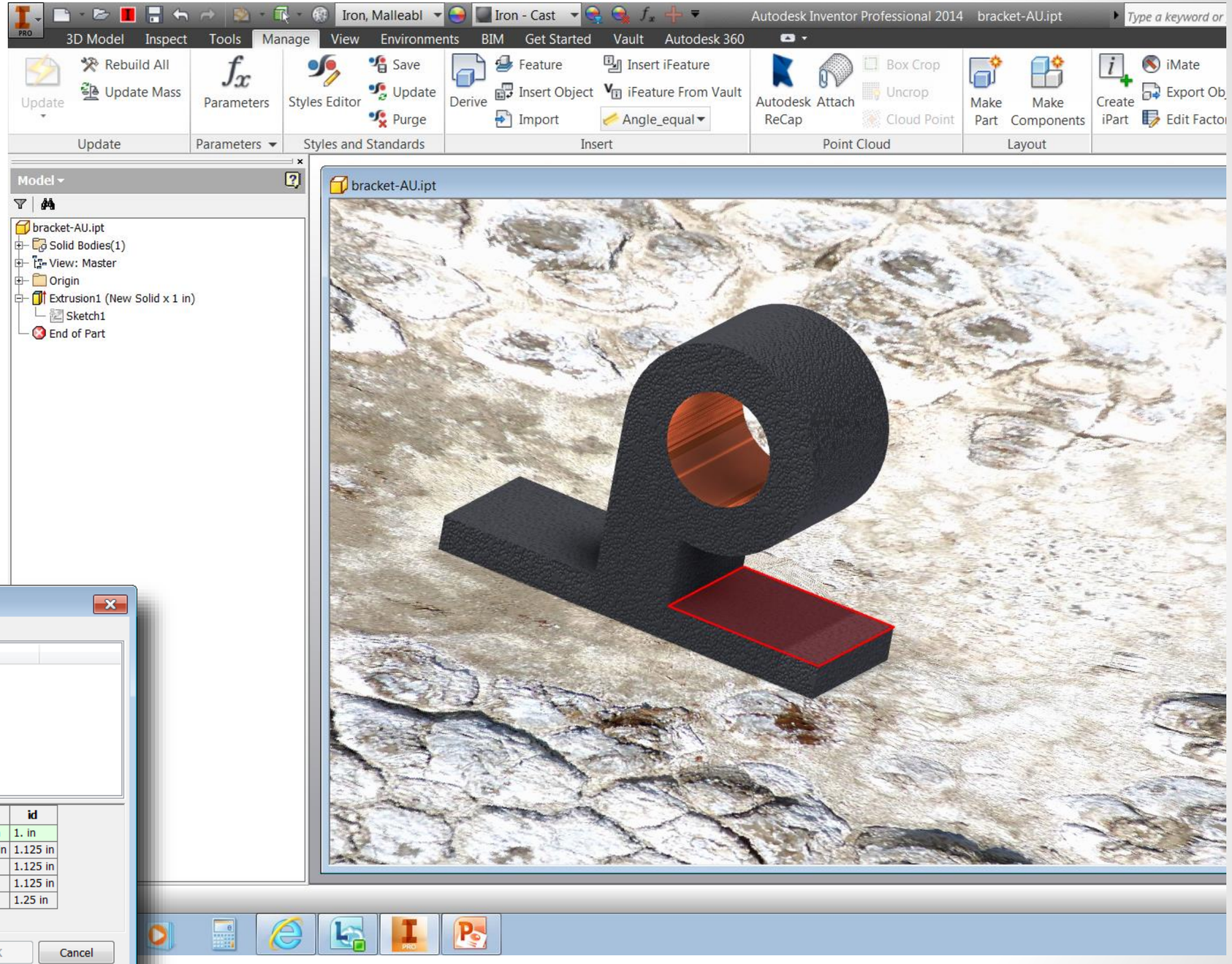


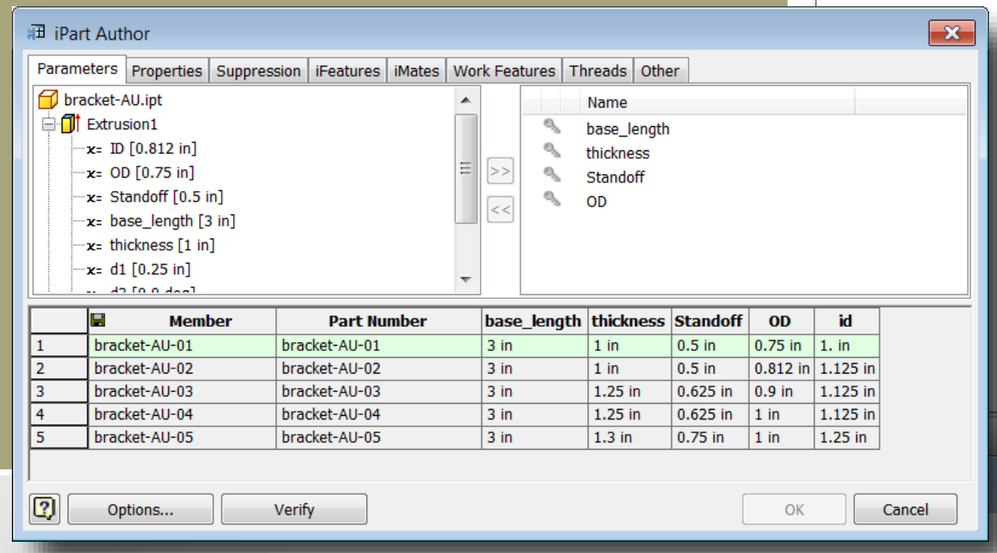
A: iPart

While iDesign and iMagic do not yet exist, iPart is one of my iLove tools.

One common question I get is when to use iPart vs iLogic

If you have a smaller set of parts that have limited number of variations iPart is my choice. If the set size is large with many variables and children, iLogic is better suited.





	Member	Part Number	base_length	thickness	Standoff	OD	id
1	bracket-AU-01	bracket-AU-01	3 in	1 in	0.5 in	0.75 in	1. in
2	bracket-AU-02	bracket-AU-02	3 in	1 in	0.5 in	0.812 in	1.125 in
3	bracket-AU-03	bracket-AU-03	3 in	1.25 in	0.625 in	0.9 in	1.125 in
4	bracket-AU-04	bracket-AU-04	3 in	1.25 in	0.625 in	1 in	1.125 in
5	bracket-AU-05	bracket-AU-05	3 in	1.3 in	0.75 in	1 in	1.25 in

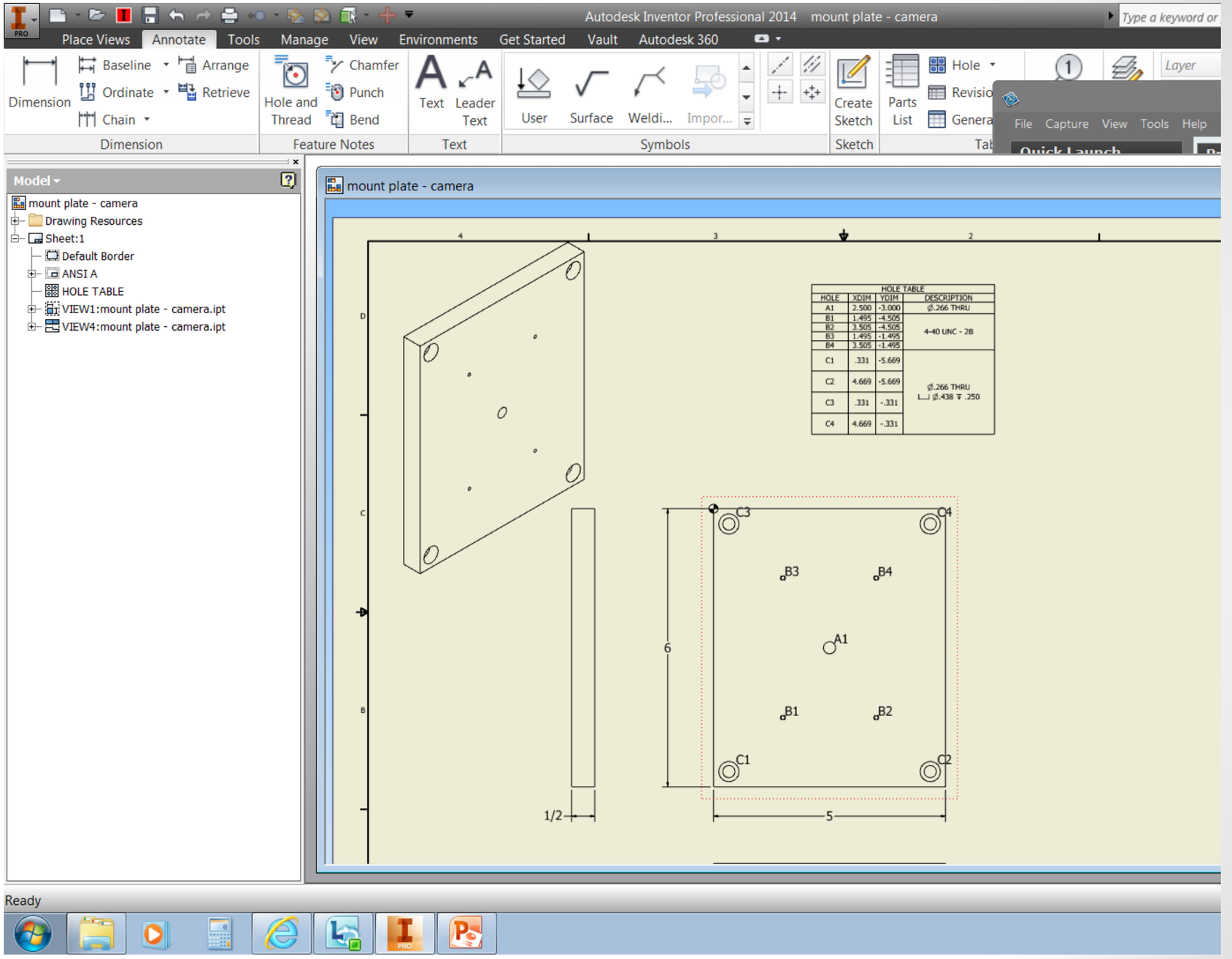
You want your hole table to look like the one on the right. Each hole type has a single description. Not line after line of the same info. How do you get this magic to appear?

A: RMC Preferences

B: Template idw/dwg setting

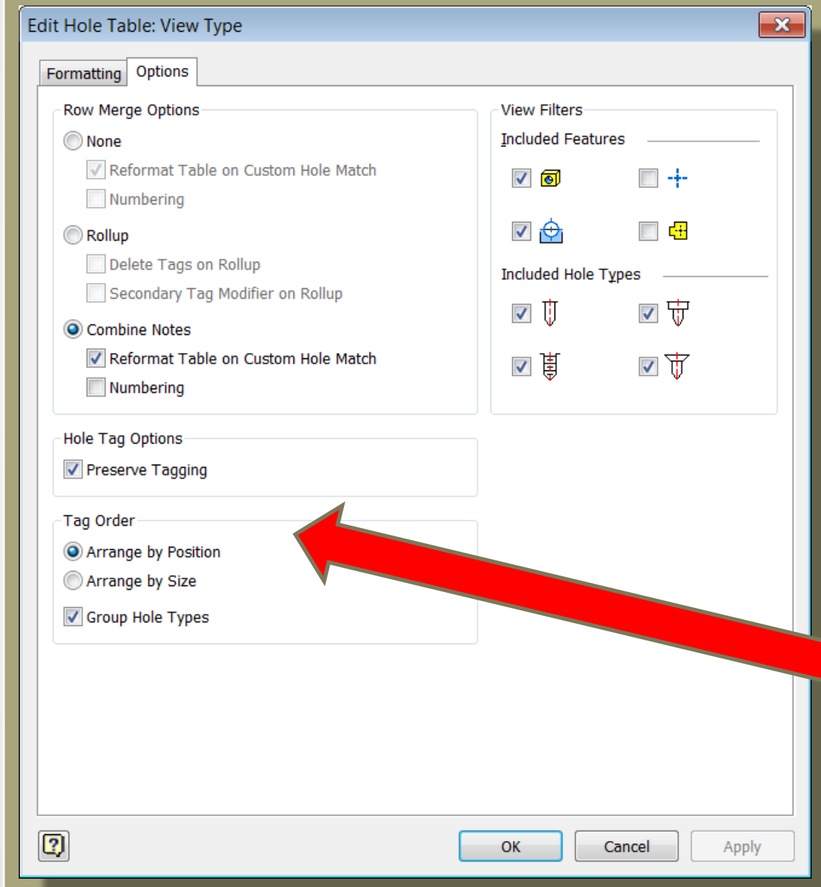
C: Styles settings - Tables

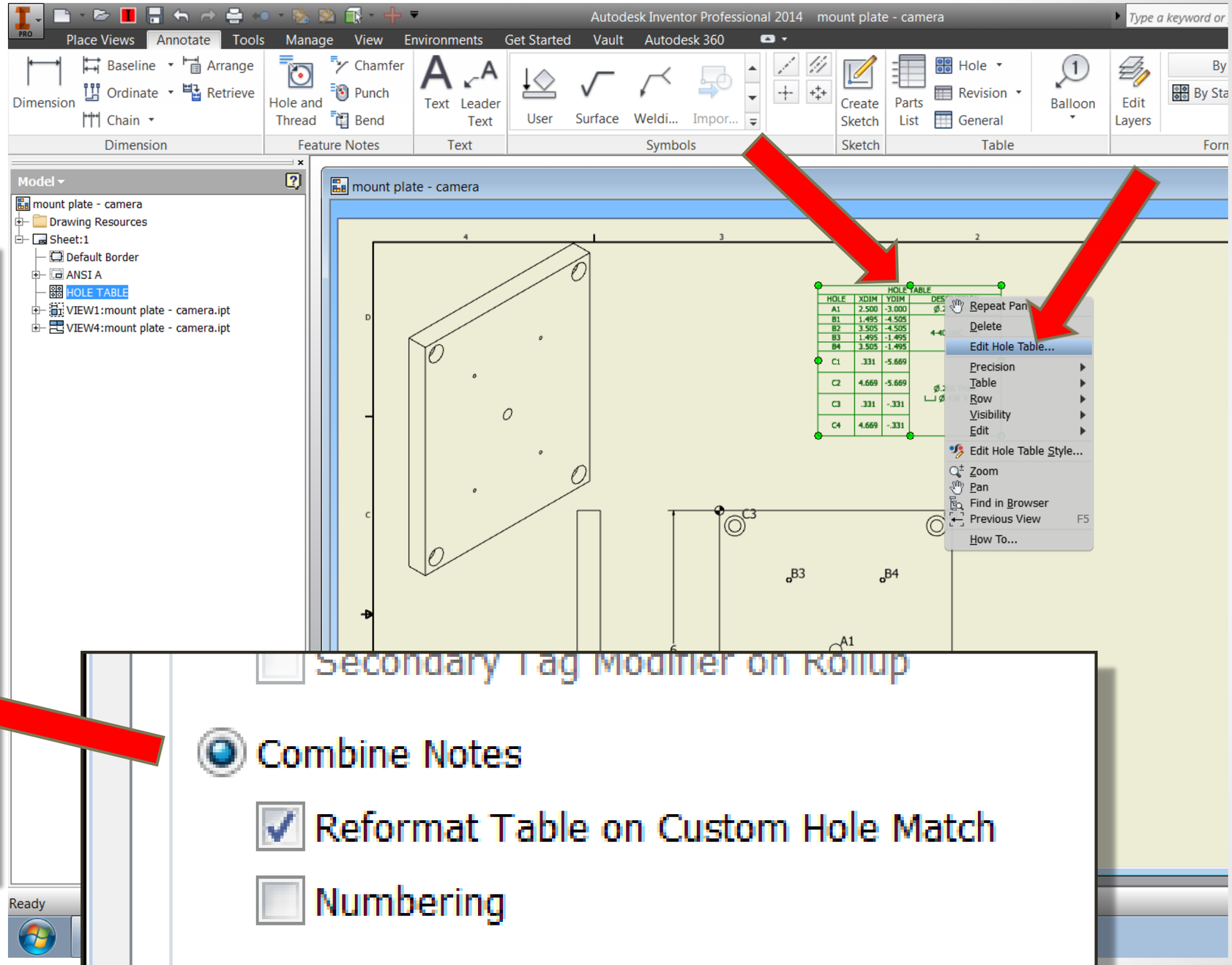
D: Hole table setting

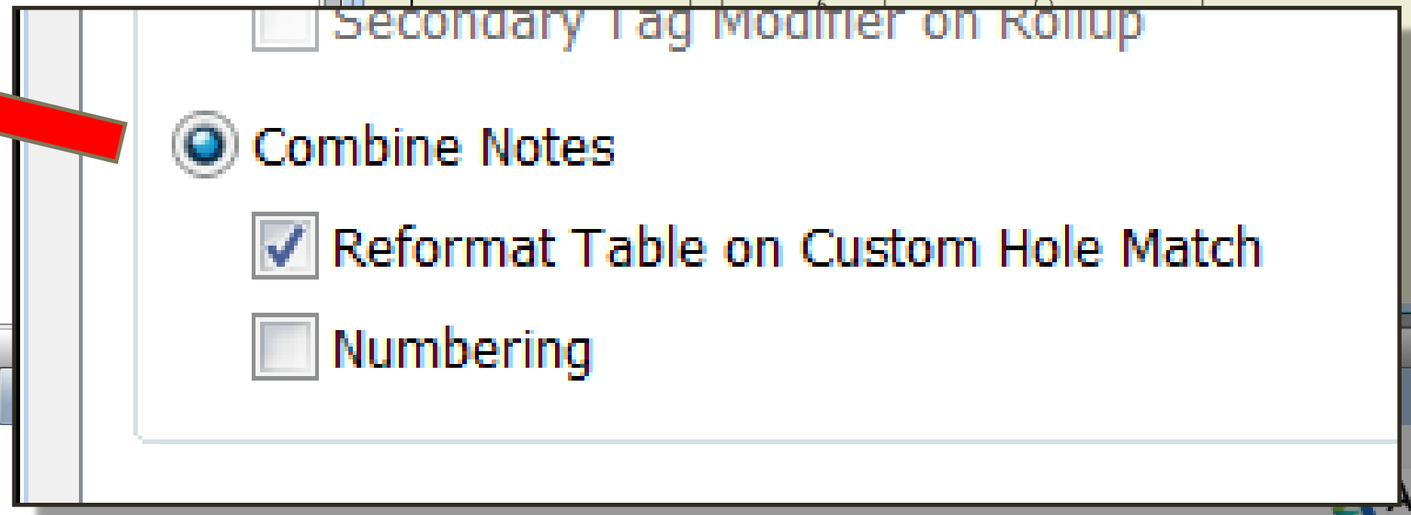


D: Edit the Hole Table

RMC on the table, then go to Edit Hole Table. In this case I chose “Combine Notes”. See below.







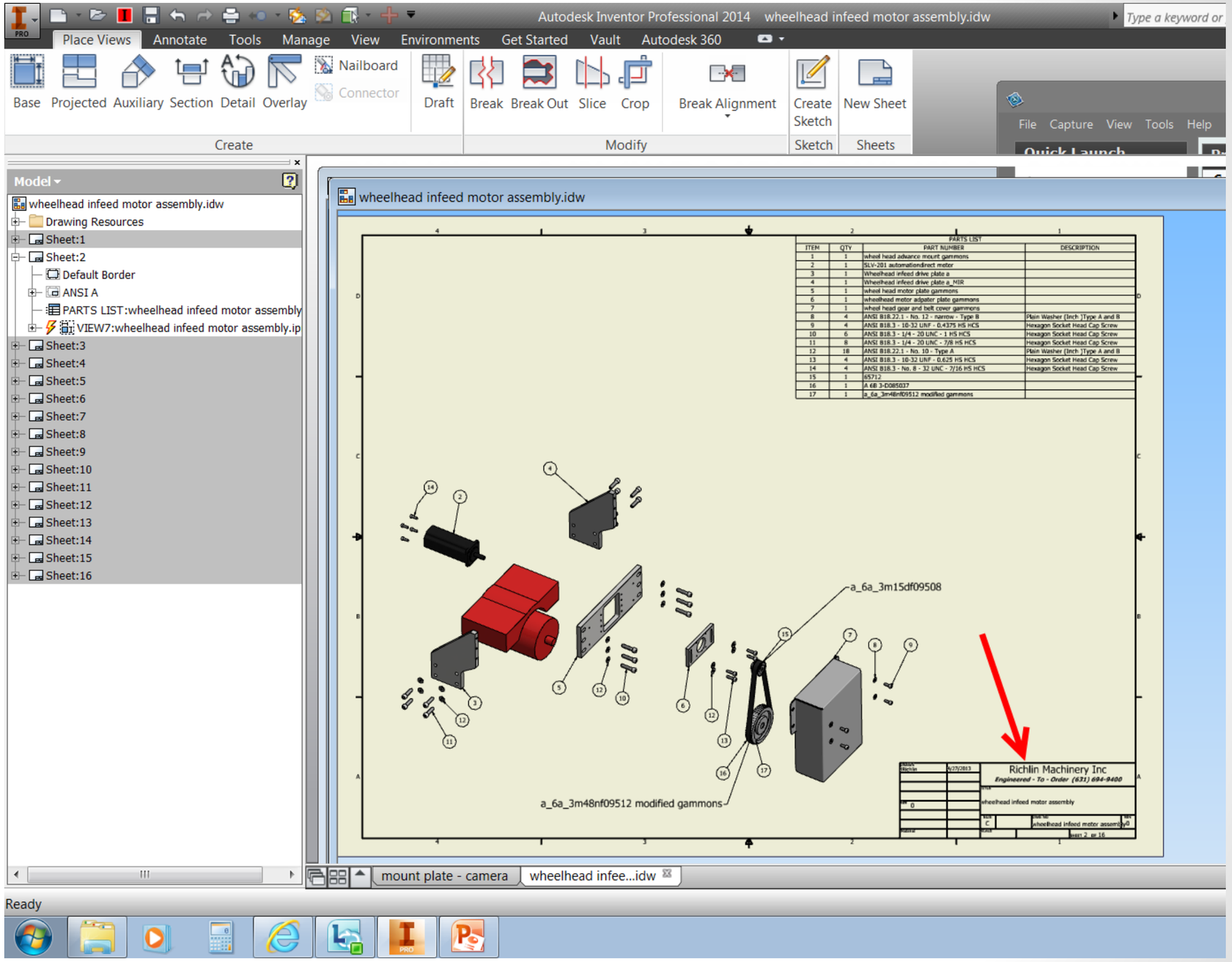
The IT at your firm got a great price on a new phone carrier, the only issue is the area code is different. Do you make the intern change all the legacy idw and dwg's? Or do you show off and get it done in one amazing feat?

A: Make the intern pay! Have him work for months changing all of your legacy document.
(if only this was an option, slave labor is banned in most states)

B: Modify the idw or dwg templates.

C: Use the Document Resources Wizard tool.

D: Vault will save the day again.

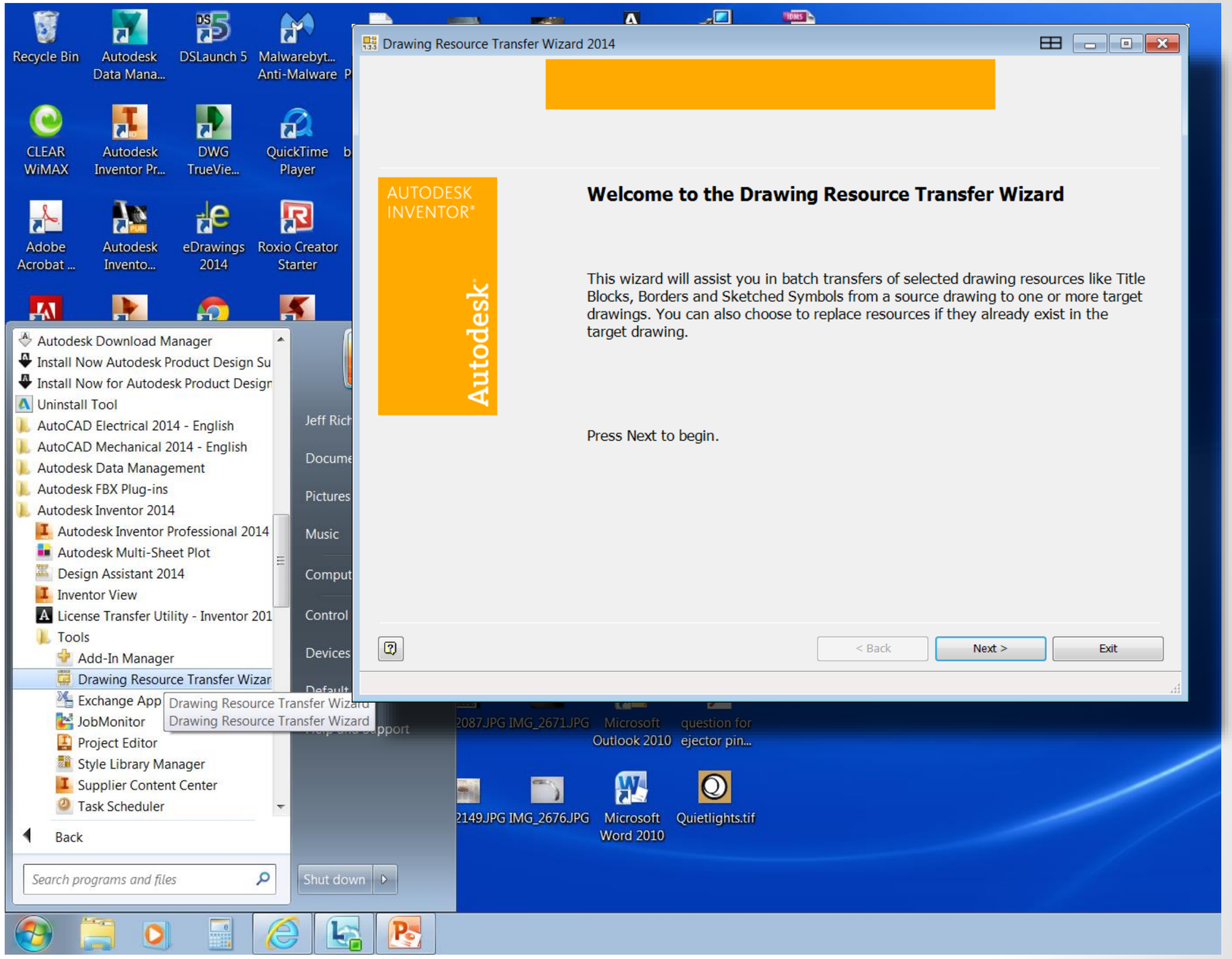


C: Use the Document Resources tool.

Believe it or not, we have had this tool available for years.

The other tools would not get it done.

Changing the template files would only have helped with future files. Not ones already created.



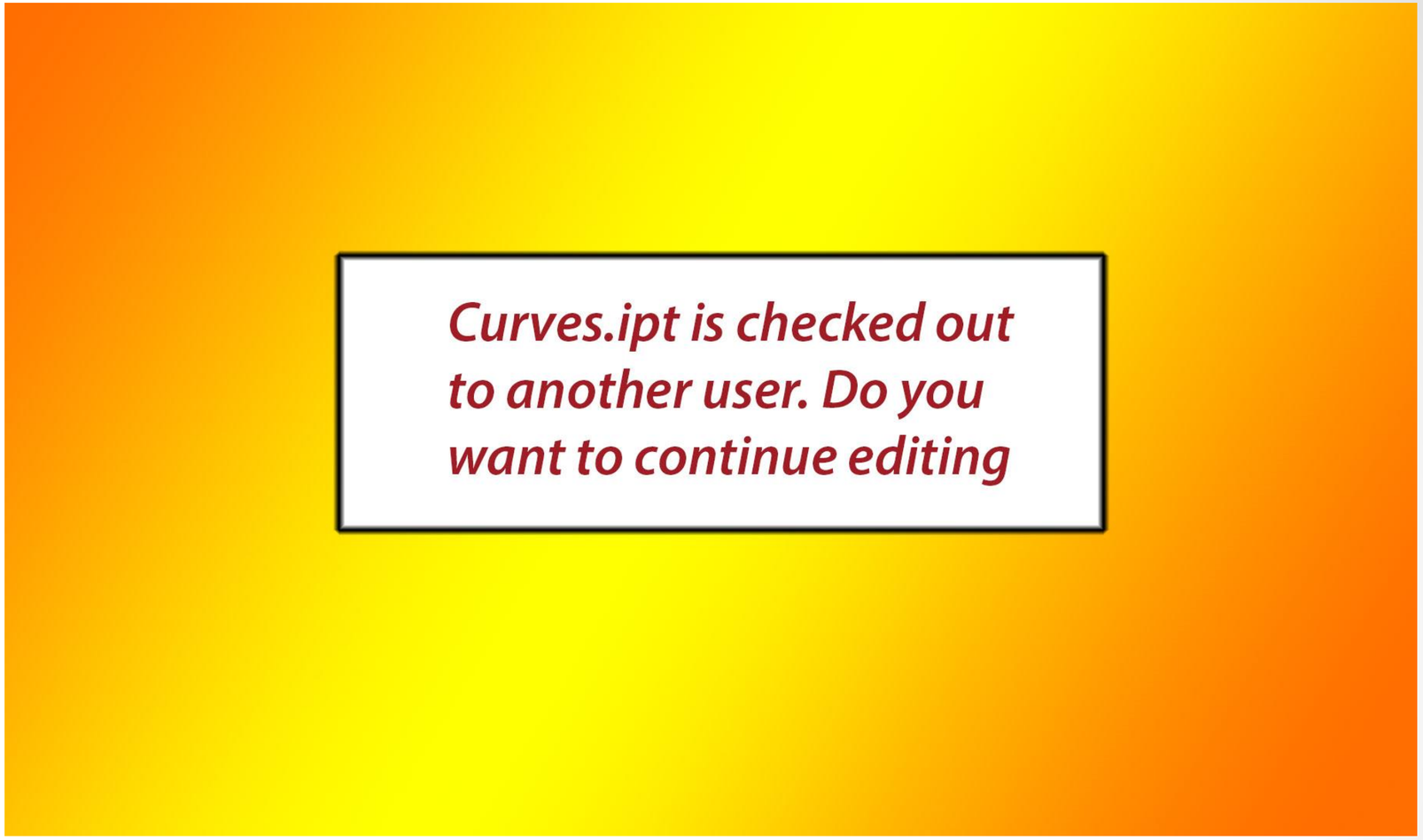
Who do you blame if you see this?

A: IT, they messed up again and moved the vault.

B: Your CAD manager. He made a change to everyone's project file and "pooched" the settings.

C: Fred was crazy busy yesterday while trying to get away on his vacation. We are now finding what havoc he has left behind.

D: Your wife, she was not happy with the Curvy part.



C: It is Fred's fault. He checked out files and never checked them back in again. We all hope it rains on his vacation!

While the other choices would all have given you problems. The key here is that the file is checked out, not that it can not be found.

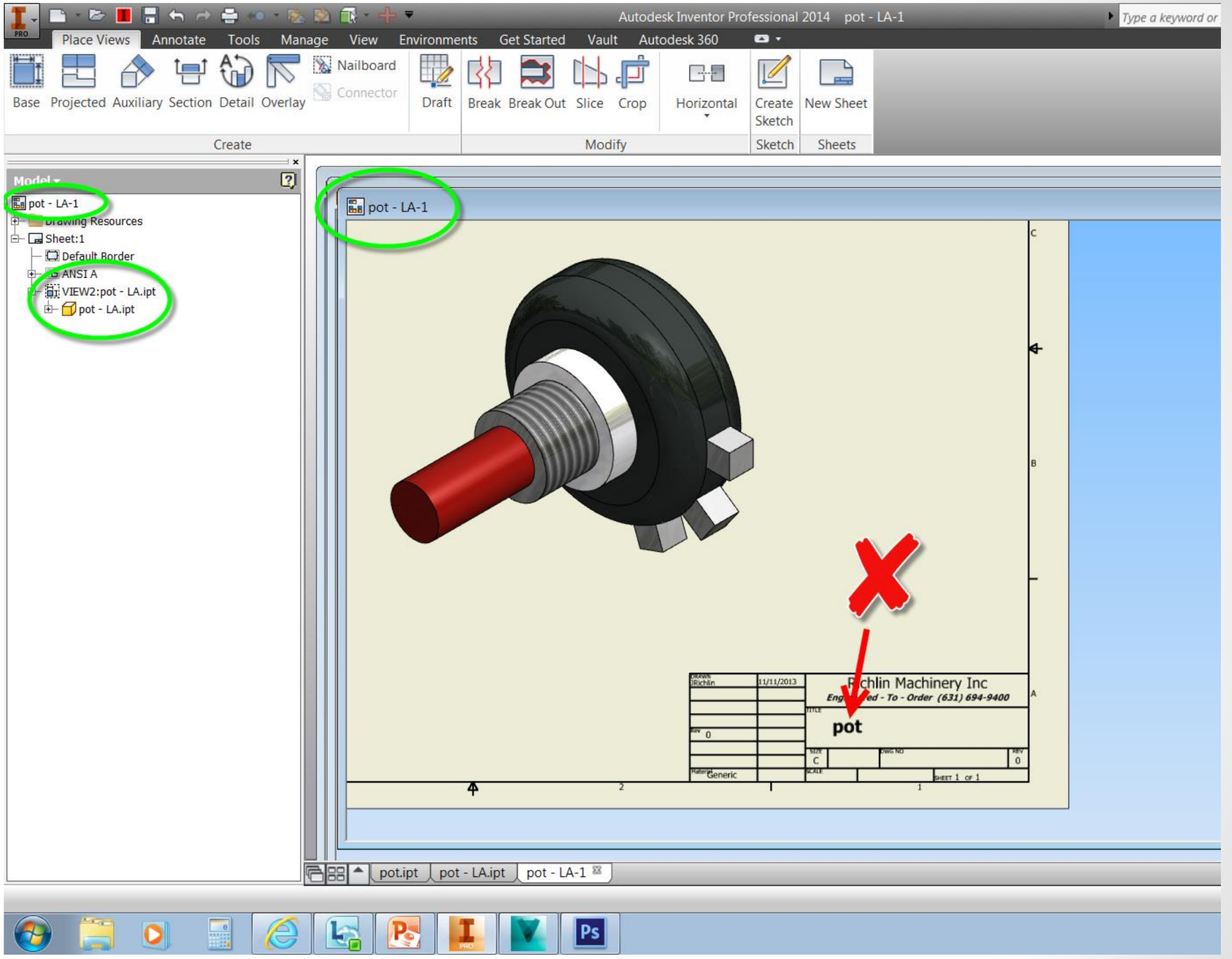
Vault has been used to copy a design. I had a pot.ipt that I copied to pot-LA.ipt However something messed up. I see on my drawing it is still named *"pot"*. Also in my parts list the old name lingers on. What do I do? What went wrong?

A: After copying a design, you must go into the iProperties and change the part number.

B: The settings in the Vault copy design dialog box needs attention.

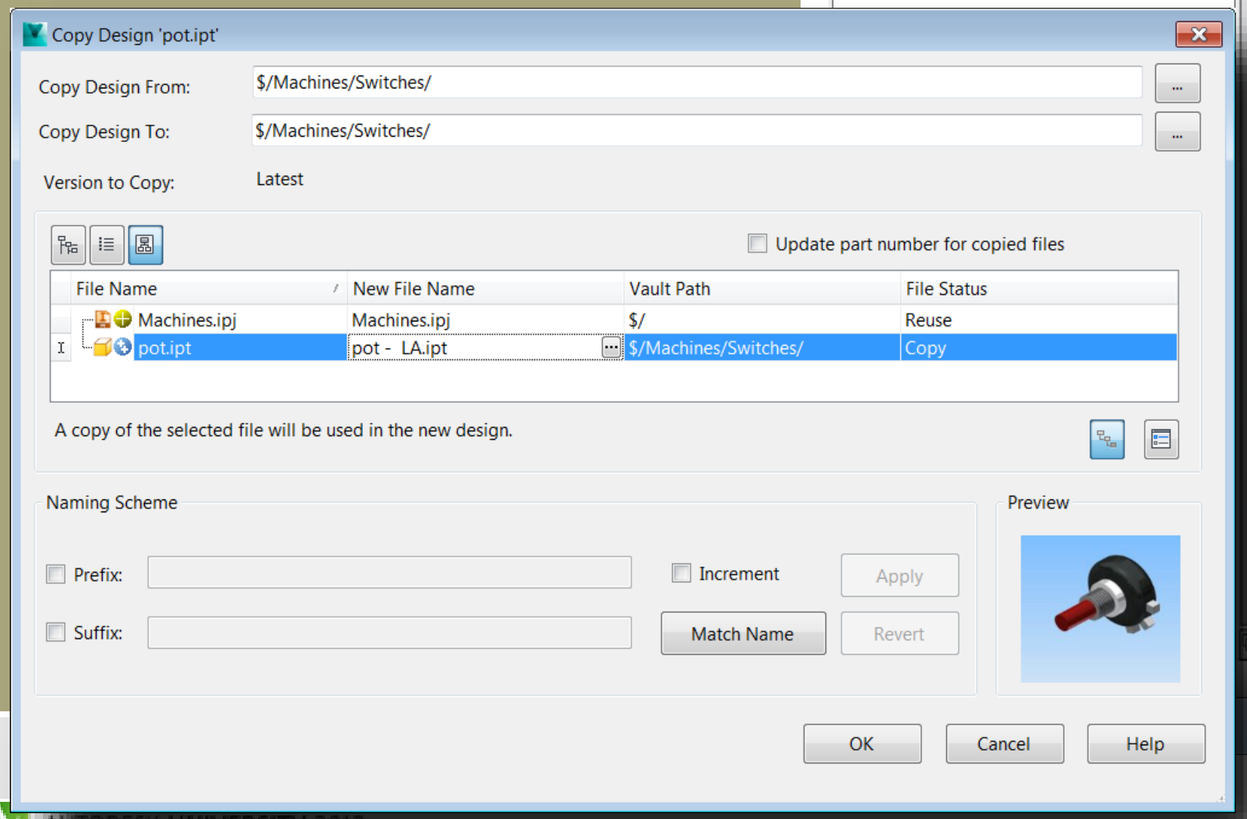
C: It is a setting in the application options dialog box. Need to find the time to find it and fix it.

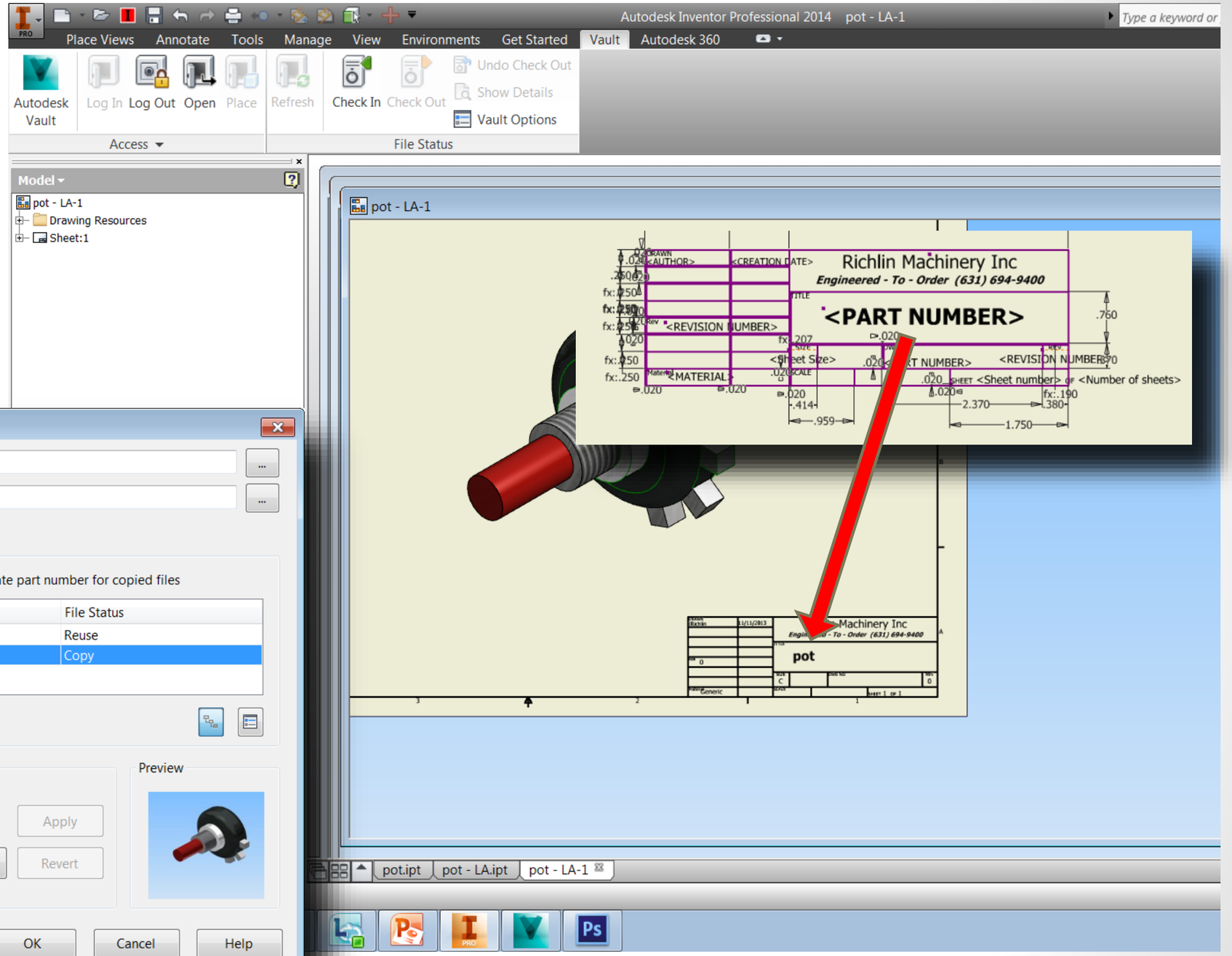
D: By the time this is found out you are already home and can't do anything about it until tomorrow, Nuts.



B: The settings in the copy design dialog box needs attention.

If this has ever happened to you I really feel your pain. This took me hours to figure out. It was after 2am and my reseller warned me that if I called him again after midnight he would block all my phone and email privileges.





AUTODESK.

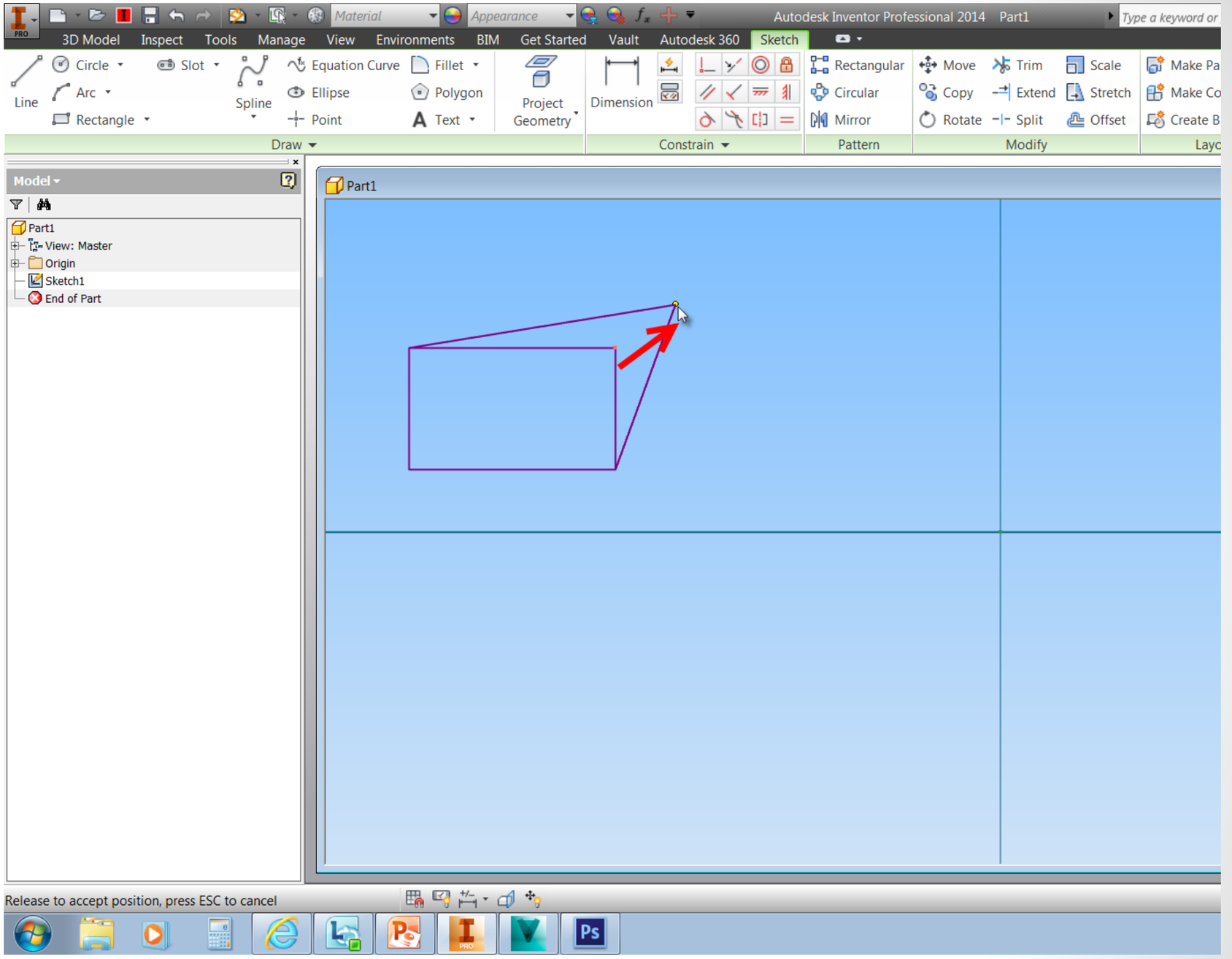
Out of nowhere my sketching tools don't work the way they used to. I draw a rectangle but it does not stay as one. I can grab a corner and make *funny* shapes. AUUUGGGGGHHHHH! Not so **FUNNY**!!!!!!!!!!!!!!!

A: Someone is having fun at your expense and has changed your project settings

B: Someone is having fun at your expense and has changed your constraint settings

C: Someone is having fun at your expense and has changed your application options

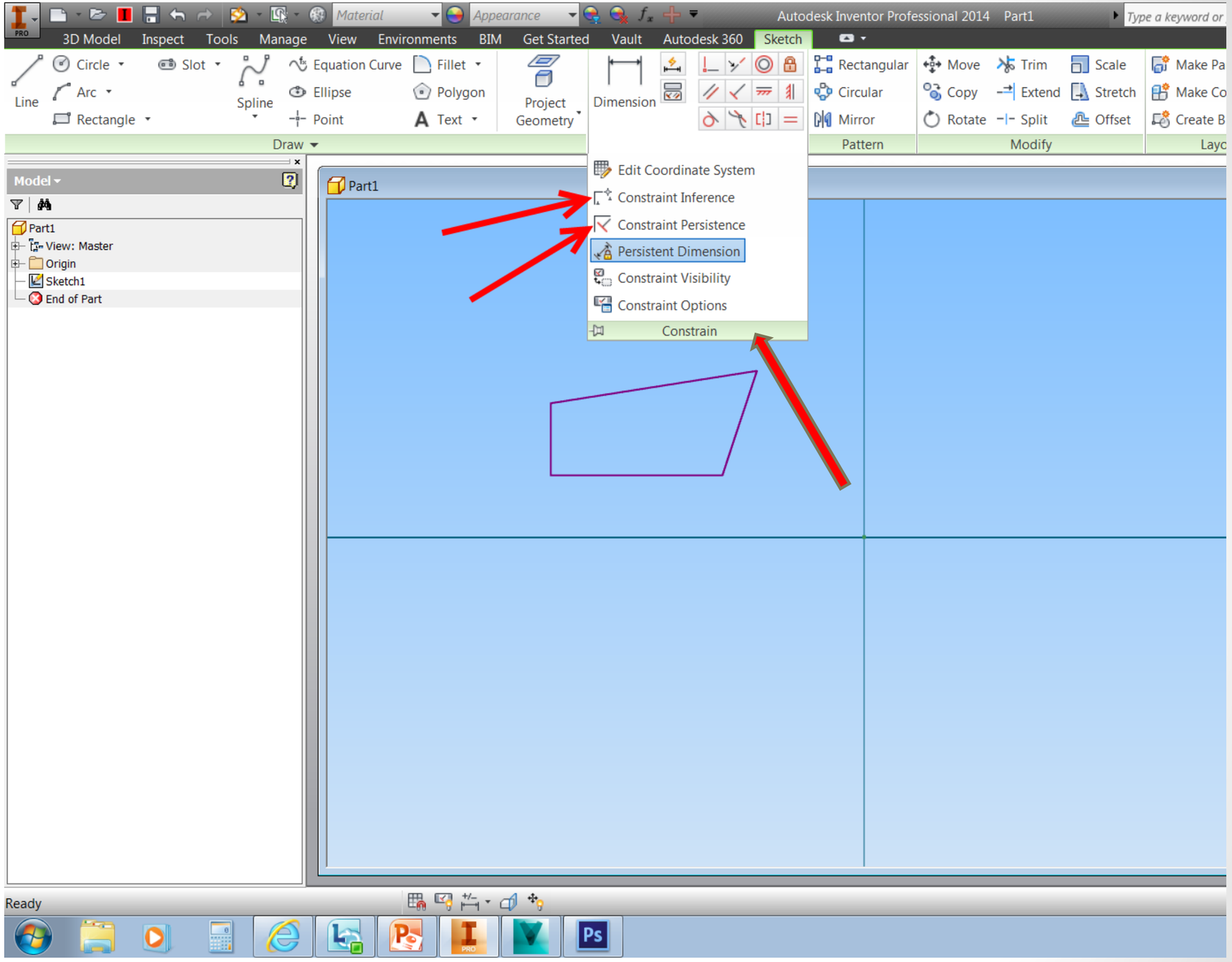
D: Someone is having fun at your expense and has changed your drawing template file.



B: Someone is having fun at your expense and has changed your constraint settings

OK, who ever hid these buttons is a sadist. The first time it happens you think you have lost your mind. After trying all else (including reloading your software) you call your reseller or CAD manager, they will have a good laugh at your expense.

Drop the Constrain tab down, turn these two states back on, have a drink, go home. Tomorrow is another day.



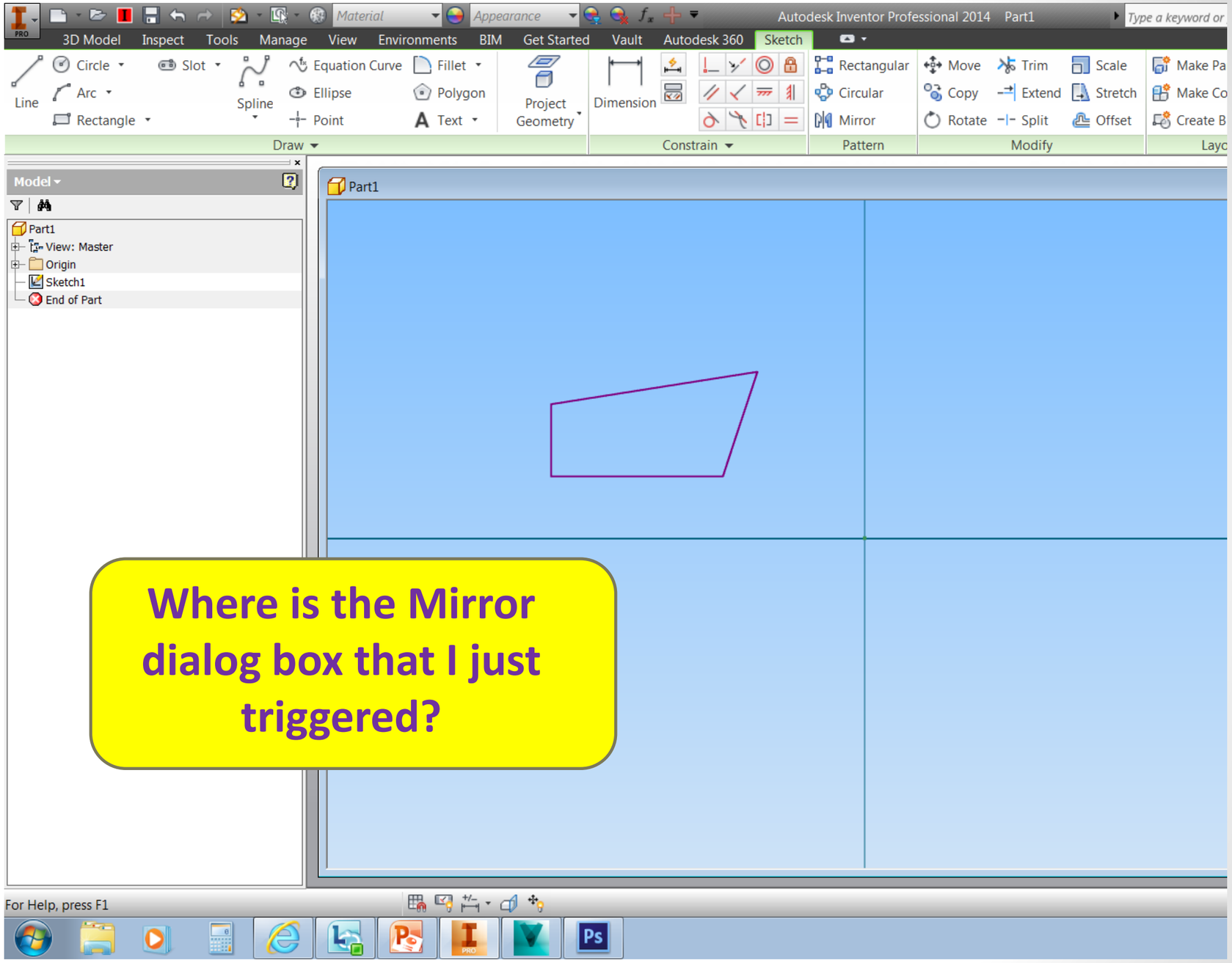
I have been working on my laptop with a projector or dual screen setup. When I go home I find out that dialog boxes are coming up where I can't see them. Effectively I am becalmed. How do I get the dialog boxes back into in my area of view??????????

A: ALT-space bar

B: It is your video card that is messed up. Go to IT and demand a new one.

C: It is an Inventor issue, reload the software.

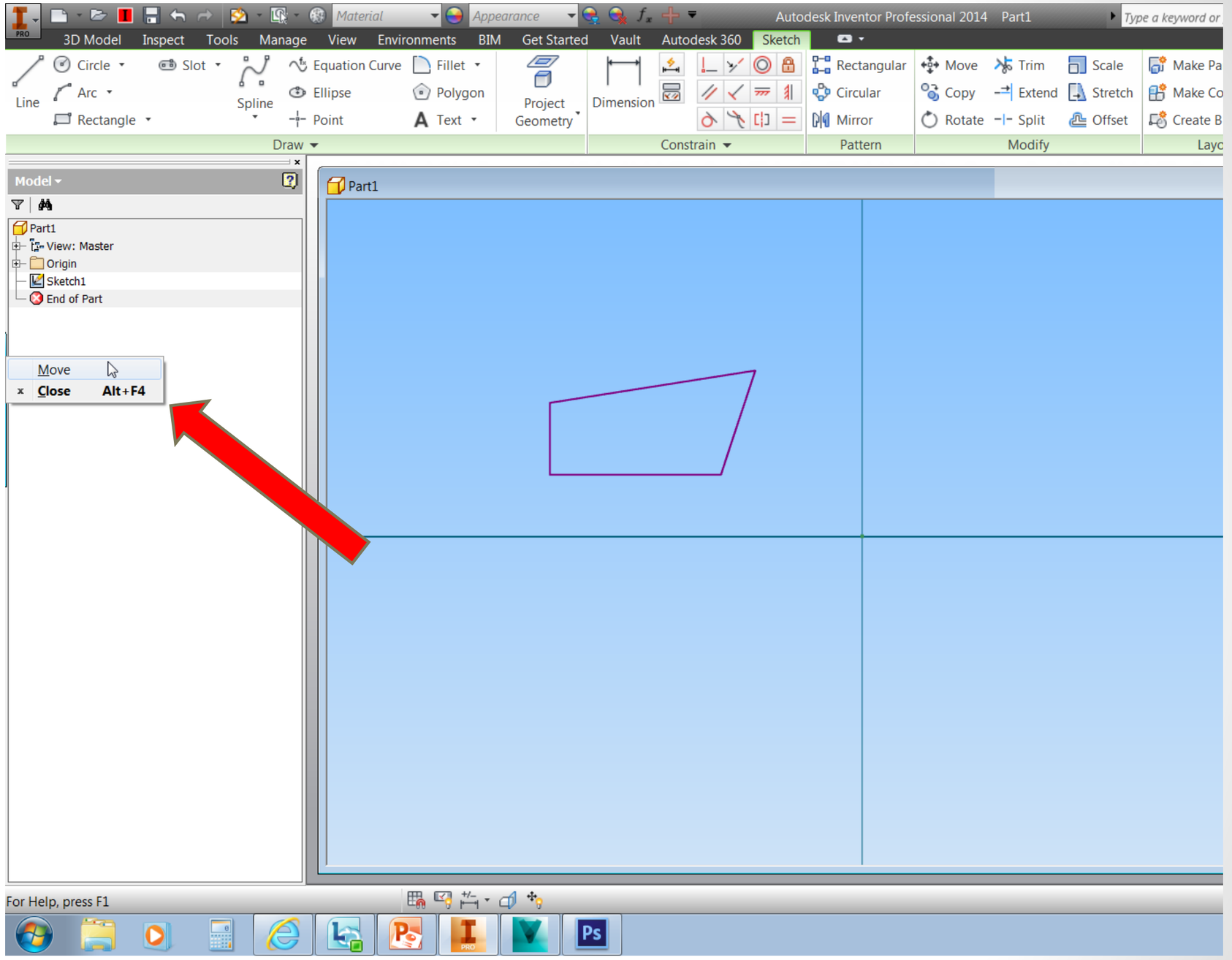
D: Log the issue with tech support, they can fix anything.



A: ALT-space bar

How many times do I wish I could have tossed my laptop into the bathtub? More than once! This was one of those times.

Just press and hold the ALT key, then the space bar. A small window will pop up if you have a dialog box off your screen. LMC the move button. Then press one of you arrow keys. The dialog box will appear under you cursor. Move it to where you want, LMC again and get back to work.



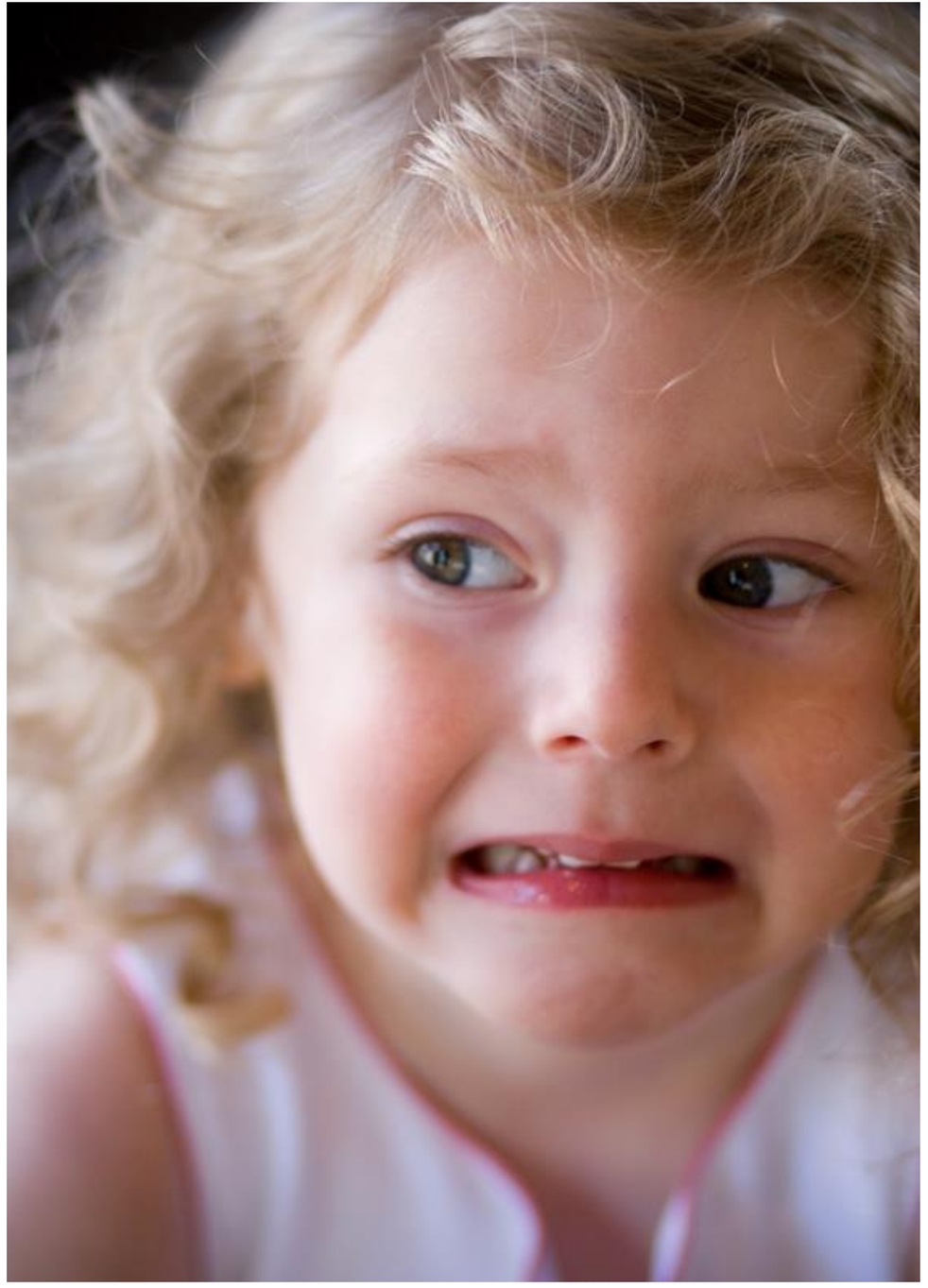
Now that the guys in logistics know we can use iProperties to help create file names, they have gone nuts. I was just asked to include the name, material, weight, size, price, delivery, designer, and warehouse location. Don't ask them why, just let them know how many characters can go into a filename.

A: Any length you want, there is no limit in windows.

B: 128 - who would want to type more?

C: 1028 - because we can.

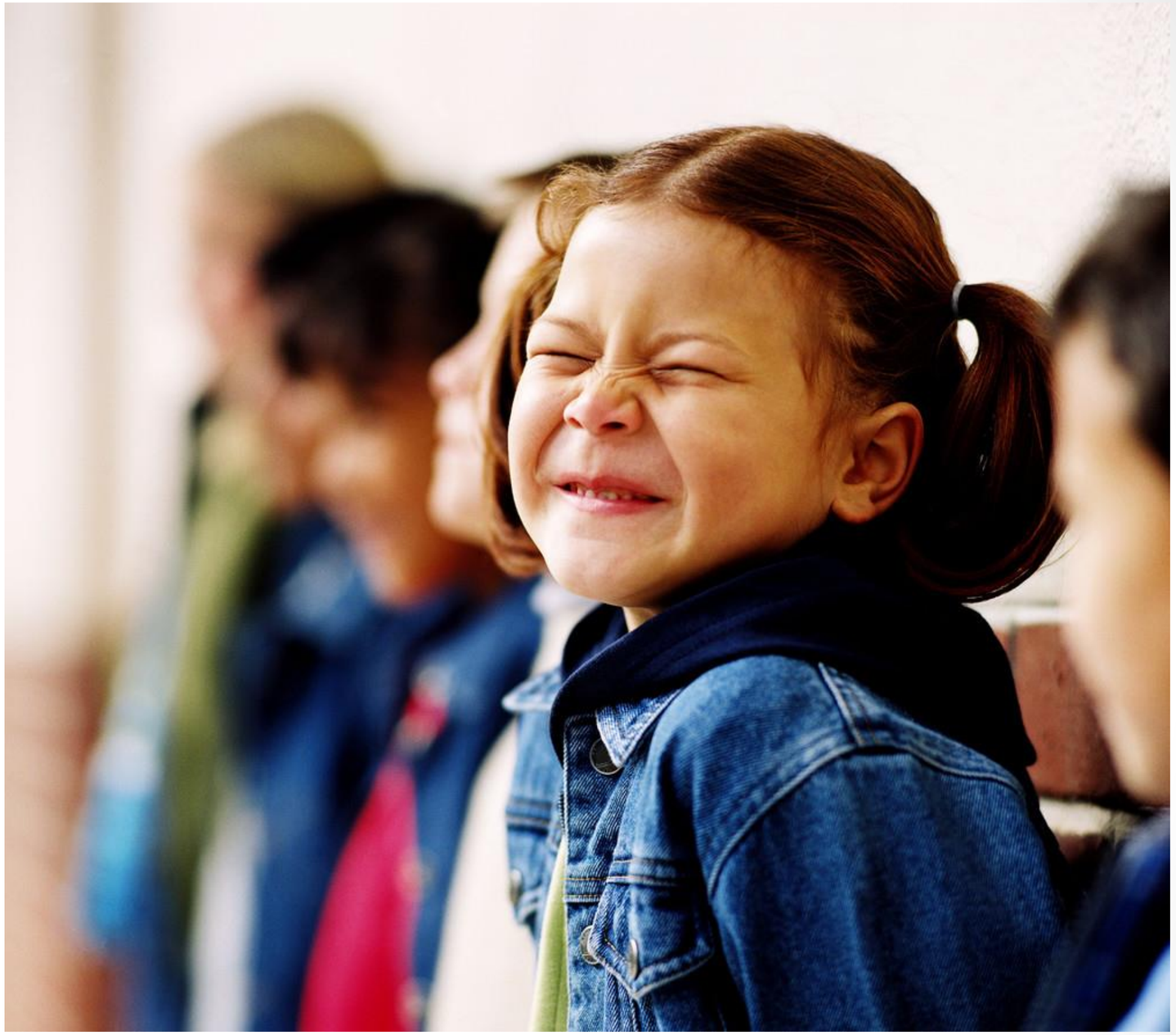
D: 256 - enough space to create a short story with a protagonist.



D: 256 - enough space to create a short story with a protagonist.

Remember, our goal is sometimes just making others happy**.**

Remember the path is included in this character count



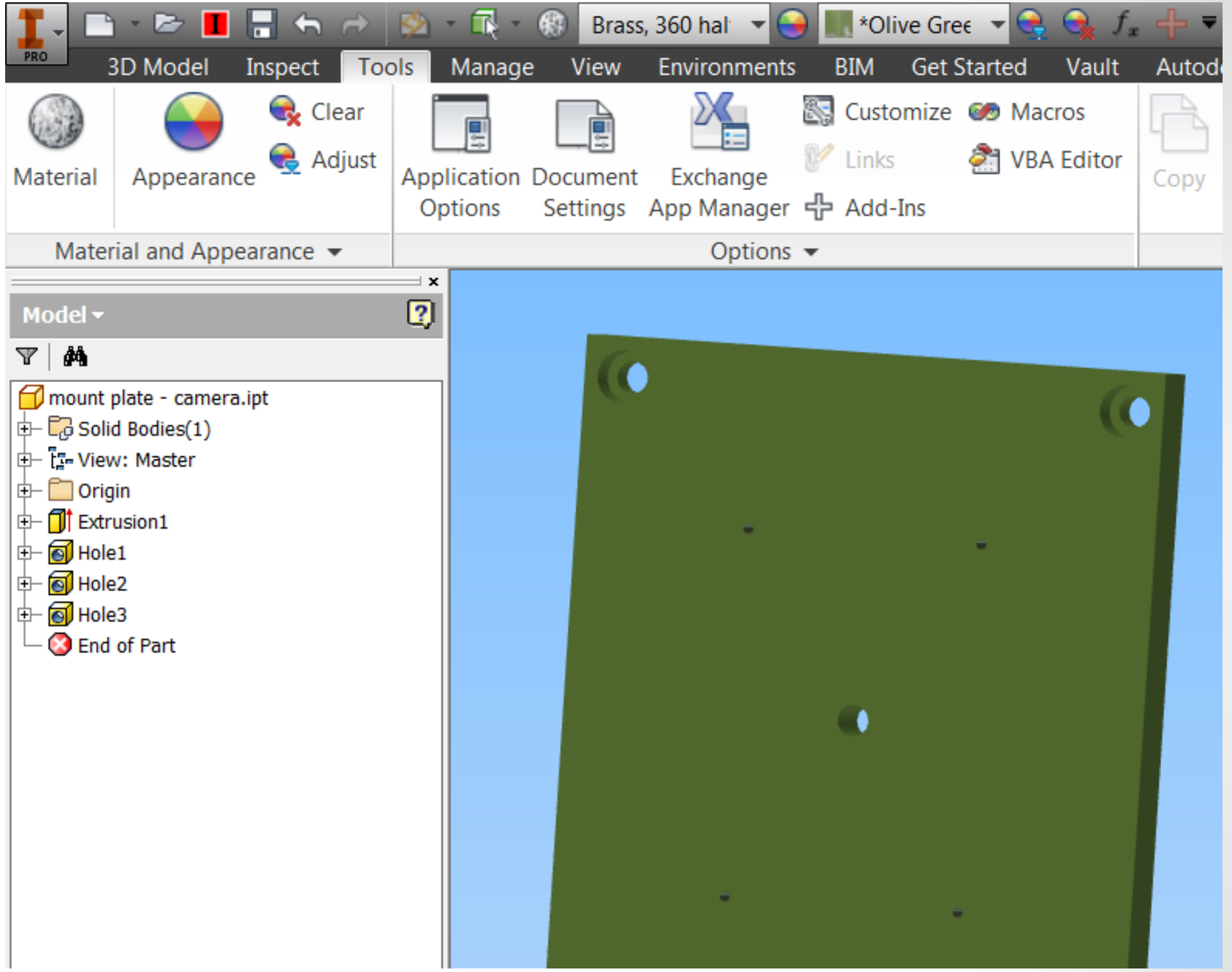
You get a part from a lazy designer. You are asked to changed the 4-40 holes to 4-48. What should the last guy have done to make this easier, or what can you do now?

A: Rename the feature from Hole # to a name that has information

B: Set your selection tool to highlight feature names in the browser

C: Hover over a feature name and see what is highlighted

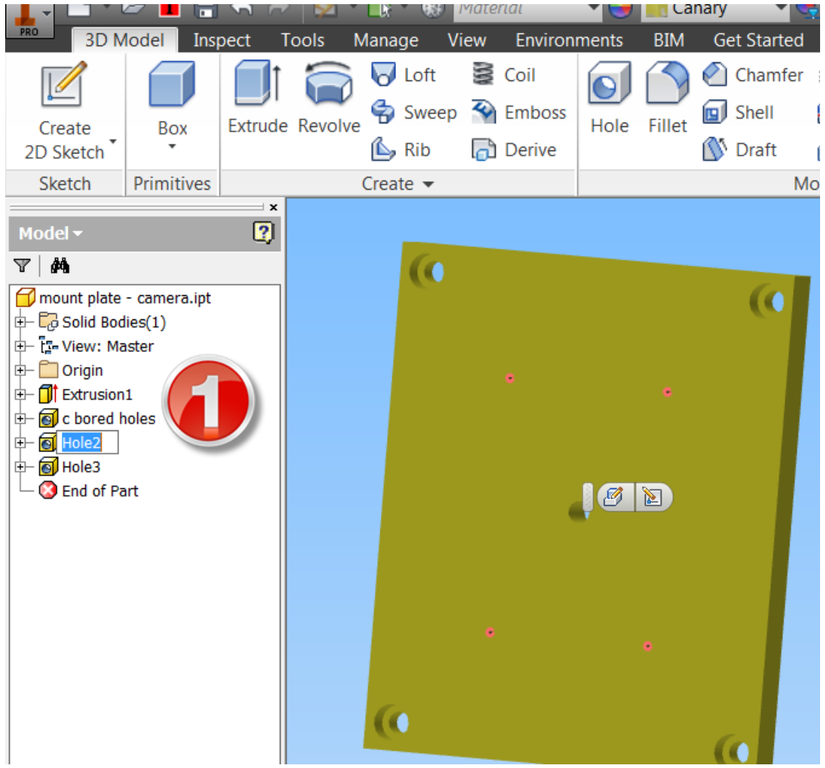
D: All of the above

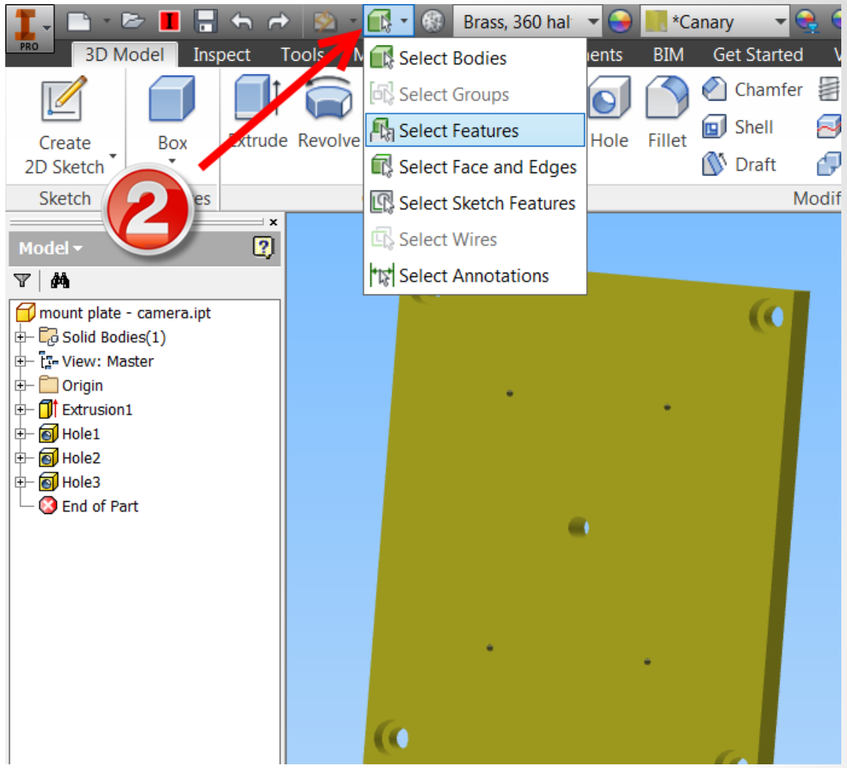


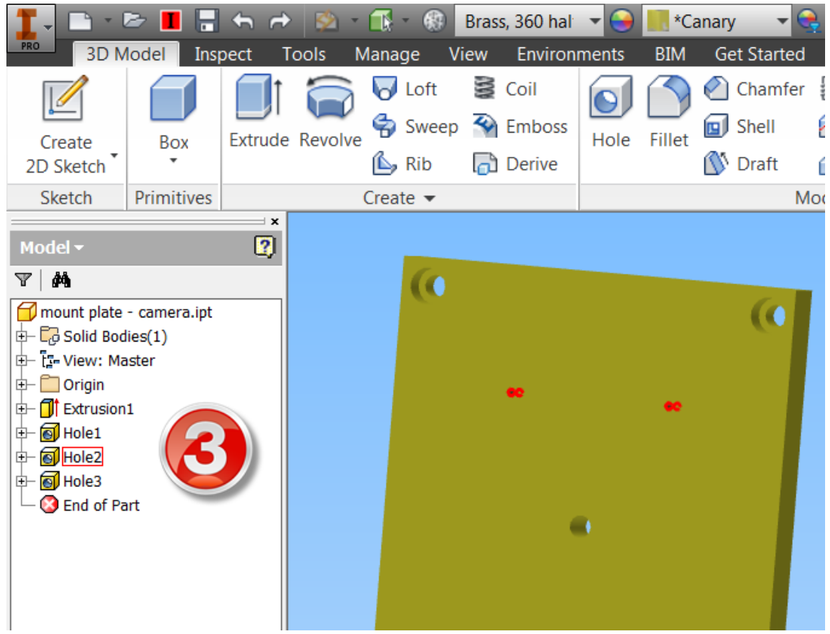
D: All of the above

~~Three~~ Four possible methods, not in any particular order:

1. Rename the feature as you would file names in Windows. Click the feature name, wait, then click it again. The feature name will turn blue and let you type in the descriptive name.
2. Use the “Select Features” choice. This will enable you to hover over the feature and the name will highlight.
3. Hover over the feature name, the hole in question will highlight
4. Bonus – see next page





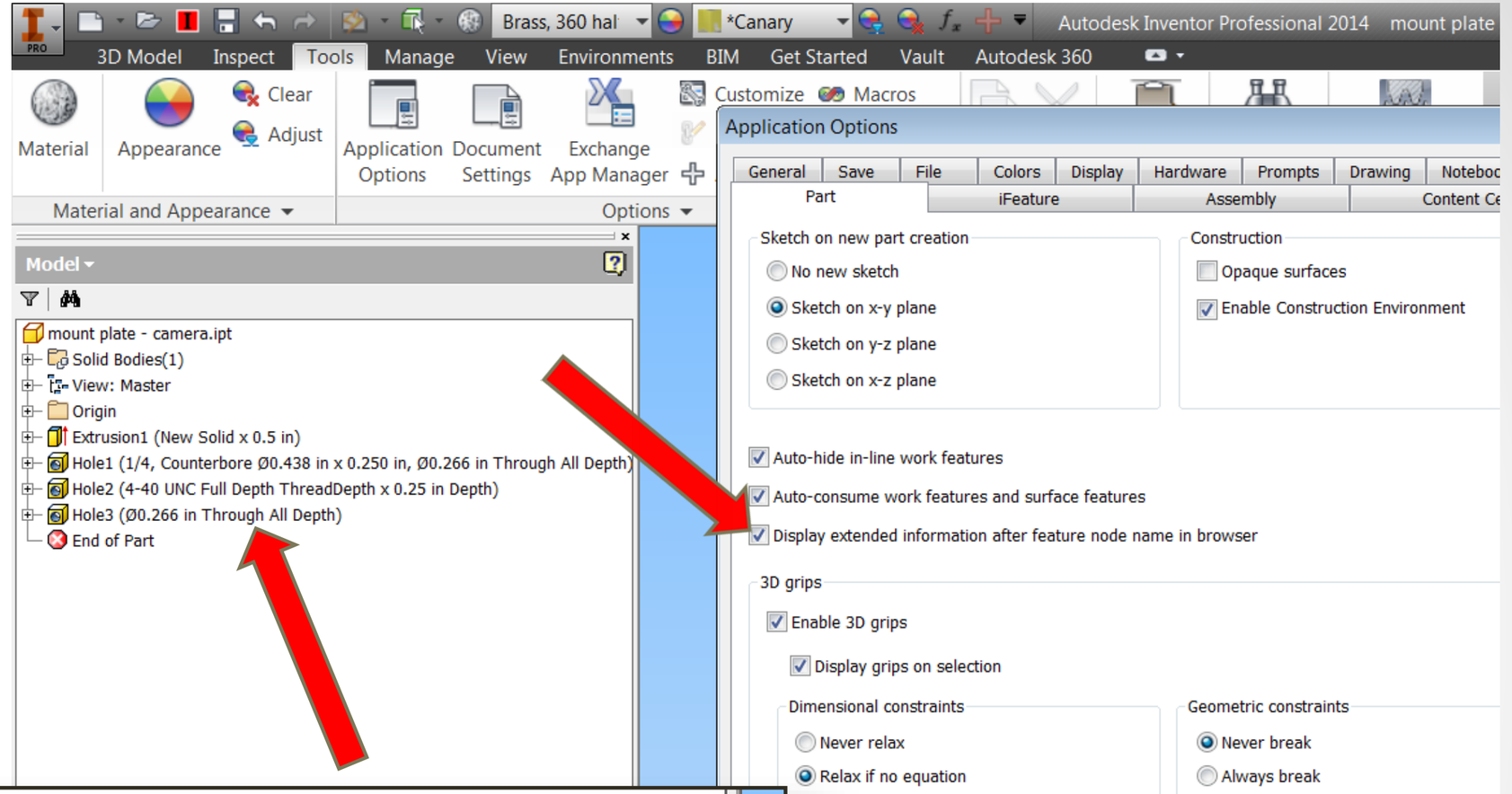


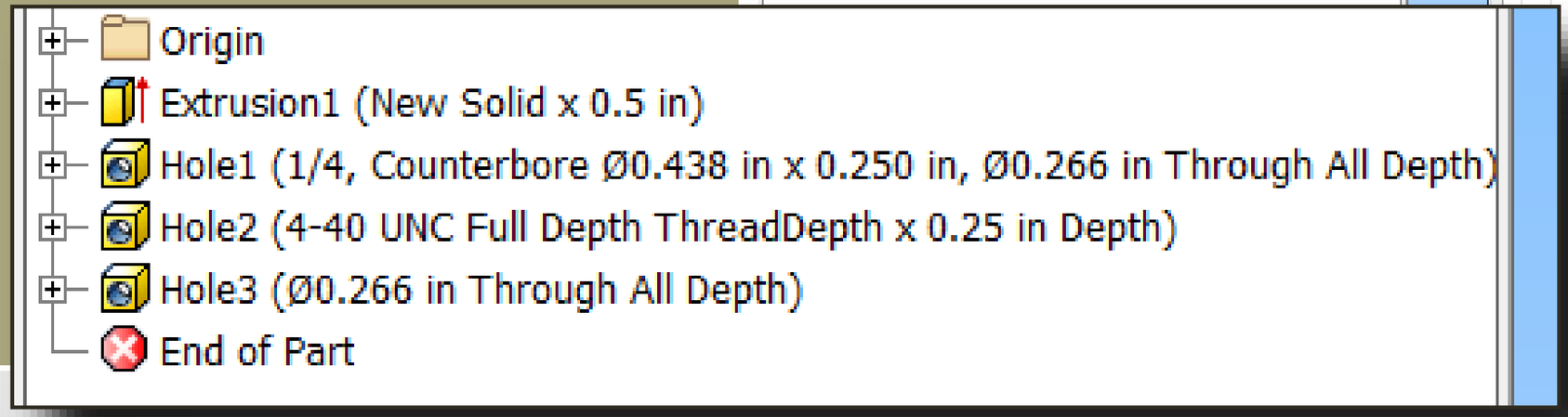
BONUS:

This is another of my favorites.

Notice in the browser the feature names have descriptive details. This saves me time.

To turn this on go to the Tools tab, select Application Options, then turn on the Display extended information check box.





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You are going along, designing with joy in your heart. Then BAM, your browser goes away. Panic, Fear, Anger!

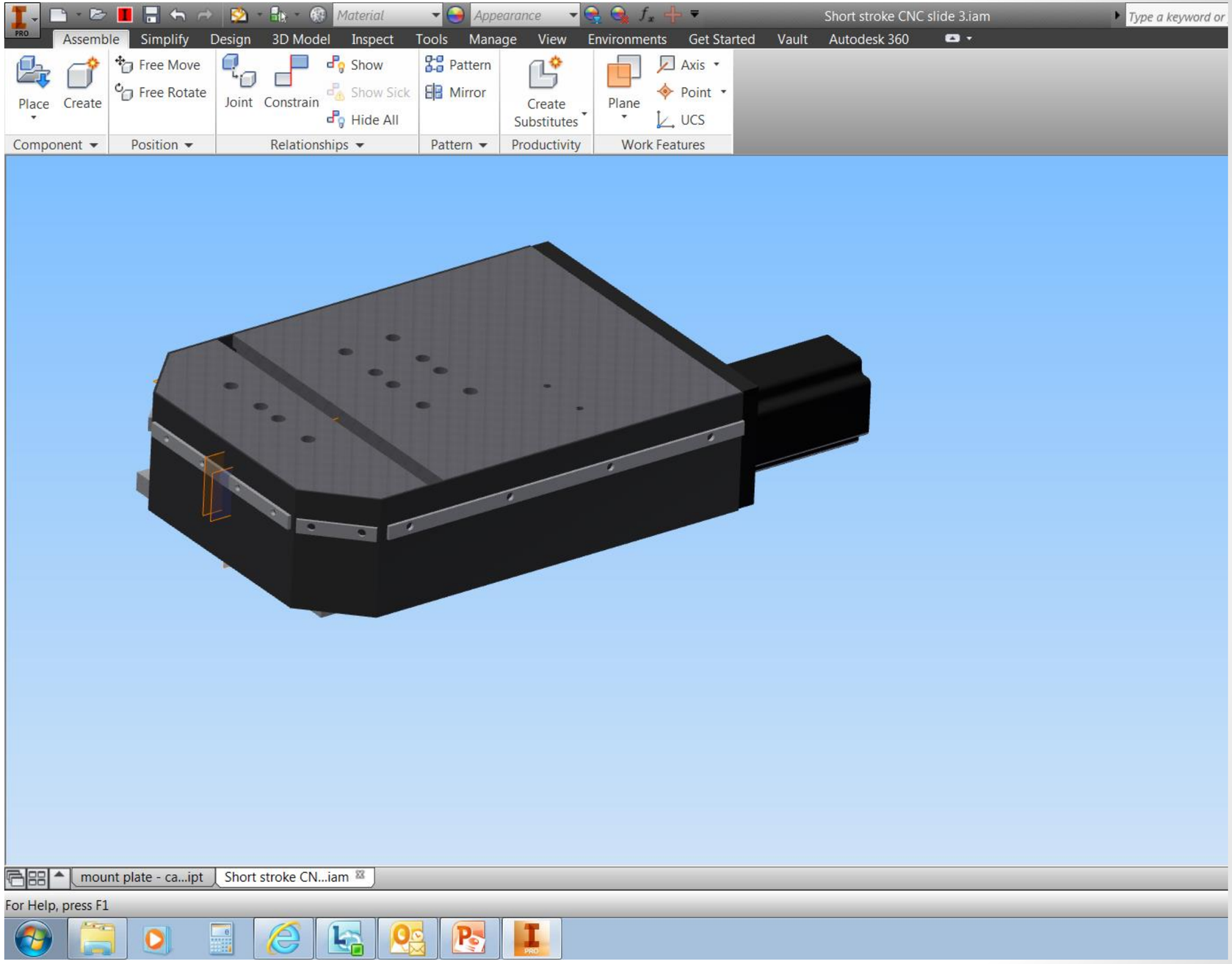
What do you do?

A: Call tech support, a reseller, your CAD manager, or even a 12 year old.

B: Give up, go home, take up knitting.

C: Go to the View tab

D: Re Boot, if that does not work, reinstall your software.



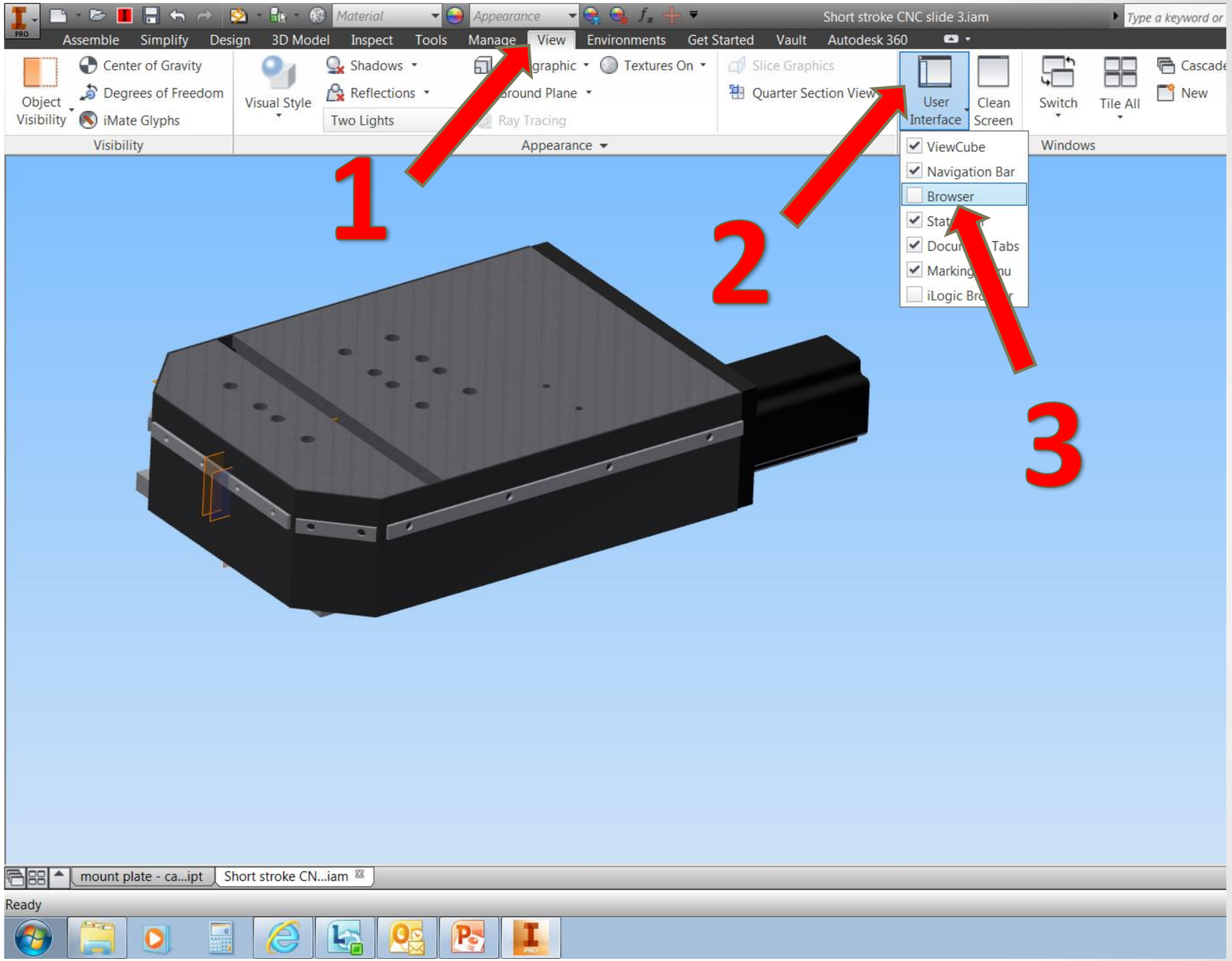
C: Go to the View tab

Go to the view tab

Select User Interface

Turn the browser back on

Get back to work



You are working on a Vaulted document. When you go to save it so you can check it in you get “Save Copy As”

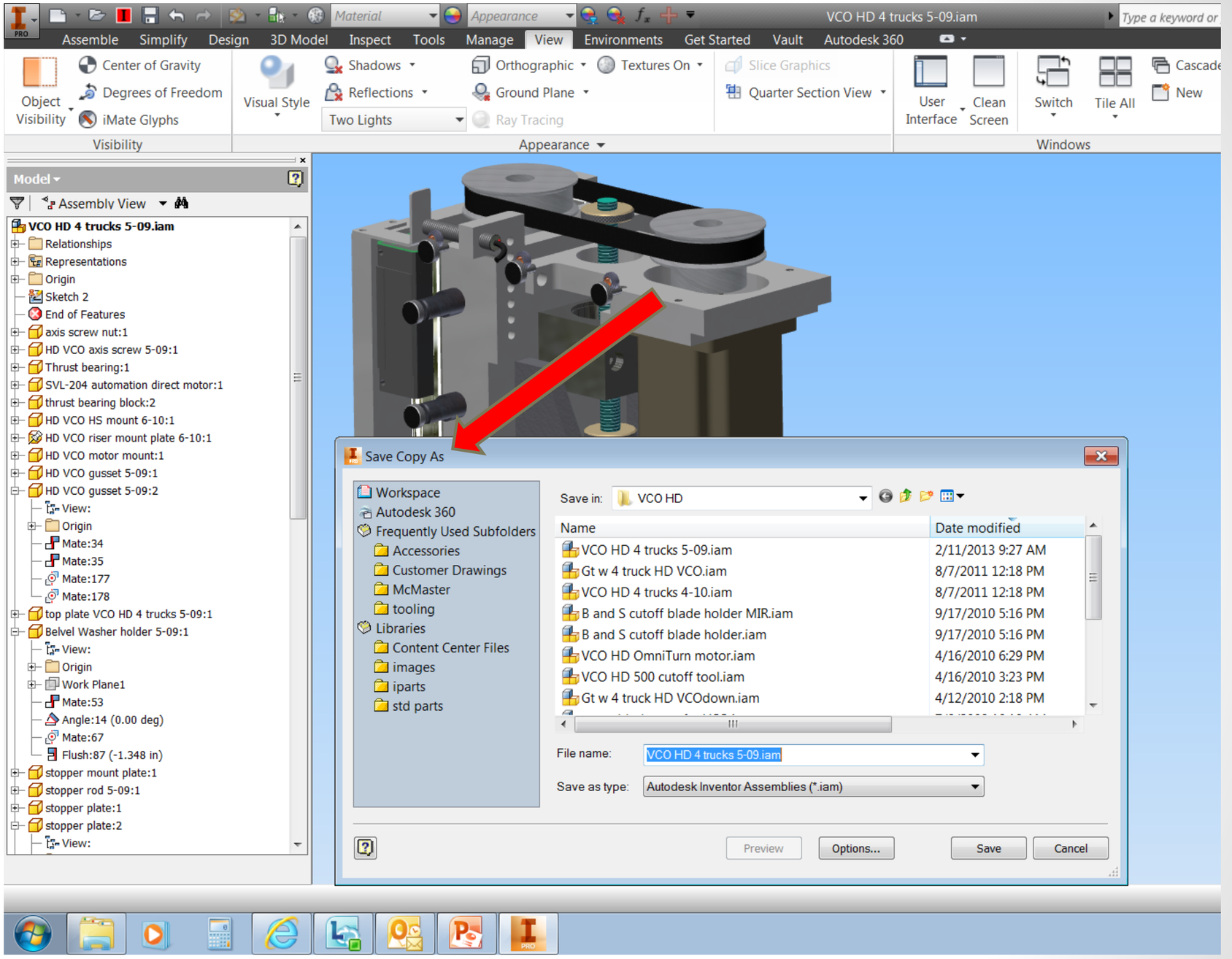
Note: You do not want a new copy, just check in the model you are working on. What do you do?

A: Don’t save, just close it. Go home, find a new job.

B: Don’t save, Check the file out.

C: Save outside the work space, then ask for forgiveness.

D: Save with a new file name, then use Vault to change it back to what you wanted.

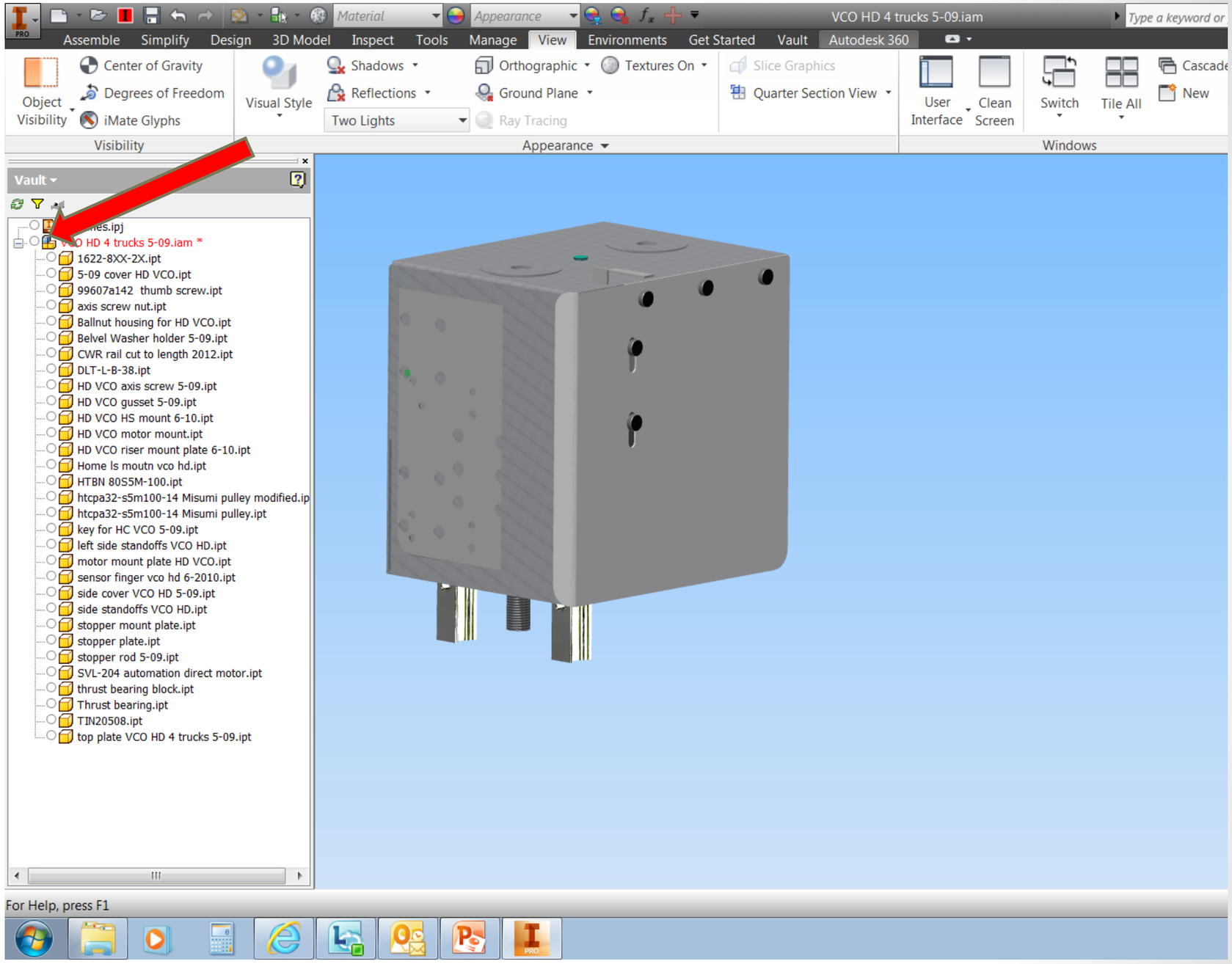


B: Don’t save, Check the file out, then save it.

Since you are being asked to save the file with a new name, it means you have not checked the file name out.

If you check the file out, you can then save the file, and then check it back in again.

When you opened this file from your workspace you were asked if you want to check the file out or it was not available for checkout. If you chose not to for some reason, or just a mistake, then do as noted above. If you could not check the file out, be careful before writing over someone else's work.



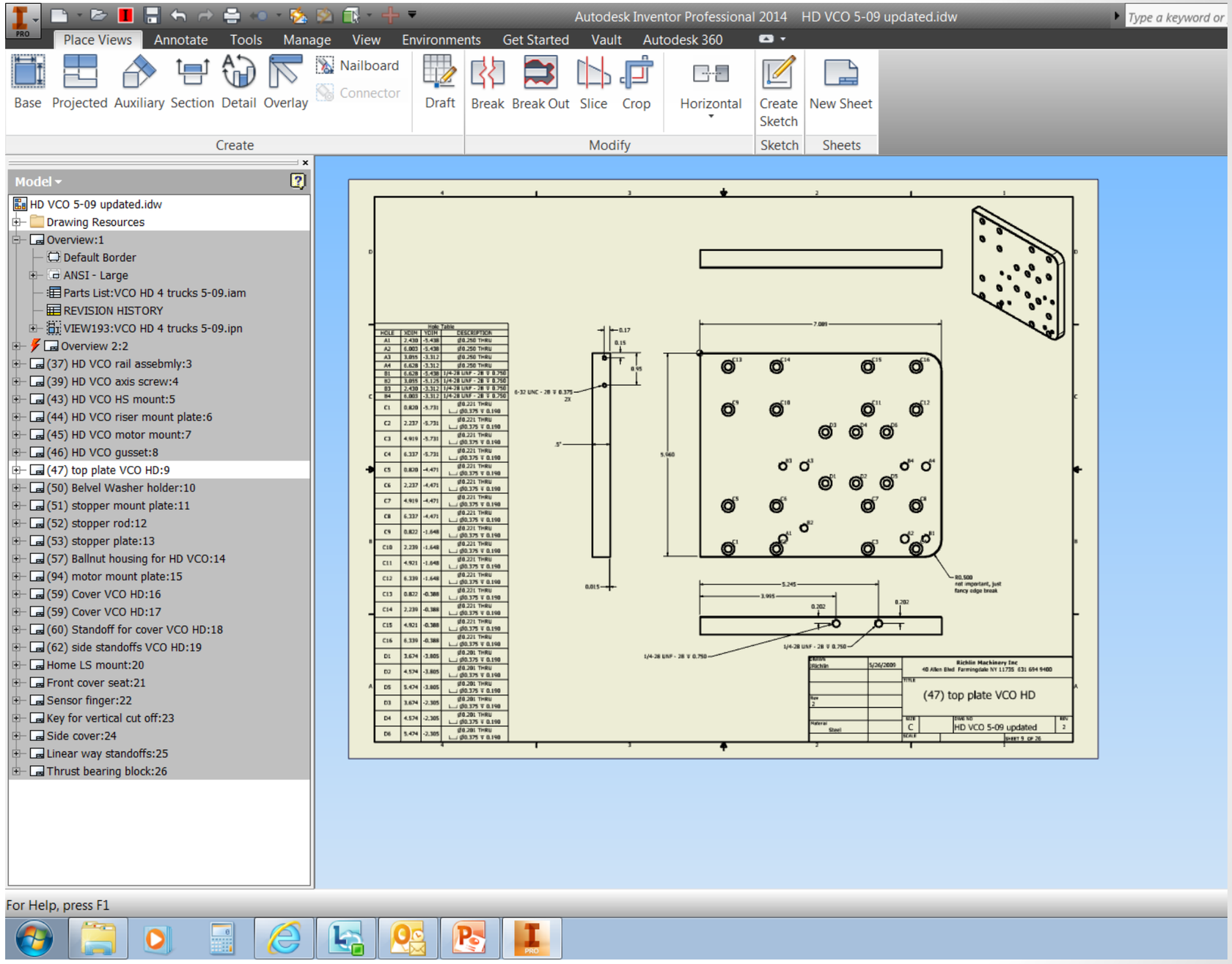
I have a vendor that needs geometry from my Inventor design for his AutoCad drawing. What do I give him so his drawing can stay current with my design without having to redraw.

A: Go through Design Review with DWF files.

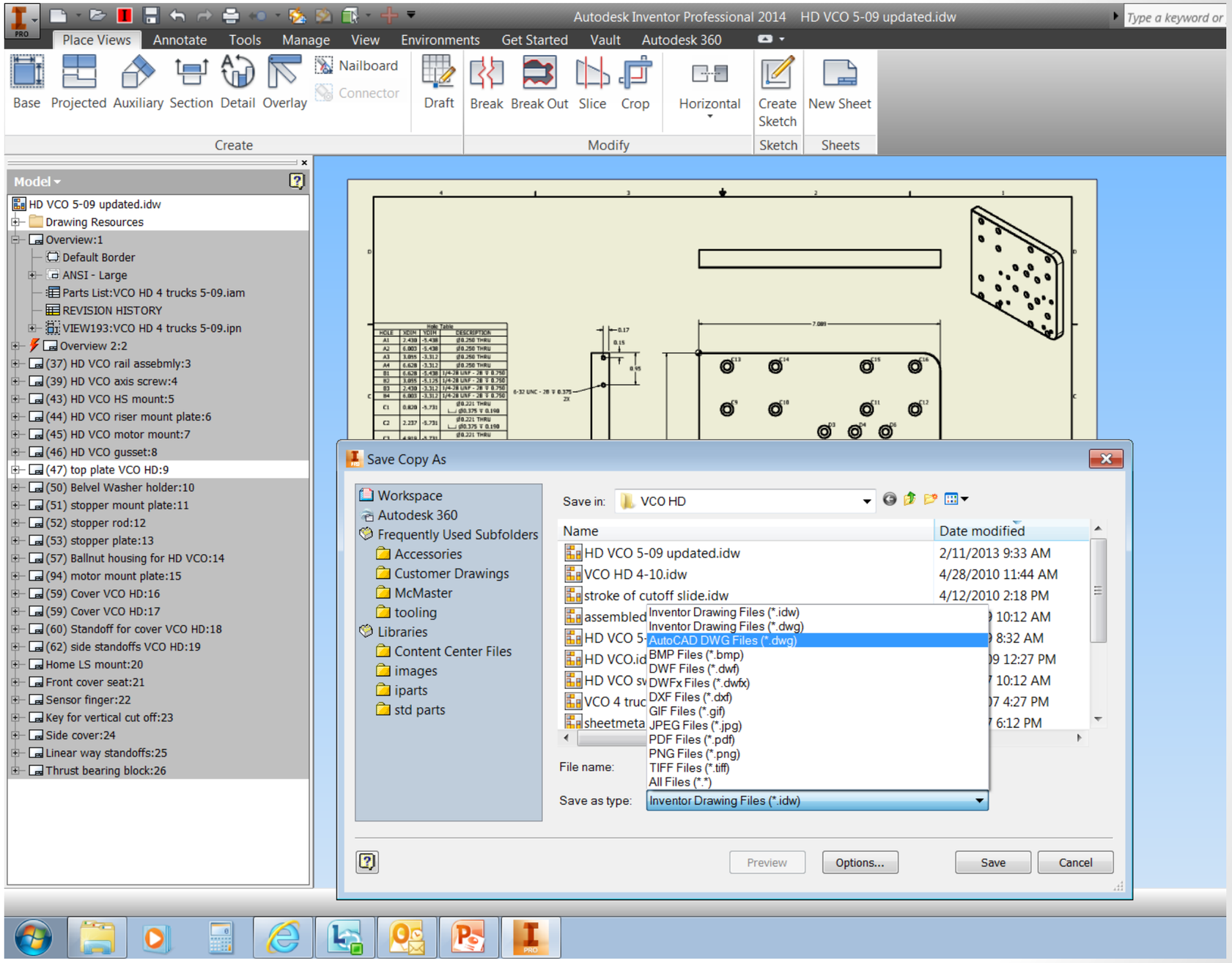
B: Save as AutoCad DXF. Both need the 2014 release.

C: Save as an Inventor IDW. Then it is a piece of cake.

D: Save as an Inventor DWG. He will need the same release as you or later.



D: Save as an Inventor DWG. He will need the same release or later.



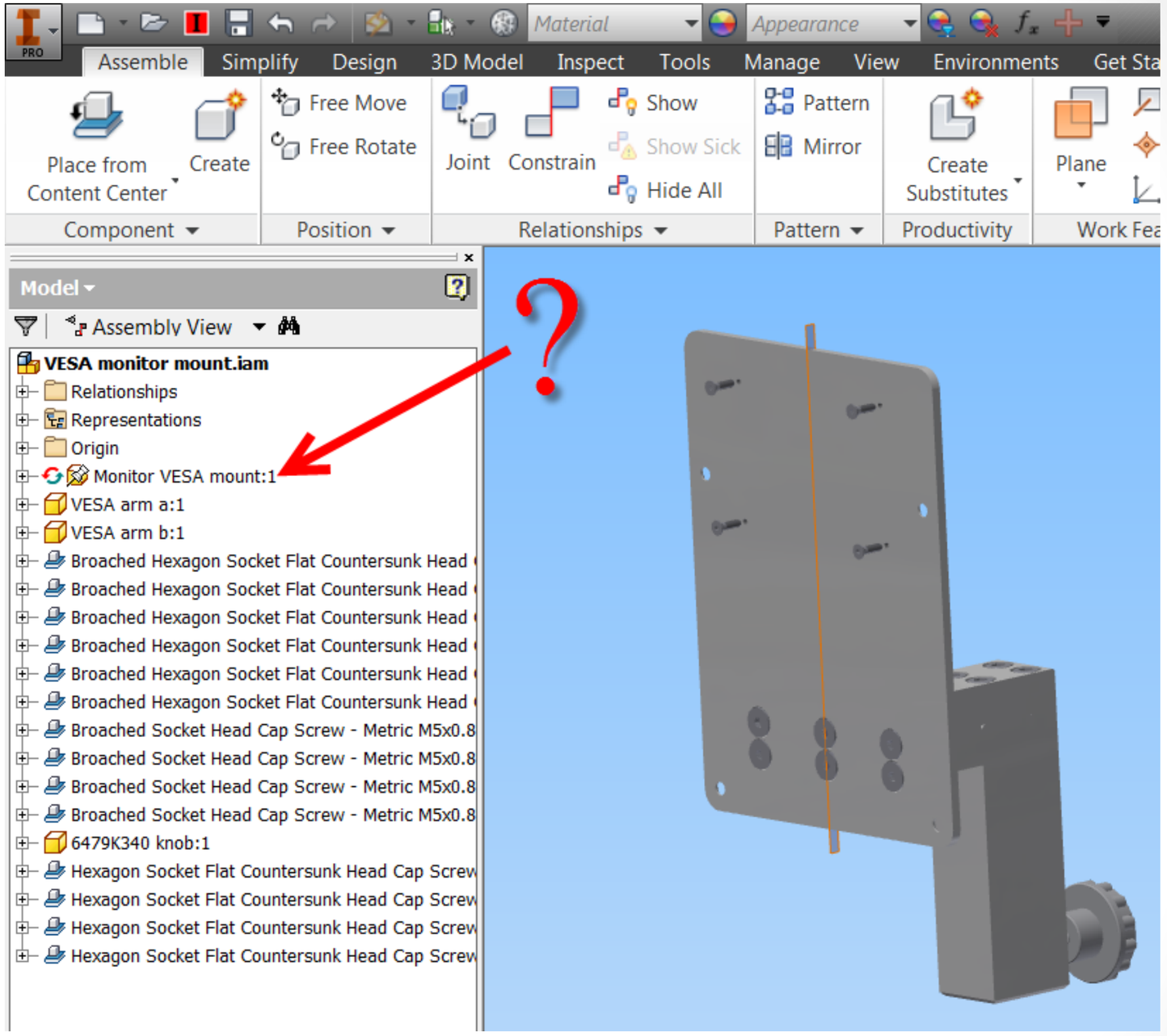
When you first create or place a file into an assembly, what is this?

A: File Name

B: Nick Name

C: Part Number

D: Description



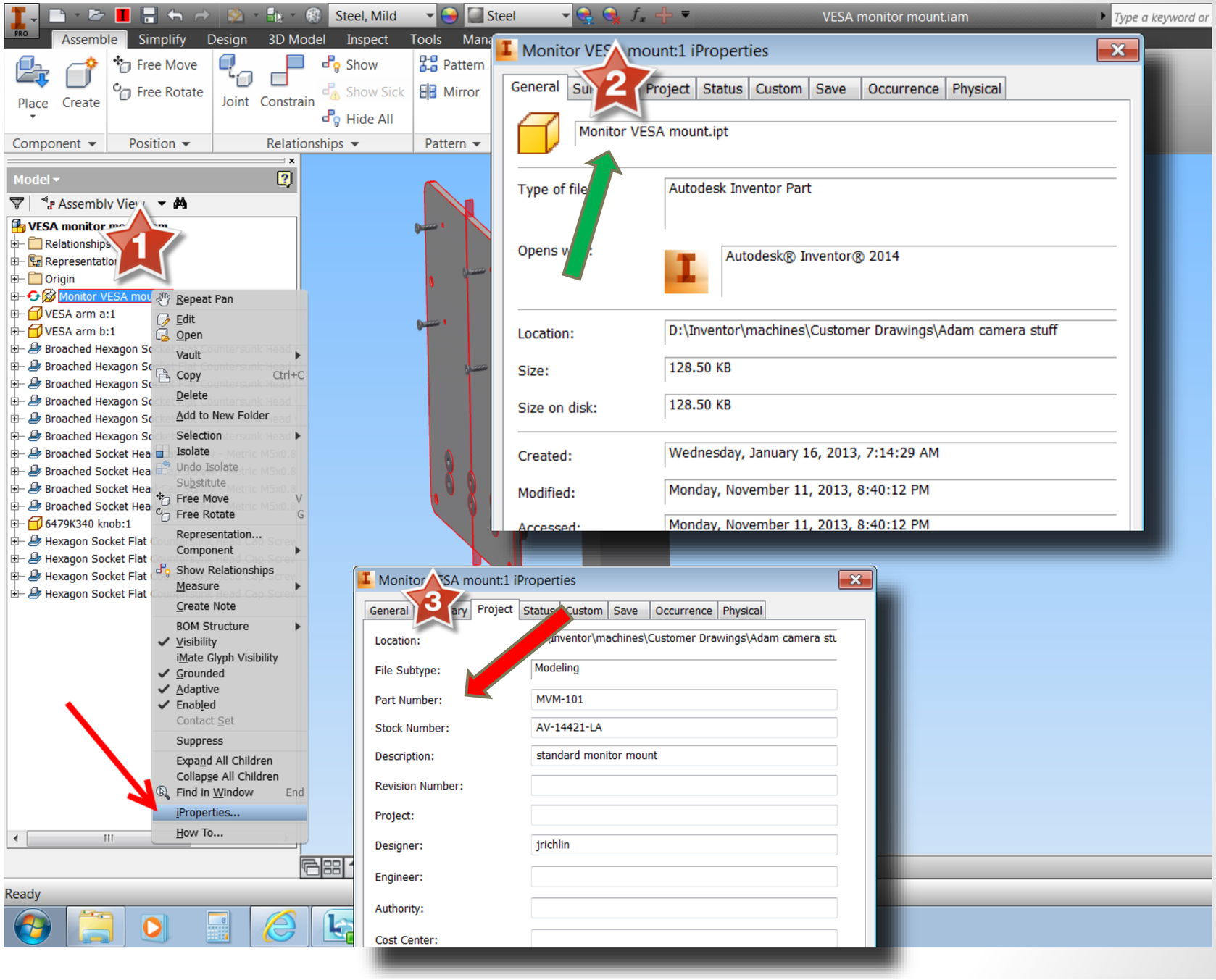
A: File Name

When the file is first created or placed the browser automatically uses the file name.

Bonus:
If you want to change it you can. In the browser you can rename the item. LMC the name, wait, then LMC again. The name turns blue and then can be changed.

This does not change the file name of the item, it is now a *Nick Name*.

Personal Bonus:
Explore the Productivity tools (assembly tab) , You can change the Browser to show Part number, filename or default.



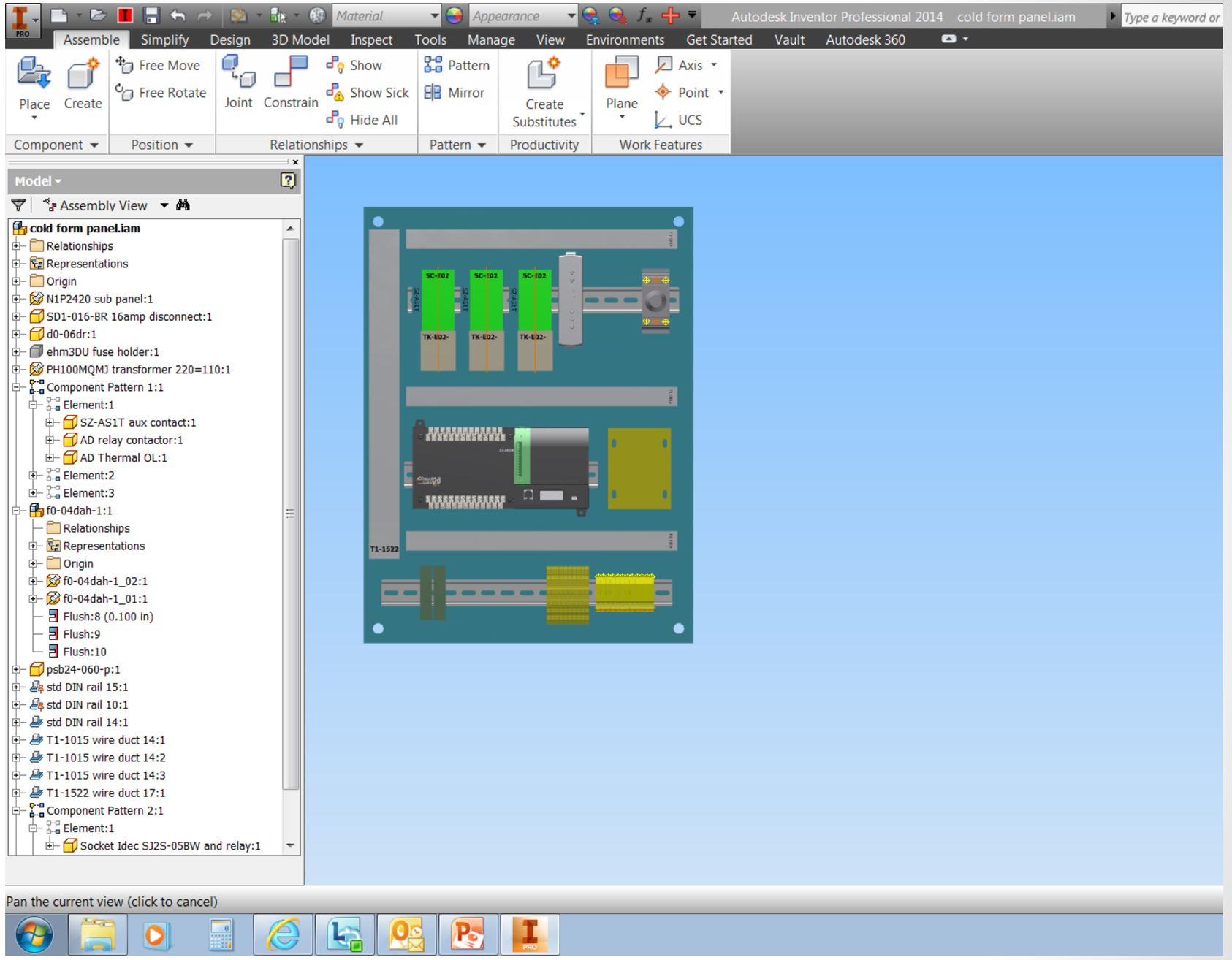
You have a complex assembly with hundreds of parts. As you were working you turn off the visibility for some. Without having to wander around the browser, how do you make them all visible again?

A: Use representation settings – set to Master

B: Use the binoculars

C: Choose File : Update : Visible : OK

D: Just close the file, then re open, all will be OK

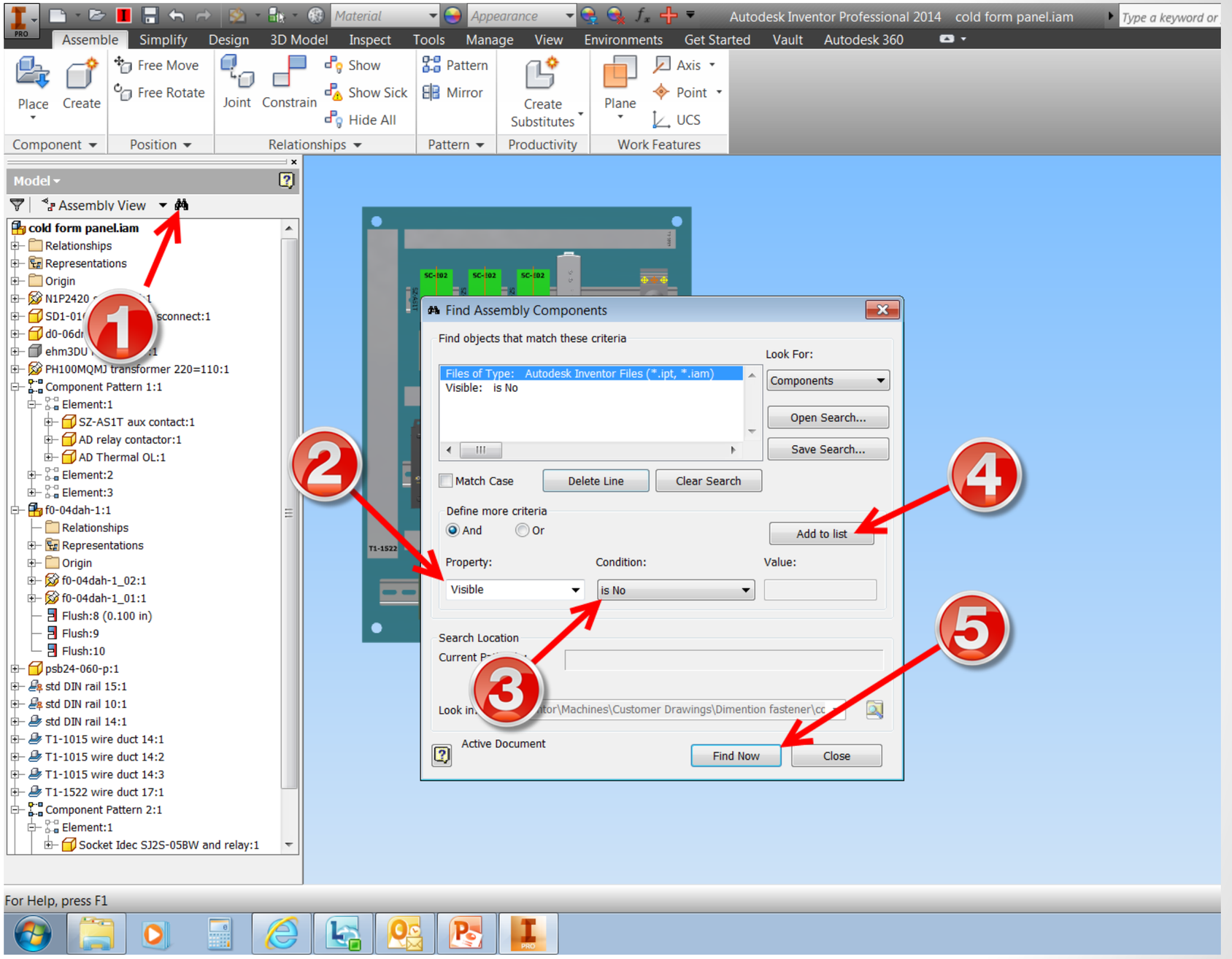


B: Use the Binoculars

Until recently I would wearing out my browser. No longer. I had a great day with my reseller while he watched me work. Much to his credit he does not laugh out loud. This was one of the very cool tips he gave me!

The steps look a little overwhelming, but don't be put off by this.

1. LMC the binoculars
2. Drop down the Property box, in this case I chose Visibility
3. Drop down the Condition box, in this case I chose "is no"
4. Add to the search list
5. Find all that items that have visibility turned off.
6. Close the dialog box
7. RMC one of the selected items to turn visibility back on for all.



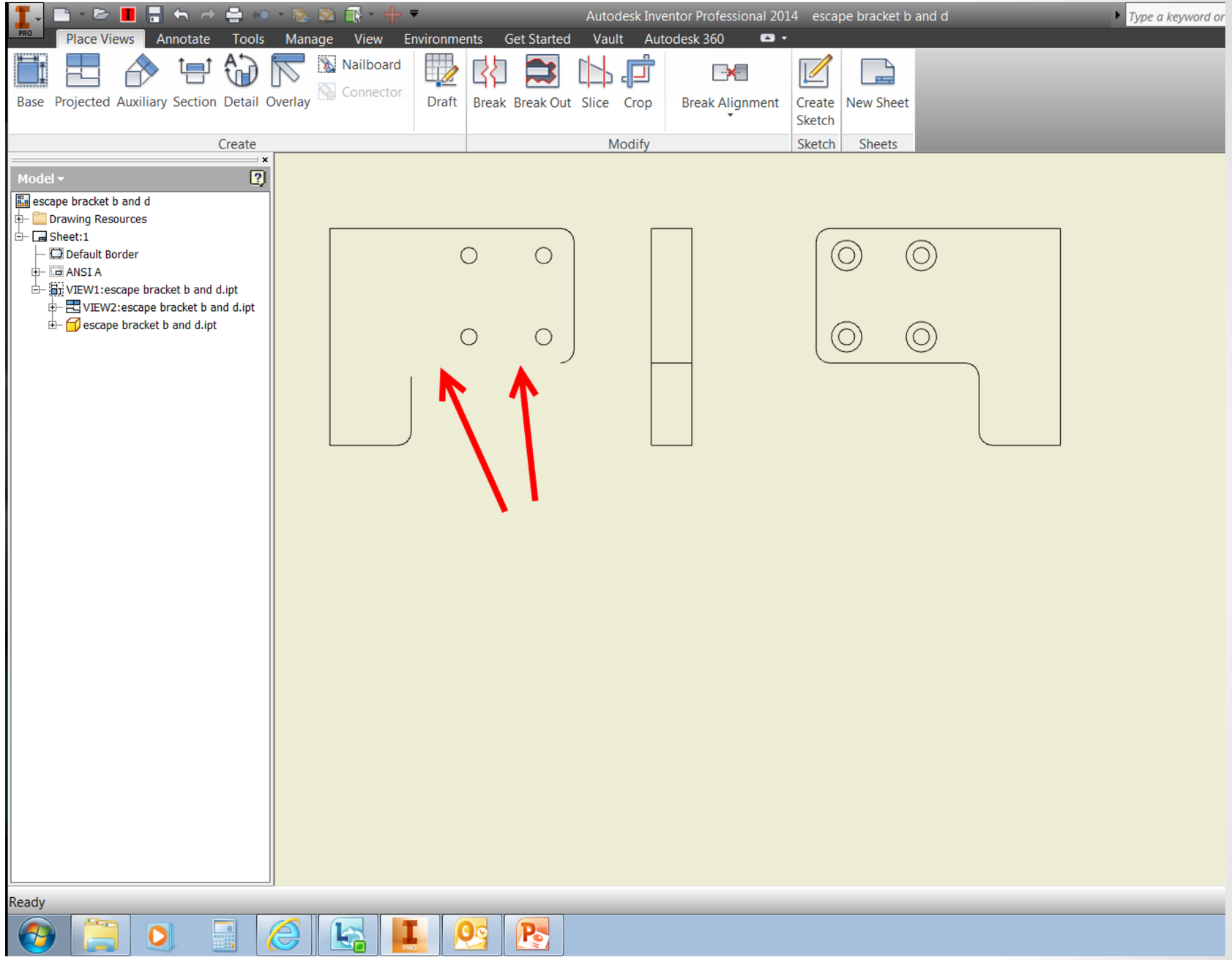
I turned the visibility off on some lines in my .dwg (or .idw). Now I realize I need some of them. How do I make them come back?

A: Close and open the drawing, they all come back by themselves

B: Styles – set to “show all”

C: RMC – Show Hidden Lines and you are done.

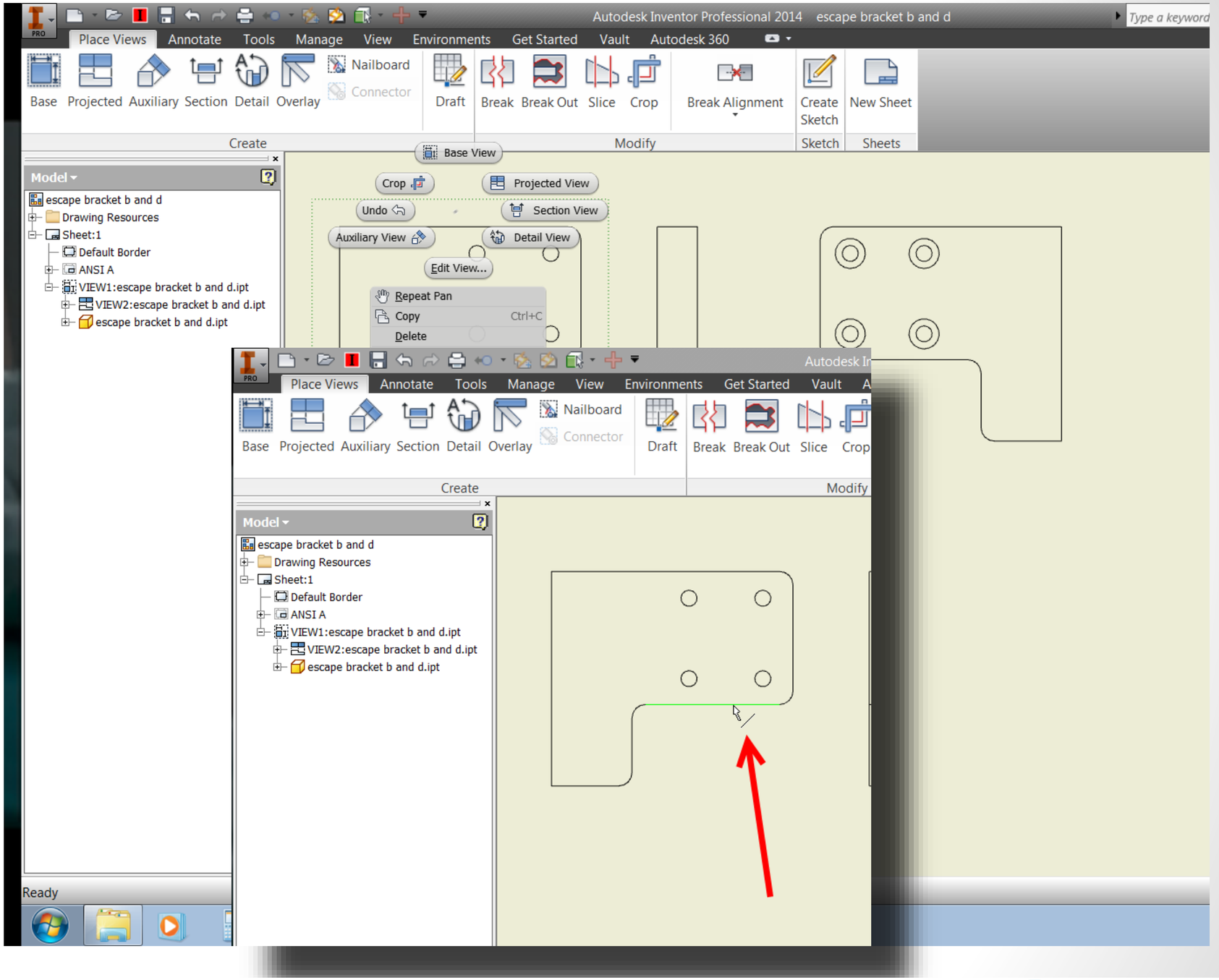
D: RMC – Show Hidden Lines, then there is one more step



D: Use Show Hidden Lines, then there is one more step

If you just show hidden lines, then when you cancel the command the lines disappear again.

While the Show Hidden Lines command is active the newly visible lines are red. LMC on the ones you want to recover, they will turn green. Then exit the command. The green will turn back to visible.



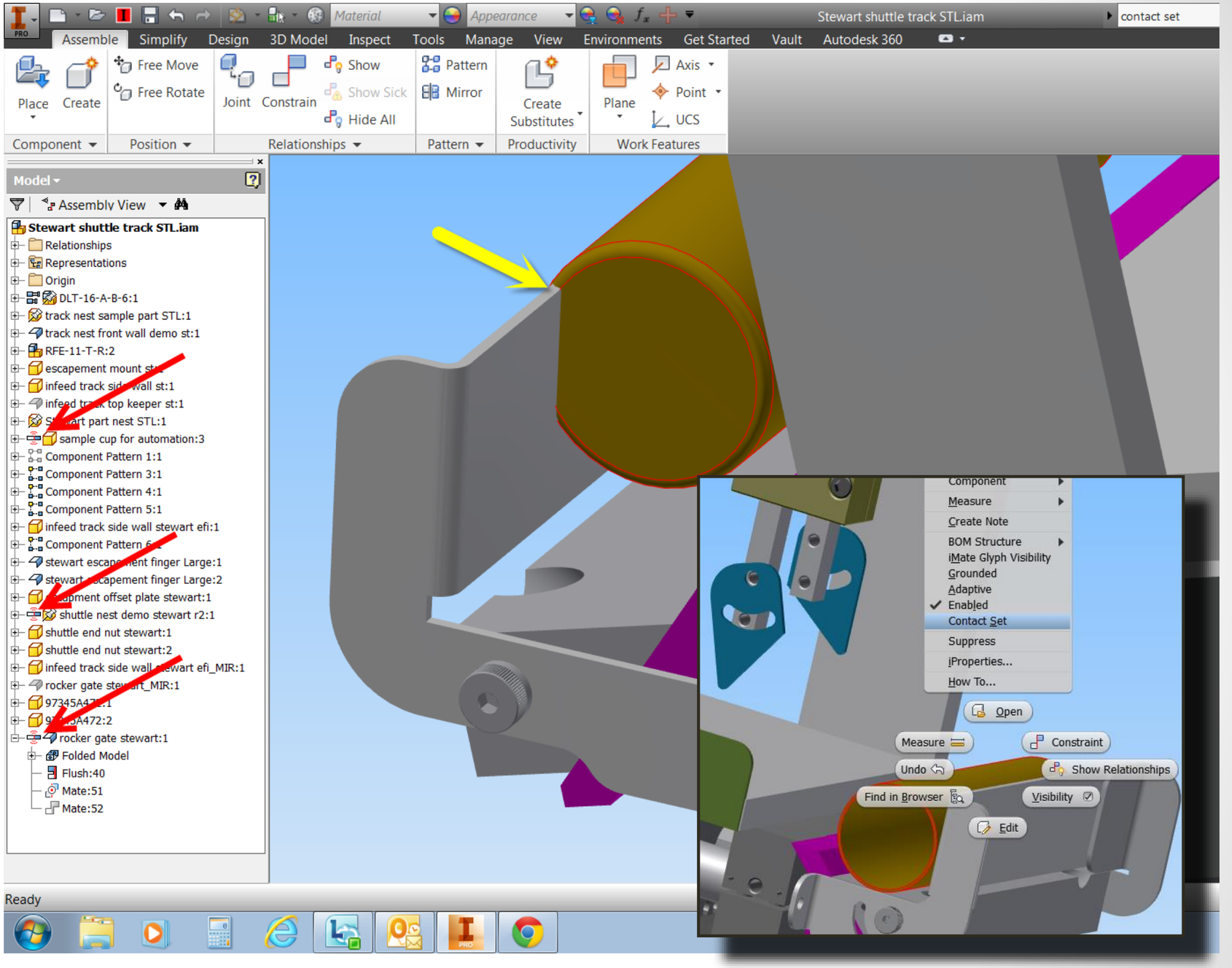
I wanted to use the “Contact Set” tool to make sure the parts don’t go where they shouldn’t. Problem, the contact set does not work. Parts are passing through other contact set parts. Why?

A: Contact Set tool is not turned on.

B: Enable Contact set function in application options.

C: The Contact Set tool is not used for this, it is used to measure surface area.

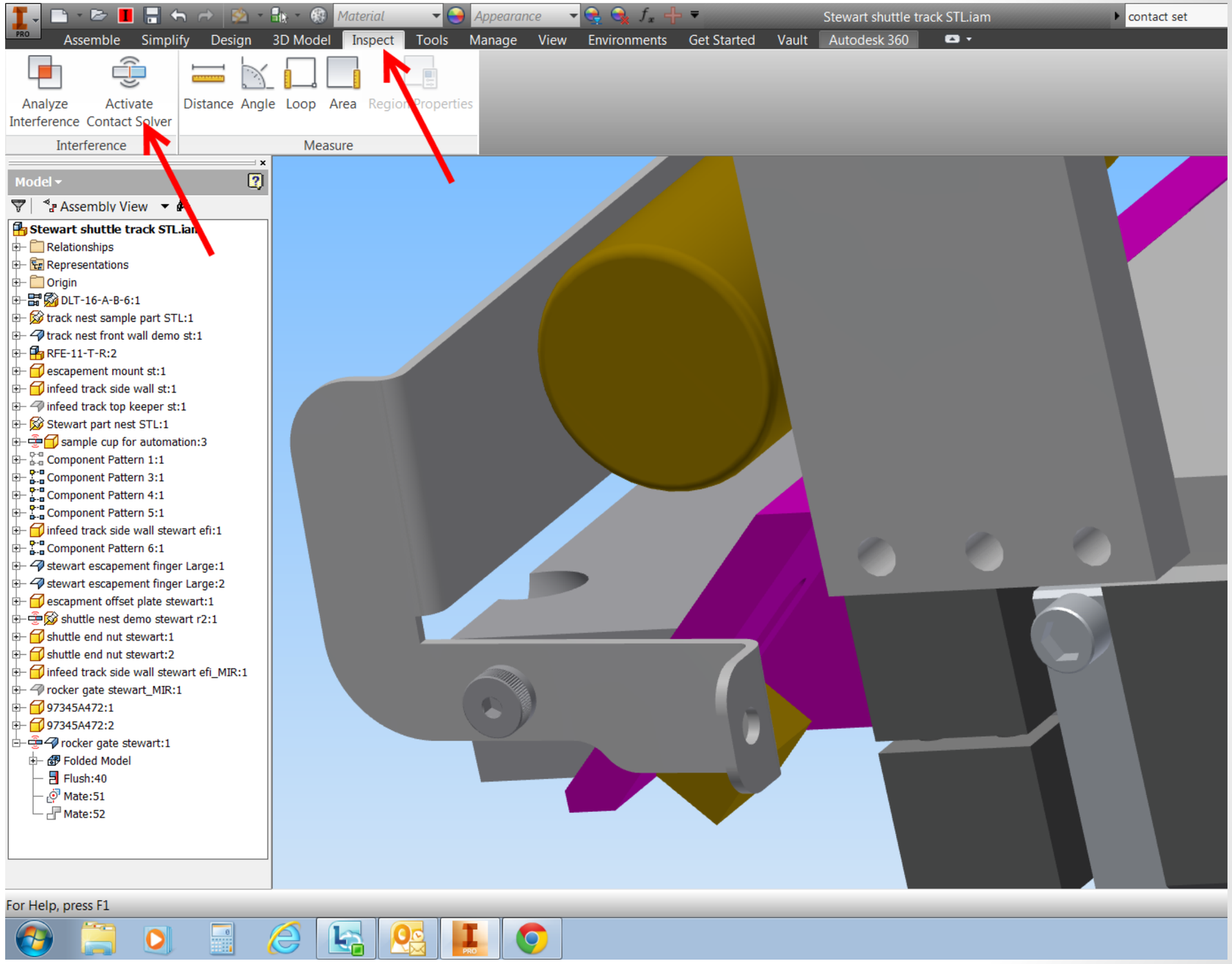
D: Contact Set tool will only work with two items at a time.



A: Contact Set tool is not turned on

After including items into the contact set you must turn the tool on.

Go to the Inspect tab, then activate the Contact Solver.



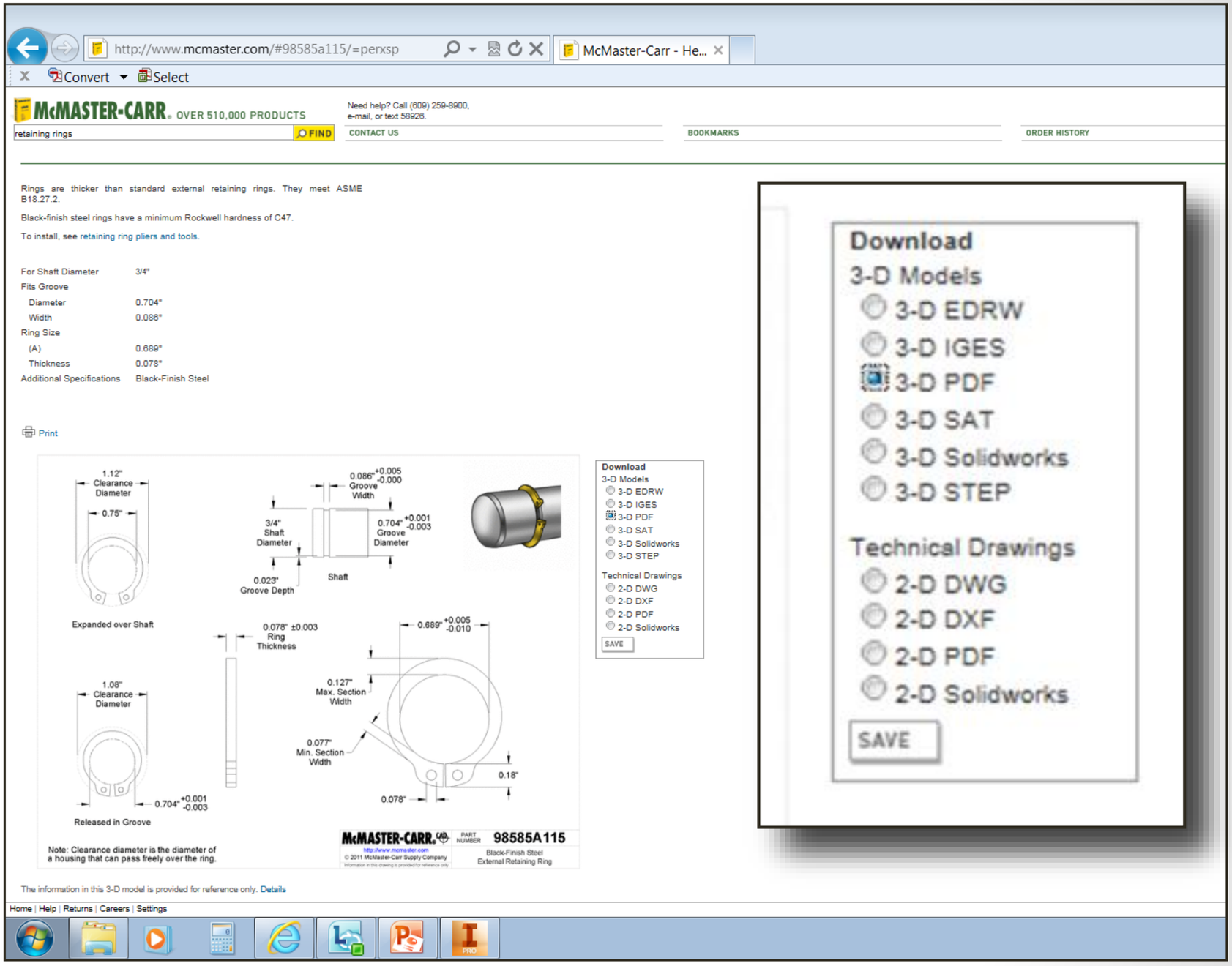
I am using a part from a vendor. I want to use the content they provide to drop into my model, The choices do not include an Autodesk 3D format. What would be the next choice?

A: Step

B: SAT

C: IGES

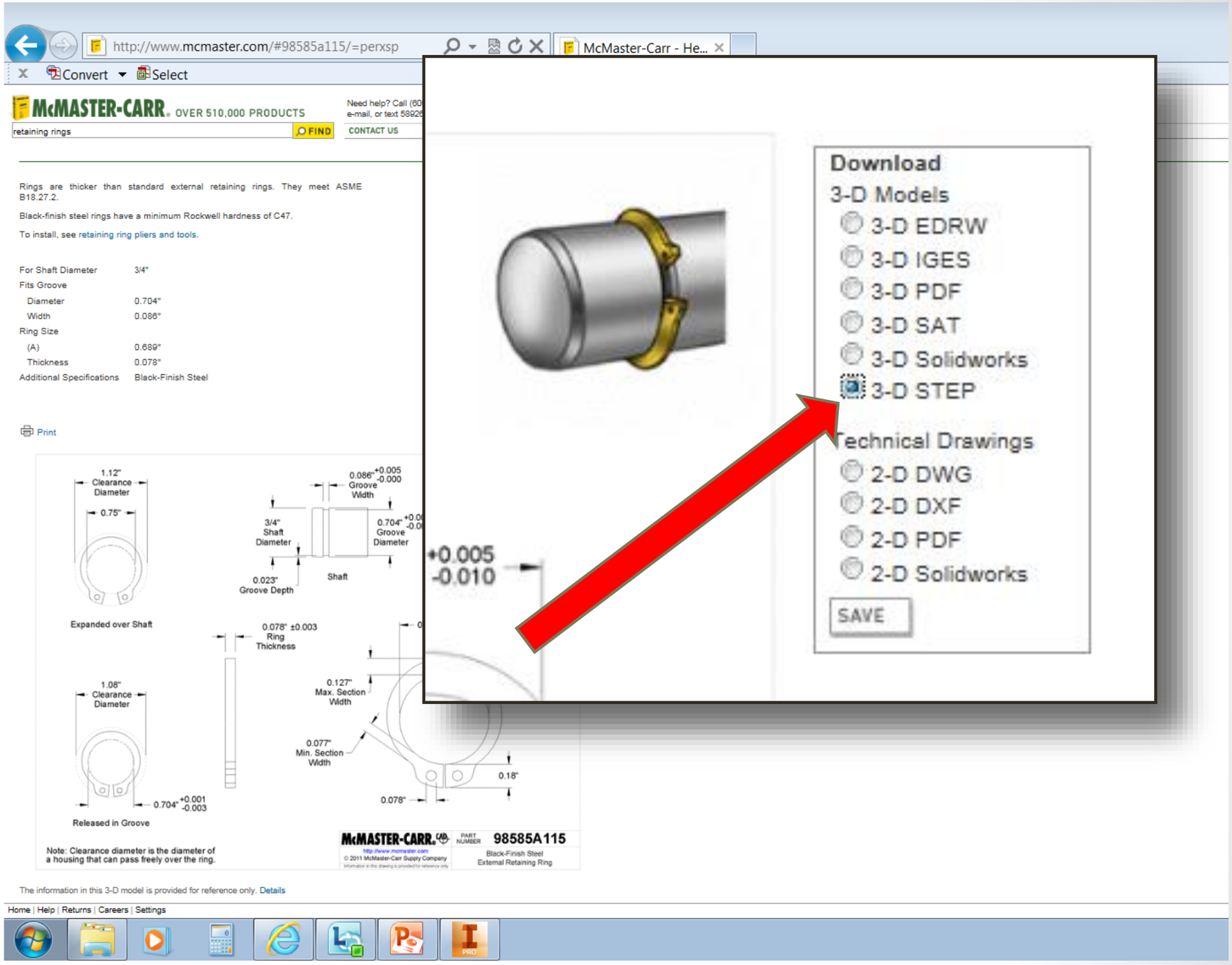
D: PDF



A: STEP

Choice in order of preference

1. Step
2. SAT
3. IGES (for surfaces)
4. ~~PDF~~



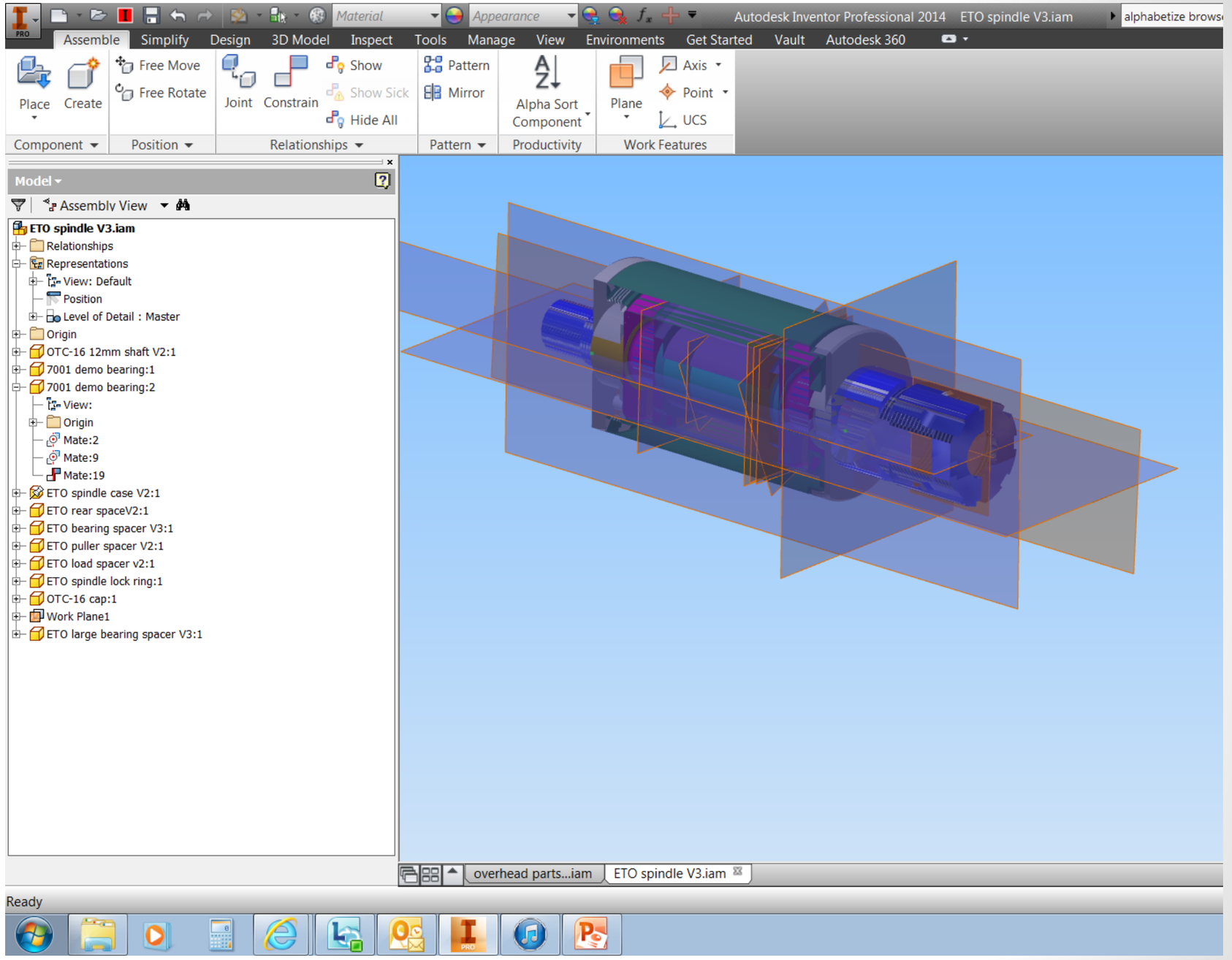
I am working on this assembly when I realize I am so overwhelmed by the planes, points, and axes. I can't see what I am doing. How can I quickly make the "noise" go away?

A: Turn off all workfeatures (planes, axes, points, ect.) one at a time.

B: Change Application options to hide workfeatures

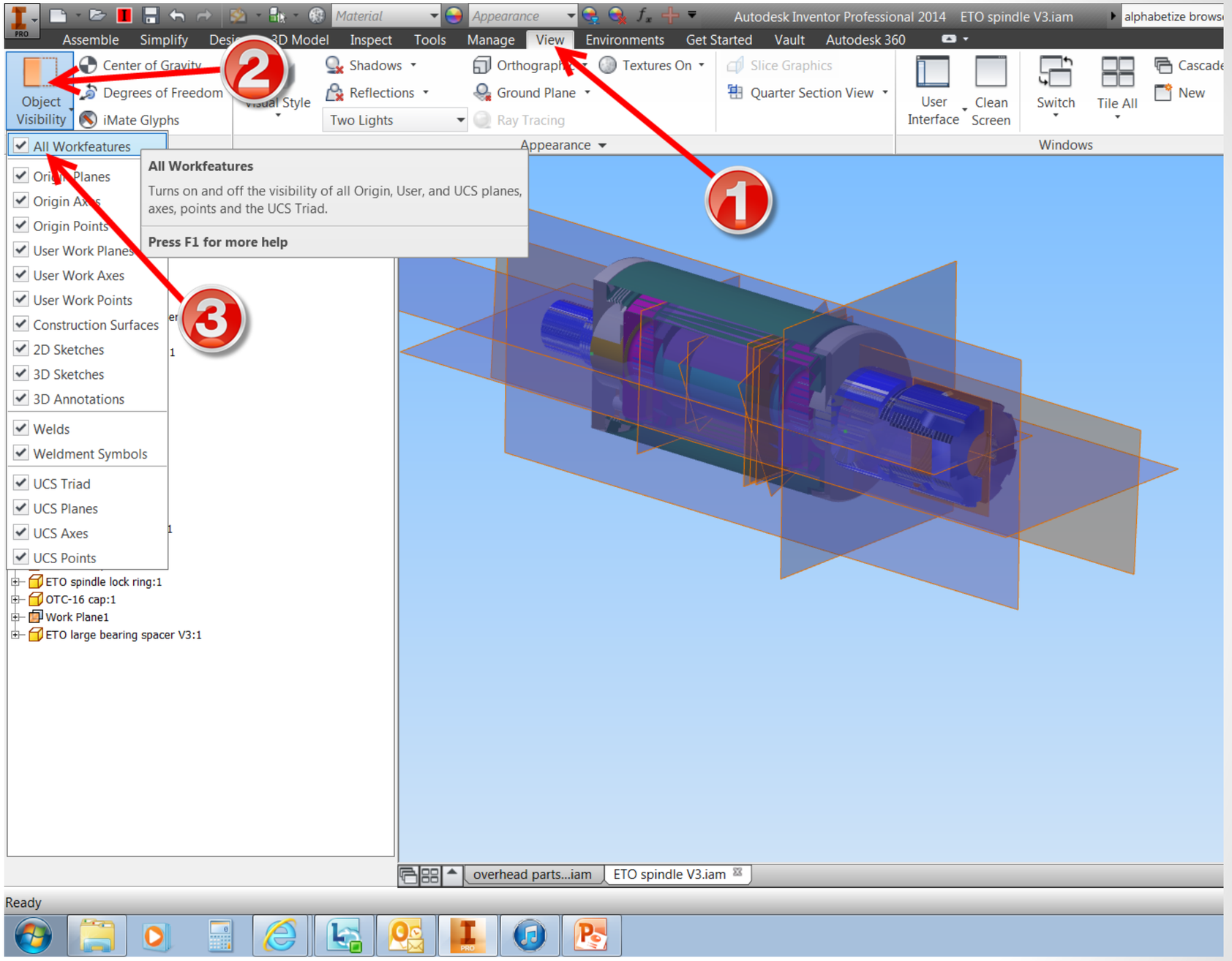
C: Object Visibility under View tab

D: Document Options – Hide workfeatures



C: Object Visibility under View tab

If you uncheck the “All Workfeatures” you are done. With the check boxes, turn workfeatures back on as you need them.



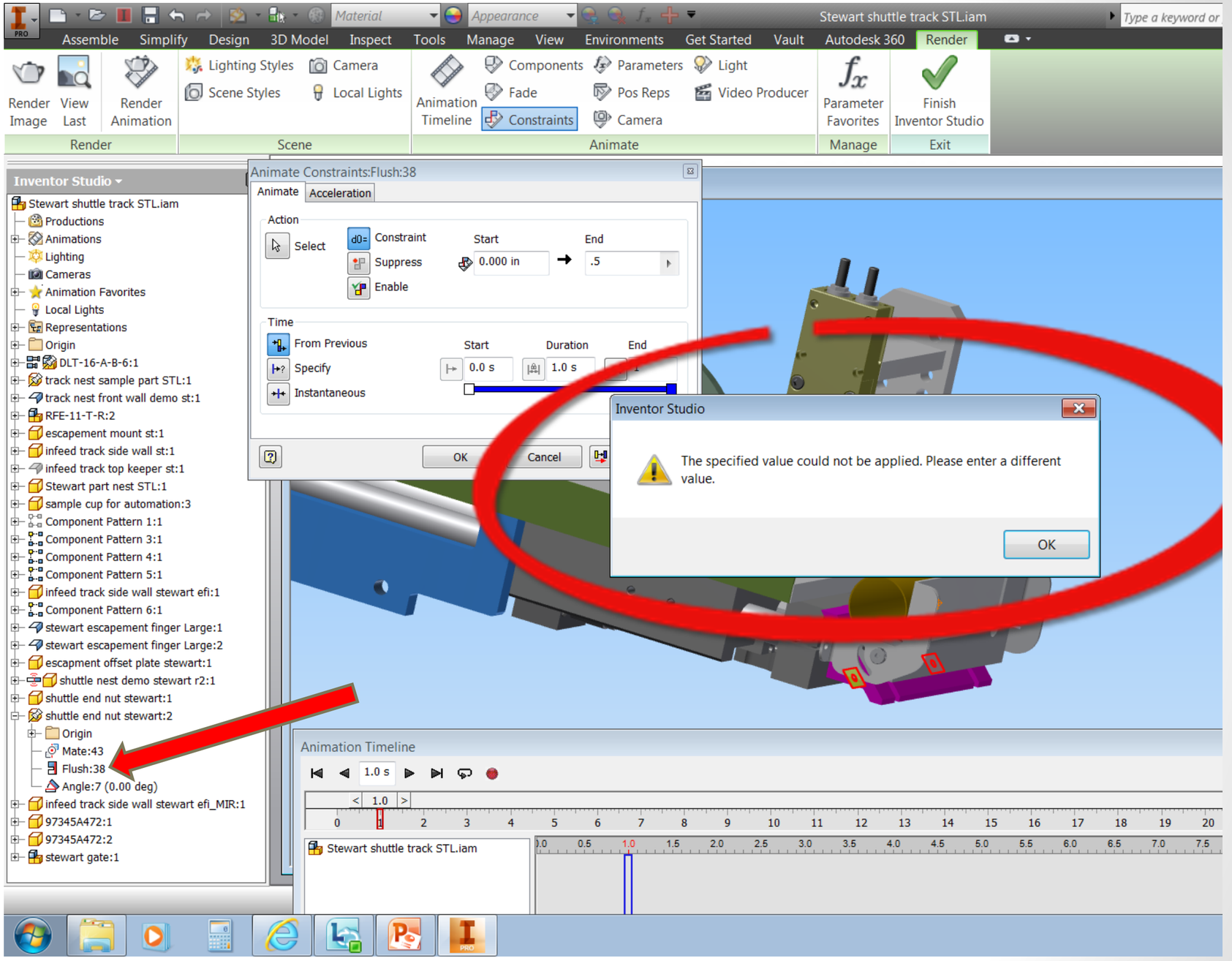
I am creating an animation of my assembly. When I try to animate a constraint I get this error. What is wrong, what do I do next?

A: This assembly can not be animated, use showcase or 3DS Max instead.

B: Change Application Options - Set animation to YES.

C: The assembly needs to be corrected before animation is possible.

D: Don't use Animate Constrains, try Animate Components.



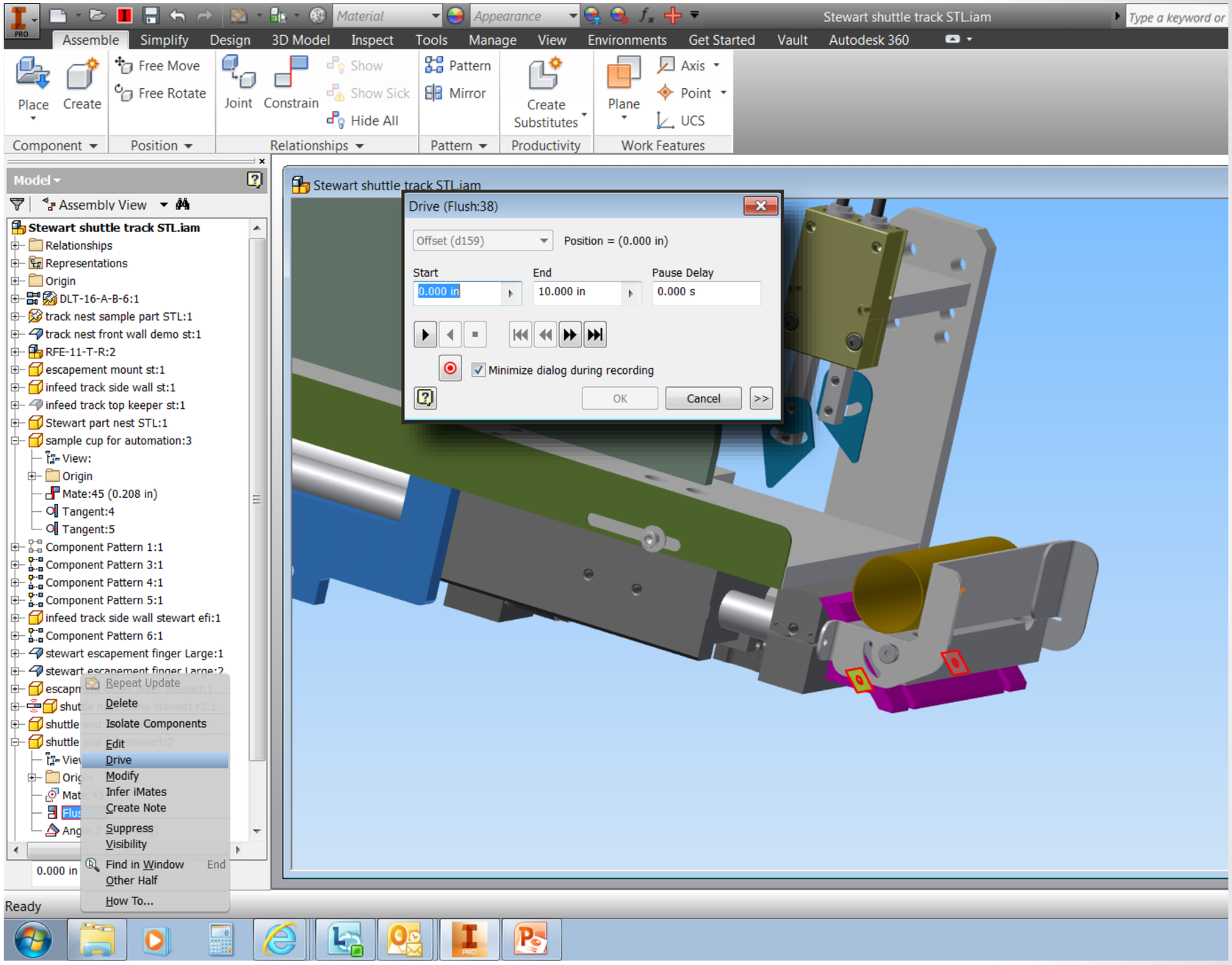
C: The assembly needs to be corrected for enable animation.

This question comes from my first attempt at animation. No matter what I did I got this error.

My assembly had constraints that would not work. Once I fixed them, I was able to go too Studio to animate and export videos.

Check your constraints with the Driven tool. If you get errors with Driven, the animation won't work.

In the assembly, RMC the constraint. Select Drive, then fill in the start and end dimension. Hit play. If that won't work, figure out why, and try again.



I want to view my browser content alphabetically. Wow, what a concept.

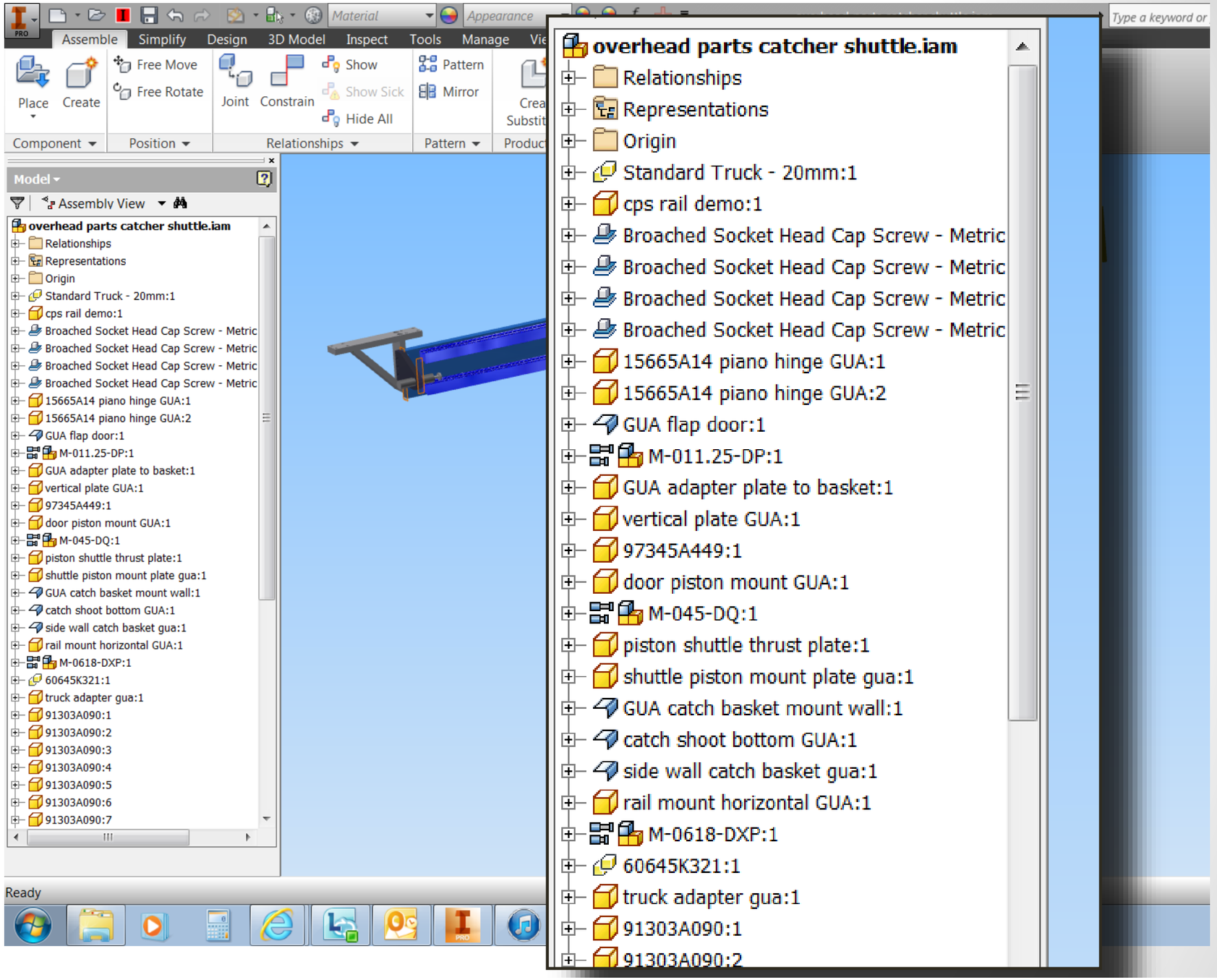
How do I do this, if I can?

A: Productivity

B: RMC - Sort

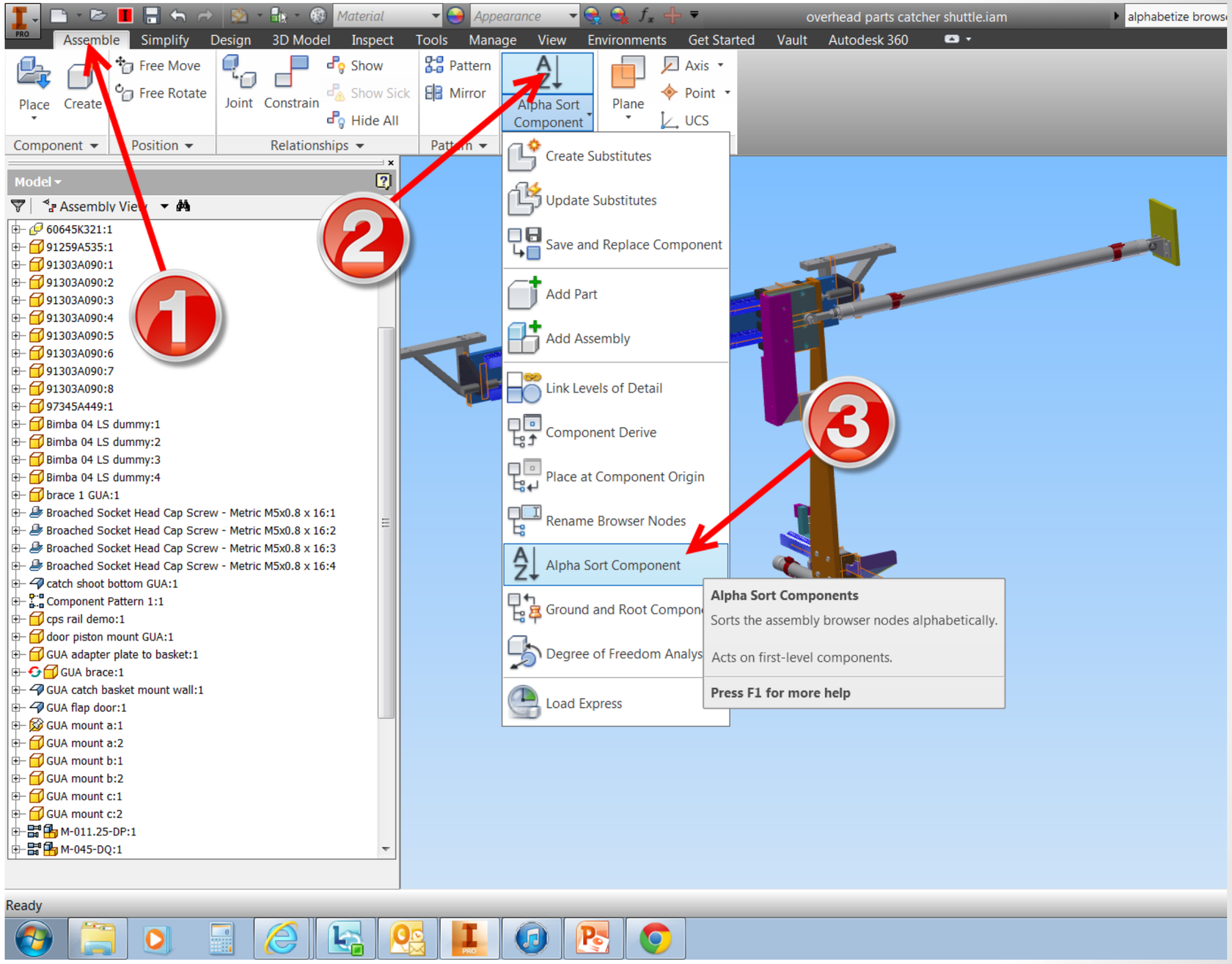
C: Can't be done yet, get AUGI to help with the wish list.

D: Simplify



A: Productivity

This tool really made my day. I don't always use it, but when I need it, AWESOME.



You are working with an assembly that your customer wants to know that the system will cost. Where could you keep data that would help create a COG (total cost of goods)?

A: Metadata

B: Engineer's notebook

C: Styles

D: iProperties

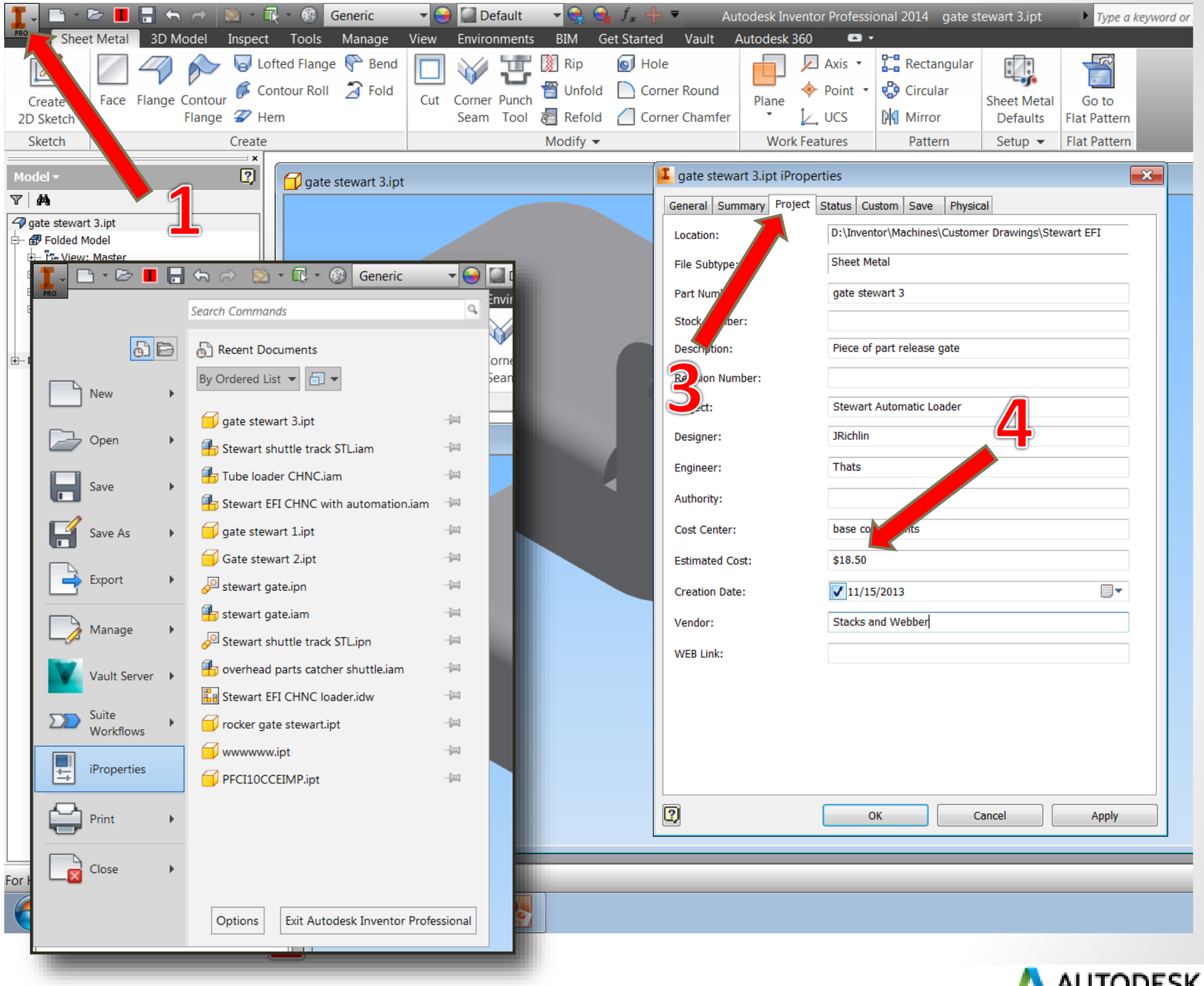
D: iProperties

iProperty data can easily be exported or used in conjunction with BOM to help create a cost of goods sold.

One way to get to iProperites is LMC the I at the top left of the Inventor Ribbon.

Select iProperties.

To get to a place to save the costing data select the Projects tab, then fill in Cost.



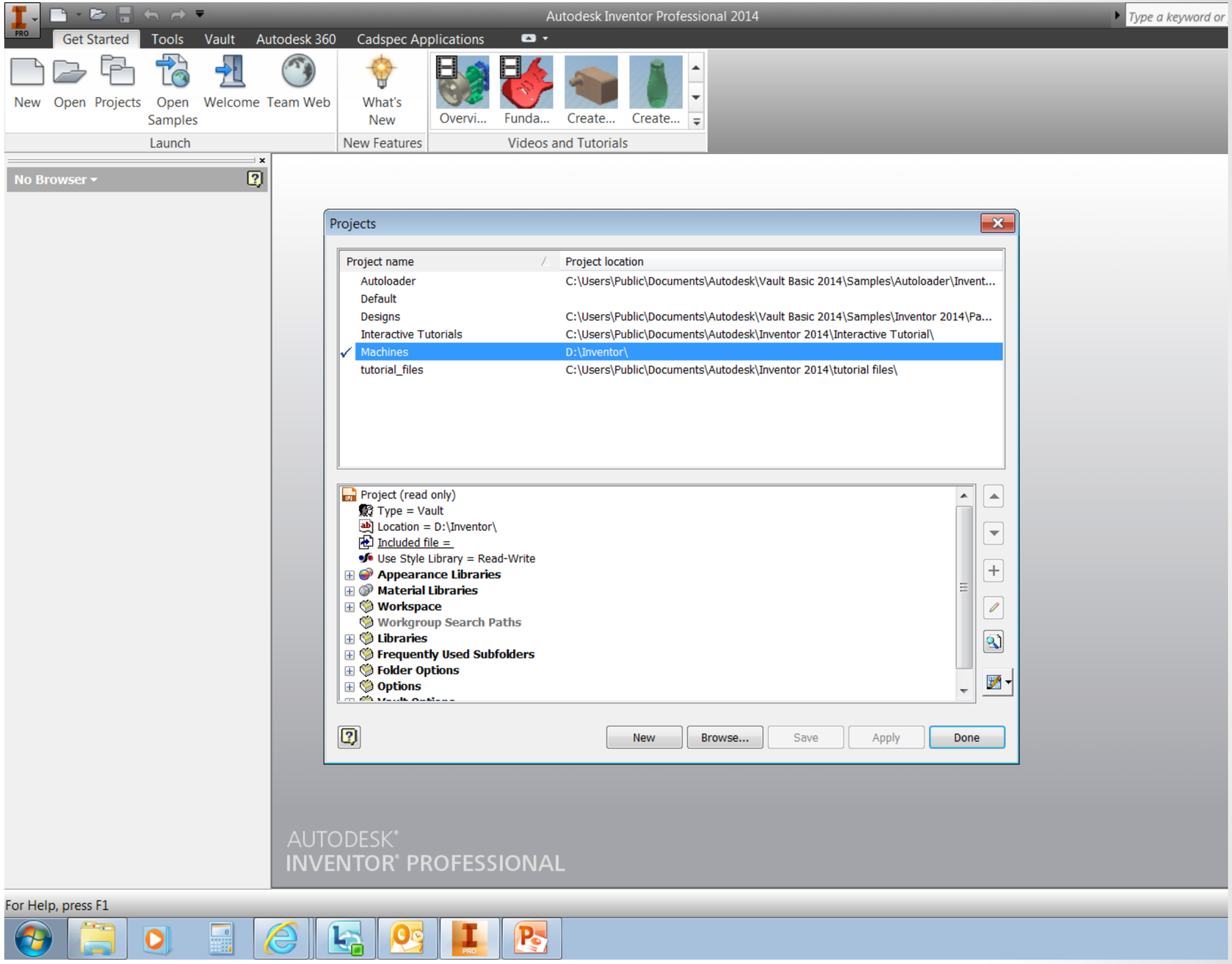
How many projects (ipj) should, or could have.

A: Definitely only one.

B: As many as you like, go crazy

C: It depends on how you are setup.
Also who you ask.

D: Nuts, all of the above.

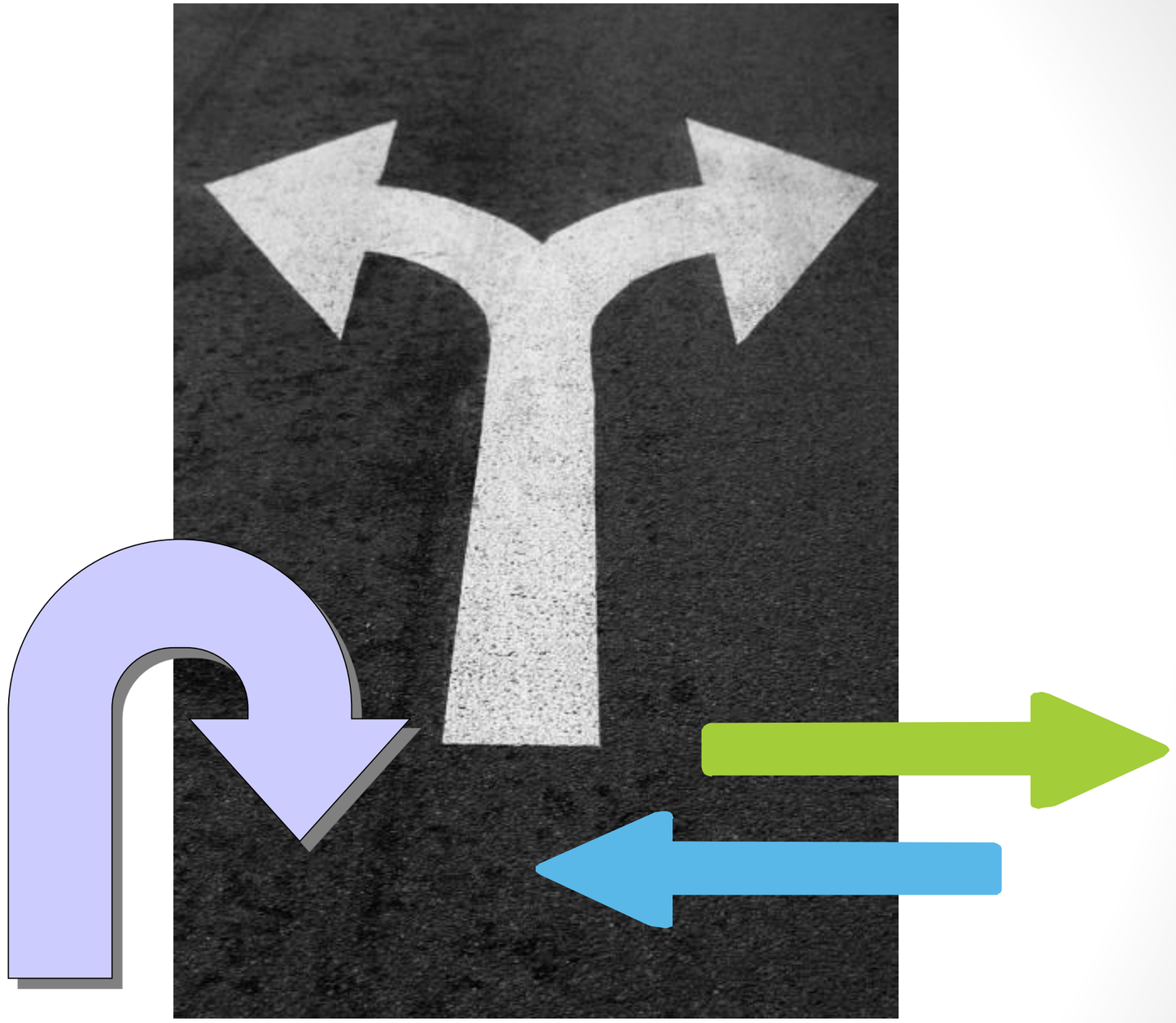


D: Nuts, all of the above.

If you are working with Vault Basic you will be working with only one.

If you are not going to vault the data, you can have as many as you like.

You can keep asking application's engineers until you get an answer you like.



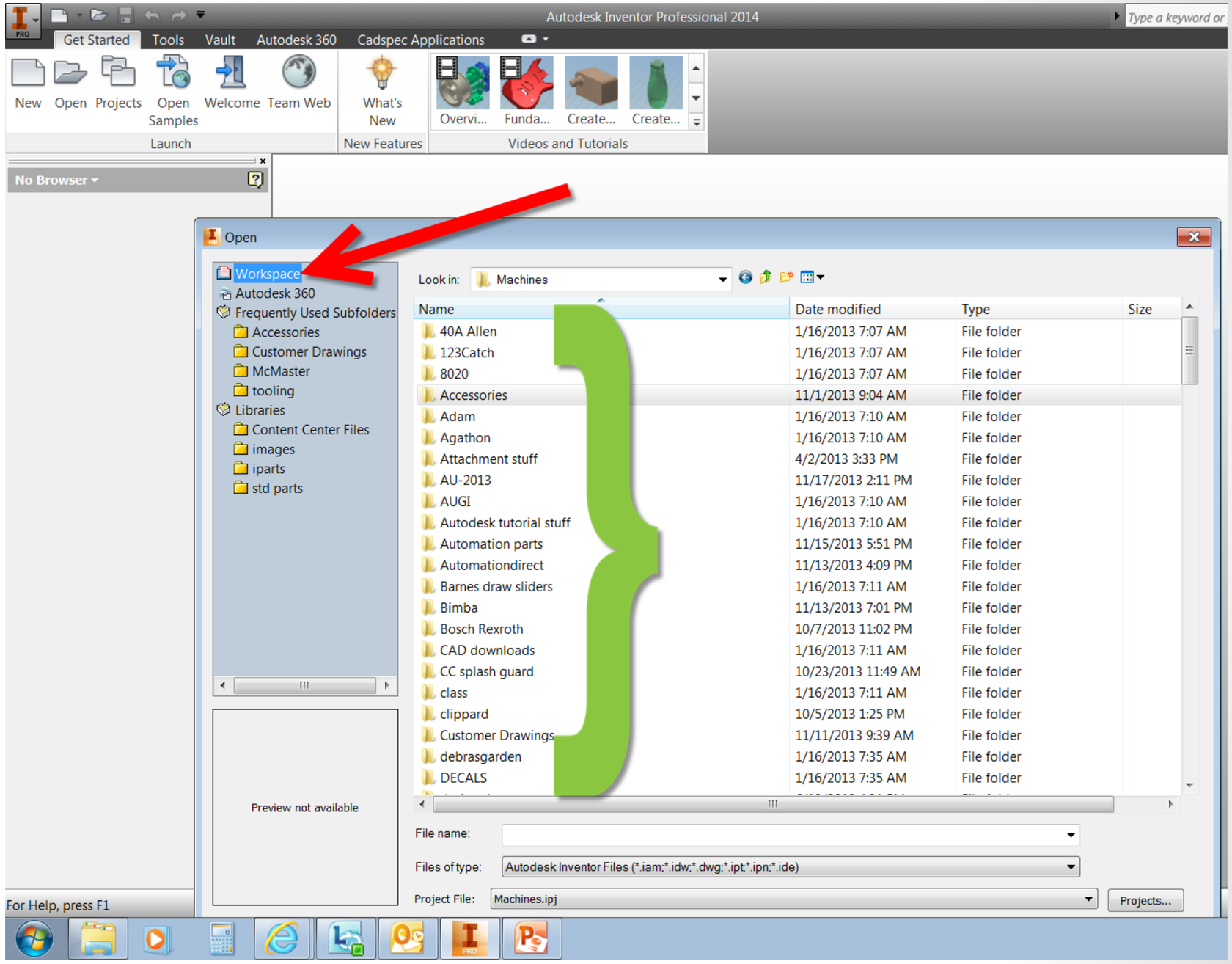
If my content is Vaulted, should I keep the workspace copies?

A: Maybe, Only if you want to.

B: Yes, Vault will not work without the workspace copies in place.

C: No, you should delete them. You should not have old copies of data that will mess up your Vault.

D: This is a trick question. The workspace is not related to your data.



A: Maybe, Only if you want to.

You do not need to keep your workspace files if you don't want to. You can open the files from the vault every time you need them. This way the files will always be the current versions.

However my work style forces me to keep my workspace data. I have a laptop and am constantly on the road. I keep files on my local drive so I can work where ever I am.

I then am careful about how I deal with data in my vault at the office. That topic is more than can be covered here.

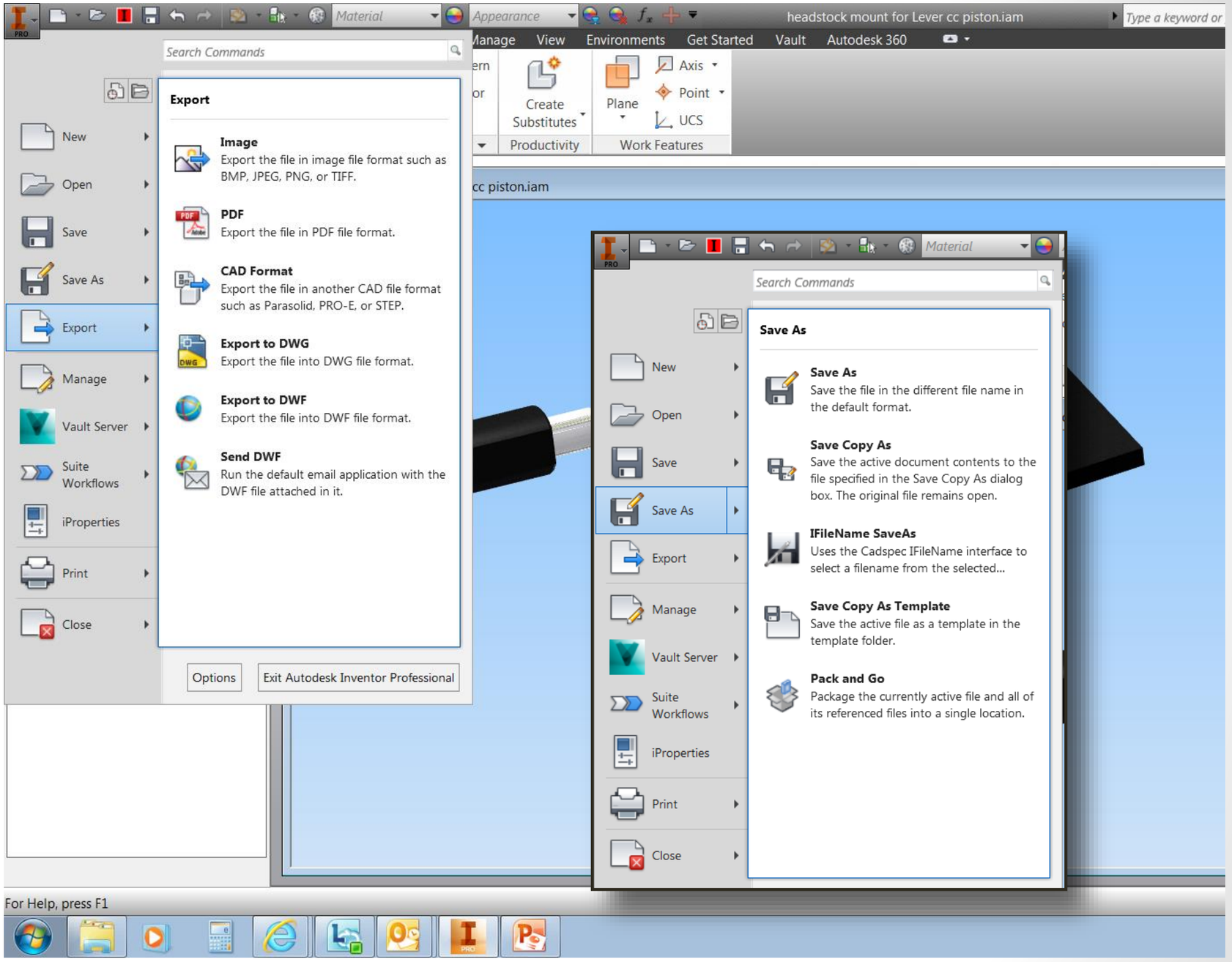
I want to share my assembly with a colleague. He is far away so I need to send him the files. When he gets the data he will be working with the files in Inventor. How do I send him the data?

A: Export a DWF complete file.

B: Trick question, you only need to send him the .iam file. What could be simpler?

C: Pack and Go

D: Export as a STEP or SAT file.



C: Pack and Go.

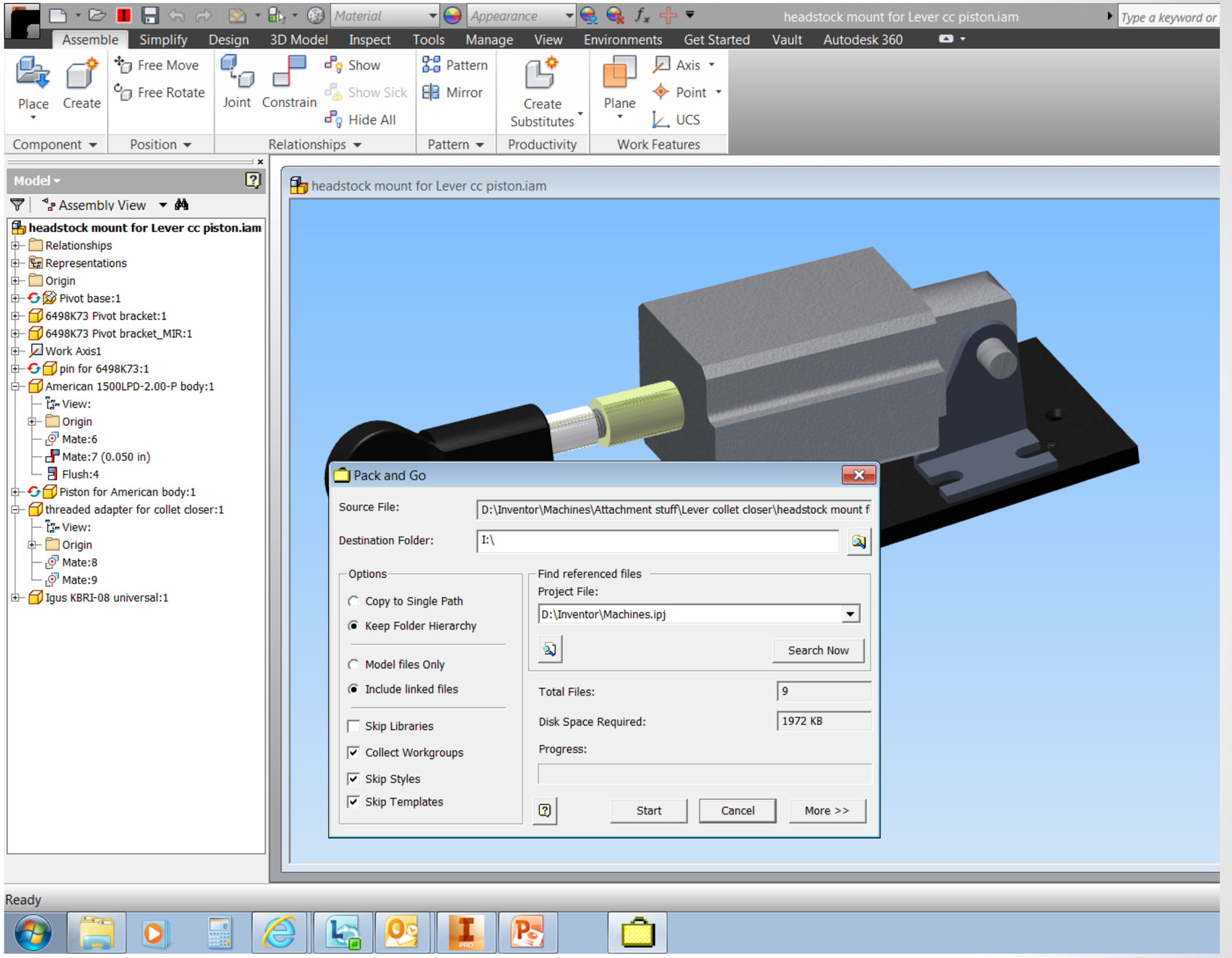
Pack and Go gets it done.

LMC the I located on the top left of the Inventor Ribbon.

Hoover over the Save As command, the additional menu opens up, select Pack and Go.

You then save as complete a copy of your assembly, parts, and linked files (if you want) as you need.

Your colleague will be able to work with the resulting copy as if working on your original.



A: JPG, TIFF, GIF

B: TSA, DEA, DOA

C: PNG, BMP, PDF

D: How foolish, none of the above.

Which files are CAD Usable 3D files?

D: NONE of those were CAD files.

Here is a list of files types that Inventor will open.

- Autodesk Inventor Files (*.iam;*.idw;*.dwg;*.ipt;*.ipn;*.ide)
- Autodesk Inventor Assemblies (*.iam)
- Autodesk Inventor Drawings (*.dwg; *.idw)
- Autodesk Inventor Parts (*.ipt)
- Autodesk Inventor Presentations (*.ipn)
- Autodesk Inventor iFeatures (*.ide)
- Alias Files (*.wire)
- AutoCAD DWG Files (*.dwg)
- CATIA V4 Files (*.model;*.session;*.exp;*.dlv3)
- CATIA V5 Files (*.CATPart;*.CATProduct;*.cgr)
- DWF Markup Files (*.dwf;*.dwfx)
- DXF Files (*.dxf)
- IDF Board Files (*.brd;*.emn;*.bdf;*.idb)
- IGES Files (*.igs;*.ige;*.iges)
- JT Files (*.jt)
- NX Files (*.prt)
- Parasolid Binary Files (*.x_b)
- Parasolid Text Files (*.x_t)
- Pro/ENGINEER Granite Files (*.g)
- Pro/ENGINEER Neutral Files (*.neu*)
- Pro/ENGINEER and Creo Parametric Files (*.prt*;*.asm*)
- Revit Project Files (*.rvt)
- Rhino Files (*.3dm)
- SAT Files (*.sat)
- STEP Files (*.stp;*.ste;*.step)
- STL Files (*.stl;*.stla;*.stlb)
- SolidWorks Files (*.prt;*.sldprt;*.asm;*.sldasm)

In a perfect world all Inventor users would have the current version. Since this is not the case, how many versions back can you save Inventor files?

A: One.

B: Two.

C: All the way back to Inventor10.

D: The Rolling Stones said it best, “You can’t always get what you want”.

D: The Rolling Stones said it best, “You can’t always get what you want”.

Sorry, there is no saving backwards. Even if you believe in Santa, it won’t happen.

The way I have worked around this is to save my parts and assemblies as STEP files. This solution does not give the recipient an assembly with history, constraints, properties and more. However they can place the model and use it as drawn.

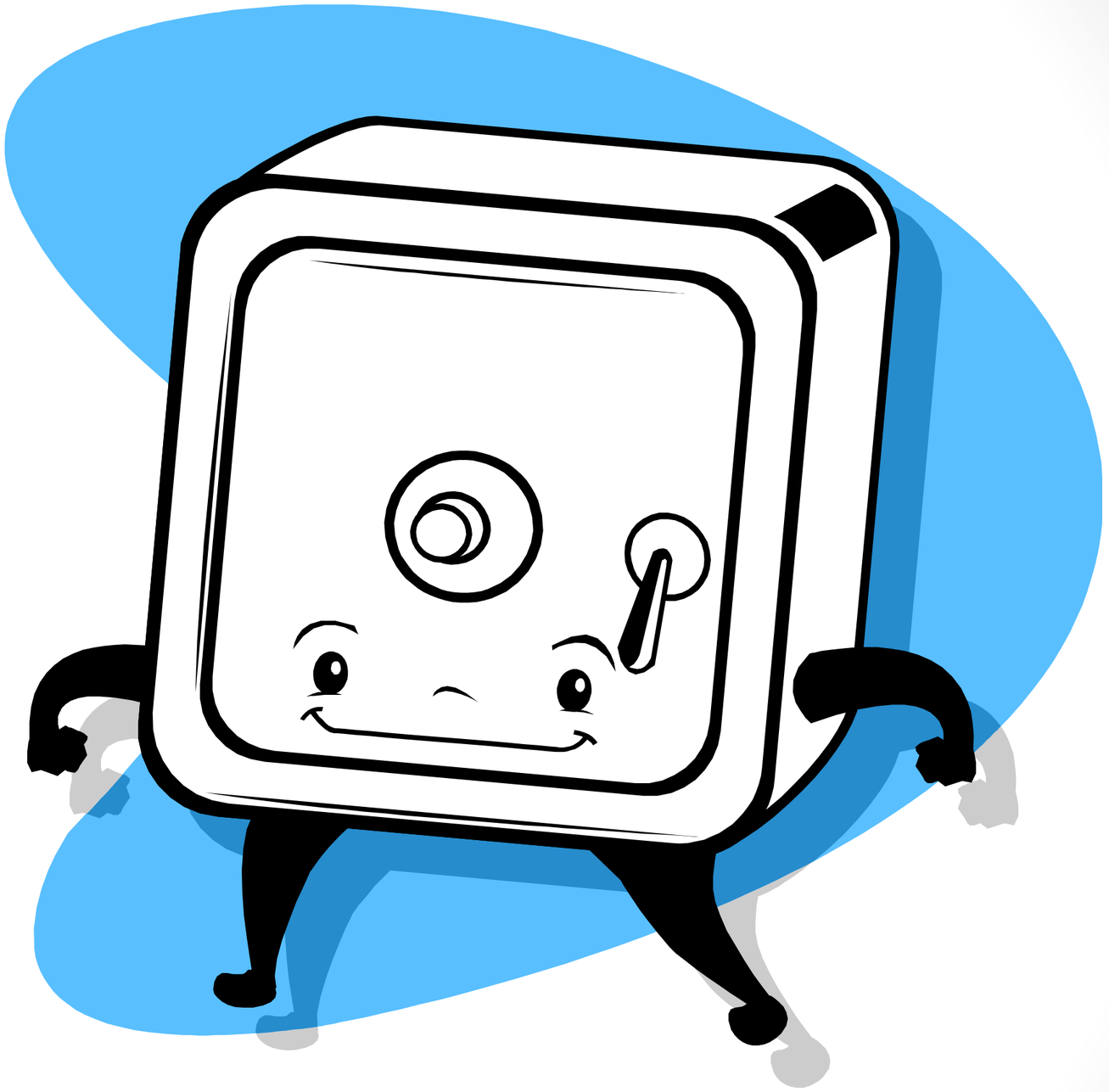
I work for a small company. We have just bought a new server for engineering (amazing but true). Now we need to move the “file store” to the new server. What can I do to save time and money?

A: Simple, copy the “file store” and just paste it in the new server.

B: Move the workspace, then tell the ADMS were the new home is.

C: Check out the entire Vault to your workspace, Create a new Vault. Check in the workspace to the new Vault.

D: This is not a good time to experiment. Call in an expert and be done without major problems.



D: This is not a good time to experiment. Call in an expert and be done without major problems.

Yes, there is enough documentation out there to help you move your file store without help. However since this is not what you do, This is a time you should pay for help. Think of it like fixing pluming in your house. If you get lucky it will go well. If it goes wrong, it can go very wrong.

If you are going to tackle the task, just make sure you backup your Vault and save the data someplace safe.

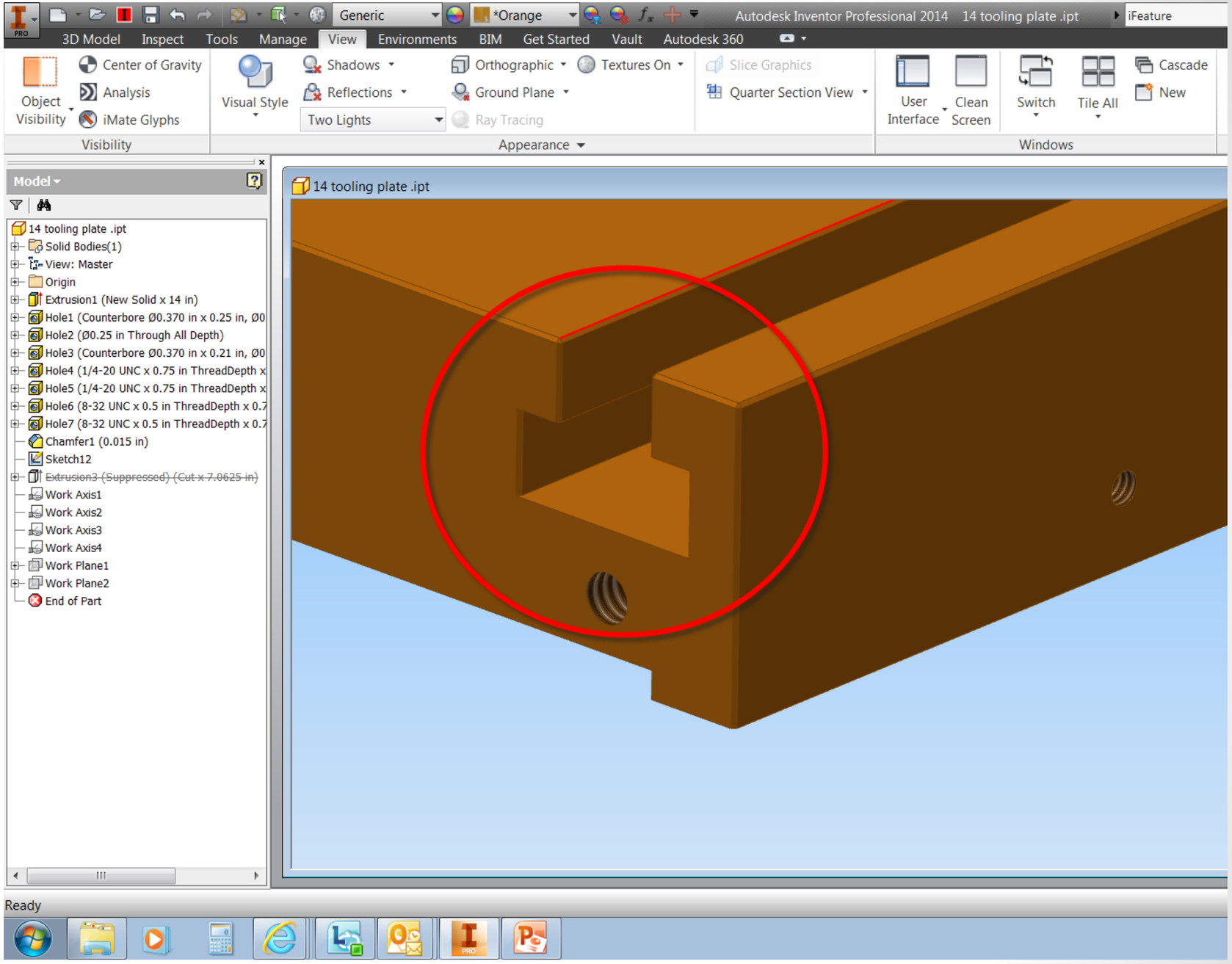
You are going to be using this T slot over and over again through out your company's tooling catalog. How do you best capture this detail so it can be used quickly and by more than yourself?

A: A fine time to learn how to use iFeatures.

B: Copy and save the sketch. Save the sketch as an ipt.

C: Vault the tooling plate, then use copy design to reuse the detail.

D: iLogic will save the day. Spend some time with the tool to find out why.

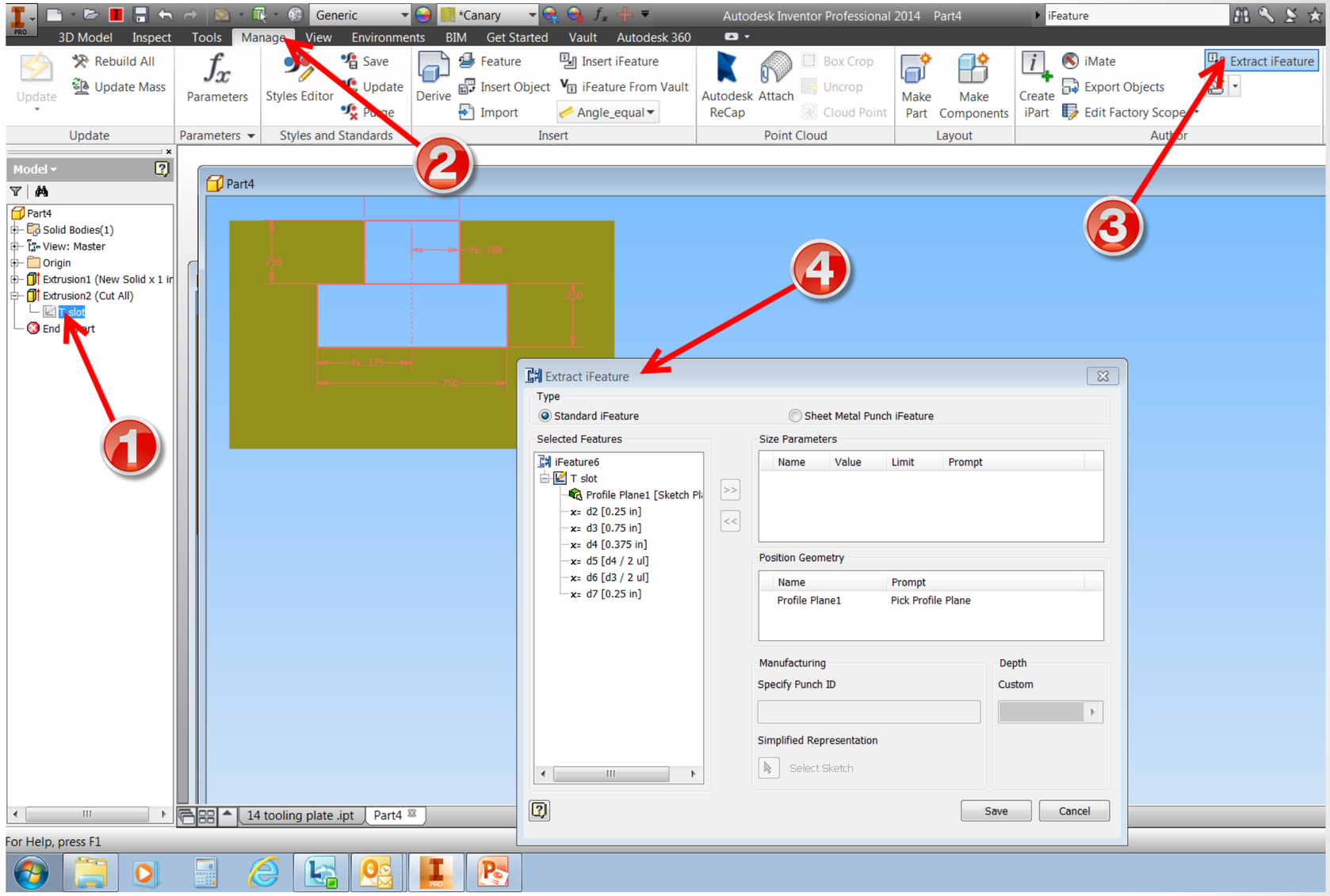


A: A fine time to learn how to use iFeatures.

iFeatures are so under utilized. Learn how to use them. **That's an order!**

For a bonus question (worth no points):

Is the feature used associative back to the iFeature?? So if you make changes to the iFeature will it update where it has been used previously?



What is the Inventor equivalent of AutoCad’s Dynamic blocks?

A: iParts

B: Sketchs

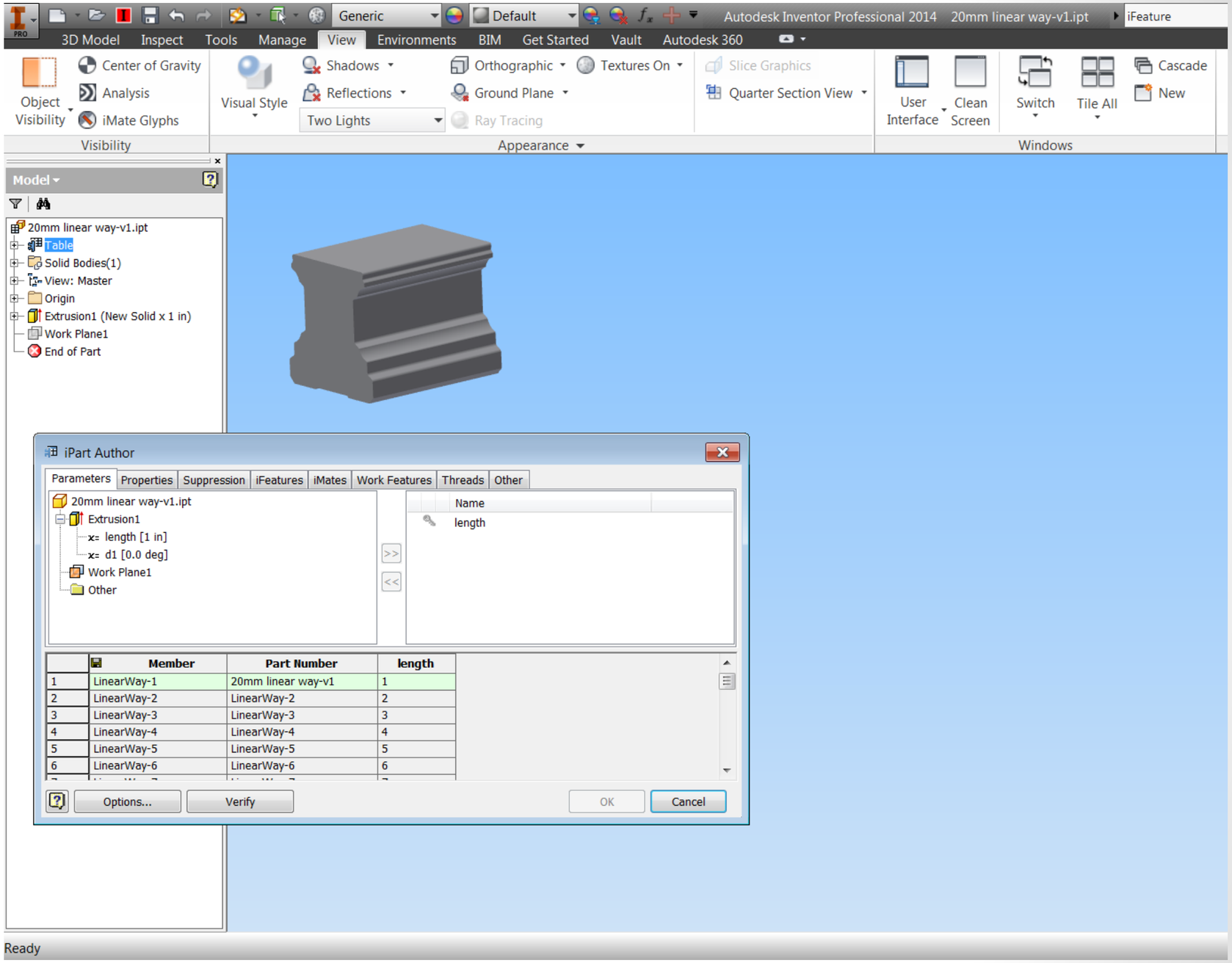
C: Sketch Blocks

D: iSketch

A: iParts

The iPart tool is so powerful in so many ways. Once I started using it I have found so many applications. Take the time to experiment and fit it in to your daily workflow.

Note:
You can also create Inventor Sketch Blocks, these are very different from iParts. These are similar to skeleton modeling



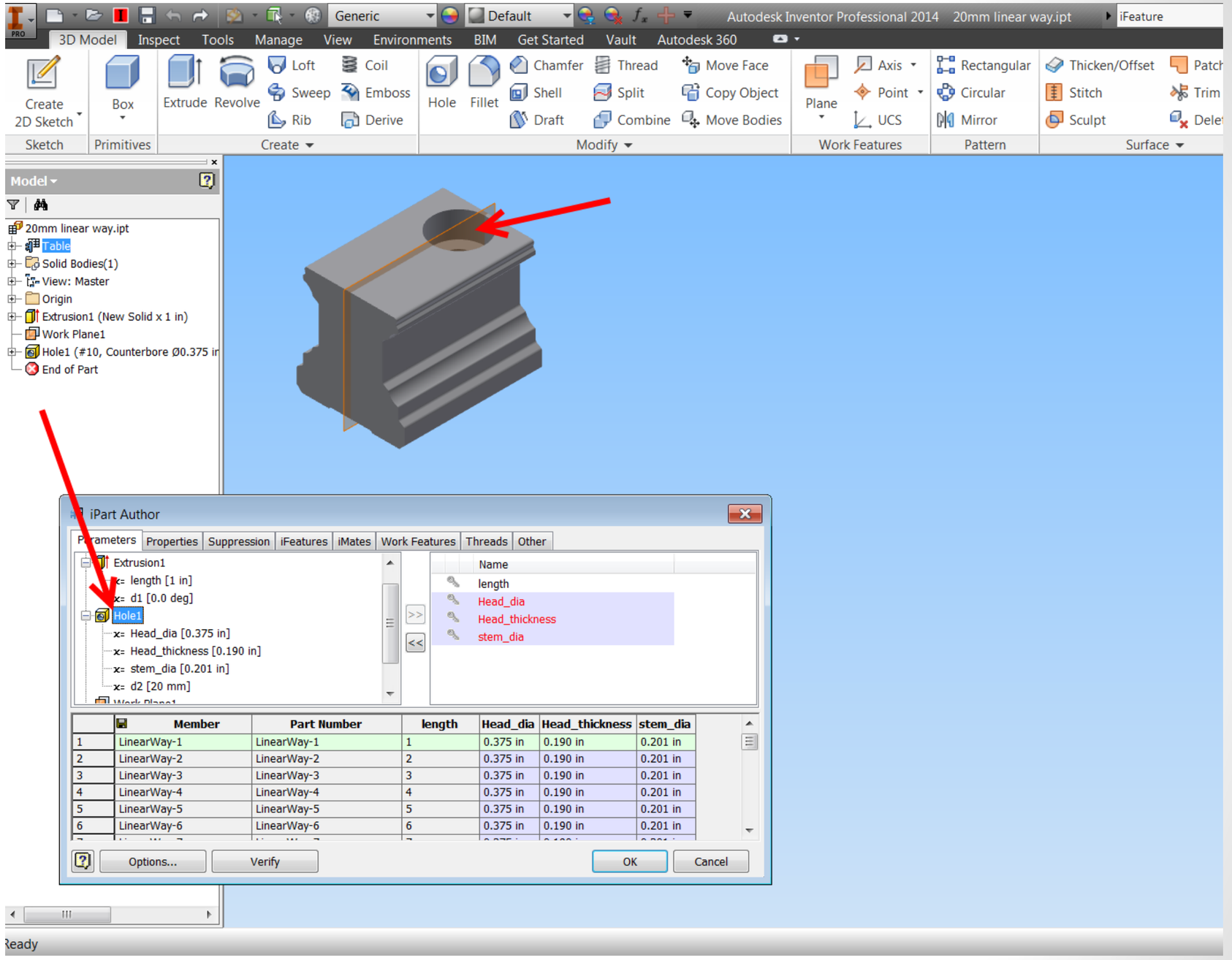
I needed to change my iPart parent. In this case I am adding a new feature, a c'sinked hole. When will the assemblies that use a child of this iPart get updated?

A: When I save the parent

B: When I open the assembly or drawing

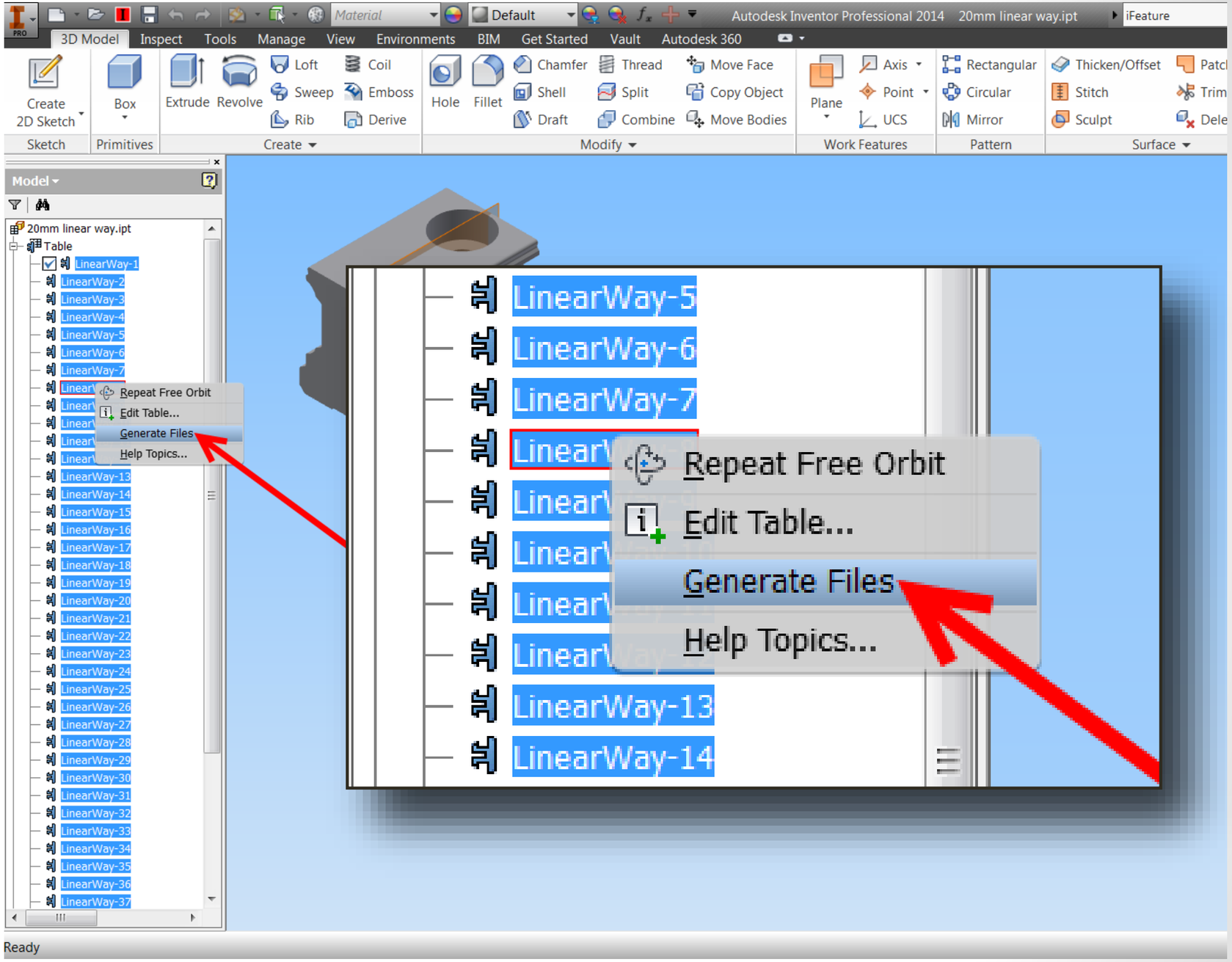
C: When I close the parent

D: When the children are generated



D: Generate Files

The children do not change until they are generated. So after you modify the parent. Select the children to be re-generated and then make it so.



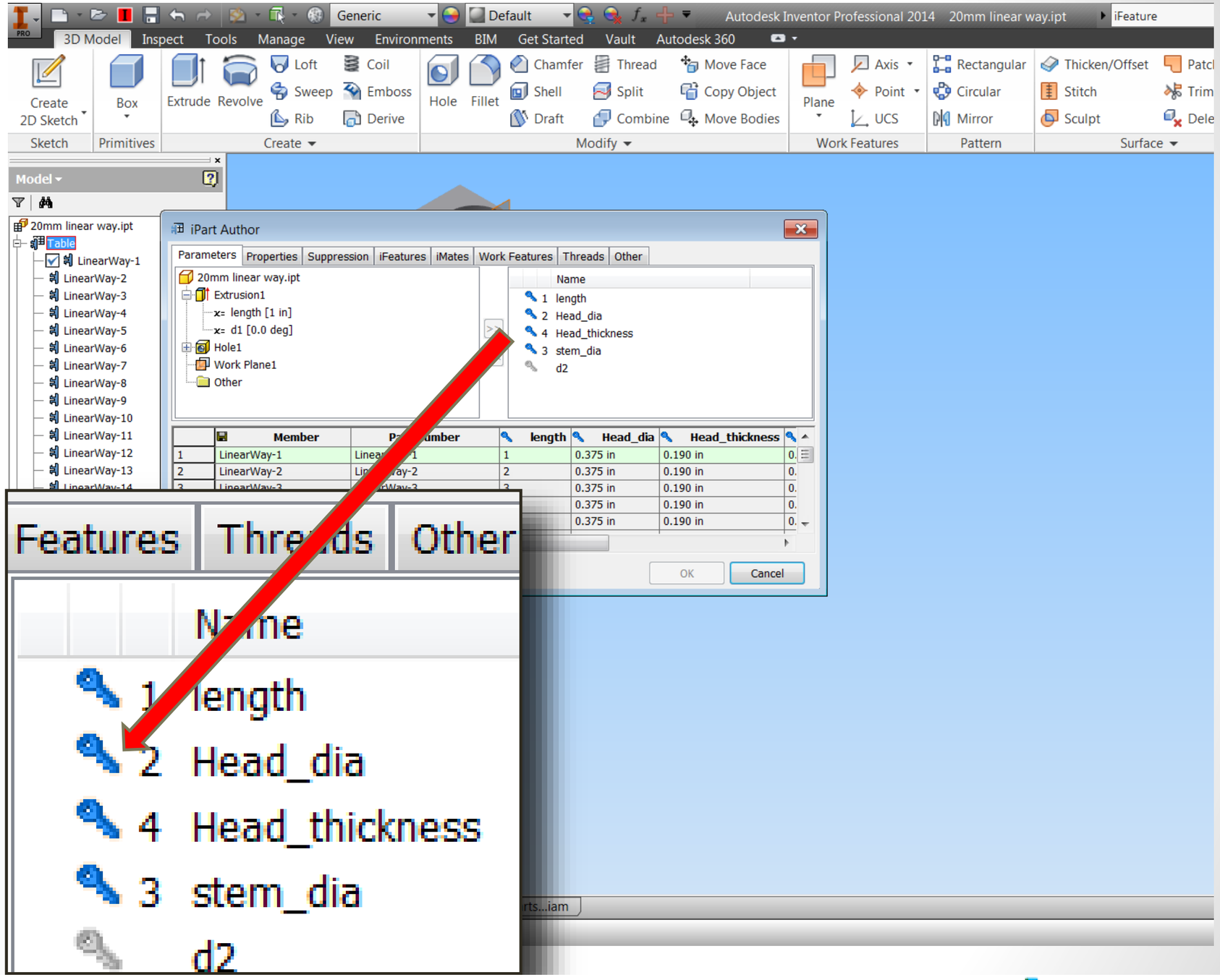
When I create an iPart, what does making a value Key do?

A: Effects what is constrained when placed.

B: Relates to iProperties. What can be saved or not.

C: Determines what details are allowed to be configured when the part is being placed.

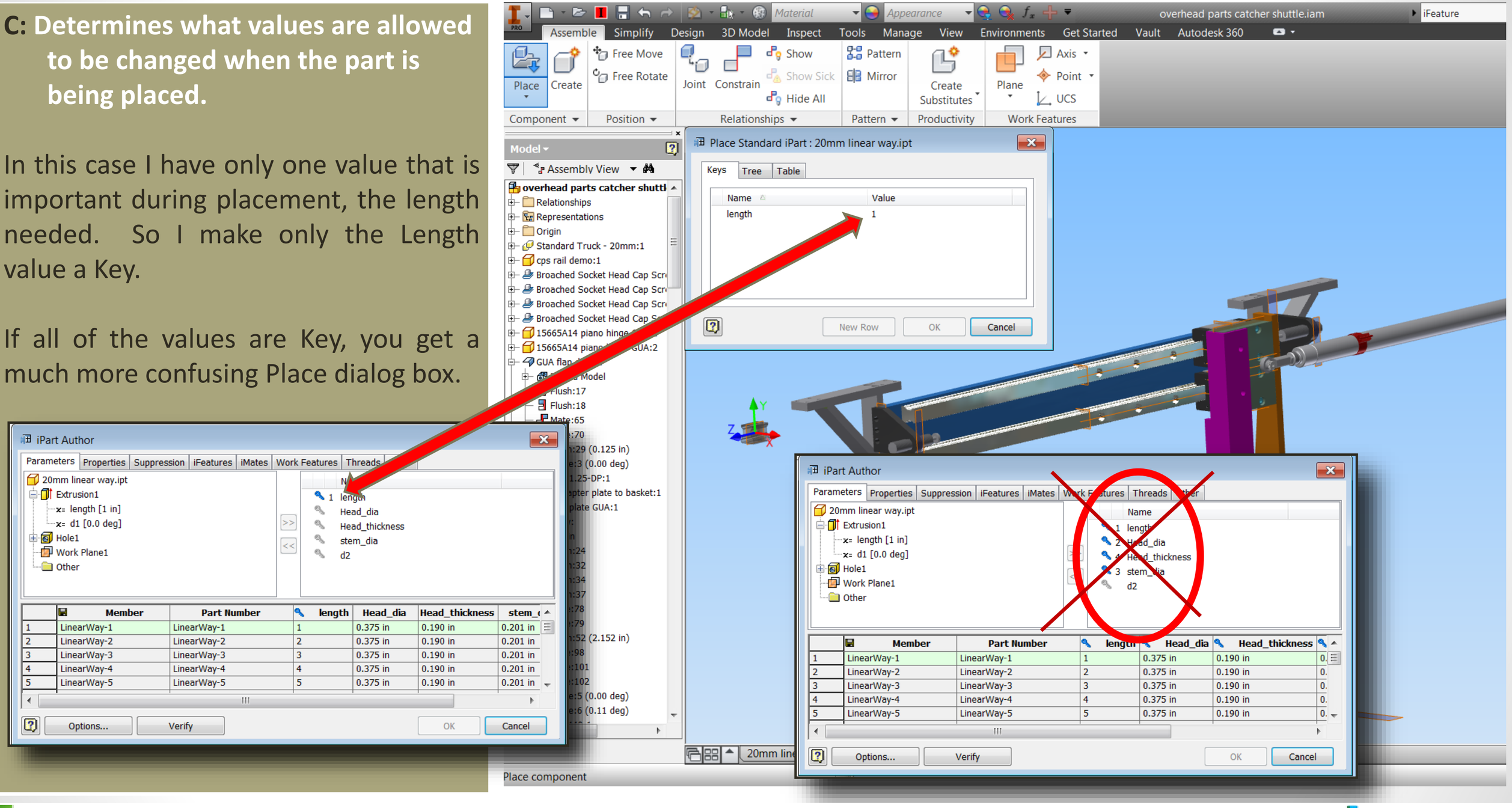
D: Enables you to sort values based on Key.



C: Determines what values are allowed to be changed when the part is being placed.

In this case I have only one value that is important during placement, the length needed. So I make only the Length value a Key.

If all of the values are Key, you get a much more confusing Place dialog box.



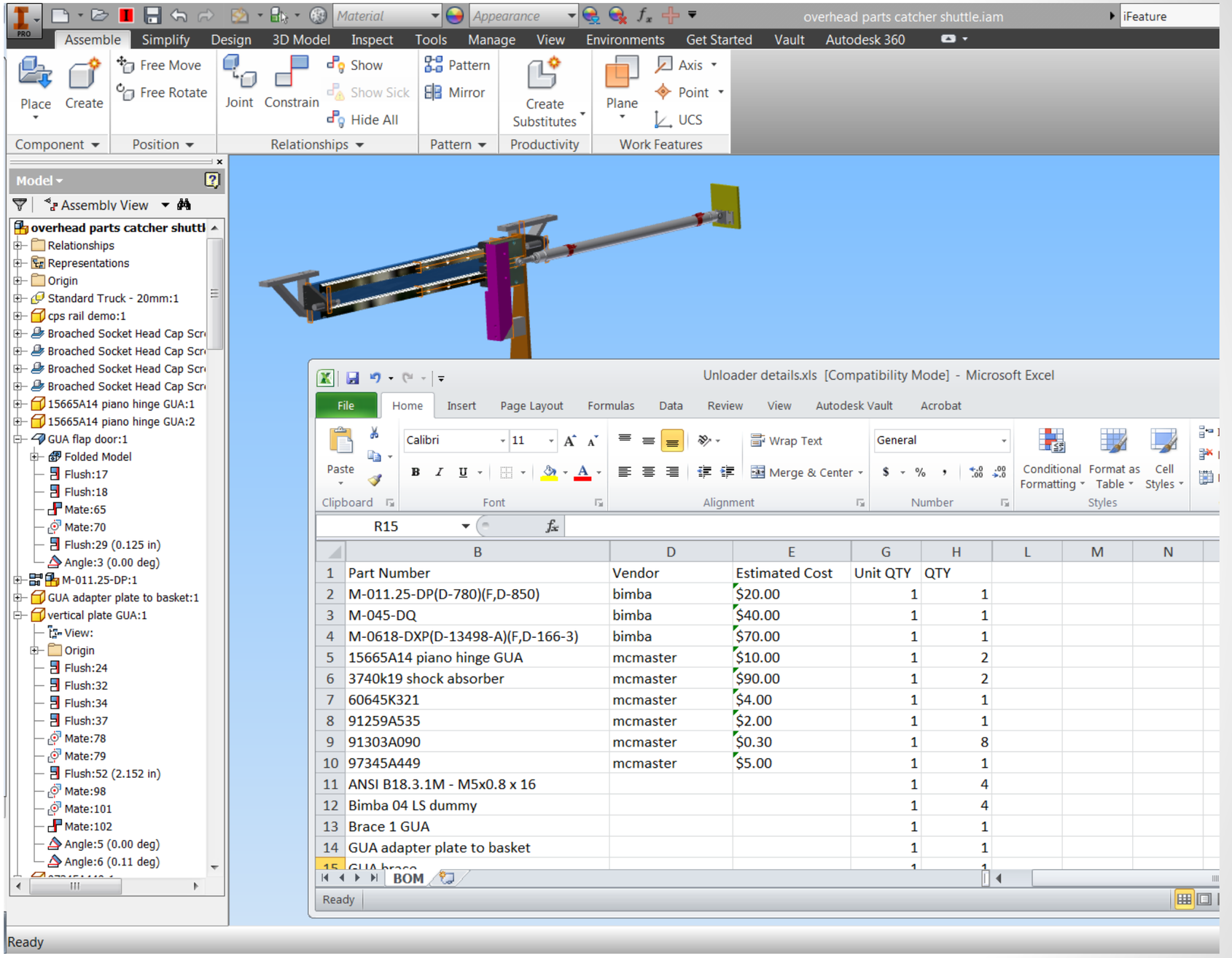
I want to give my purchasing department a spread sheet with items needed, estimated cost, and possible vendor. How do I generate this spreadsheet?

A: Vault comes to the rescue again.

B: BOM and iProperties.

C: Create a drawing, then use the Parts List tool.

D: Hire an intern, have him go through the assembly.

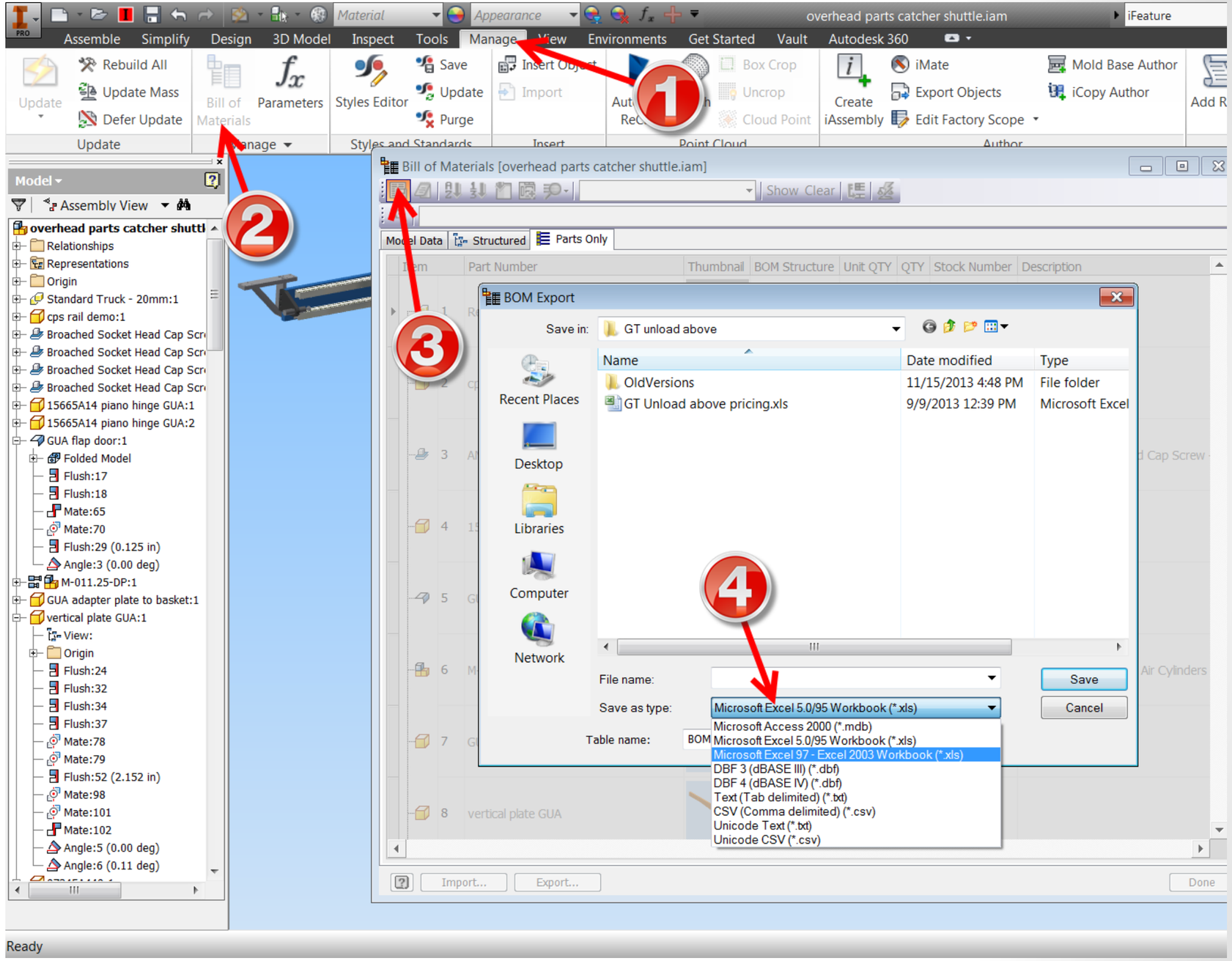


	B	D	E	G	H	L	M	N
1	Part Number	Vendor	Estimated Cost	Unit QTY	QTY			
2	M-011.25-DP(D-780)(F,D-850)	bimba	$20.00	1	1			
3	M-045-DQ	bimba	$40.00	1	1			
4	M-0618-DXP(D-13498-A)(F,D-166-3)	bimba	$70.00	1	1			
5	15665A14 piano hinge GUA	mcmaster	$10.00	1	2			
6	3740k19 shock absorber	mcmaster	$90.00	1	2			
7	60645K321	mcmaster	$4.00	1	1			
8	91259A535	mcmaster	$2.00	1	1			
9	91303A090	mcmaster	$0.30	1	8			
10	97345A449	mcmaster	$5.00	1	1			
11	ANSI B18.3.1M - M5x0.8 x 16			1	4			
12	Bimba 04 LS dummy			1	4			
13	Brace 1 GUA			1	1			
14	GUA adapter plate to basket			1	1			

C: BOM and iProperties

After you add the iProperty columns you want, then export your BOM as an Excel spreadsheet.

1 – go to the Manage Tab
2 – Select the Bill Of Materials
3 – Select BOM Export
4 – Select the format to export



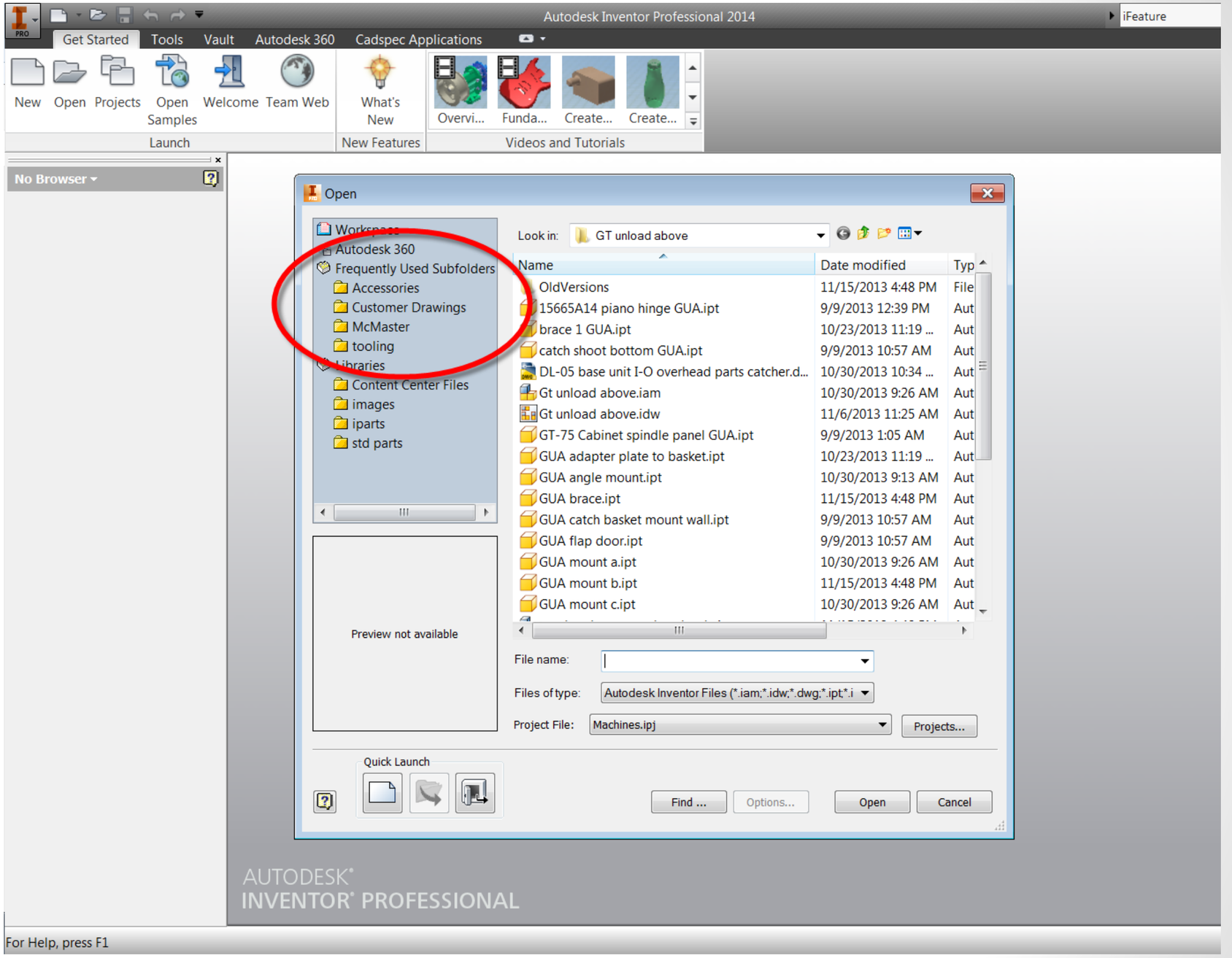
Frequently Used Subfolders are great for getting around your work workspace faster. No longer having to wander around opening folder after folder. Where do you go to set them up?

A: Application Options

B: Document Settings

C: Project File

D: Windows operating system



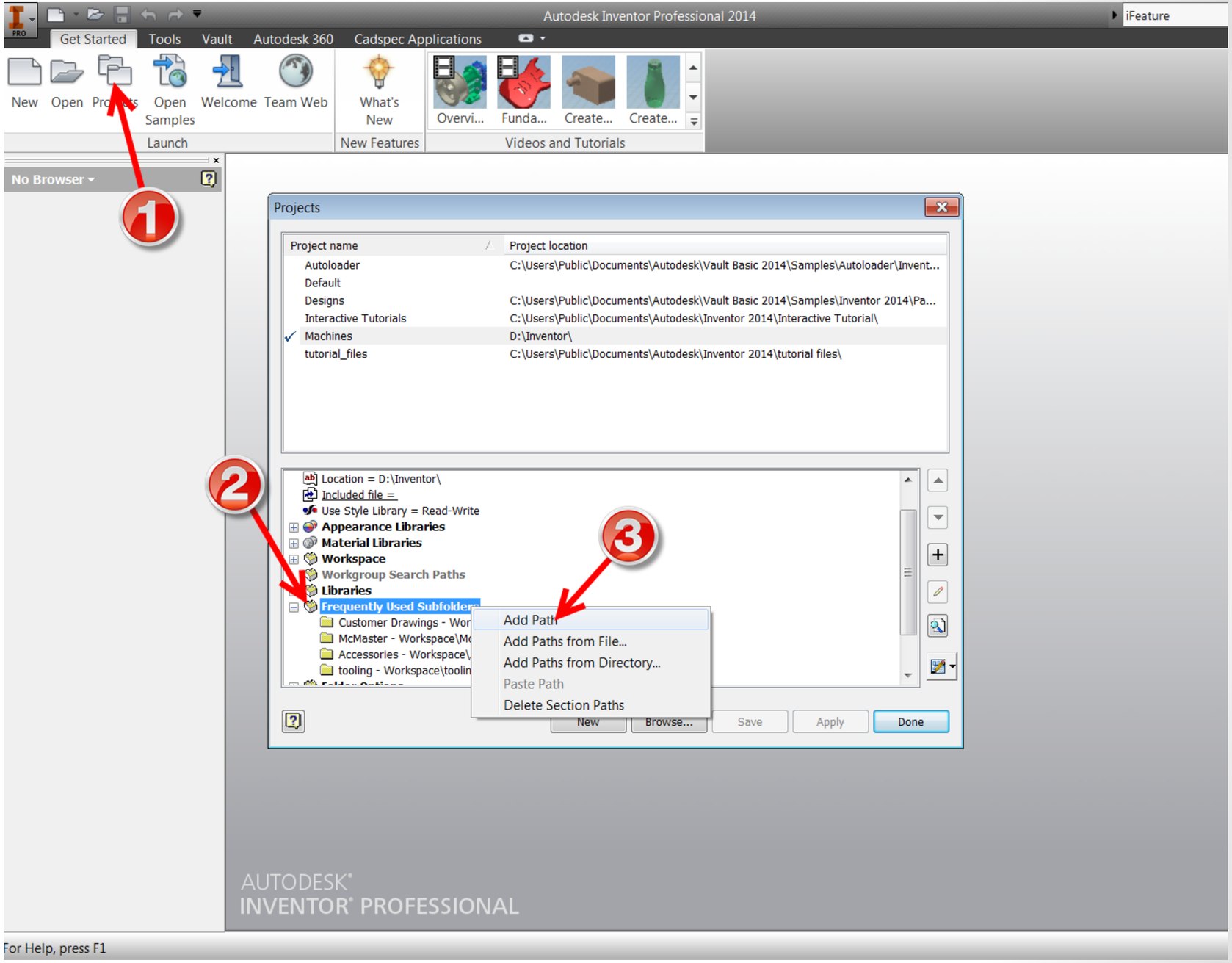
C: Project File

The Project file is where the Frequently Used Subfolders are setup.

If your project file has been vaulted and checked in, you must first check it out.

RMC the Frequently Used Subfolder heading, then select the modification to the folders.

Once you create a new folder, make sure you give it a name other than the default “folder”.



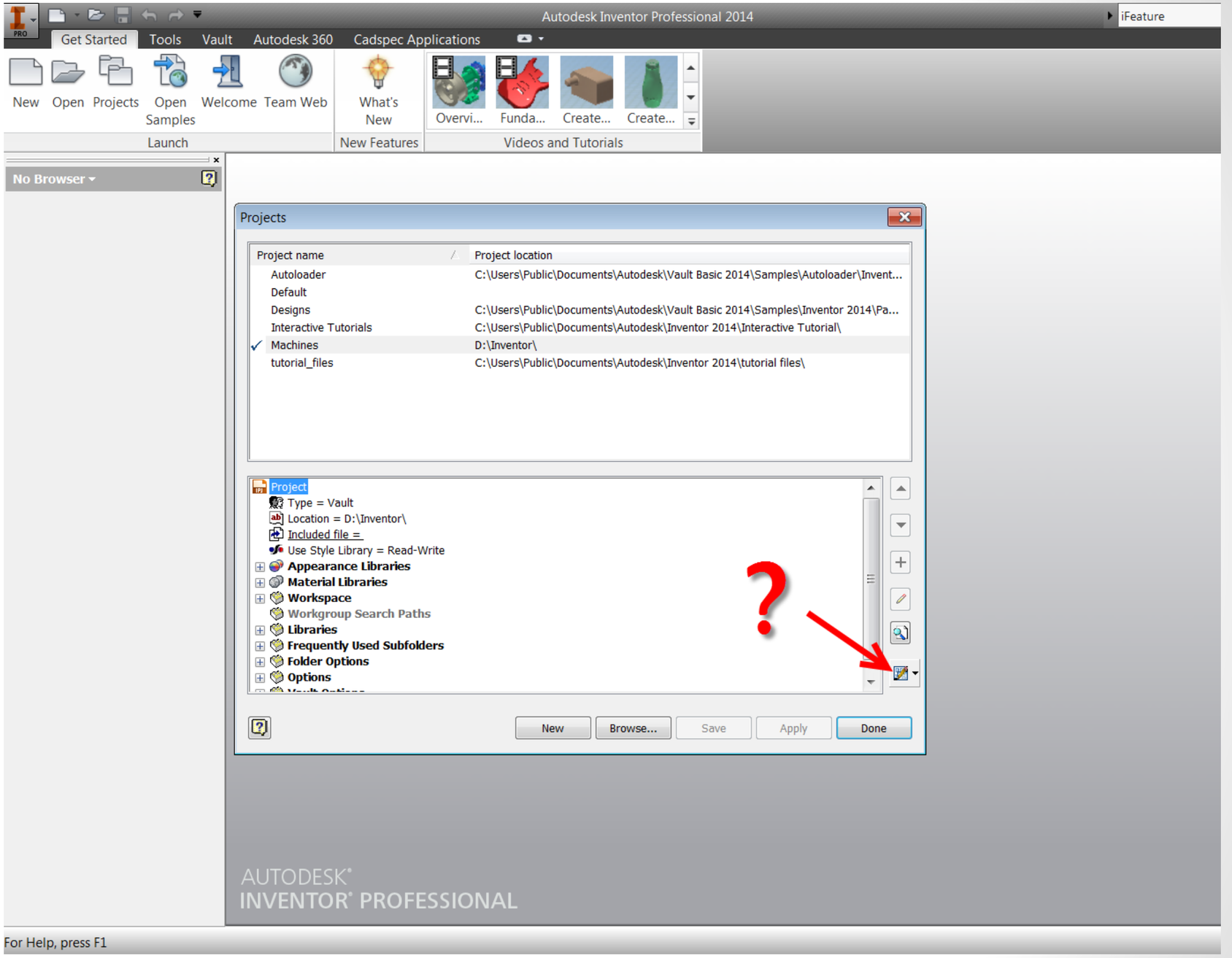
In the Projects dialog box, what does this button do?

A: Gets you to the Configure Content Center Libraries dialog box

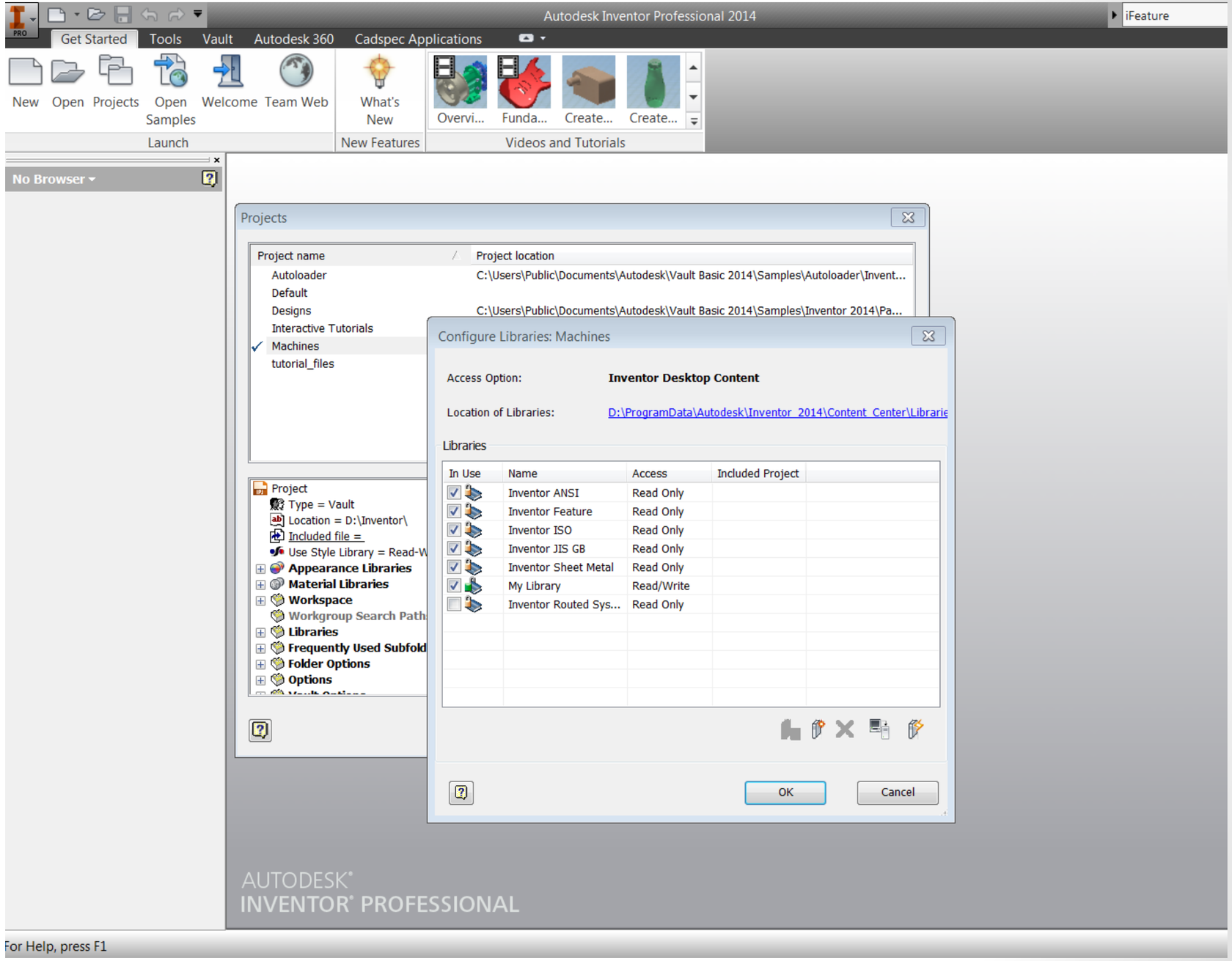
B: Gets you to the Configure <u>your</u> libraries dialog box

C: Share your Project file with the rest of your team

D: Launches a counter attack on the marketing team of the company.



A: Gets you to the Configure Content Center Libraries dialog box



What spreadsheet programs can you use to work within Inventor?

A: GoogleDocs

B: OpenOffice

C: Excel

D: All of the above

A: Excel

One of those times when the choice has been made for you.

You have a customer that is trying to move faster than he should. He needs about 30 different adapters. The thread stays the same for all, the heads vary.

He knows the head dimensions, just the thread form has not been finalized.

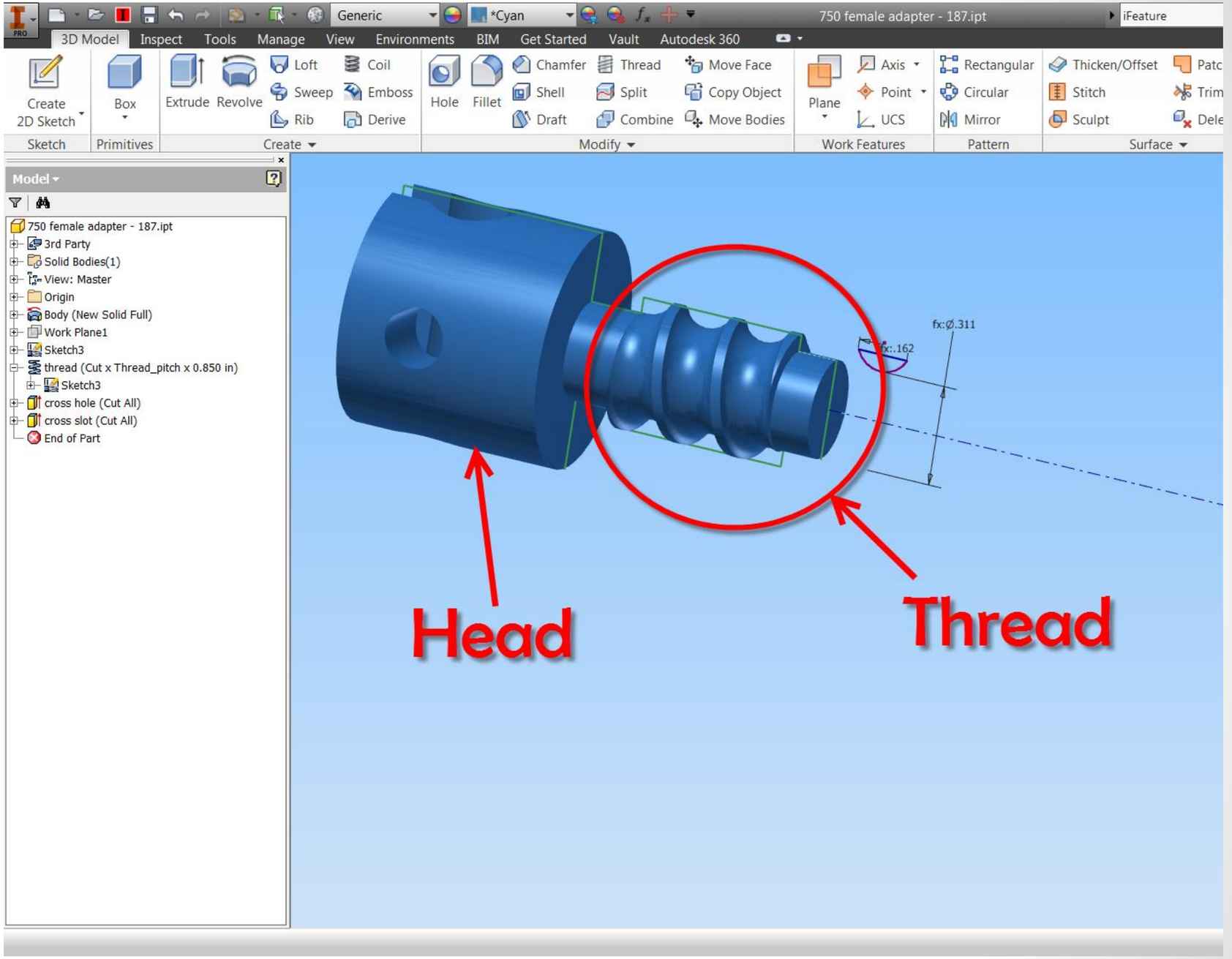
Since he is in a rush, he wants you to start now. Draw all the parts with the same thread, then fix them all later when that details are finalized. From the tools below, what could help you get it done in a timely manor.

A: Design Accelerator

B: Excel

C: Design Assistant

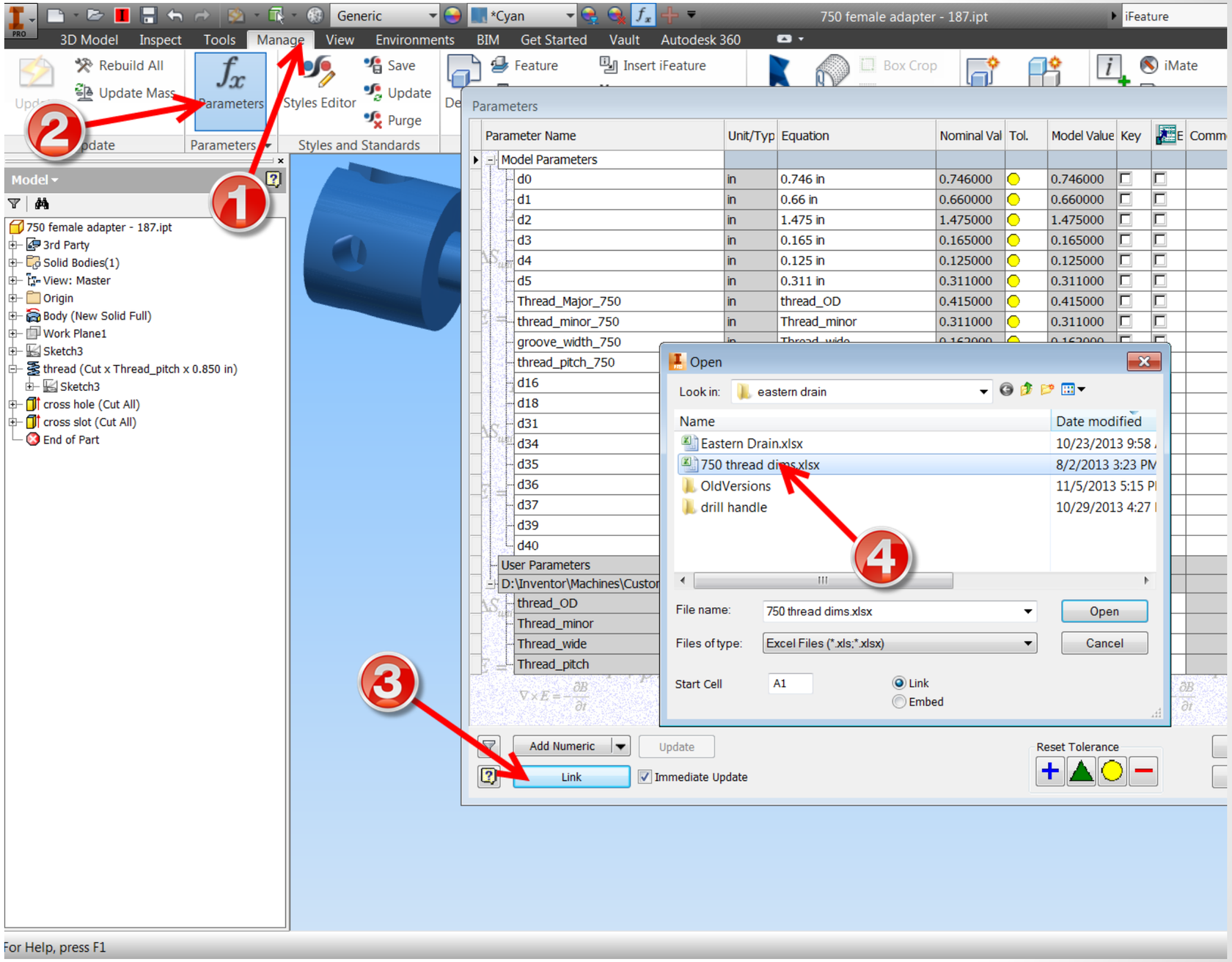
D: Vault



B: Excel

Parameters come to the rescue.

You can setup User Parameters that are linked to a spreadsheet. If the dimensions of the thread form are all linked to the same spreadsheet all the parts can be changed at one time.



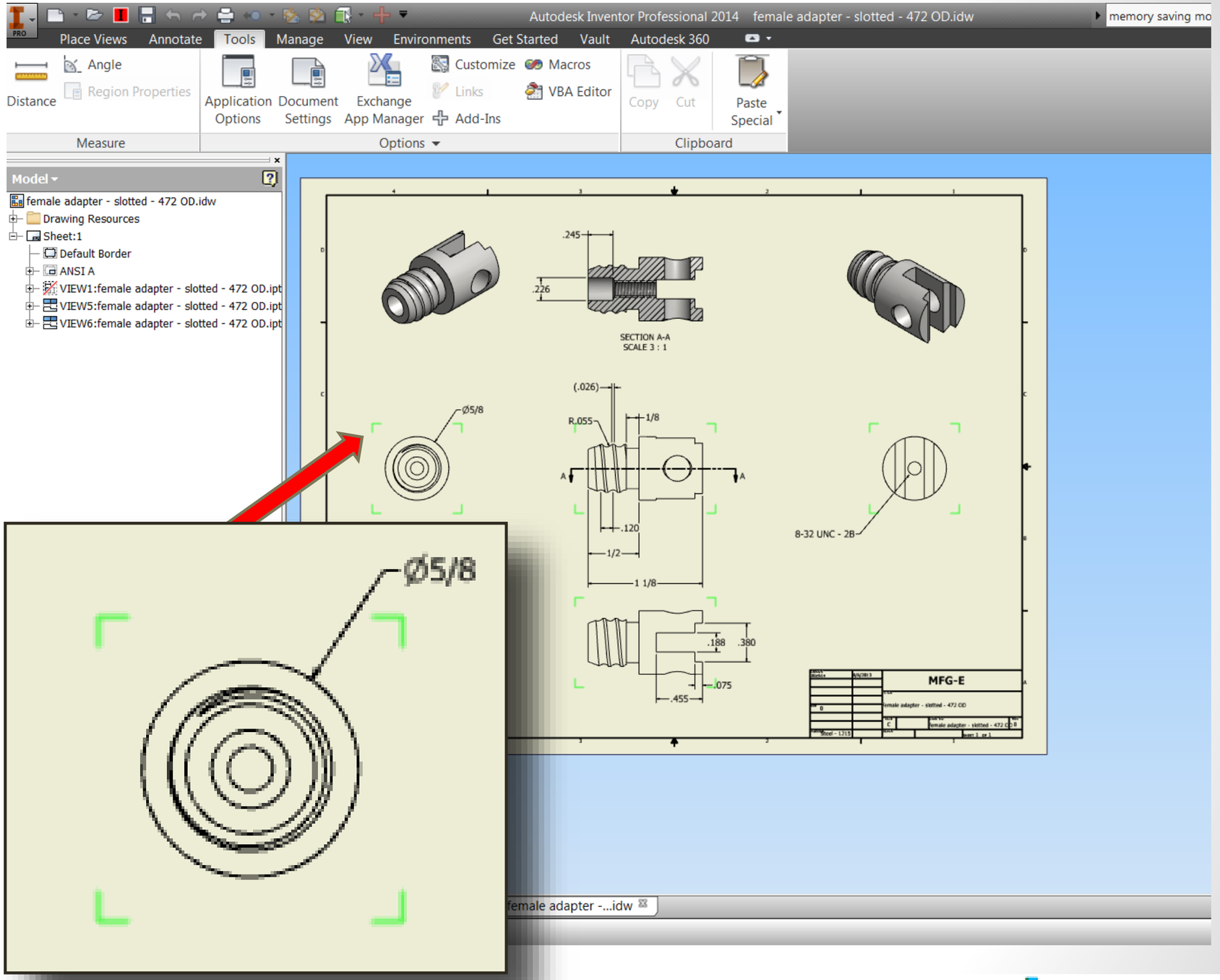
The green corners around the drawing views denote that the image is in Raster View Only or not completely calculated. The views are now not as sharp and clear as they could be. How could this happen?

A: Setting in Application Options.

B: Document Settings setting.

C: Drawing View Setting.

D: Could be any or all of the above.



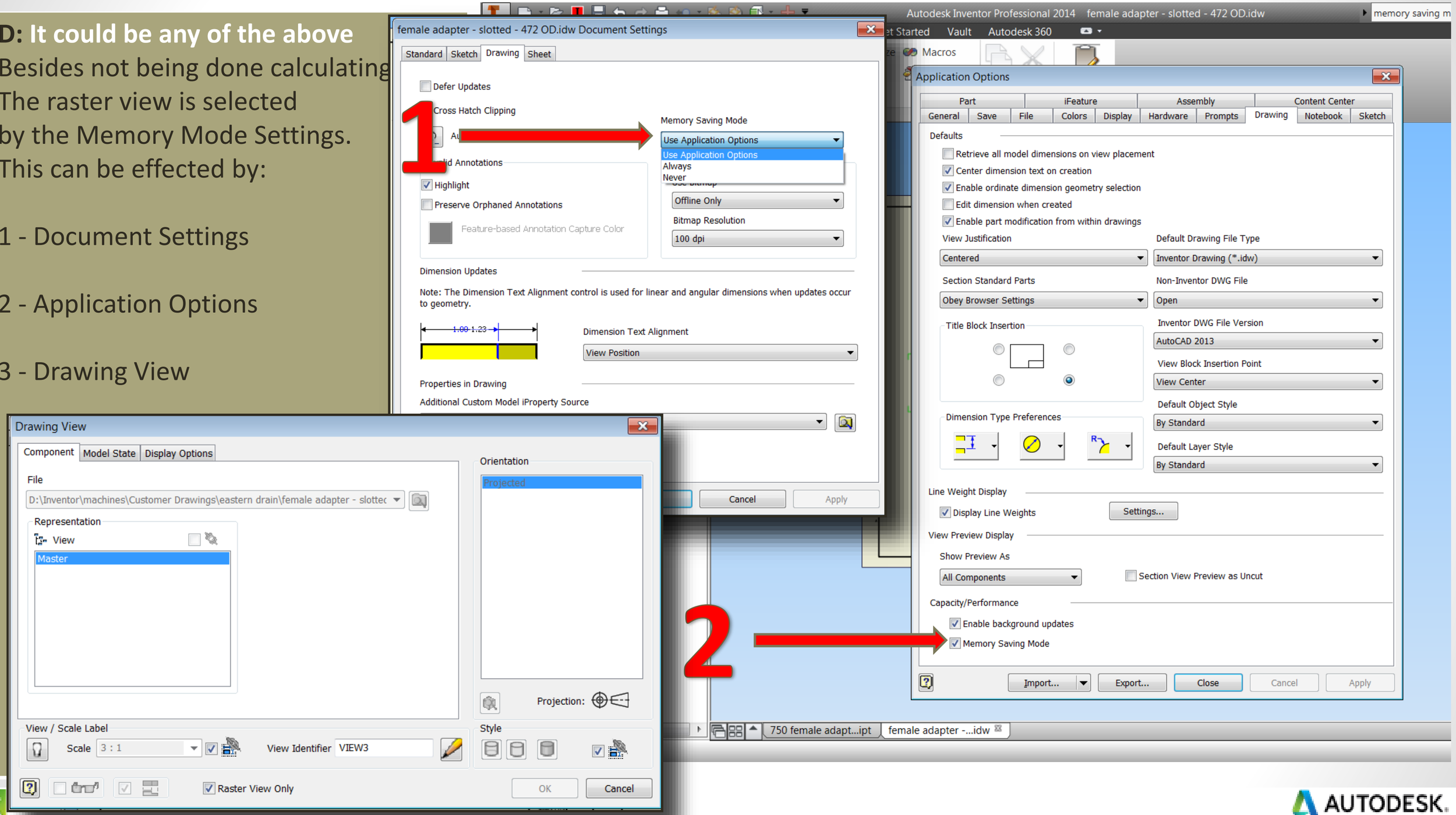
D: It could be any of the above

Besides not being done calculating
The raster view is selected
by the Memory Mode Settings.
This can be effected by:

1 - Document Settings

2 - Application Options

3 - Drawing View



I would like to get a ***quick*** reverse engineered version of a sample piece.
How can I get this done in a few minutes?

A: Send it out for a Laser scan

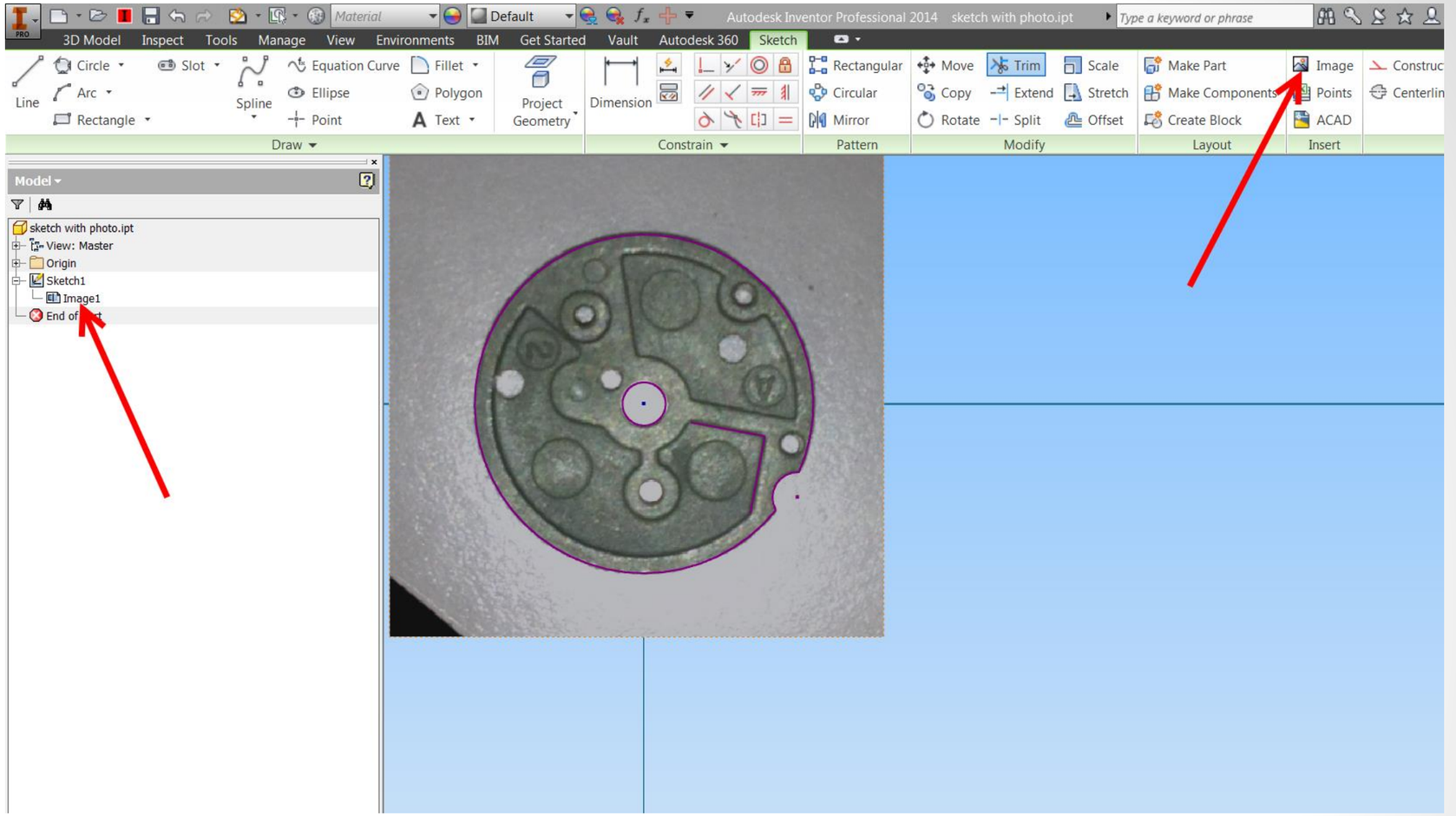
B: Send it out to a company with a CMM to get the geometry data

C: A Smart Phone and a sketch

D: Graph paper and a ruler.

C: A Smart Phone and a sketch

While in a sketch you can import an image. Then trace over the photo, scale, and extrude. Done in minutes!



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The questions and answers will be available on the AU class website, they will also Be available for download at: http://www.engineeredtoorder.com/au2013.htm

Thank you to the companies who donated prizes:

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