IM463454

2D + 3D: Get the best of both with AutoCAD and Autodesk Inventor

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Learning Objectives

- Discover how to reuse AutoCAD 2D data inside Inventor
- Discover how to reference 2D AutoCAD drawings into Inventor as a design reference
- Discover how to bring 2D data from Inventor back into AutoCAD
- Discover how Inventor DWG drawings can be opened directly in AutoCAD, or converted into 2D AutoCAD DWG format

Description

I don't have time to learn 3D modelling!
All my legacy data is 2D!
My customers demand 2D drawings from me!
I know there's a better way to get my work done and help my colleagues downstream - but I'm under too much pressure to spend time working this out!

2D and 3D workflows don't have to be mutually exclusive. Your best option could be a combination of both.

You can have the flexibility of 2D AutoCAD when you need it, and the automation power of Autodesk Inventor when you're ready.

In this presentation, you'll see how you can move from 2D to 3D at your own rate. We'll show you how to reuse your legacy 2D data in a 3D workflow, and how to repurpose your 3D data as 2D drawings for your customers.

AutoCAD and Inventor work better together for 2D + 3D workflows.

Supporting documents

Don’t forget to watch the presentation and download the dataset that comes with the handout.

http://cadso.co/IM463454_DOWNLOAD

Paul Munford

Paul Munford is a laugher, dreamer, bon vivant, CAD geek and Technical Marketing Manager for Autodesk in the UK.

Paul 's background in manufacturing items for the construction industry gives him a foot in digital prototyping and a foot in Building Information Modeling (BIM).

Paul was a speaker at Autodesk University for the first time in 2012, and he says it's the most fun anyone can have with 250 other people in the room.

Luke Mihelcic

Luke has been involved with design, engineering and analysis since 1995. Today, he is the Technical Marketing Manager for the core manufacturing solutions at Autodesk,

Luke’s career started in telecommunications designing mobile production equipment for TV and radio. He has taught design and engineering at Pittsburgh Technical Institute, and prior to joining Autodesk,spent over 10 years as an application engineer for an Autodesk reseller and Blue Ridge Numerics.

Luke’s various roles as educator, end-user and provider give him a unique perspective on identifying, understanding and helping solve design and engineering challenges.

At Autodesk, Luke is responsible for the development, creation and implementation of relevant content and tools that help users understand and utilize the Autodesk Design and Manufacturing Portfolio.

Contents

Introduction

Is there still a place for 2D drafting in modern design and manufacturing?

Well – depending on your processes and technology, the answer may be ‘Yes’ or ‘No’, but for many of us – it’s going to be somewhere in between!

You don’t have to choose between 2D and 3D. We’d like to show you some tips, tricks and workflows to use 2D drafting in AutoCAD and 3D parametric design in Inventor together.

Perhaps you are transitioning from one product to the other? Perhaps you are coming from AutoCAD to Inventor, and you want to reuse or repurpose some of the work you did inside AutoCAD inside Inventor?

Or perhaps you are collaborating with colleagues who can only provide you with 2D details, and you need to convert them to 3D?

Maybe it’s the other way? Maybe you work fully in 3D inside Inventor, but your clients or customers are asking for 2D drawings as a deliverable?

Maybe you want to keep your survey drawings and site layouts in 2D, but you want to coordinate with mechanical designs that have been modelled in 3D?

However you want to use 2D and 3D design, there is a workflow to help you!

IDW vs DWG

Did you know that you can open **Inventor** drawings natively in **AutoCAD**?

Autodesk Inventor was created by **Autodesk** in 1999. When it was first launched, it was decided to re-write the rule book and create a brand-new drawing format for Inventor called '.IDW'

Of course, it was soon realised that customers expected Inventor to be able to open and work with AutoCAD data, and DWG was added as a drawing format.

So – what's the difference between IDW and DWG?

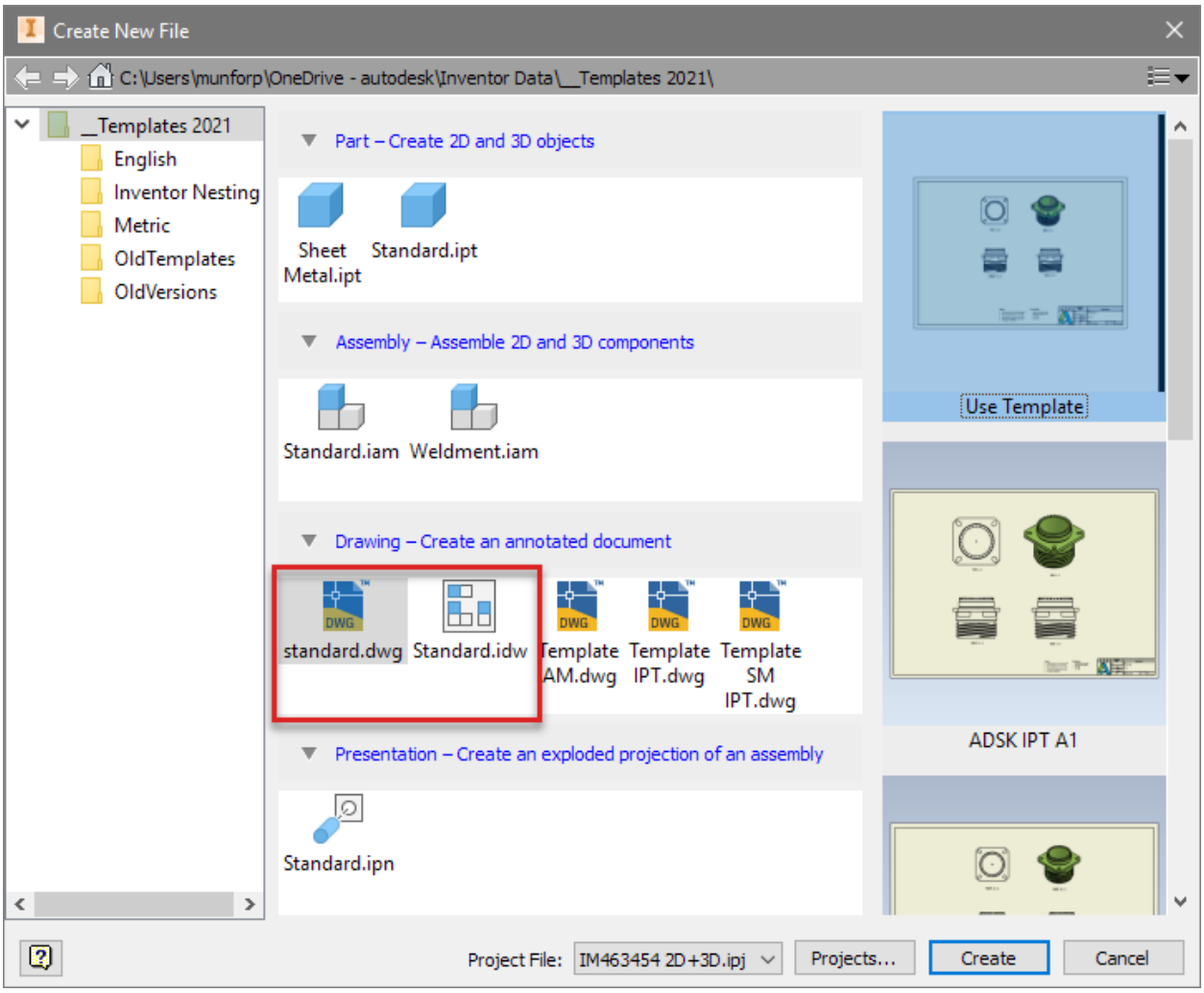
Well – IDW files are native to Inventor. The file size is a little bit smaller and they load a little bit faster. The downside of IDW is that there are very few compatible programmes for viewing or exchanging IDW data.

DWG files are native to AutoCAD, the file size will be a little bit bigger than IDW and they may take fractionally longer to open than IDW. The advantage of DWG is that the files will open natively in Inventor and AutoCAD, and there are many other programmes and viewers that are compatible with DWG data.

How do I choose which file format to use? Well, if you never share data with others, use IDW. If you ever need to share data with colleagues using AutoCAD or many other CAD systems, you'll find DWG is acceptable almost anywhere!

So, how do we create DWG files with Inventor?

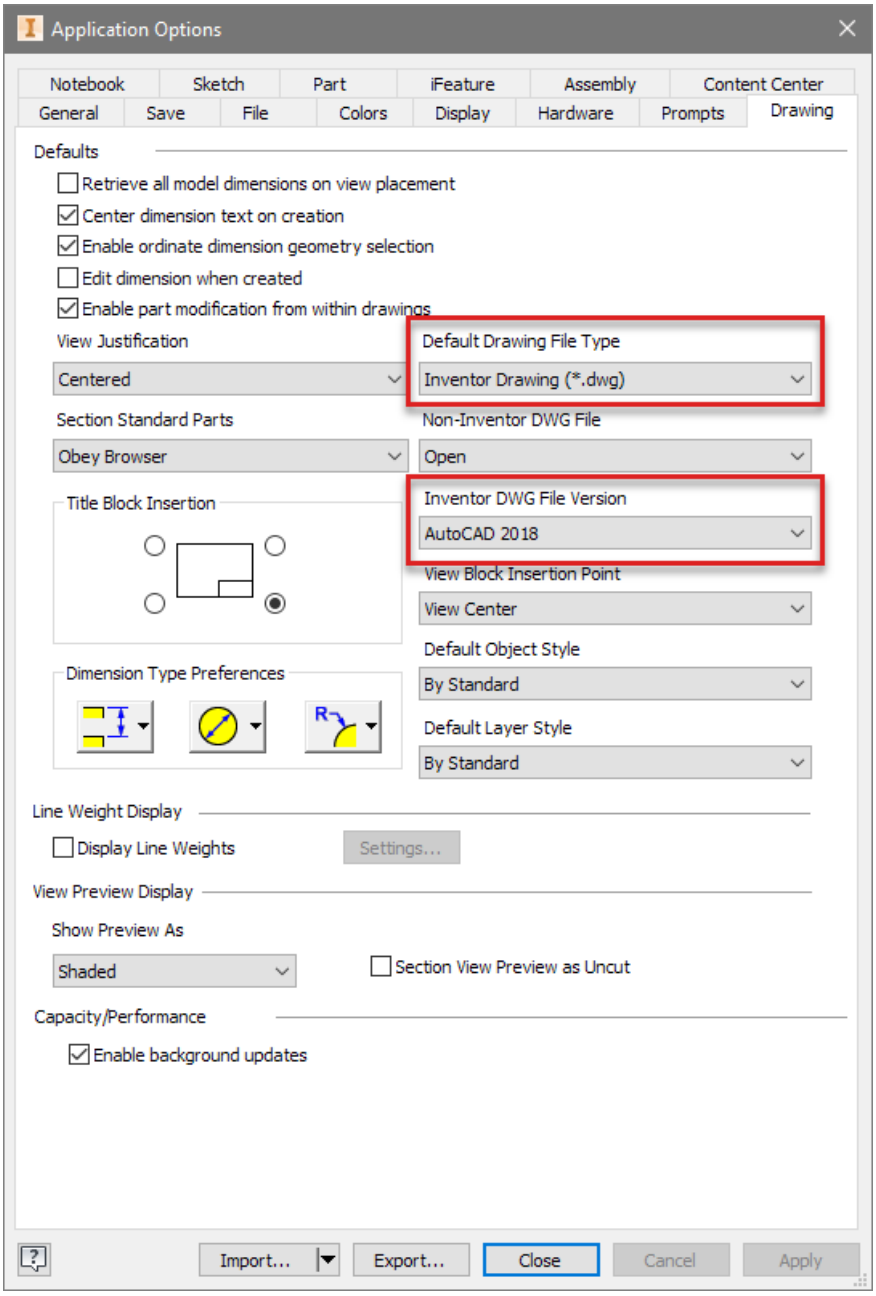
Inventor comes by default with both IDW and DWG templates. To create a DWG file in Inventor you just need to pick the DWG template.



Setting DWG as the default drawing type in Inventor

If you want to permanently select DWG as your format you just need to select it in the application options:

Tools (Tab) > Options (Panel) > Application Options (Button)



In the application options dialog, look under the ‘Drawings tab’ at the ‘Default drawing Type’ drop down.

Note that you can also set the DWG file version you want to use here.

Creating drawings in Inventor using the AutoCAD DWG file format follows the exact same process as IDW, there is no loss of functionality – nothing is missing.

To open the drawing in AutoCAD, you just need to close it In Inventor and open it in AutoCAD – it’s that simple!

So, what can we do with our Inventor DWG now that we have it in AutoCAD?

You’ll notice that you can’t change the Inventor model while you are in AutoCAD. That’s because the Inventor design model isn’t in AutoCAD – only the drawing is. So, what else can we do?

Well, we can plot the drawing, take measurements and add items such as revision clouds and notes. This makes AutoCAD a great tool for anyone who wants to review our design electronically, but who doesn’t want the overhead of installing or learning how to use Inventor just to look at drawings.

But, what if you need 2D AutoCAD geometry? That’s also possible, and we’ll get to it soon!

Insert Inventor views into AutoCAD drawings

We can't edit the geometry in an Inventor DWG directly, but we can reuse 2D views. To do this, open an Inventor DWG file in AutoCAD and then switch to model space.

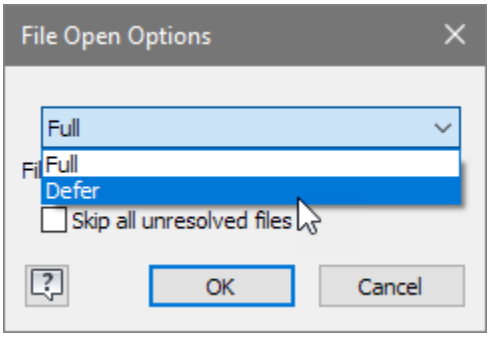
GASP There's nothing there! And that's OK. Unlike AutoCAD, Inventor has a modelling environment that is separate to its drawing environment. It references the model into the drawing a bit like AutoCAD uses x-references.

But, if we open the AutoCAD block palette, we can see that all the model views that were created in Inventor are saved in AutoCAD as AutoCAD blocks. In fact, this is why an Inventor DWG is bigger than an Inventor IDW file, because it contains two sets of data: Inventor data and AutoCAD data.

We can now insert the Inventor view into the AutoCAD model space as a block. An example of when this could be useful is in facility or job site layout. We can add an AutoCAD block that represents the footprint of our 3D model into our 2D layout. If we change the Inventor model, we just need to open the DWG file in Inventor to update the drawing, and then we can open it in AutoCAD again to update the view block.

Just be aware that if you copy and paste the view block into another drawing, the link will be lost and the block will no longer update. If you need this workflow, we encourage you to take a look at Autodesk Factory design utilities, which we will show you toward the end of this presentation.

Try it!



1. In Inventor, start the open command and navigate to **'00 Inventor Drawing.dwg'** from the dataset.
2. Select the file with a left click and select the 'options' button in the open dialog.
3. Select 'defer' updates, then click 'OK' top open the drawing.
   (This saves us having to share the full Inventor assembly!).
4. Take a look at the drawing in Inventor, then close the file in Inventor without saving.
5. From the dataset, open **'00 Inventor Drawing.dwg'** in **AutoCAD**.
6. Navigate to Paper Space.
7. Take some measurements, add some annotations or plot the drawing. What can you do with an Inventor DWG drawing in AutoCAD? What can't you do?
8. Close the file without saving it.

Extra credit

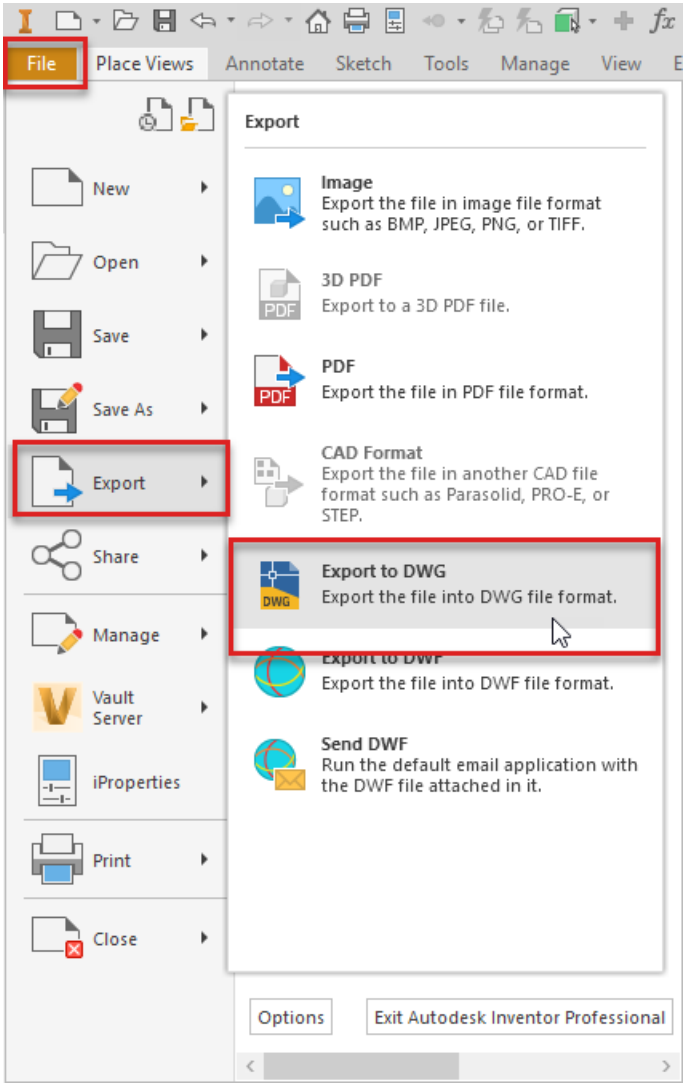
9. Double click over a viewport to enter it – what do you notice?
10. Exit the viewport and switch to model space – what do you see?
11. Start the 'insert' command (or open the block palette). Notice that the Inventor views are available to you in AutoCAD as blocks. Insert a view/block into Model Space.
12. Close the file without saving it.

Convert an Inventor DWG into an AutoCAD DWG (with Inventor)

OK. So we've learned that we can create drawings in DWG format in inventor and open them directly in AutoCAD – but what we can do with that drawing is limited.

Let's imagine that we've been asked to deliver a native, editable, 2D AutoCAD drawing as a deliverable – how do we do that?

To create an editable DWG file, we'll need to export the drawing.

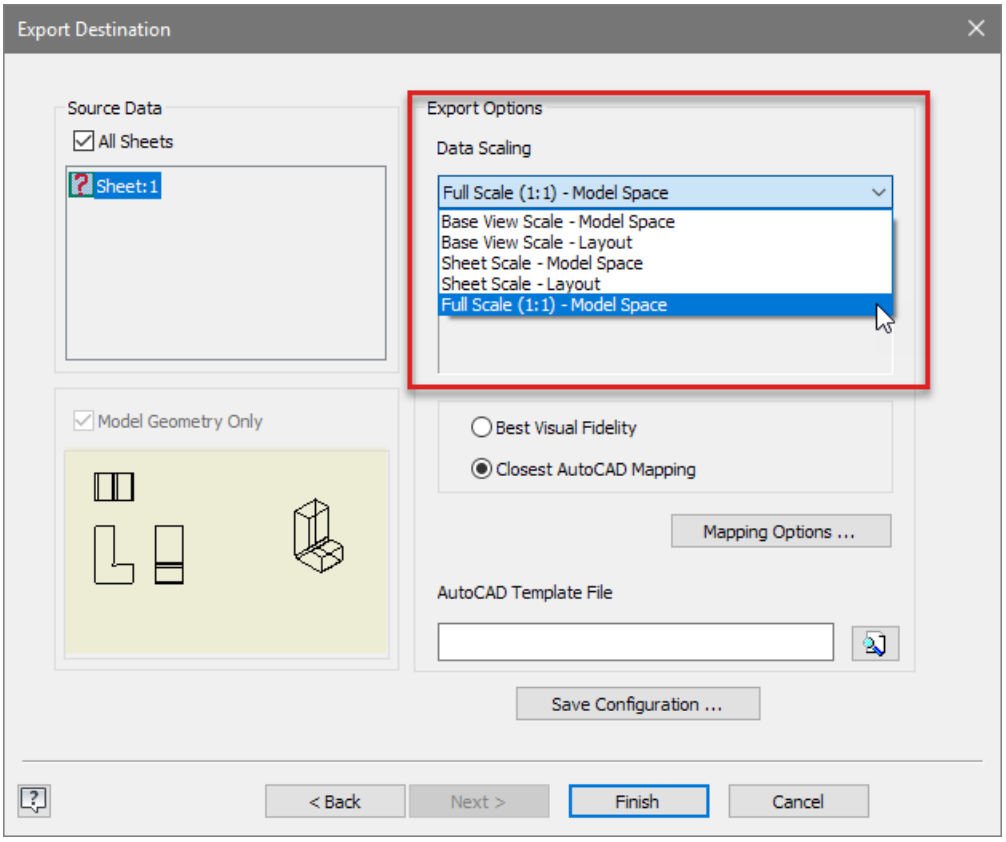


To export an Inventor DWG to AutoCAD, Go to the Application menu
(The orange tab with 'File' written on it)

Navigate to 'Export, then 'Export to DWG'.

In the dialog, note that you can save your configuration and use it again.

Click Next.



Look for ‘Export options’ > ‘Data Scaling’. Your Inventor drawing is drawn to scale on a virtual sheet of paper. The data scaling option allows you to choose how you want your drawing to come into AutoCAD.

You can choose whether the geometry will come into AutoCAD at the same scale as it was in Inventor or at 1:1 scale. You can also choose if you want the drawing to appear in AutoCAD’s Model or Paper Space.

When you are ready, click ‘finish’.

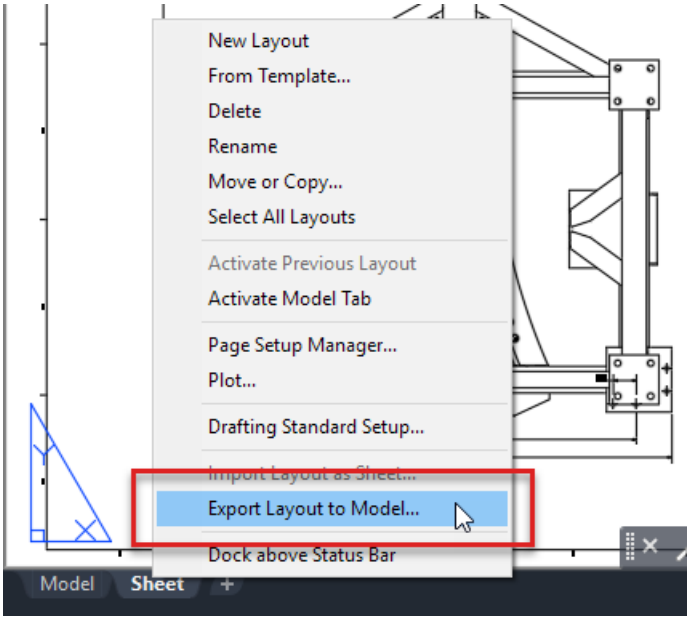
When you open the file in AutoCAD, all geometry will now be in an editable 2D format.

For details of all the export to DWG options available to you, check out the help: https://knowledge.autodesk.com/support/inventor/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/Inventor-Help/files/GUID-0BA0F188-8990-4054-BBEB-41C38F6982F2-htm.html

Convert an Inventor DWG into an AutoCAD DWG (with AutoCAD)

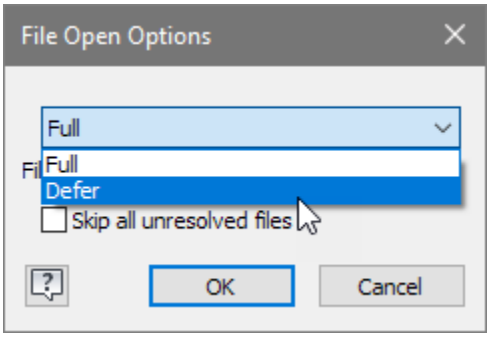
If you only have AutoCAD, or you are just more familiar with the AutoCAD Interface, you can also convert drawings from Inventor DWG to AutoCAD DWG from inside AutoCAD. We will do this with the 'EXPORTLAYOUT' command.

In AutoCAD, open an Inventor DWG file. Navigate to paper space. Right click over your layout tab and choose the option 'Export Layout to Model'. Follow the prompts to create your editable 2D version of the Inventor drawing.



Notice that you don't get any option to scale the Inventor drawing or remove the title block. You may need to scale the geometry if you want it to be a 1:1 – and do a little clean up.

Try it!



1. In Inventor, start the open command and navigate to **‘00 Inventor Drawing.dwg’** from the dataset.
2. Select the file with a left click and select the ‘options’ button in the open dialog.
3. Select ‘defer’ updates, then click ‘OK’ top open the drawing.
   (This saves us having to share the full Inventor assembly!).
4. Go to: Application menu (File) > Export > Export to DWG
5. Click the ‘next’ button
6. Under ‘Export options’ > ‘Data scaling’ pick ‘Full scale (1:1) Model space.
7. Click finish and save your file. Close ‘00 Inventor Drawing.dwg’ without saving.
8. Open your exported file in AutoCAD to see what you’ve got.

Extra credit

1. From the dataset, open **‘00 Inventor Drawing.dwg’ i**n AutoCAD.
2. Navigate to paper space.
3. Right click over the paper space layout tab and pick ‘Export Layout to Model’.
4. Name and save your file.
5. Close ‘00 Inventor Drawing.dwg’ without saving.
6. Open your exported file in AutoCAD to see what you’ve got.

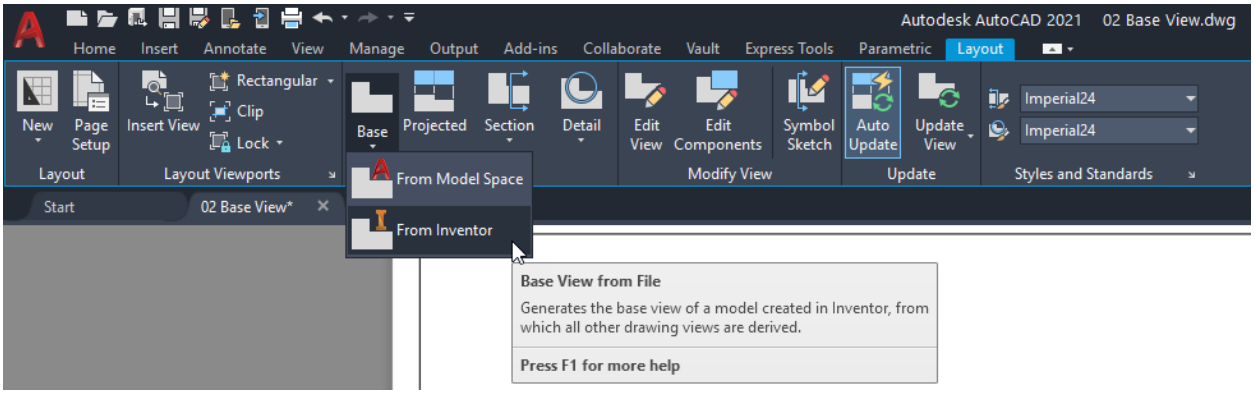
Tip*: There are example export files in the ‘05 Examples of Exported files’ folder. Compare the export from Inventor to the export from AutoCAD. Compare the location of 0,0,0, the scale of the geometry, the layers used, etc. Which method suits your workflow best?*

Create drawings of Inventor models in AutoCAD

There's one more method of using AutoCAD and Inventor together that can give us the best of both worlds. At some companies, the tasks of 3D modelling a design and the creation of 2D drawings are done by different people.

This is an example where each user can use each program for what it's best for: Inventor for 3D parametric modelling and AutoCAD for 2D drafting.

To do this we use a command in AutoCAD called 'Base View'.



To create a drawing in AutoCAD from an Inventor model, we'll use a command called 'Base View'. Go to an AutoCAD drawing layout and start the base view command. You'll notice that you can chose an AutoCAD model, or you can choose a model from Inventor!

If you've used the drawing environment in Inventor, the base view commands will look pretty similar. If you've used the drawing environment in Fusion 360, it will look even more familiar, because the Fusion drawing environment comes from AutoCAD!

We can add views, sections, and details of the model, and add dimensions and notes. We can't edit the model from here – we'll need to do that in Inventor.

For more information on the base view command in AutoCAD – check out the help here: https://knowledge.autodesk.com/support/autocad/learn-explore/caas/CloudHelp/cloudhelp/2021/ENU/AutoCAD-Core/files/GUID-FD856E3D-9A74-415D-83BA-7AC1C2FFA235-htm.html

I can also recommend J.C Malitzke's Autodesk University class: 'Create Compelling 2D Sections, Details, and Auxiliary Views from AutoCAD 3D Models'

Try it!

1. In AutoCAD, open '**02 Base View.dwg'** from the dataset.
2. Navigate to: Layout (Tab) > Create view (Panel) > Base (Button) – Select on the bottom half of the button and choose 'From Inventor' from the drop down.
3. Optional – select Inventor project file. See Tip Below.
4. In the 'Select file' dialog, browse and select the Inventor part file '**02 Base View.ipt'**
5. Move your cursor across the layout. Left click to place your view, select 'eXit' to confirm.
6. The Base View command is still running, continue placing orthographic or isometric views by left clicking on the sheet.
7. Right click and choose 'Enter' to complete the command.

Extra credit

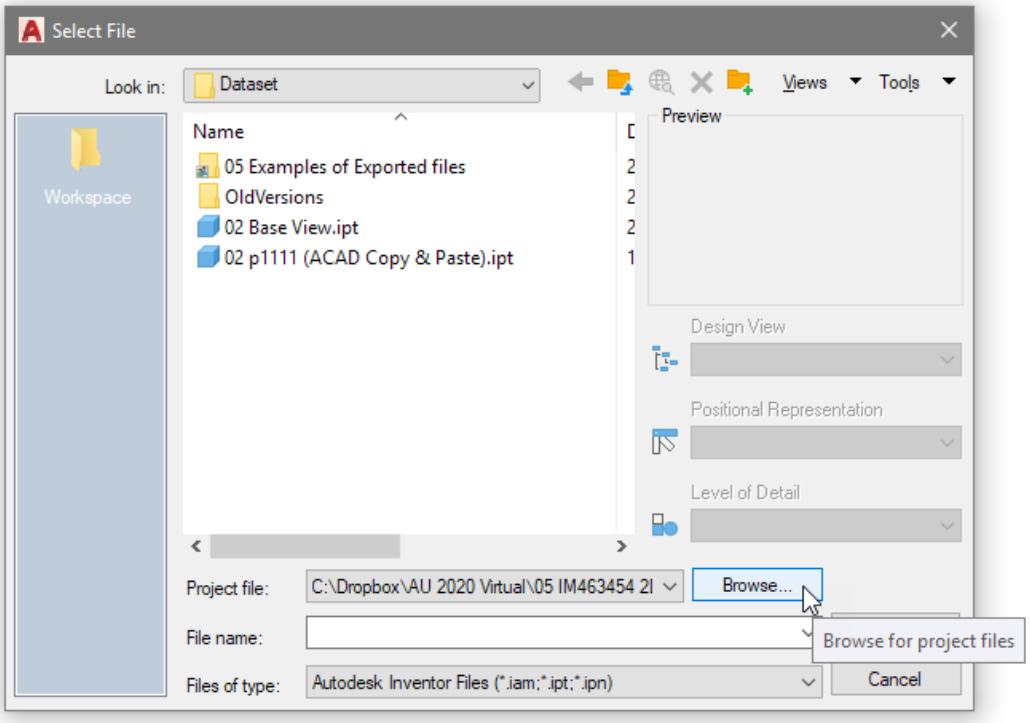
If you like, you can open a completed drawing created with the 'BaseView' command: '02 Base View Complete.dwg'

7. Create section, detail, and Isometric views in AutoCAD
8. Change the view type to 'Shaded'
9. Add Center marks and Center lines in AutoCAD
10. Add dimensions and annotations in AutoCAD

***Tip**: When you first use the Base View command, you'll have the option to select an Inventor project file to help manage your data.*

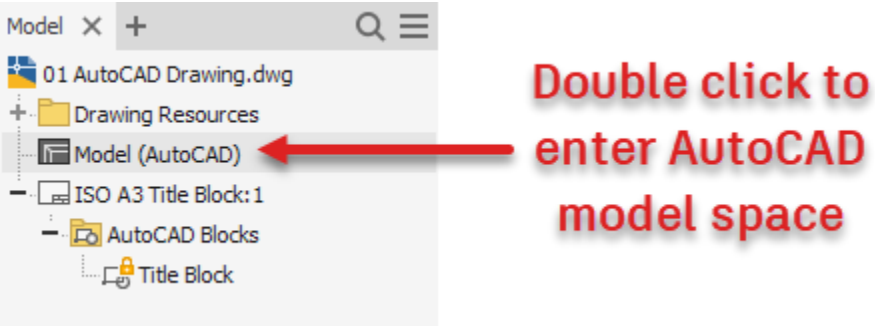
If this isn't set, click 'Browse' and browse to the location of the data set for this class.

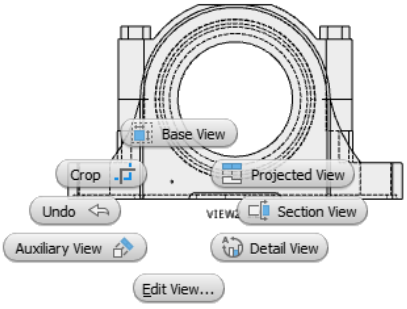
Select the file 'IM463454 2D+3D.ipj' and click open to set the Inventor project file for this class current.

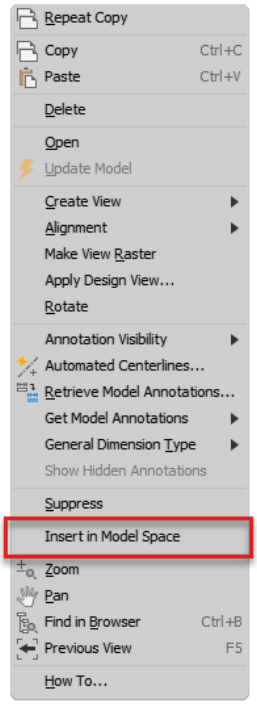


Open AutoCAD drawings in Inventor

You can open an AutoCAD dwg file directly in Inventor. It will open as an Inventor drawing. You can add views from your Inventor model to your AutoCAD drawing, to create a hybrid drawing which uses the best of both packages.



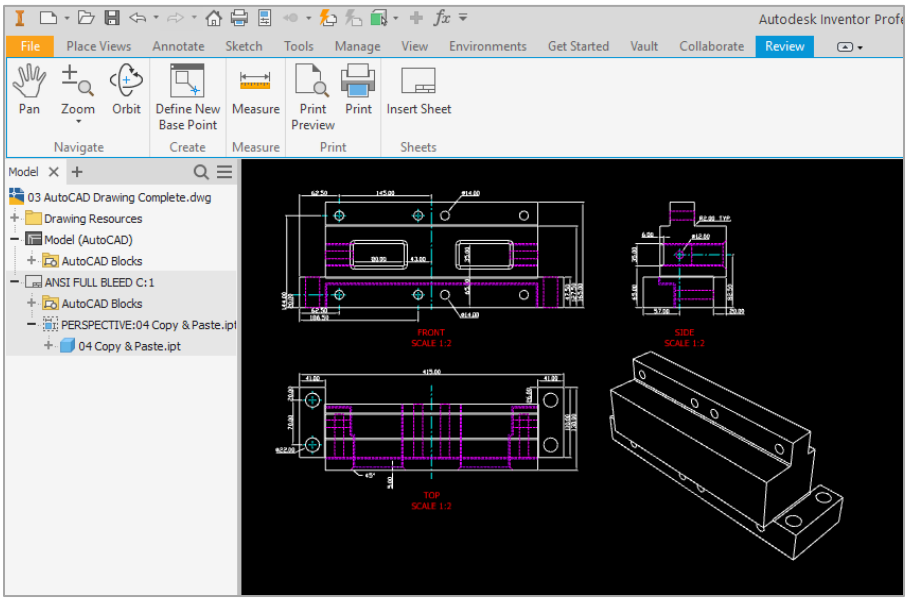




You'll notice that an AutoCAD drawing opened in Inventor has a model space layout. You can view the model space geometry that was created in AutoCAD, but you can't edit it.

If you have an inventor view in your AutoCAD drawing, you can add it to model space. Right click on the view boundary (Or the view node in the browser) and select 'Insert in Model space'.

This only works for a single view. To insert multiple Inventor views into AutoCAD's model space, you'll need to open the drawing in AutoCAD.



Try it!

1. In Inventor, open '**03 AutoCAD Drawing.dwg'** from the dataset.
2. Notice that the perspective view was created in Inventor. All other views where created in AutoCAD.
3. In the browser, double click left click on the 'Model (AutoCAD)' node to enter model space.
4. You can see the AutoCAD geometry, but you can't interact with it.
5. Double left click on the 'ANSI FULL BLEED C:1' node to go back to the drawing.
6. Right click on the boundary of the 'PERSPECTIVE NTS' view
   (Or right click on its node in the browser).
7. Select 'Insert in model space'. Click OK in the warning dialog.
8. Enter model space again to see the result.

Extra credit

If you like, you can open a completed drawing:
'03 AutoCAD Drawing Complete.dwg'

11. Save and close the drawing in Inventor.
12. Open the drawing in AutoCAD.
13. Inspect the inventor view – the end result is the same as insert the view as a block in AutoCAD.
14. Add dimensions and annotations in AutoCAD

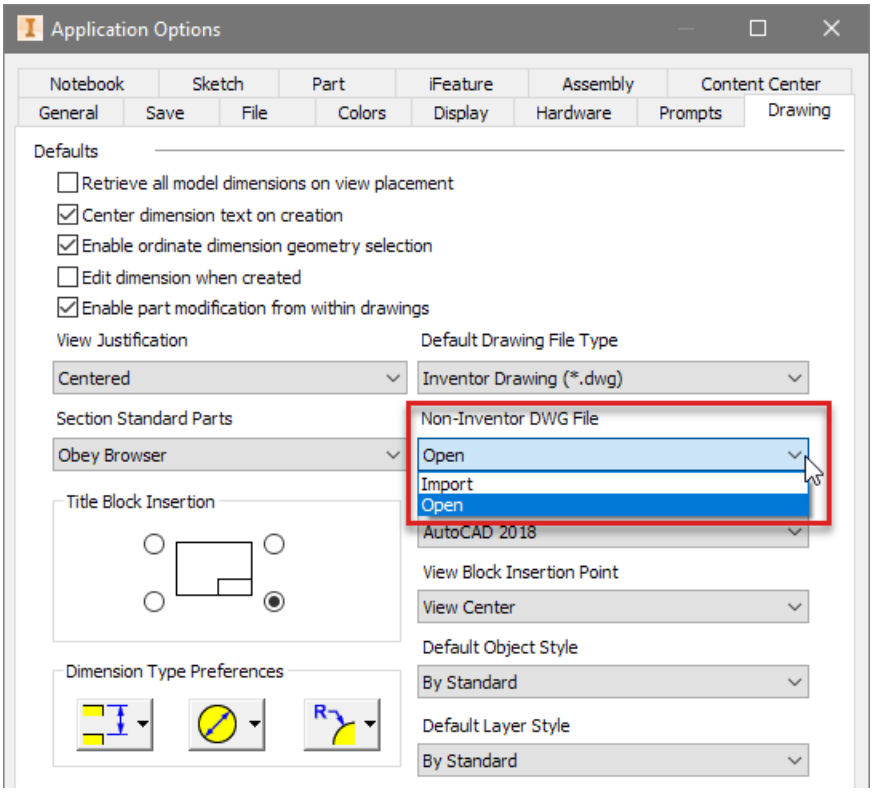
***Tip**: Inventor can 'Open' or 'Import' an AutoCAD drawing.*

(We'll find out more about the import options later in the handout).

To set the default behaviour, navigate to:

Tools (Tab) > Options (Panel) > Application options (Button)

Go to the drawing tab, and set the default behaviour under 'Non-Inventor DWG file'



Copy and Paste from AutoCAD to Inventor

We've learned that we can open AutoCAD drawings in Inventor, and we can add Inventor views to an AutoCAD drawing, but we can't edit the AutoCAD geometry using this method… So how can we re-purpose AutoCAD geometry in Inventor?

The simplest way is with copy and paste.

- Open a drawing in AutoCAD.
- Select some geometry (Include an AutoCAD dimension).
- Right mouse click and choose 'Clipboard > Copy' (or use CTRL+C on your keyboard).
- Open an Inventor part file.
- Start a new sketch.
- Right mouse click and choose 'Paste' (or CTRL+V on your keyboard).
- Left mouse click to place your geometry.
- Notice that AutoCAD dimensions have become Inventor parametric dimension constraints.

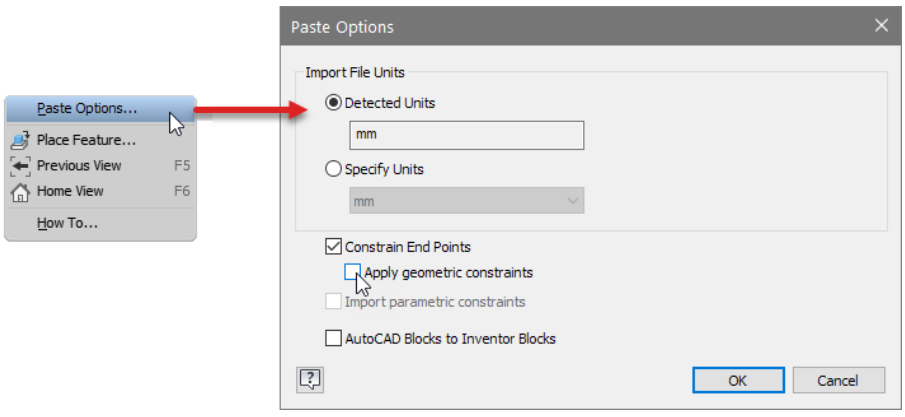
Simple!

The downside of copying and pasting from AutoCAD is that you may need to spend some time making sense of your geometric and dimensional constraints in order for your part to update predictably as you change parameters (please don't just apply a 'Fix' constraint to everything – you'll regret it later!).

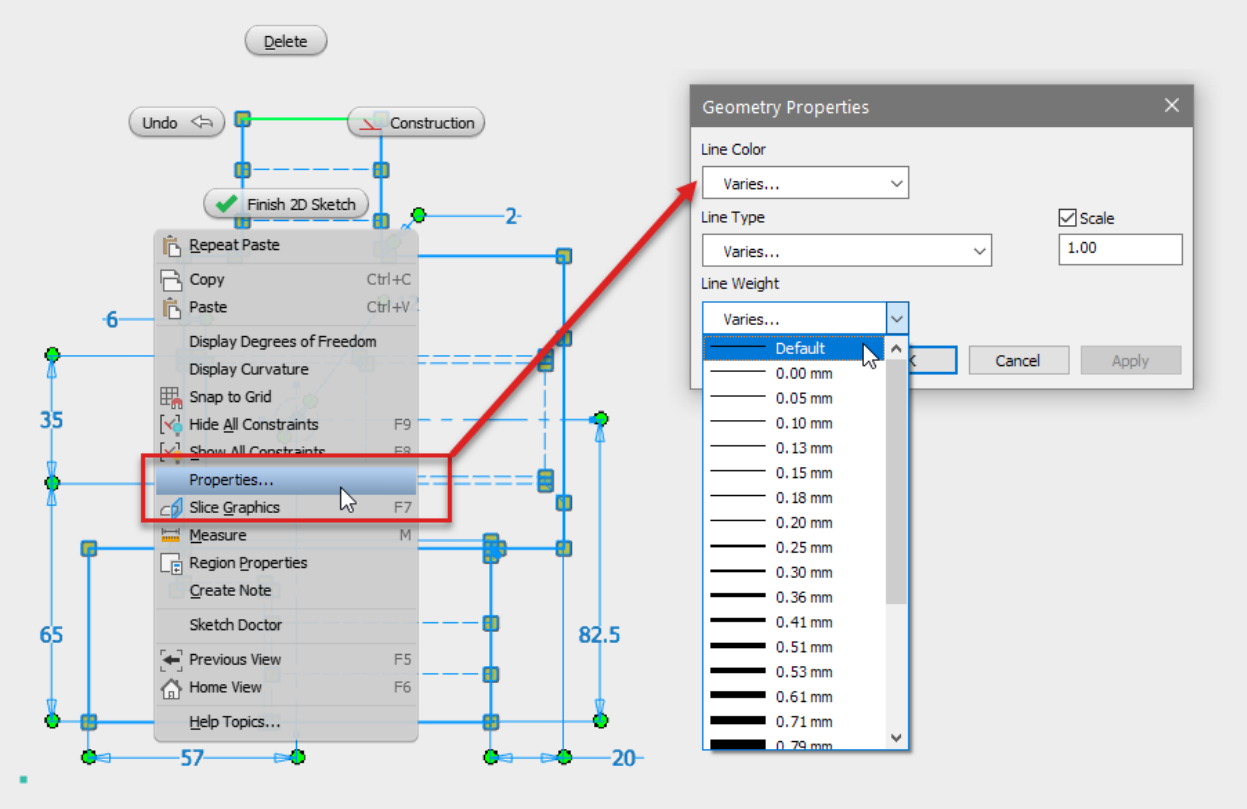
See the next section for another technique that may help.

***Tip**: Before you left mouse button click to place your copied geometry, right mouse button click. This will bring up the 'Paste Options' dialog, with more options for units and automatically applying constraints.*

We'll use this option in the next exercise.



***Tip**: AutoCAD layer properties such as colour, line weight and line type will be brought into your Inventor sketch. I find this unhelpful because it overrides Inventor's use of line colour to denote constrained status, and it makes Inventor's construction and centreline geometry difficult to distinguish.*



To remove these overrides in AutoCAD, temporarily move everything to the '0' layer, and make sure that all properties are 'By layer' before you copy them over to Inventor.

To remove the overrides in Inventor, select the geometry, right click and choose 'Properties'. Use the drop-down menus to set the overrides to 'Default' and un-check the (Line type) 'Scale' box.

Use AutoCAD blocks as Inventor Sketch blocks

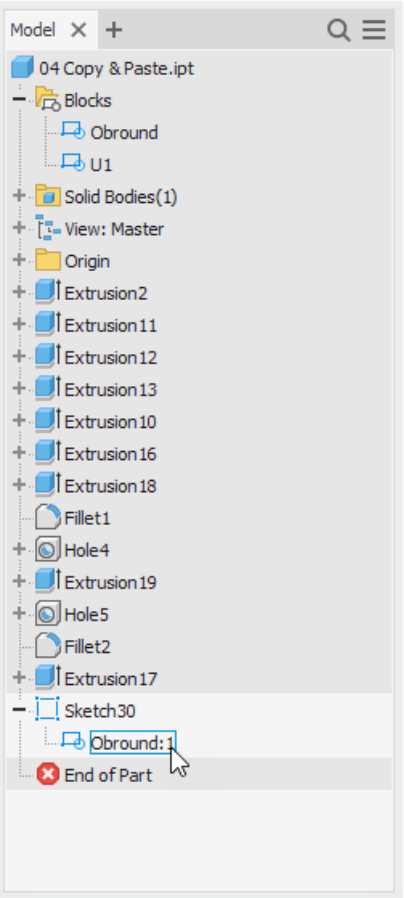
In the last section, we saw how we can copy and paste geometry from an AutoCAD drawing into an Inventor sketch.

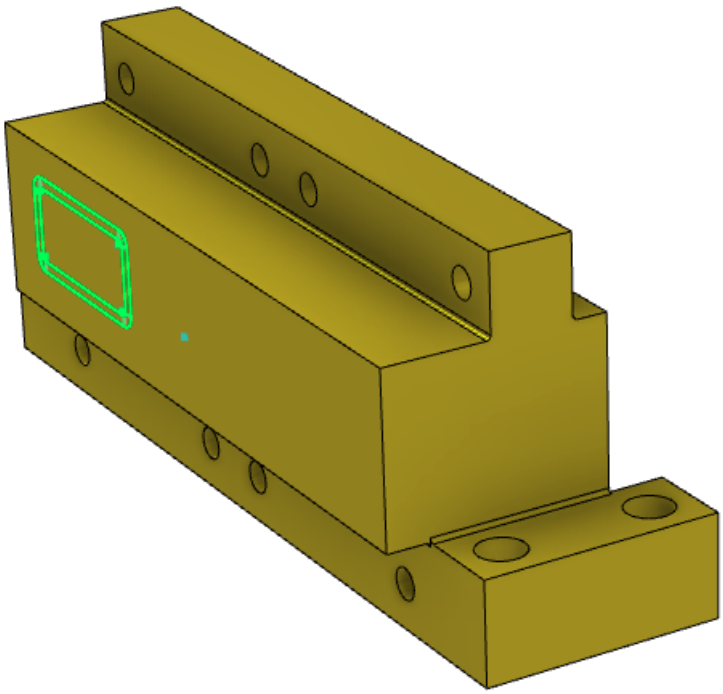
The downside was that we needed to do some extra work inside Inventor to constrain the geometry.

The method I prefer takes a little more work in AutoCAD, but saves a ton if time in Inventor.

My preferred method is to convert the AutoCAD geometry into a block first, and then copy and paste the block into Inventor.

In Inventor, the AutoCAD block becomes an Inventor sketch block. This groups the AutoCAD geometry into one lump, and we only need to apply one or two constraints to fully constrain each block.





Try it!

1. In AutoCAD, open **'04 Copy and Paste.dwg'** from the dataset.
2. From the front view, select the 'opening' on the left. This is an AutoCAD Block called 'Obround' (see image below).
3. Right click and choose: Clipboard > Copy.
4. In Inventor, open **'04 Copy and Paste.ipt'** from the dataset.
5. Start a new sketch on the face of the part.
6. With the sketch active, right mouse button click and choose 'paste'.
7. Before you place the contents of the clipboard, right mouse button again and choose 'past options'.
8. Make sure that 'AutoCAD blocks to Inventor blocks' is selected – click the OK button.
9. Left mouse button click to place the contents of the clipboard.

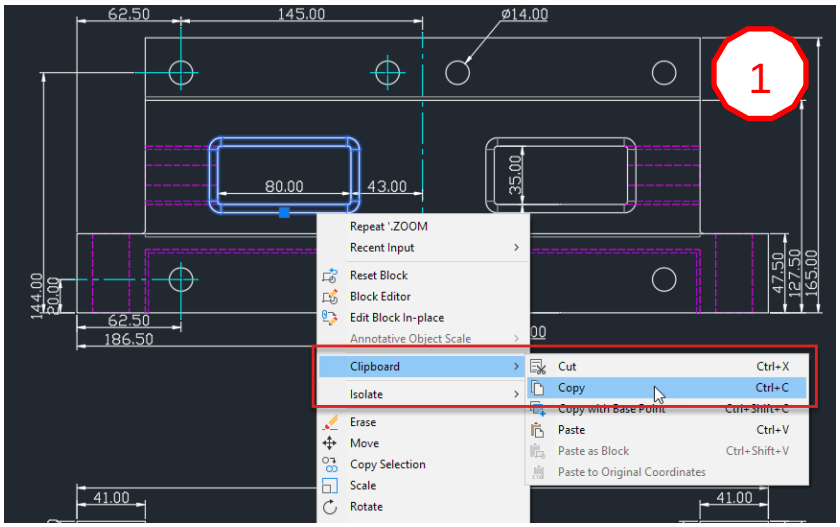
***Tip**: Notice that a new 'Blocks' folder has been added to the top of the model browser. This is where your Inventor sketch block definitions will be stored.*

Notice that your new sketch has a sub-node called 'Obround' – this is an instance of the Obround block.

If you edit the block definition, all instances of the block will update. This is a great way of sharing geometry across multiple Inventor sketches on multiple planes.

Extra credit

- Find the Obround block definition in the blocks folder.
- Right click on the Obround block definition and choose 'Place Block' to place another copy of the Obround block.
- Constrain the Obround blocks.
- Exit the sketch and use the 'Extrude' tool to create a through all cut extrusion, using the Obround blocks geometry.
- Right click on the Obround block definition and choose 'Edit Block' to edit the block definition.
- What happens to the instances of the Obround block?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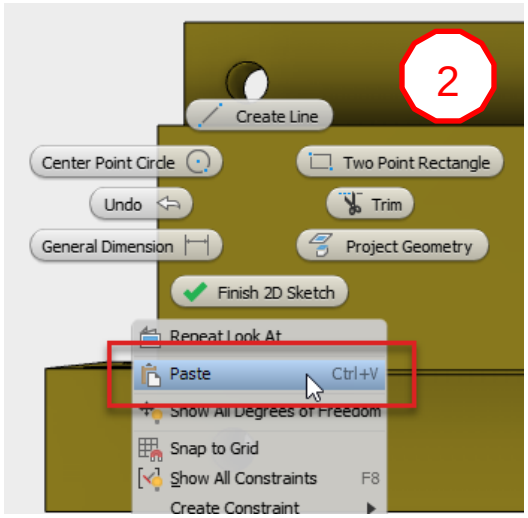
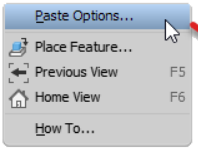
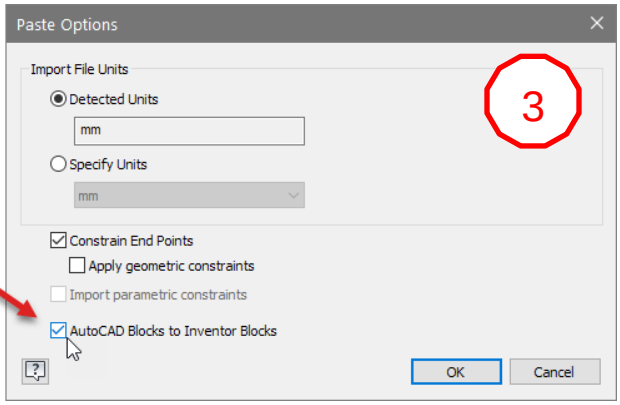
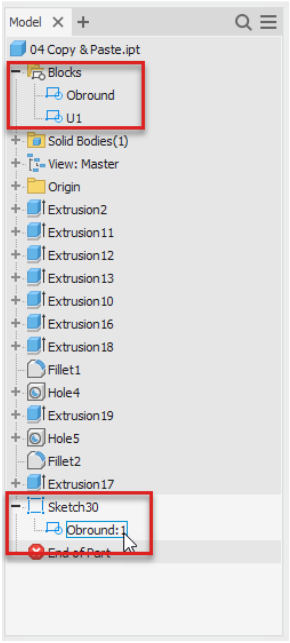
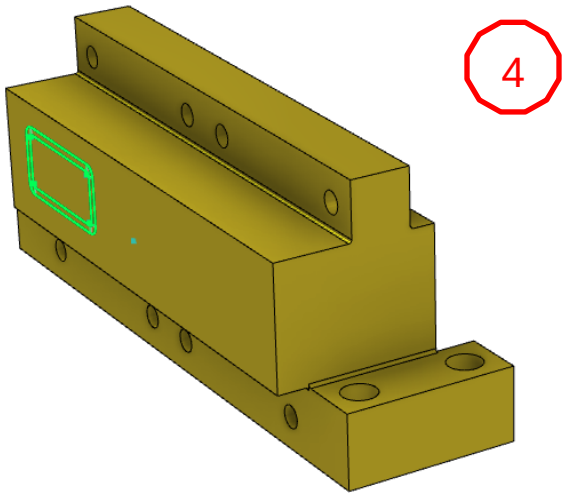
144.00
20.00
62.50
186.50
47.50
127.50
165.00
41.00
41.00
Repeat 'ZOOM
Recent Input
Reset Block
Block Editor
Edit Block In-place
Annotative Object Scale
Clipboard
Isolate
Erase
Move
Copy Selection
Scale
Rotate
Cut Ctrl+X
Copy Ctrl+C
Copy with Base Point Ctrl+Shift+C
Paste Ctrl+V
Paste as Block Ctrl+Shift+V
Paste to Original Coordinates


2
Create Line
Center Point Circle
Two Point Rectangle
Undo
Trim
General Dimension
Project Geometry
Finish 2D Sketch
Repeat Look At
Paste Ctrl+V
Show All Degrees of Freedom
Snap to Grid
Show All Constraints F8
Create Constraint


Paste Options...
Place Feature...
Previous View F5
Home View F6
How To...


Paste Options
3
Import File Units
Detected Units
mm
Specify Units
mm
Constrain End Points
Apply geometric constraints
Import parametric constraints
AutoCAD Blocks to Inventor Blocks
OK
Cancel


Model
04 Copy & Paste.ipt
Blocks
Obround
U1
Solid Bodies(1)
View: Master
Origin
Extrusion2
Extrusion11
Extrusion12
Extrusion13
Extrusion10
Extrusion16
Extrusion18
Fillet1
Hole4
Extrusion19
Hole5
Fillet2
Extrusion17
Sketch30
Obround:1
End of Part


4

Import an AutoCAD file into an Inventor sketch

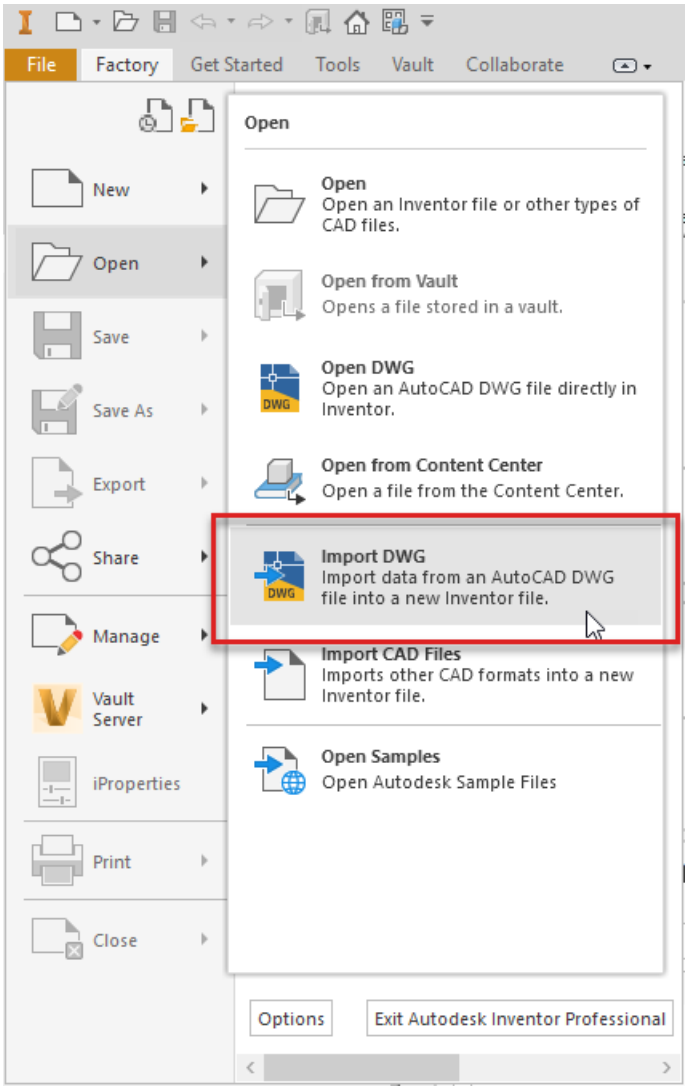
Now you know how to get AutoCAD geometry into Inventor piecemeal – just copy and paste right!

Next we'll learn how to bring the whole AutoCAD drawing into Inventor at once.

Earlier in the handout, we discussed that you can *Open* an AutoCAD drawing in Inventor, or *Import* an AutoCAD drawing into Inventor. We even discussed an Inventor application option to set the default action. In this section we'll learn the difference.

When we Import an AutoCAD drawing into Inventor, we will be copying all the geometry from an AutoCAD DWG file and placing it into a sketch in Inventor.

There are two approaches here, we can import the AutoCAD file directly and let Inventor create the part file for us, or we can start with a sketch in a part file open, and then import the AutoCAD file.



To **Import an AutoCAD drawing into a new Inventor part file**, go to the application (File) menu and choose 'Open' > Import DWG.

The DWG/DXF wizard will open, and you'll have the opportunity to use a saved configuration. Click 'Next'.

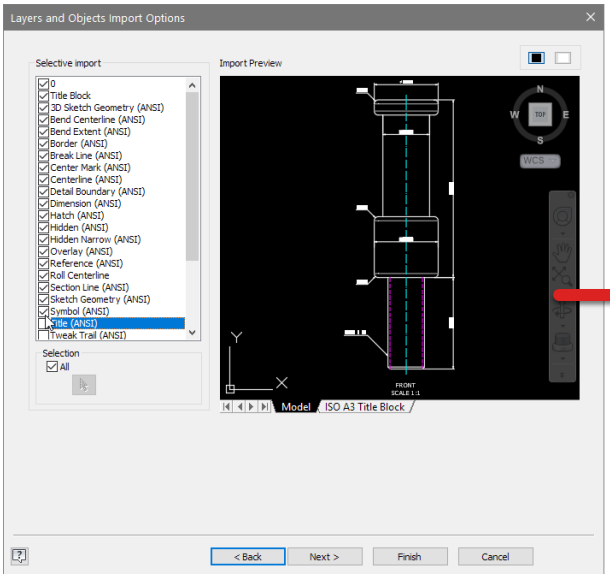
In the Layers and object import options, we can filter the objects that will be imported by layer.

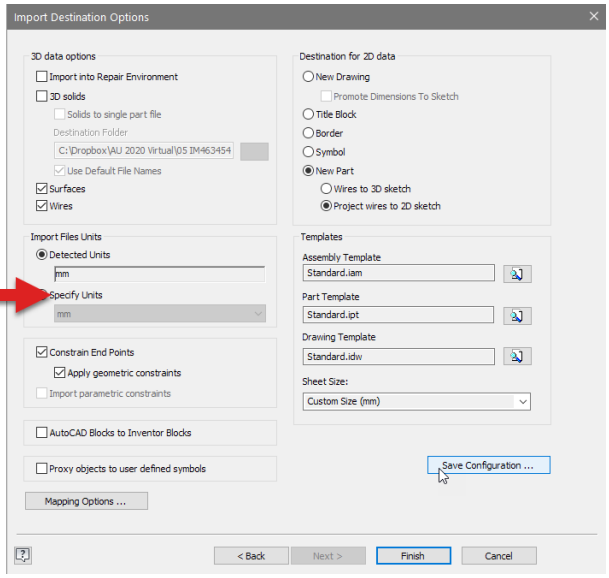
There is a handy preview to help you visualize what you will be importing.

Click Next.

In the import destination options dialog, you have more choices of what to import, what units to use, what the destination is for your geometry and what template you'd like to use.

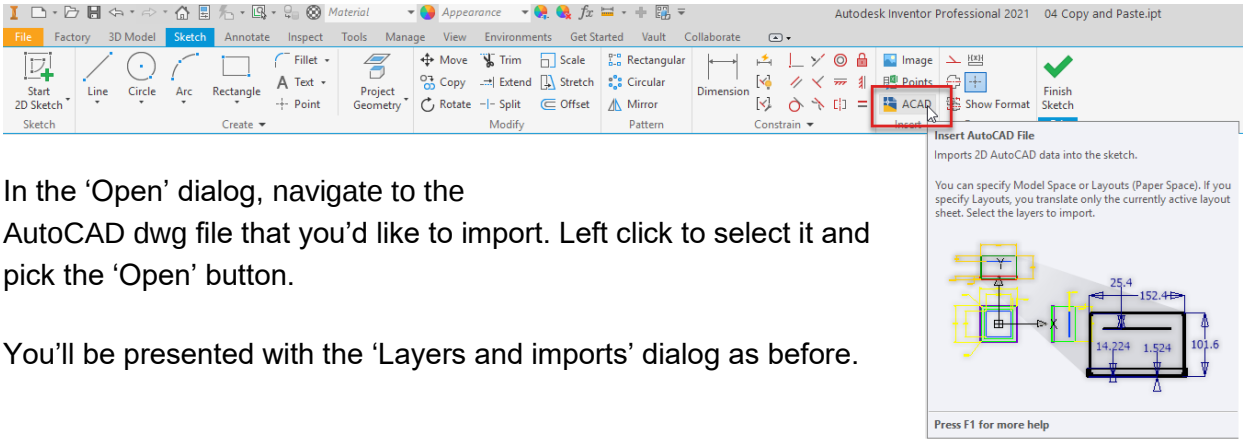
You can also save all these choices as a configuration, to save you time next time you go through this process.





To **import an AutoCAD drawing directly into an existing sketch**, first open your part file, then edit your sketch then navigate to:

Sketch (Panel) > Insert (Tab) > ACAD (button)

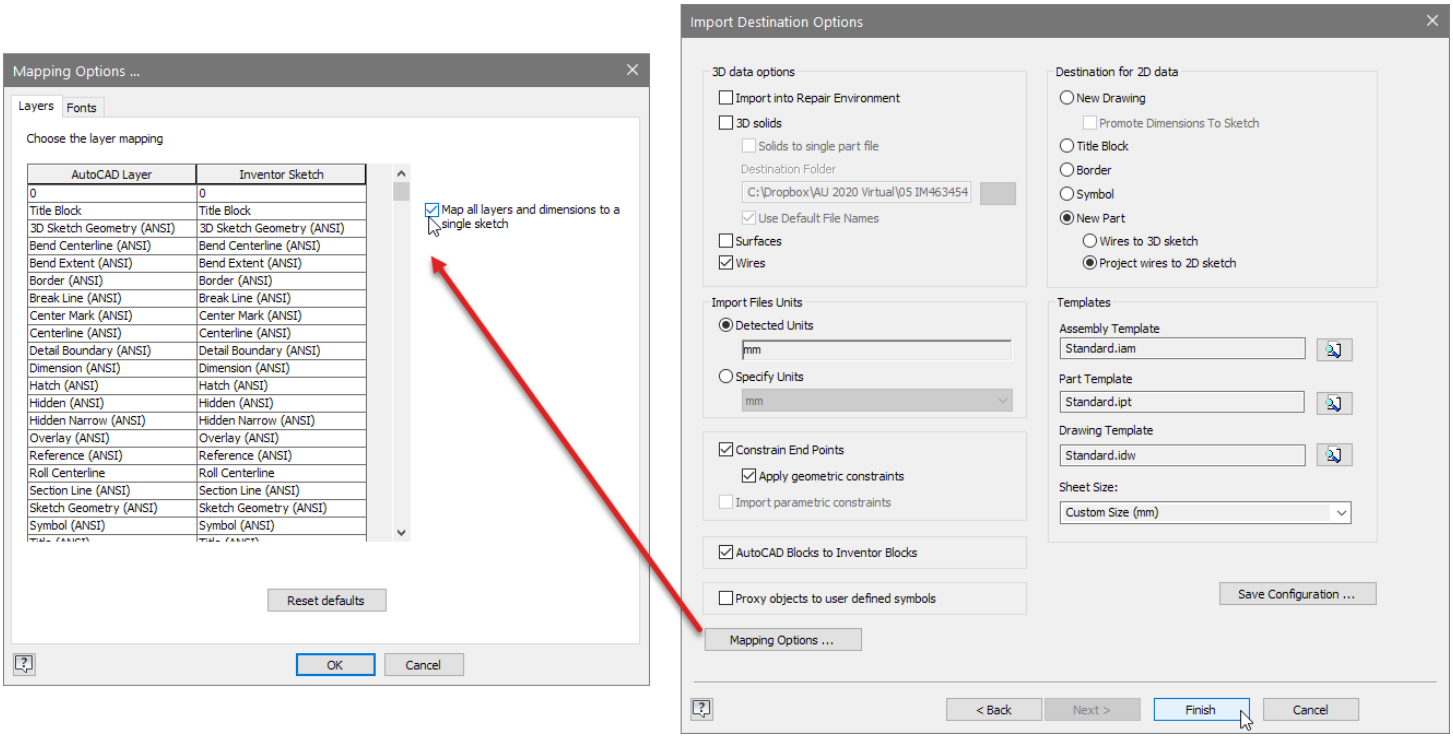


In the 'Open' dialog, navigate to the AutoCAD dwg file that you'd like to import. Left click to select it and pick the 'Open' button.

You'll be presented with the 'Layers and imports' dialog as before.

Try it!

1. In Inventor, navigate to the Application (file) menu and from the 'Open' flyout choose 'Import DWG'
2. In the 'Import' dialog, navigate to '**05 Import AutoCAD.dwg'** from the dataset, select it with a left click and click on the 'Open' button.
3. In the DWG/DXF file wizard dialog, click the 'Next' button.
   (Be patient while the AutoCAD file loads…)
4. In the 'Layers and Objects Import Options' dialog, remove the check mark from the following layers:
   0
   Title Block
   Defpoints
   Viewports
5. Click the 'Next' button.
6. In the 'Import Destination Options' dialog, set the following:
   Destination for 2D Data – Select '**New Part'** and '**Project wires to 2D Sketch'**
   3D data options – check **Wires**, uncheck all other options
   Check '**Constrain end points'**
   Check '**Apply Geometric constraints'**
   Check '**AutoCAD Blocks to Inventor Blocks'**
7. Click the **Mapping options** button (Bottom left of the dialog)
8. In the 'Mapping options…' Dialog, check '**Map all layers and dimensions to a single sketch**'
9. Click the OK button to close the 'Mapping Options' dialog.
10. Click the 'Finish' button to close the 'Import Destination Options' dialog.
11. In the 'Autodesk Inventor Professional – Data Import' dialog, click 'Accept'.
12. A new part will be created, containing a single sketch, which contains the model space geometry from your AutoCAD file.
13. If you like – you can save your file for future reference. You'll find a complete example of this exercise in the '05 Examples of Exported files' folder, it's called '05 Import AutoCAD DWG Complete.ipt'.



Extra credit

- Edit your sketch. Look at the constraint indicator in the lower left of the interface. How many dimensions/constraints are required to fully constrain your sketch?
- The AutoCAD annotation dimensions have been converted to Inventor parametric dimension constraints. What happens if you edit one of them?
- Open a new part, create a sketch and use the 'Insert AutoCAD file' (ACAD) Command to import an AutoCAD drawing – is the result different?

***Tip**: Notice that you can also import AutoCAD paper space geometry into an Inventor drawing file. This could be a great way to reuse an AutoCAD title block and border in an Inventor drawing template.*

Link an AutoCAD file into Inventor with AnyCAD

'AnyCAD' is Inventor's method of linking to files created in other CAD packages. Unlike an import, AnyCAD preserves the reference to the original file. You can edit the CAD file in the original authoring application, and when you save the file, you'll see the updates inside Inventor.

To reference an AutoCAD file into an Inventor sketch using AnyCAD, you'll need to start with an Inventor file open. In a part file, Navigate to:

3D Model (Tab) > Insert (Panel) > 'Import' Button

Browse to the file you want to import, select it with a left click and click on the 'Open' button.

Next, Inventor will ask you for a work plane or surface to place the referenced AutoCAD file onto, and an insertion point.

You'll now see the DWG file as an additional node in your browser.

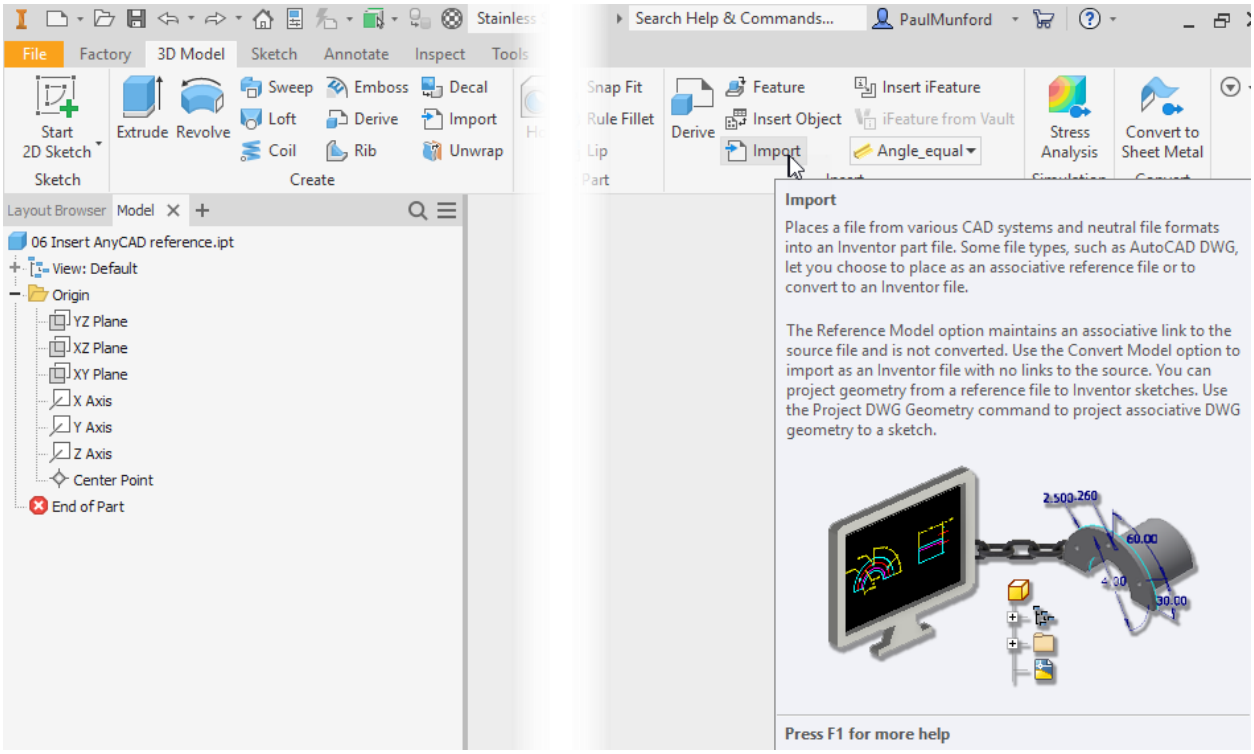
Unlike an import you can't use the AutoCAD geometry directly. You'll need to create a new sketch on the same plane that you placed the AutoCAD reference onto, and project geometry using the 'Project DWG Geometry' command.

Unlike an import, you don't edit the geometry itself, or its dimensions. To edit the geometry you open the DWG file in AutoCAD, make your edit and save the file. When you return to Inventor, the reference will update (you may need to perform a local update).

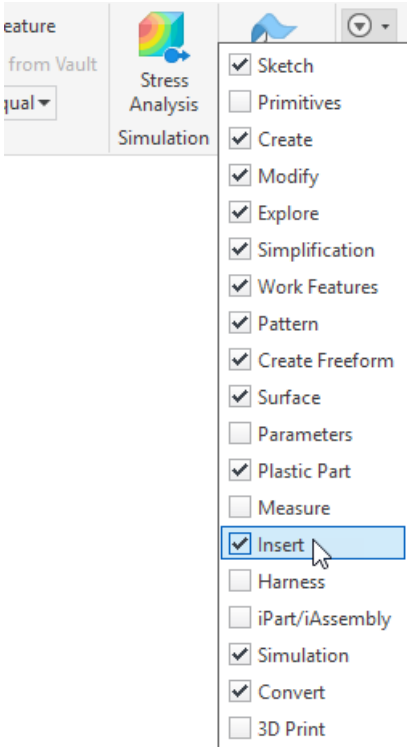
The big advantage to referencing AutoCAD files into Inventor, is that the link remains 'live', meaning – if you update the AutoCAD file, the Inventor file will update as well.

This allows you to maintain the AutoCAD file as the single source of truth but take advantage of a 3D model in Autodesk Inventor for rendering, simulation, CAM or Bill of Materials.

*For more information on **AnyCAD**, I recommend Mike Thomas's Autodesk University class 'AnyCAD and the Exchangeability of Inventor'*
https://www.autodesk.com/autodesk-university/class/AnyCAD-and-Exchangeability-Inventor-2019

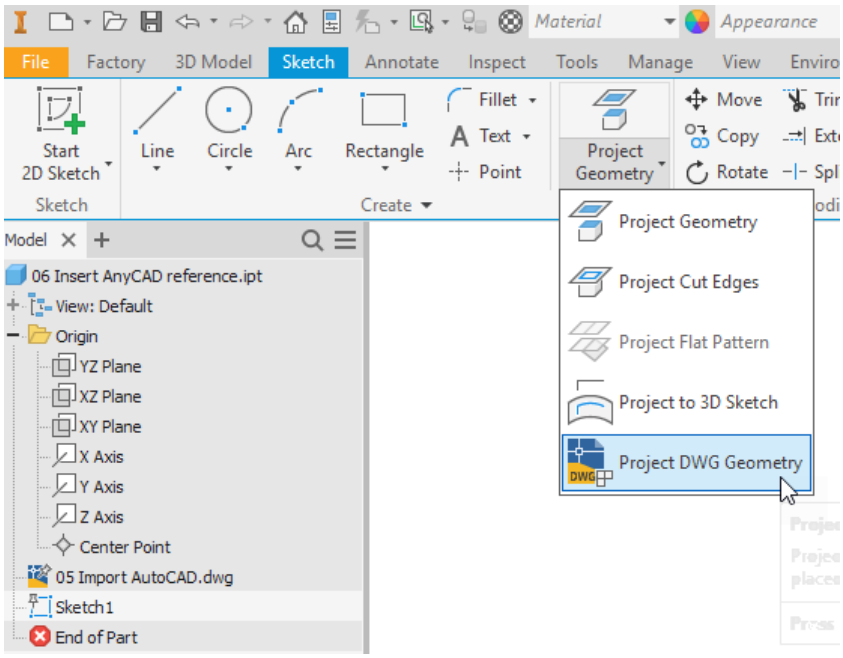


***Tip**: If you don't see the panel you are looking for, click on the 'Black triangle in a circle' drop down at the end of the ribbon, and make sure that the panel you want to see is selected.*



Try it!

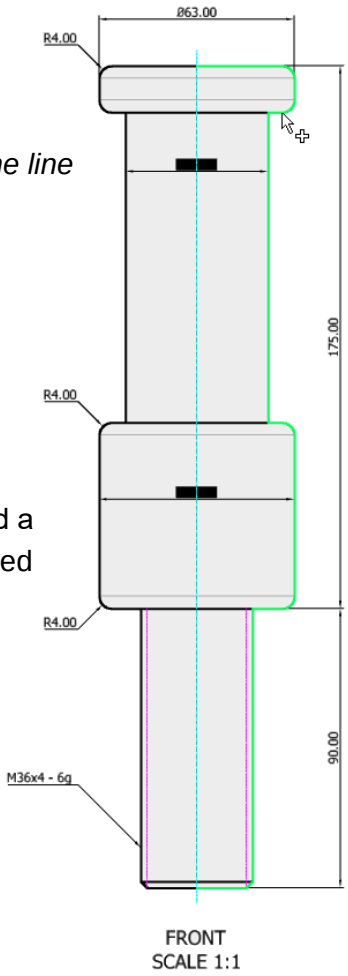
1. In Inventor, open the part file ‘06 Insert AnyCAD reference’.
2. Navigate to:
   3D Model (Tab) > Insert (Panel) > ‘Import’ Button
3. In the ‘Import dialog’ - browse to, and select ‘05 Import AutoCAD.dwg’
4. Click ‘Open’
5. Select the XY plane to place your DWG
6. Select the 0,0 (Center point) to locate your DWG.
7. In the dialog, read the warning – and click ‘OK’
   ***TIP**: Click ‘Prompts’ and select ‘Do not show this message again ever’ if you don’t want to see this message every time.*
8. Select the XY plane in the browser
9. Click on the in canvas ‘Create sketch’ button
10. Navigate to:
    Sketch (Panel) > Create (Tab) > Project Geometry (Button) - But don’t click on it!
11. Click on the bottom half of the ‘Project geometry button (It has a black downward pointing arrow on it)
12. Select ‘Project DWG Geometry’



13. Project the centreline of the AutoCAD reference, and the loop of geometry that creates the left hand profile.
14. Right click and choose ‘Cancel’ to complete the command.

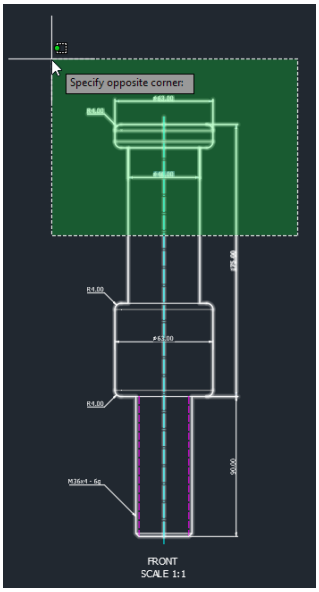
    ***TIP**: Format the projected centreline as an Inventor Centreline line type to save clicks in the next step.*

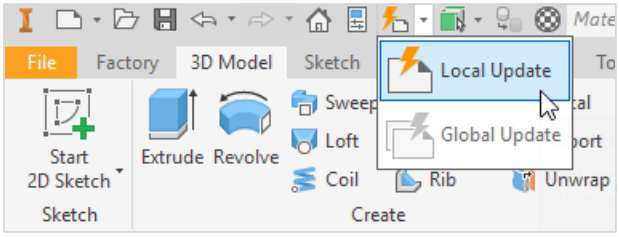
15. Click on the green ‘Check’ mark at the end of the sketch tab to complete the sketch.
16. Navigate to:
    3D Model (Tab) > Create (Panel) > Revolve (Button)
17. Select the profile and the axis and click ‘OK’
18. You now have a 3D shape!
19. If you like – you can save your file for future reference. You’ll find a complete example of this exercise in the ‘05 Examples of Exported files’ folder, it’s called ‘05 Import AutoCAD DWG Complete’.



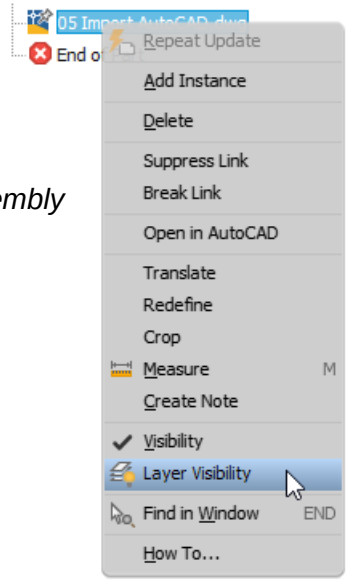
Extra credit

- In Inventor, open '05 Import AutoCAD DWG Complete'.
- In AutoCAD, open '05 Import AutoCAD.dwg'
- In AutoCAD, stretch the top half of the geometry and save the file
- In Inventor, click on the 'Local Update' button in the QAT (It looks like a piece of paper with the corner folded, and a lightning strike symbol).
- Did the AnyCAD reference update?





***Tip**: Right click on the AutoCAD DWG Node in the browser for more options, such as Layer visibility, cropping of the DWF file and suppressing the link.*



***Tip**: You can also insert AnyCAD referenced DWG files into an Assembly sketch. This is great for constraining (or jointing) components onto a layout.*

Factory design utilities

Inventor AnyCAD for AutoCAD DWG files is a great workflow for converting 2D AutoCAD drawings to 3D Inventor parts. Another useful workflow is for site layout.

You can lay out your facility or job site in AutoCAD, then AnyCAD reference this into an Inventor part or assembly file. You can insert the layout part into your Inventor assembly model and constrain your Inventor models to the AutoCAD layout.

The AutoCAD layout is now controlling the position of the Inventor components. You can have the best of both worlds by keeping the items you want in 2D and model only the items you really want in 3D.

This is pretty cool , but there is one final way that you can share AutoCAD and Inventor data that's even cooler – and that's factory design utilities.

If you subscribe to the Product Design and Manufacturing collection, then you'll already have AutoCAD and Inventor. I hope that we've shown you plenty of ways that you can work with AutoCAD data inside Inventor, and Inventor data inside AutoCAD. I hope that you can see that you don't have to choose between the two products and that you can use the best of both tools in your workflow.

The Product Design and Manufacturing collection also includes Factory design utilities, a set of tools to help you design, optimise and digitise your facility. Factory design utilities includes Recap for point cloud scan, Navisworks for model aggregation and clash detection, Process Analysis for process optimisation, and Autodesk Vault for asset data management – but we will concentrate on the link between AutoCAD and Inventor,

Factory design utilities connects AutoCAD and Inventor in a seamless, bi-directional manner that allows you and your team to collaborate on your projects in real time.

Factory includes an asset library that you can add your custom designs to in both 2D and 3D format. This allows you to work in either AutoCAD or Inventor concurrently. When you add 2D assets to your AutoCAD drawing, they will automatically sync over to Inventor as 3D assets.

After editing your design in 3D in Inventor, any changes will be sync'd back to 2D in AutoCAD.

Factory design utilities is a great tool if you have a distributed team working on facility layout and machine design and you need to collaborate effectively taking into account each other's changes as the project unfolds.

If you are a Product Design and Manufacturing collection subscriber, you already have Factory design utilities available to you, so take a look at it and see if it can help your company's workflow.

Conclusion

There are many ways that you can work with 2D AutoCAD data and 3D Inventor data. You don't have to choose one tool over another, you can use the right tool for the right job at the right time.

You can work with colleagues who use a different CAD system than you and share data and remain coordinated on your projects without a lot of manual work.

In summary:

- Inventor DWG files can be opened in AutoCAD for measuring, mark-up and review.
- Views from Inventor DWG files can be re-purposed in AutoCAD.
- Inventor DWG files can be converted to AutoCAD DWG files for editing in AutoCAD.
- AutoCAD geometry can be Copy and Pasted, Imported or Referenced with 'AnyCAD' into Inventor.
- Factory Design Utilities supports bi-directional sync between AutoCAD and Inventor.

Please try the examples from the data set that comes with this class and let me know how you integrate these workflows into your design and manufacturing process!

Thanks

Thanks very much to Christa Prokos for proofreading this handout. Any mistakes remaining are my own!

Further resources

'In the machine' Blog:
https://inthemachine-autodesk.typepad.com/blog/2010/07/official-autodesk-inventor-podcast-episode-42-using-dwg-files.html