

MFG463506

It's STILL all in the Details – Tips & Tricks of Detailing in Inventor

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Learning Objectives

- Inventor Drawing Environment - Views
- Standard and Custom Symbols
- Detailing Drawings – Dimensioning Tips & Tricks
- Modeling for Drawings - Design Views & Level of Details

Description

Do you wish detailing your drawings took far less time? Do you wish there were quicker, more efficient ways to get your detail drawings completed? This class will go over some tips and tricks you can use in creating your drawings as well as best practices, setting standards and items you tend to use often, creating and using custom symbols, efficient dimensioning, using Design Views and Level of Details to help speed up the detailing process and file activation.

Speaker(s)

Jim has 40 years in the mechanical design field covering industries of automotive, power generation, industrial machinery, consumer goods, avionics, and hazardous waste removal. Jim has been using Inventor software since Release 3 and was the first commercial implementation of Vault Workgroup software. Awarded Autodesk's, Expert Elite status each year since 2014, an Autodesk Inventor Certified Professional and an Autodesk Certified Instructor. Jim has also been a speaker at Autodesk University 2015 thru 2020 as well as a Speaker Mentor for AU2017 thru 2020.

Radu has over 20 years in the mechanical design using Autodesk software and several years in implementing ERP and Data Management products. Currently he's working in the IT division at Benson Industries, a premier custom curtainwall and external cladding subcontractor in the USA and Asia. Before this employment, Radu has worked for Imaginit Technologies for over 10 years, one of the world's largest providers of design technology solutions to the engineering community. Radu is an Autodesk Inventor Certified Professional, Implementation Certified Expert and an Autodesk Certified Instructor since 2007. He's also been a speaker at Autodesk Accelerate 2019.

Preface – The Legal Details of the Details

You may or may not realize, each drawing you create is actually a legal document, a contract per se. The drawing communicates all the required information needed to manufacture the product depicted. Add this document to a Bill of Materials, Quality Inspection requirements, ECO's, etc. and you have a legal, binding contract on how that product must be created.

Consider if the part is created incorrectly, liability can be assigned to the responsible party for not adhering to the drawing specifications, etc. But if those specifications on the drawing are incorrect, then the liability falls on those responsible for the drawing/design and who signed off on such. Granted, if you're a detailer, you're pretty well in the clear on the liability issue, but if you're an Engineer or higher up the approval chain, you are no doubt on the hook for the design you signed off on.

If a catastrophic failure happens in the field where such a design is used, this legal liability falls onto the top manager and trickles down to the Engineer on record for that design. Consider the ignition issues we saw in the news from General Motors, each person who signed off on the design as brought into court from the CEO to the Engineer.

Just one of the many reasons you should never check your own work nor sign off as an Engineer if in fact you are not a degreed Engineer, regardless if you happen to have an in-house title that claims such or even if you stayed at a Holiday Inn Express last night.

Section #1 - Inventor Drawing Environment

Views

Once you've created a new drawing to detail your 3D model, the next logical step may be to create the Views needed to not only depict the model, but to specify how to inspect and verify the physical part. Once your basic views have been created, you might need to create secondary, specific views in order to clarify the design. These views can be any number or combination of Section, Break Out or Detail views. While these may seem obvious, users tend to have issues creating them.

1.1 Manipulating Views

Views, once placed, can be manipulated in a number of ways. You can **Align** a view to another either in the Horizontal or Vertical directions. Simply select the view you want to align, Right Mouse Button (RMB) AND SELECT "Alignment", select the option of how you want to align the view, then select the view to align it to.

To **Rotate** a view after placement, select the view, RMB and select "Rotate". In the "pop-up" menu, specify how you want to base the rotation, whether it be by Edge, Absolute Angle or Relative Angle. Specify either Horizontal or Vertical and which direction of rotation (Counter-clockwise or Clockwise) by the applicable radio button, lastly, select "OK".

To change the **Scale** of a view, Select the view, RMB, select Edit, adjust the scale in the Scale field of the "pop-up" menu, select "OK"

To change the **Name** of a view, Select the view, RMB, select Edit, select the Label field of the “pop-up” menu, specify the view name you want, Toggle the View Visibility button, select “OK”

1.2 Section Views

Section Views are used to display the component or components in a cutaway style to clarify internal portions of the component(s). To create a basic Section View:



Section

1. Select the icon from the View/Create panel
2. Select the view that you want to section
3. “Draw” the sectioning line to represent the “cut” through the component(s)
4. RMB, select “Continue”
5. Set/Select the options as required in the Section View dialogue box
6. Select the placement location for the view using the LMB

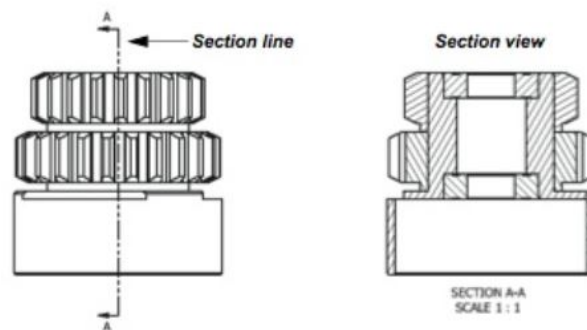
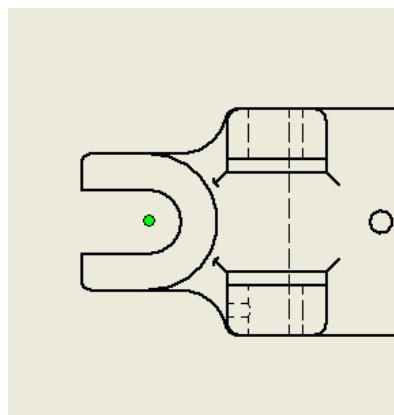
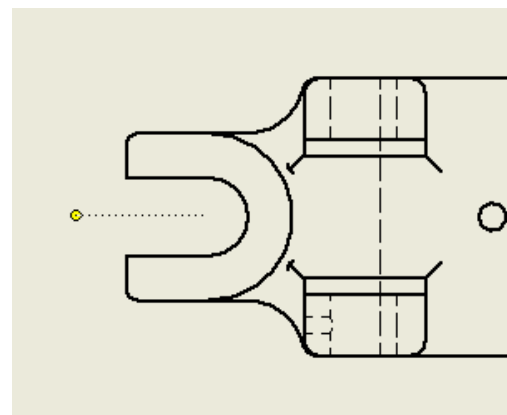


FIGURE 1: SECTION VIEW

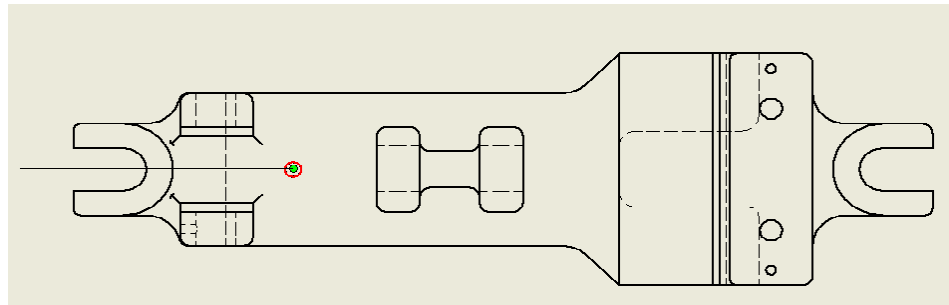
Key hint #1 on Section Views – positioning the mouse over an entity end point, midpoint, center, etc. will give you a green glyph (Figure A & C), pulling off that will give you a dotted line (Figure B) for guidance and placing the section cut location with such, will line the cut with that point (Figure D & E).



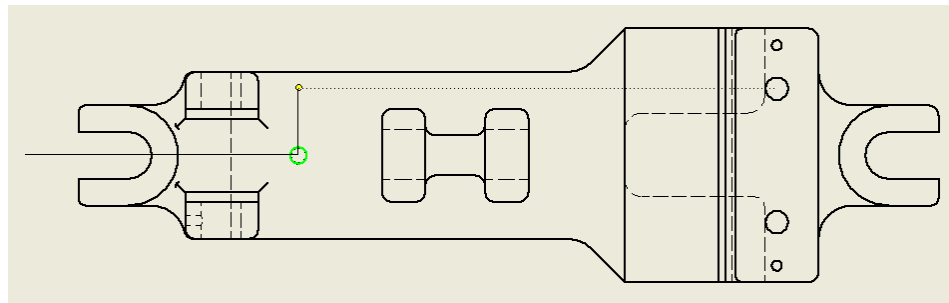
(Fig A)



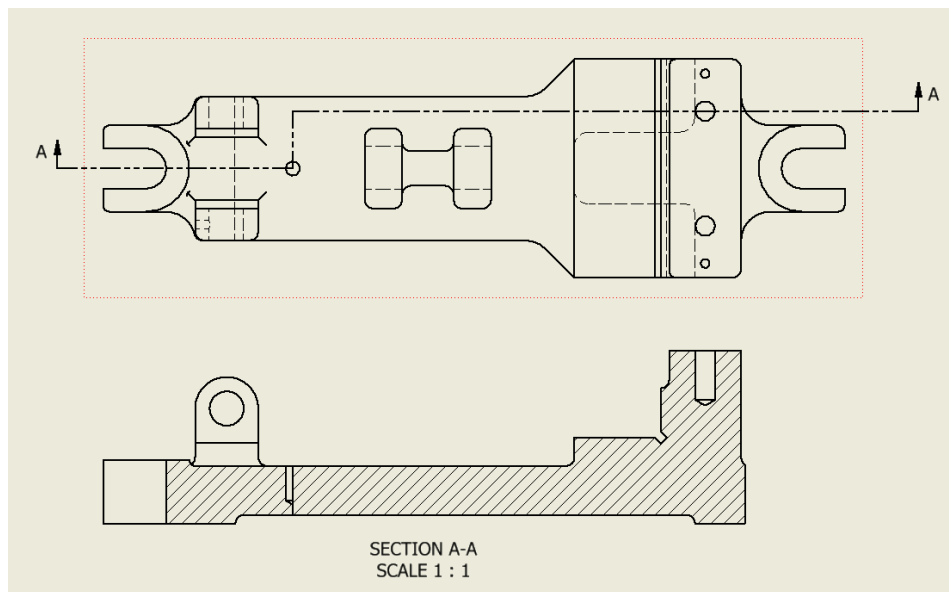
(Fig B)



(Fig C)



(Fig D)



(Fig E)

Key hint #2 on Section Views – Do not create a sketch line to locate your section view cut line. The Section View command creates its own sketch in the process. Place the section View cut line by “eyeballing” it, then once the command is done, RMB on the Section View cut line and select Edit. This will put you into Sketch mode where you can now constrain the cut line to the exact location you need.

1.3 Break Out Views

Break Out Views are used to display the areas or components that may be hidden by other components of the assembly model. To create a basic Break Out view:

1. Break Out views require a sketch to define the “cutaway”. Be sure to active the view and make the sketch associative to that view. You can select the view within the browser as well.
2. Select “Start Sketch” or with the view active, type “S” to start sketch mode
3. Sketch in the desired profile for the break out area, finish the sketch



4. In the Modify panel, select the Break Out view icon: Break Out
5. Select the view - if there is only one sketch, the sketch will be automatically selected; if there are multiple sketch profiles, select the required profile manually
6. Specify the depth type in the dialogue box pop-up
7. Click “OK”

1.4 Detail Views

Detail Views are used to enlarge an area of a parent view in order to display the details more clearly. To create a basic Detail View:



1. In the Create menu, select the Detail View icon: Detail
2. Select the parent view
3. Set the required options in the dialogue box
4. Select a point on the parent view that will specify the center of the detail view
5. Move the cursor away from the center location, select a location to specify the size of the area
6. Select the placement point of the Detail View

1.5 View Based Sketching

View based sketches are typically used for creating cut boundaries to be used in Break Out Views as specified above. They are also used when creating a Section View, the Section Line is an automatically generated view-based sketch. You may have numerous other reasons or uses to create a view-based sketch, but the process is the same regardless. Just be sure you select the applicable view or have it activated, otherwise the sketch you create will be a sketch of the drawing itself and thus will not adjust with the view as you may intend, such as moving, etc. To create a Sketch Based View:

1. Select the Sketch command in the ribbon, then select the desired view, or
2. Select the desired view and type "S" on the keyboard
3. You'll see the origin crosshairs signifying you are in sketch mode for that view
4. Create your desired sketch
5. RMB, select "Finish Sketch"

1.6 Manipulating Views

Views, once placed, can be manipulated in a number of ways. You can **Align** a view to another either in the Horizontal or Vertical directions. Simply select the view you want to align, Right Mouse Button (RMB) AND SELECT "Alignment", select the option of how you want to align the view, then select the view to align it to.

To **Rotate** a view after placement, select the view, RMB and select "Rotate". In the "pop-up" menu, specify how you want to base the rotation, whether it be by Edge, Absolute Angle or Relative Angle. Specify either Horizontal or Vertical and which direction of rotation (Counter-clockwise or Clockwise) by the applicable radio button, lastly, select "OK".

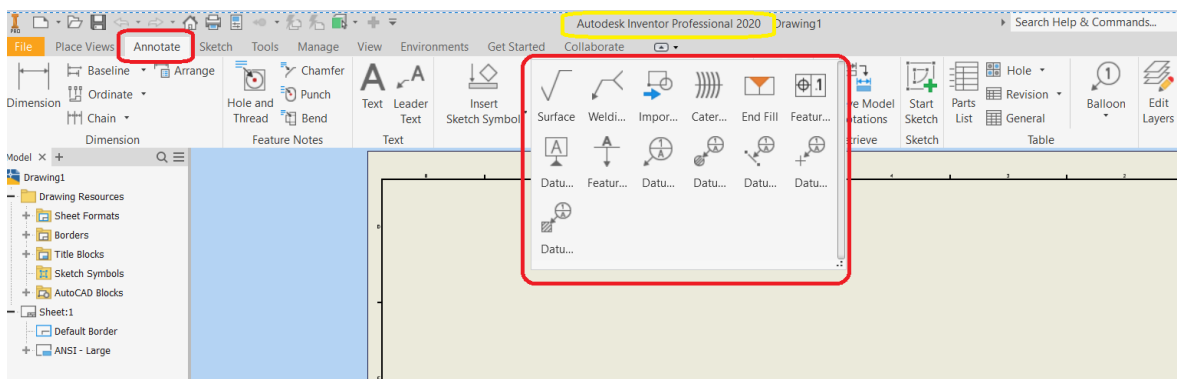
To change the **Scale** of a view, Select the view, RMB, select Edit, adjust the scale in the Scale field of the "pop-up" menu, select "OK"

Section #2 – Standard and Custom Symbols

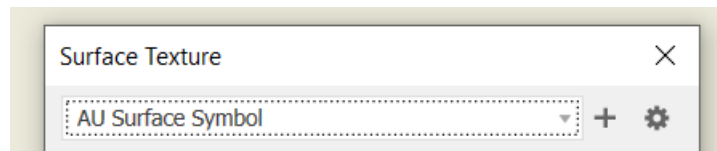
2.1 Standard Symbols

They are predefined annotations such as surface texture, features control frame, welding or datum identifier. You can copy existing symbols and paste them in your drawings as new symbols to quickly and easily create new symbols. The active style determines the symbol appearance; for further details see the Style and Standard Editor.

Create a new drawing using your template and add a couple of standard symbols, a surface texture symbol and welding symbol, see below.



Click the plus sign and save a preset of the surface texture symbol, see below.



If you create another drawing, you can reuse your preset symbol without the need to add the details again.

2.2 Custom Symbols

(Be sure to download the included dataset)

Local sketch symbols are stored in the Drawing Resources Sketch Symbols folder in the browser. Symbols created in the current drawing can be saved locally or to an external symbol library. Symbols created in a template file are available locally in all drawings using that template. Symbols added to and stored in a sketch symbol library can be accessed by multiple users and computers.

You can save all your sketch symbols to a sketch symbol library, right-click on the Sketch Symbol node in your Model browser and select Save All to Symbol Library.

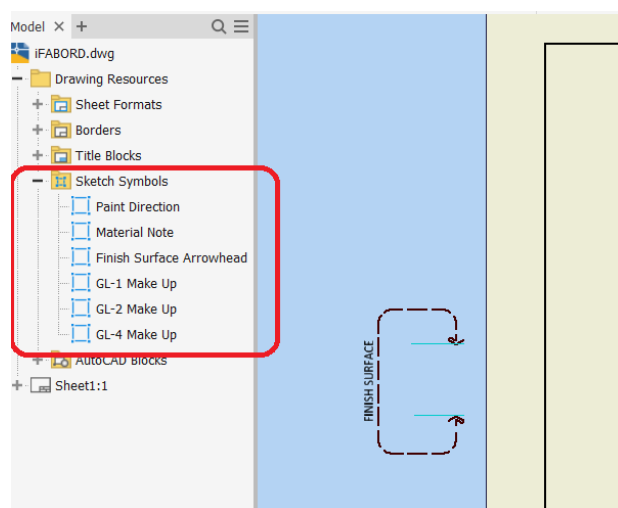
If no library exists, select the Create a New Library icon to create a new external library. Select the folder icon to create a subfolder within the selected library if desired.

Note: Sketch Symbol Library location is saved by default to C:\Users\Public\Documents\Autodesk\Inventor 2020\Design Data\Symbol Library. You can add, delete, or rename libraries here.

Two sketch symbols cannot be saved to the symbol library with the same name; actually, a library is a drawing.

A sketch symbol can contain geometry, text, iProperties, prompted text and imported images. All of these are going to be demonstrated below.

Open iFabord drawing (from the included dataset). Notice the saved sketched symbols created and stored in the template.

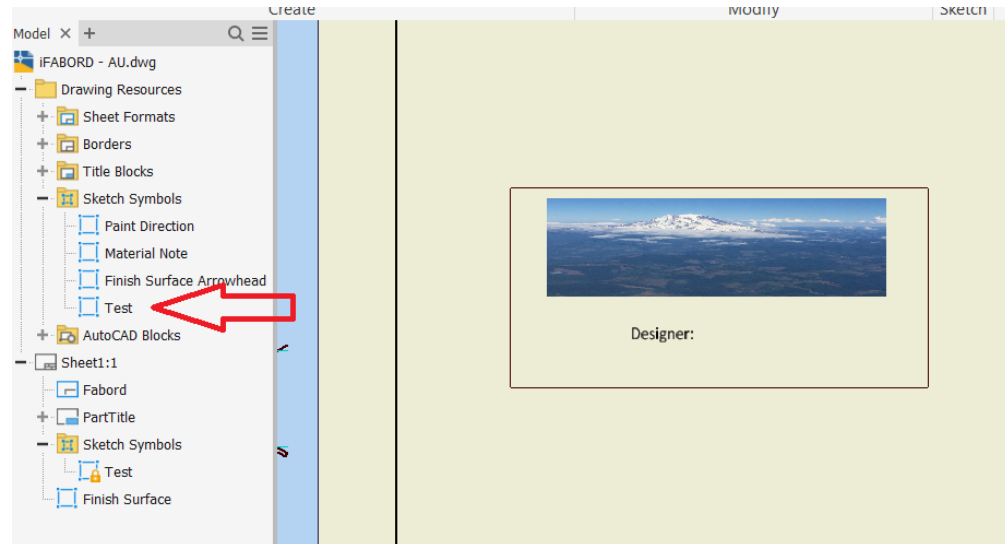


The Finish Surface symbol, that is not in the folder, was created in a separate sketch and it's going to be deleted if there is no need for it.

Creating a new symbol:

While in the iFabord template, right click on Sketch Symbols folder in the browser and pick <Define a new symbol>

Add some geometry, a text linked with an iproperty from a model and picture. Name the symbol and notice that it is saved in the Sketch folder, see below.



Add a Base View and notice how the Designer field updates accordingly with model iProperty.

Insert existing symbols from the Sketch folder and see how they behave.

If you need to update a symbol, pick <Edit Symbol> from the right click menu.

Open iTagbord template (from the included dataset) and notice other symbols that are created and saved.

DTL is the Detail number and SHT the Sheet number where people need to look for other drawings. (FAB and SHOP drawings).

Bonus: Revision Cloud

It is not a default function in Inventor, but can be added by using the Revision Cloud add-in. Follow the steps provided by this link:

<https://knowledge.autodesk.com/support/inventor/troubleshooting/caas/sfdcarticles/sfdcarticles/How-to-create-revision-cloud-in-Inventor.html>

If this link becomes non searchable, open any search engine and type: <How to create revision cloud in Inventor>. Make sure that you are reading the solution suitable for your current release.

Unfortunately, revision cloud is not coming in the upcoming release so better get used with the installation. For more fresh details please vote for it if you need Autodesk to implement it.

Follow the steps provided by this link:

<https://forums.autodesk.com/t5/inventor-ideas/inventor-revision-cloud/idi-p/3739886>

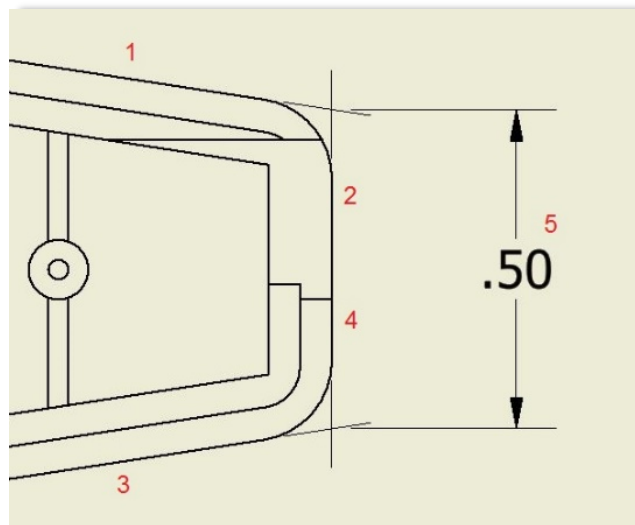
If this link becomes non searchable, open any search engine and type: < inventor idea revision cloud>

Section #3 – Detailing Drawings – Dimensioning Tips & Tricks

Dimensions, Text and Notes are the bulk of your detail drawings, knowing how to use the numerous variations will make detailing your drawings go a lot smoother and look more professional. Majority of your dimensions will use the default “Dimension” command. The Inventor application will “know” the type of dimension based on your selections. With that in mind, there are a few little tricks for specific dimensions which we will tackle a few here:

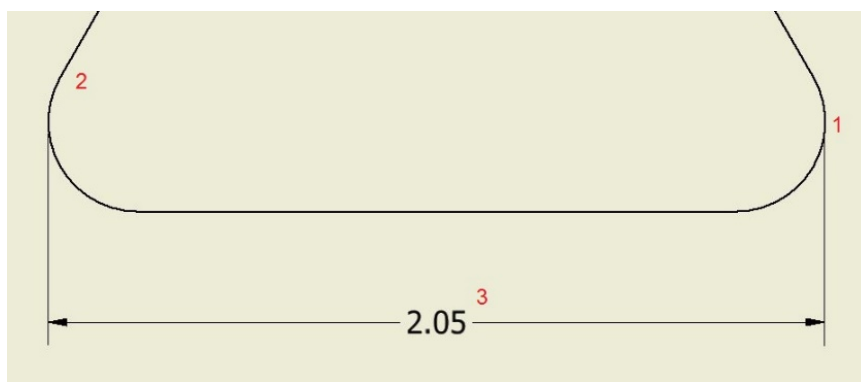
3.1 Intersection Dimensions (based on the provided example)

Using the default Dimension command, select the first reference line (1), RMB and select “Intersection” in the pop-up menu, select the second reference line (2), move the mouse over the third reference line (3) and select, RMB and select “Intersection” again and select the same (second) reference line (4), move mouse off to side to locate dimension and click (5) in the graphics window to place.



3.2 Tangent Dimensions (based on the provided example & pre-2020)

Using the default Dimension command, select the first circular reference (1), **Note:** it'll default to a radial or diametric dimension, simply ignore this and move to select the second circular reference (2), but don't be click happy here. This takes some finesse and a careful eye. With the mouse over the second reference edge, move the cursor towards the (in the example shown) top portion until you see the "tangent" glyph (you'll originally see the default dimension glyph showing a dual arrowhead line in the horizontal or vertical orientation) and a yellow circle at the cursor arrowhead, select there, you'll see the tangent dimension previewed, position and click in the graphics window to place the dimension (3).



3.3 Model Dimension



Using Model Dimensions offer numerous benefits, from not having to add a dimension manually on the detail sheet to having the ability to edit the model from the detail sheet. You have two choices on how to "import" these from the model into your detail drawing:

- At view creation – on the Recovery Options tab (*Display Options prior to 2020*), select the "All Model Dimensions" check box. This will bring over all model dimensions for that view. You can then arrange them as needed for clarity and even select the ones you don't want to include. This is the best way if for example you want to bring over most of the model dimensions...it takes less steps to deselect a few than to select a bunch one by one.
- After the view has been created – On the Annotate tab, on the Dimension menu, select "Retrieve" and select the Model dimension you want to insert into said view. This option is available on both the Ribbon Menu and by RMB

3.4 Ordinate Dimensions in Detail Views

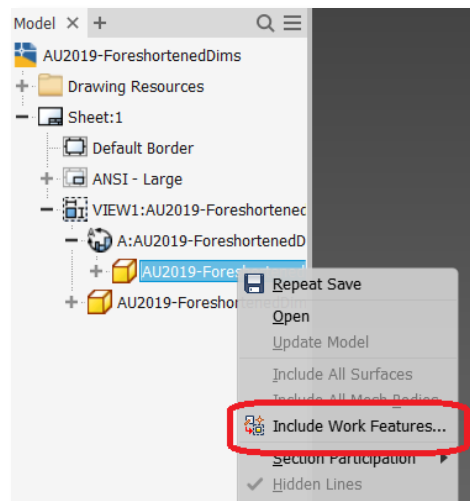
Creating an Ordinate dimension, especially on a Detail View, has always been a bother for users, more so for those just starting out with Inventor. Add to that, it's a "use it or lose it" type of process that is easily forgotten if not used often.

Let's look at a few examples:

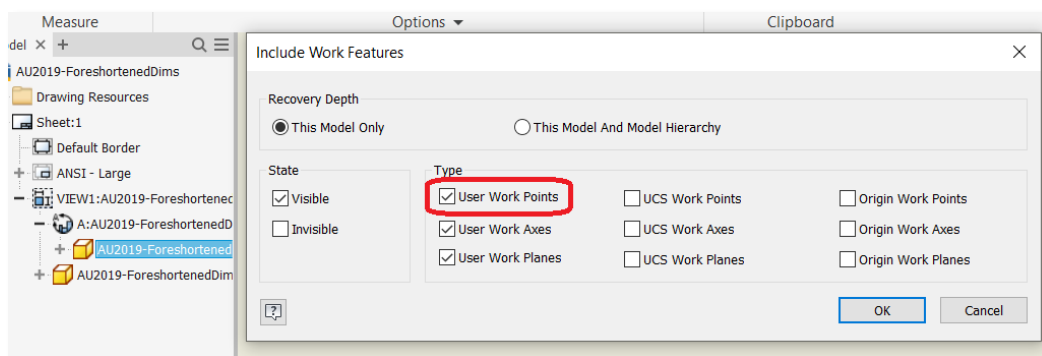
First and foremost, you'll need to know where the origin point is on the part. Is it at the edge you want the ordinate dimensions to be based from? Add a work point to the model at this location.

Once you've decided or added the work point to be used for the "zero" location of the Ordinate dimension, you'll need to make it visible in the Detail view for selection during the dimensioning command.

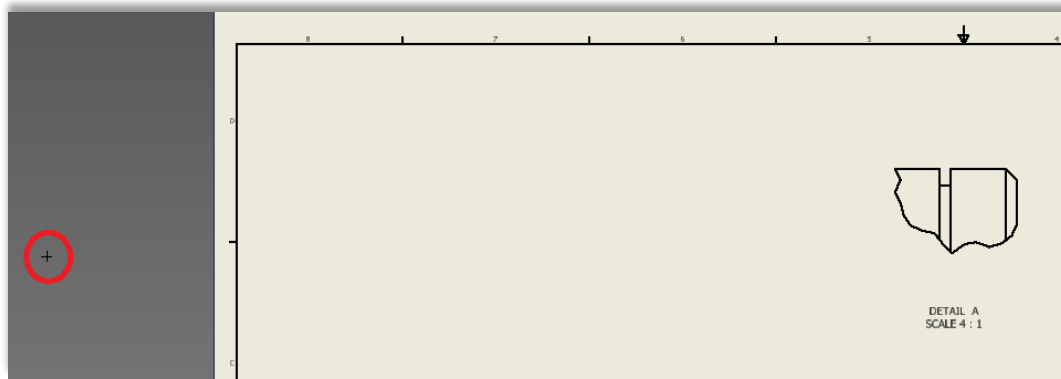
To do this, in the browser tree, expand the node for the Detail view you are placing the dimensions in. RMB on the model in the browser tree and select "Include Work Features"



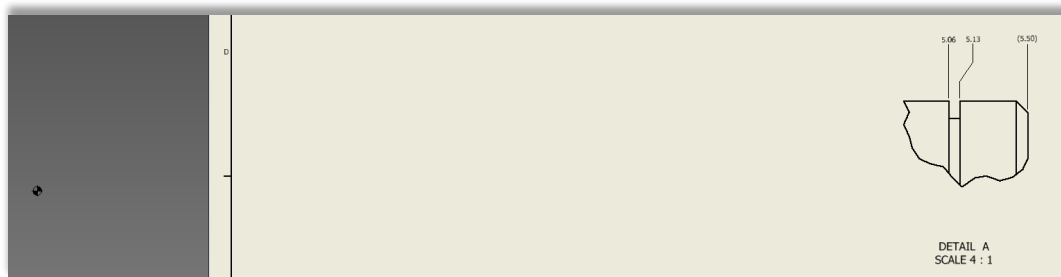
This brings up the following menu:



Click “OK” and the Work Point will now be visible for that view

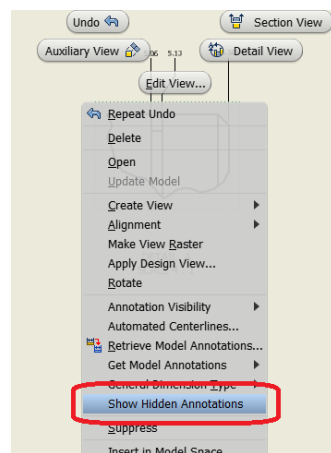


You can now start the Ordinate Dimension command and select the work point as the start (Zero) of the dimension:



To hide the work point, simply RMB and deselect “Visibility”.

To get the work point back again after the fact, RMB in the view and select “Show Hidden Annotations”



Section #4 – Modeling for Drawings (Design Views & Level of Details)

There are two main sub sections, View Representations and LOD (Level Of Detail) Representations

4.1 View Representations

View Representations control component visibility and transparency, sketch and work feature visibility, component selection status (enabled or not enabled) and others. Turning off the visibility of components do not remove them from your system's RAM, therefore View Representations are not used to manage memory.

Also, can be used as a filter in the drawing's part list.

4.2 LOD Representations

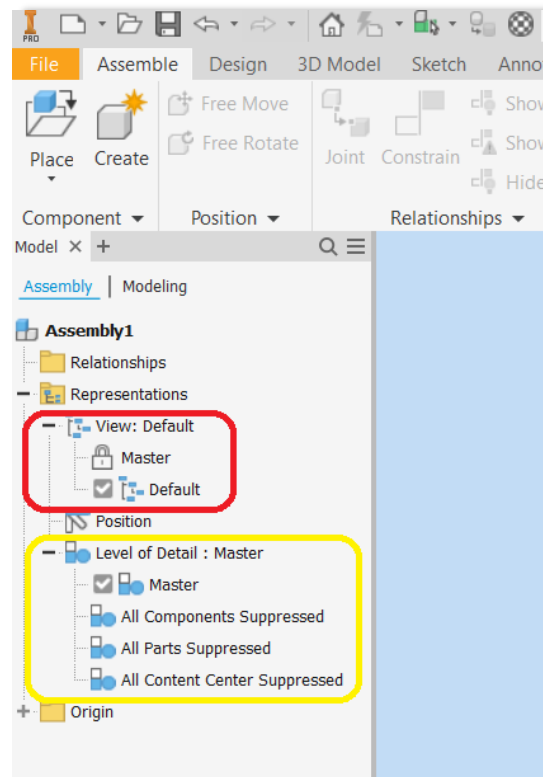
Level of Detail Representations improve capacity and performance. They suppress unneeded components or replace many parts with a single part representation to reduce memory consumption and to simplify the modeling environment. Save the representation with a name and activate it for modeling tasks or select it for creating drawings.

Level of Detail Representations control only suppression states of components and are used to manage memory by not loading the parts into RAM. They cannot be used to filter the drawing's part list.

Use the capacity meter to manage memory consumption. To improve performance when working on a large assembly, use the capacity meter to determine the current state of memory consumption. When you want to reduce memory consumption, determine if you can suppress some components and create a new Level of Detail representation or use a Substitute Level of Detail to reduce memory consumption even further.

Practice 1

Open a new assembly and notice the default view representations.



Close it and open Unit 52.iam (from the included dataset).

Create a new drawing based this file. In the “Unit” assembly file, make the glass transparent (RMB on the model in the browser tree and select “Transparent”) and notice if it updates in the drawing view(s). Depending on your view settings, it may or may not update. You can test this with Visibility as well.

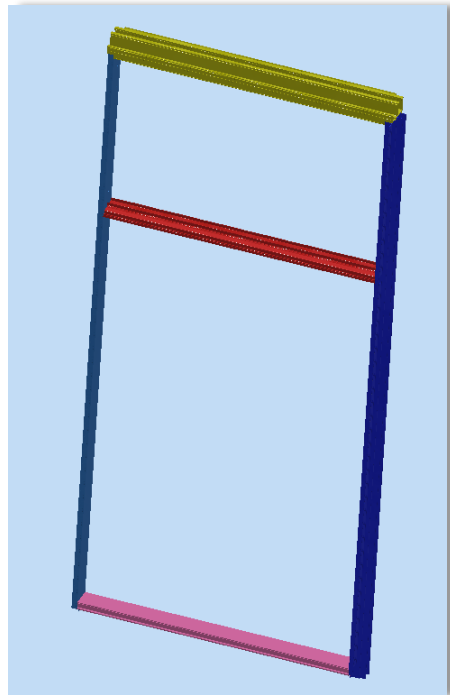
When a component is set to Transparent in the assembly, transparency is carried over in associative drawing views. Drawing view transparency overrides are disabled in these associative views. To manage component transparency in the drawing view you must turn off the view associativity.

Close all files and don't save anything.

Practice 2

Open the Unit 52 assembly again (from the included dataset).

Create a new View Representation by expanding the Representations folder in the model browser, right clicking on the View node and selecting “New”, name it “Frame”. Turn off visibility of the components until it looks like the example below:

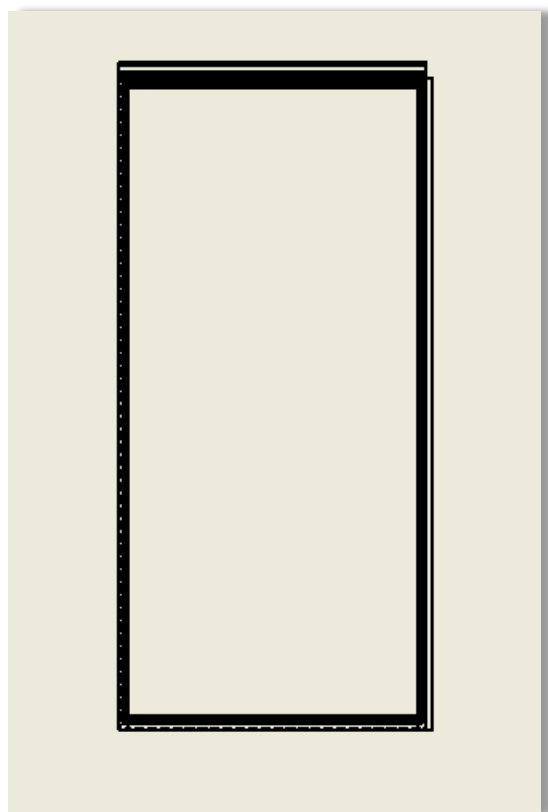


Save the file.

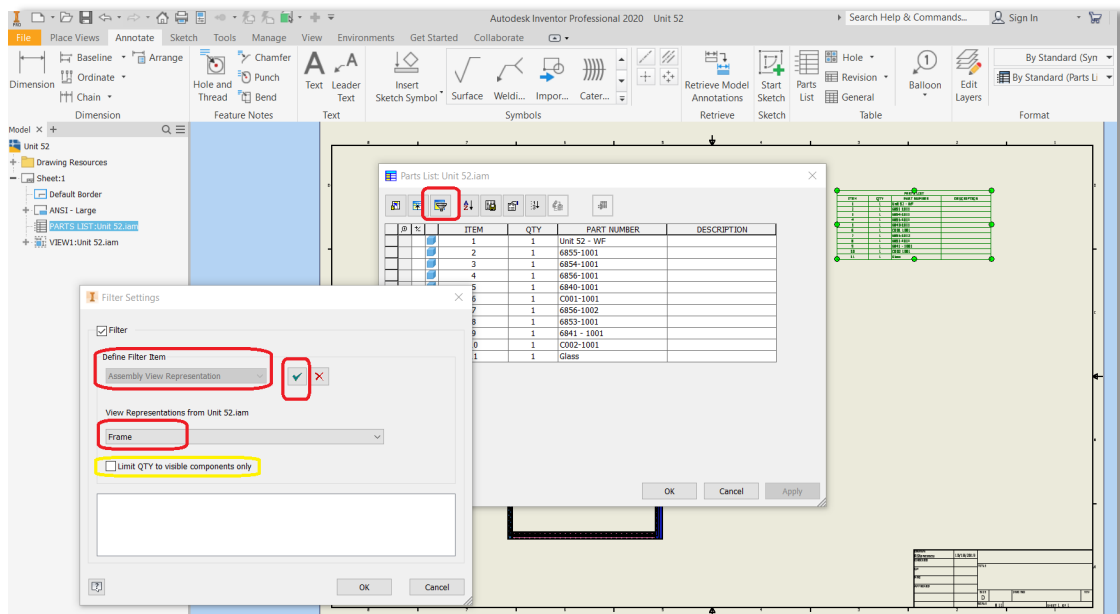
Switch between Default and Frame and notice the difference.

Create a new drawing and add a view with Frame view rep.

In the assembly while Frame is active, turn visibility off for the “6853” component. Return to drawing and notice that view updates accordingly, see below:



Add a Parts List and notice that it shows all components. You need to add a filter to display only the parts of Frame view rep. Edit the Parts list, click on Filter and select the Frame view rep, please see below highlighted in red.



If more parts are added in the assembly, they will show up in the Parts List; there is Lock option for Frame, so it will prevent the view rep of changing and part list to update. To lock a view representation, right click on the view and pick <Lock>.

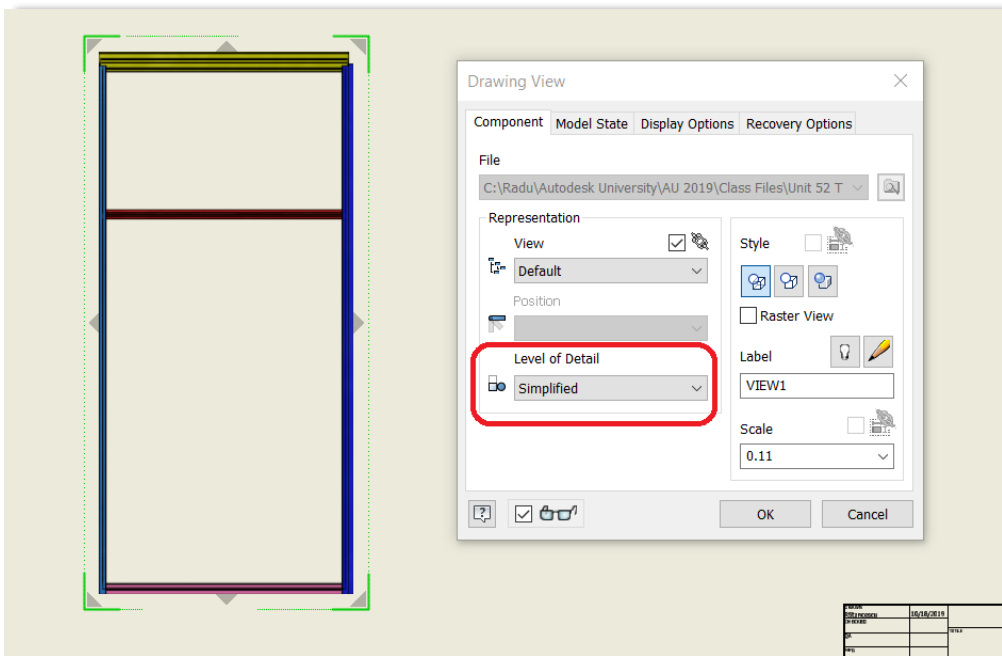
For an accurate display of quantities in Parts List, consider selecting <Limit QTY to visible components only, see highlighted in yellow above.

Practice 3

Notice the Capacity Meter showing 11 and 12. (all drawings must be closed)

Create a new LOD by expanding the Representations folder in the model browser, right clicking on the Level of Detail node and selecting "New Level of Detail", suppress five parts and name it Simplified. You cannot suppress parts and save the assembly without creating a LOD. Capacity Meter shows 6 and 7.

Create a new drawing and add a view for assembly using Master LOD. You can always edit the view and switch to Simplified LOD, see below.



You can close the assembly for a better visualization of capacity meter numbers change.

Tips:

1. If you add more parts while being in Master LOD, then Simplified LOD updates too with same parts. There is no Lock option as in view rep.
2. If you add a Parts List with Simplified being active in view drawing, then it shows all parts. A parts list always references only the Master Level of Detail representation. Suppression or Unsuppression in any Level of Detail representation is never reflected in a parts list.
3. You can copy a View representation to a Level of Detail representation to remove invisible components from memory by suppressing them. Use these steps to create a Level of Detail from a View representation. A new Level of Detail is created which duplicates the name and component visibility status from the View representation.

You can also copy a LOD to View representation if desired.

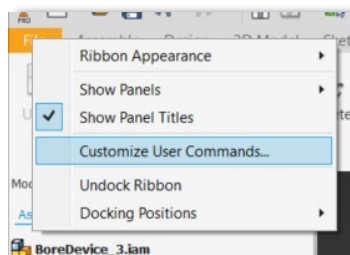
Bonus Section – Customizing Inventor

One of the easiest ways to customize your Inventor environment is by changing the settings in the Applications Options/Document Settings dialogue box. Adjusting a few of these settings can also reduce file load times of very large files as well as help in the navigation around the application based on your personal preferences. *(Refer to my AU 2019 class: CADZilla Returns for more information)*

Some of the items you can customize:

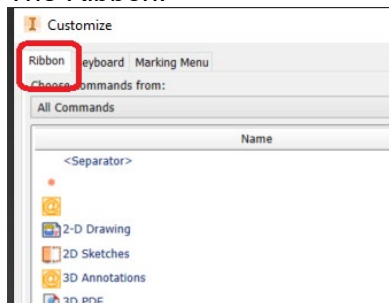
1. Grid Lines
2. Auto-projecting part origin & edges
3. Place and ground first component in an assembly
4. Increase or decrease the Undo file size
5. Show My Home (or not) *[if you notice lag, clear the preview files]*
6. Change the color and image of your background
7. Reverse the Zoom behavior (to be like AutoCAD)
8. Drawing File Type (.idw or .dwg)
9. Enabling dimension edit on creation (big time saver)
10. Change the units of the file (inch to mm or vice versa)

To customize user commands, right click on the File tab and select “Customize User Commands...”

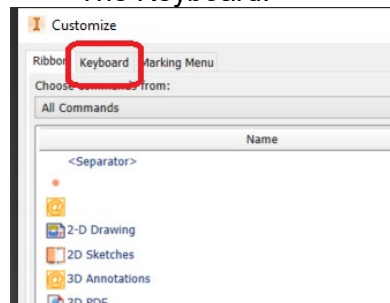


The following pop up menu allows you to customize:

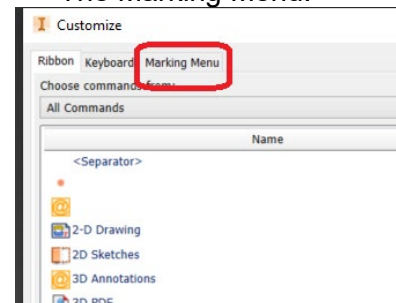
The Ribbon:



The Keyboard:



The Marking Menu:



The Customize menu also allows you the options to Import/Export these settings.

NOTES: