It’s all in the Details – Best Practices for Detailing Drawings

Jim O’Flaherty – Applications Expert - MSD

Twitter: @CadZilla01

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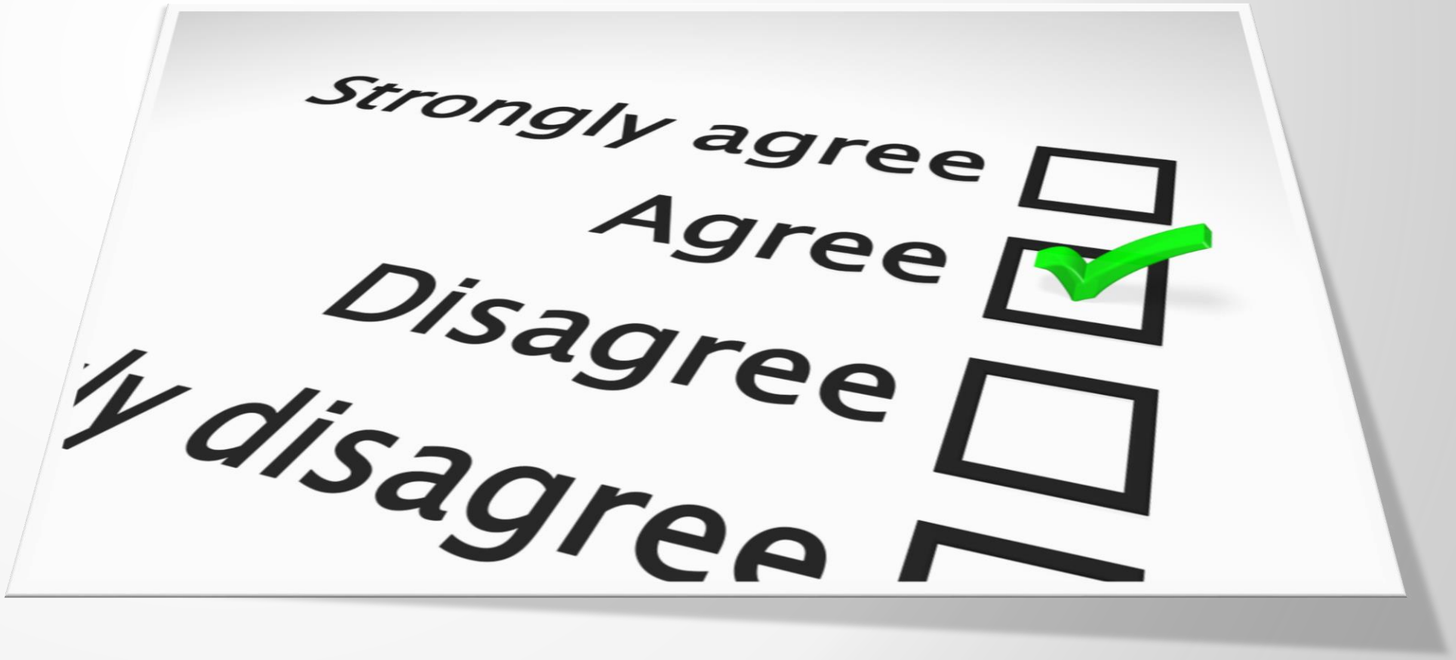
- Via the Survey Stations, email or mobile device.
- AU 2016 passes awarded daily!
- Give your feedback after each session.
- Give instructors feedback in real-time.



Who am I?

- James “Jim” O’Flaherty
- Applications Expert - Manufacturing Solutions Division
- Started career in 1980 using a drafting board and a slide rule
- Began on Autodesk Inventor @ Release 3
- Autodesk Certified Inventor Professional
- Autodesk Certified Inventor Instructor
- Autodesk Expert Elite
- Other CAD software? (those shall remain nameless)

Class summary

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?

We will cover 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.

Key learning objectives

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

- Navigate & Customize your local Inventor environment
- Create Custom, Commonly Used Symbols
- Set & Enforce In-House Standards
- Detail Drawings More Efficiently
- Use Design Views/Level of Details

Preface – The Legal Details of the Details

The legalities of a Mechanical Drawing

Legally speaking drawings are in fact legal documents because they communicate all the needed information about what is to be created for the customer. Thus it is part of the contract as is the purchase order along with other documents such as ECOs, specifications, etc.

If the product is produced incorrectly, liability can be assigned to the responsible party based on the drawing and whether or not the product was created per the drawing specification.

If those specifications are found to be wrong, liability falls squarely on those who signed the approvals for the drawing, i.e. Engineer, Director of Engineering, QA, etc.

Documents show GM rejected better ignition switch in 2001
Drawings Of Two Options Given To Investigators
GM Is Set to Face Criminal Charges Over Ignition Switches
General Motors will likely be prosecuted over ignition switches linked to deaths
Document Shows GM Engineer Approved Ignition Switch Change

The legalities of a Mechanical Drawing

Drawings are becoming more of an inspection document in the current era of 3D modeling where the 3D database is “read” by the machine as opposed to the machinist, practically eliminating human error of misinterpretation. This 3D model and associated drawing(s) are the “perfect” part, the tolerances you provide determine if the part passes or fails Quality Control inspection and whether the part goes out to the customer or the recycle bin.

How many of you use a variation of dimensioning standards? (ANSI/ASME/DIN/JIS/etc.)

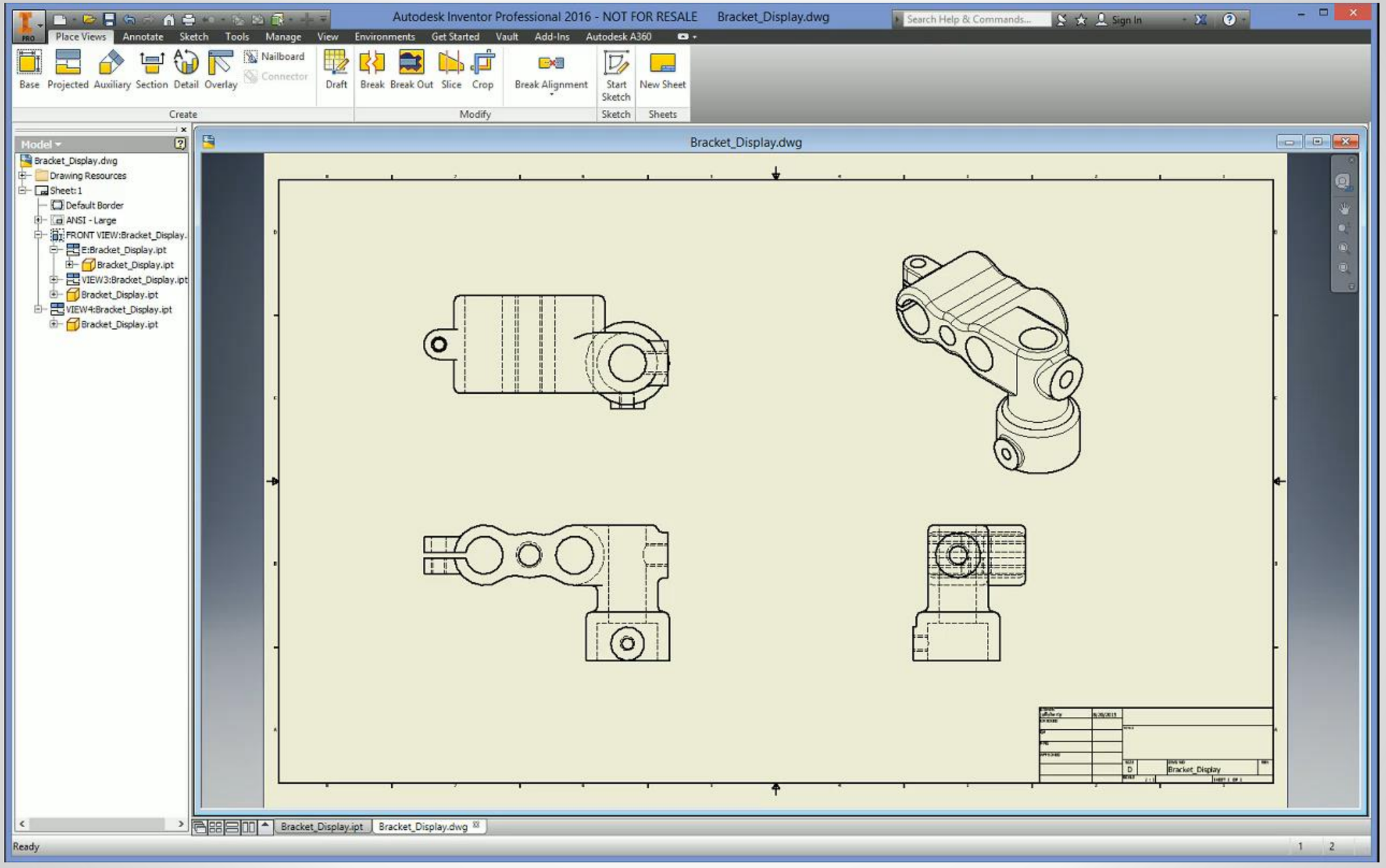
How many of you use GD&T? (Geometric Dimensioning & Tolerancing)

For more info on GD&T and how it can help, see me after this class

Section #1 – Views

Views

- Manipulating Views
  - Align
  - Rotate
  - Scale
  - Name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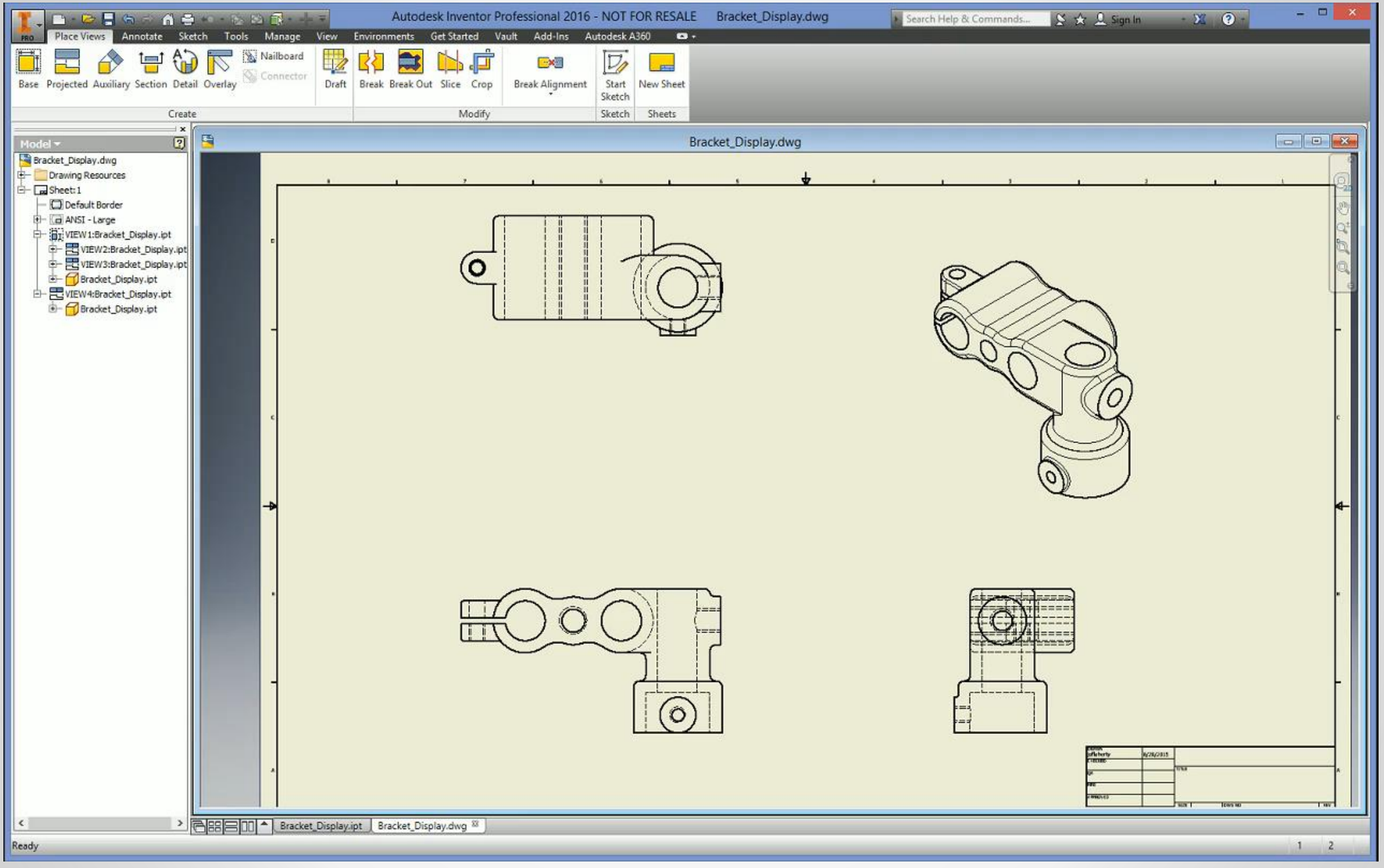
Views

- Section Views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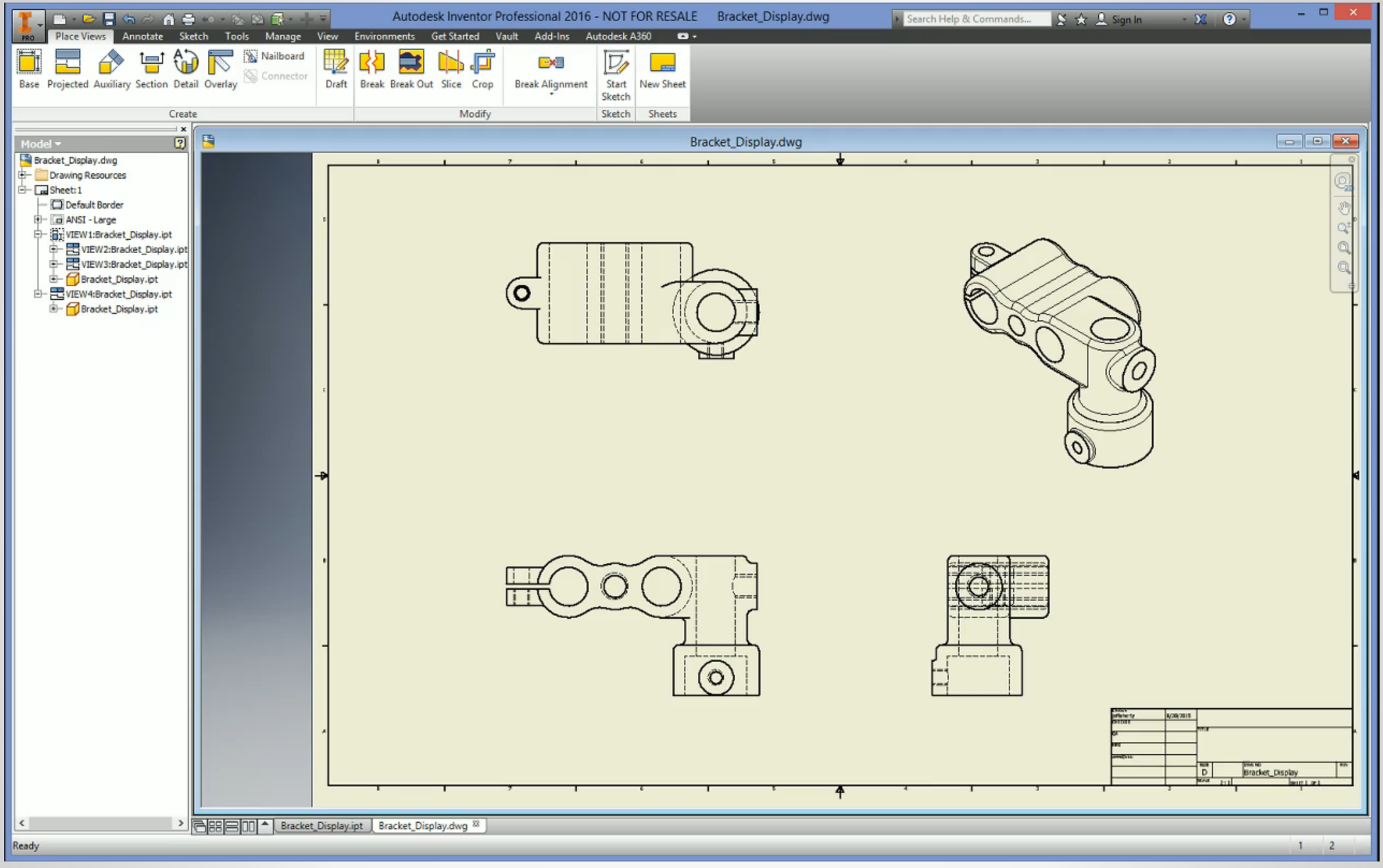
Views

- Detail Views & Break Out Views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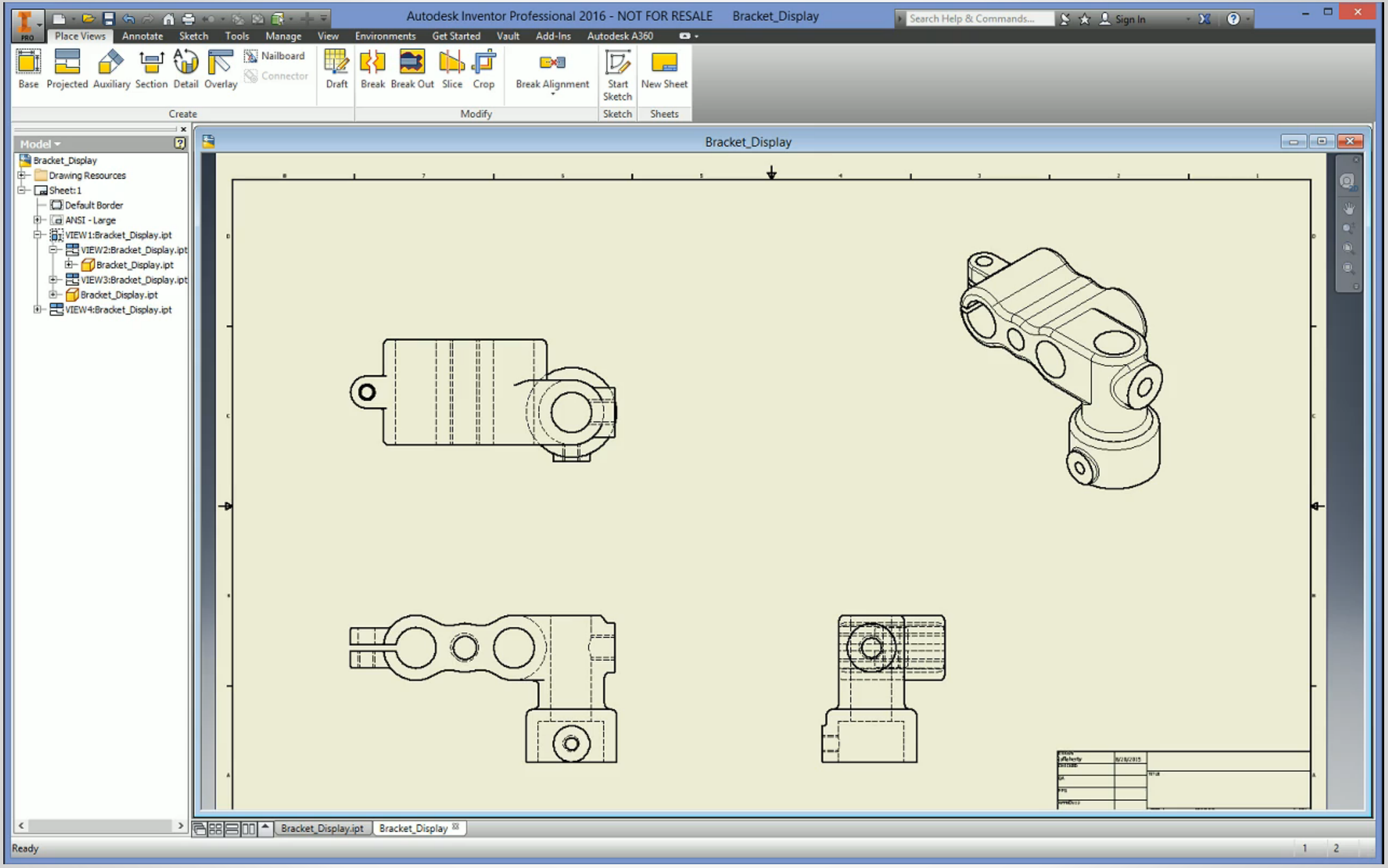
Views

- View Based Sketching
  - Select the view you want to sketch in and then select the Sketch command (keyboard “S” or Ribbon Menu)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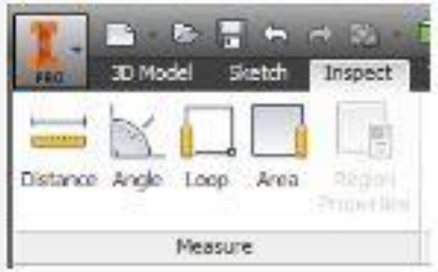
Section #2 – Measure Tool

Measure Tool

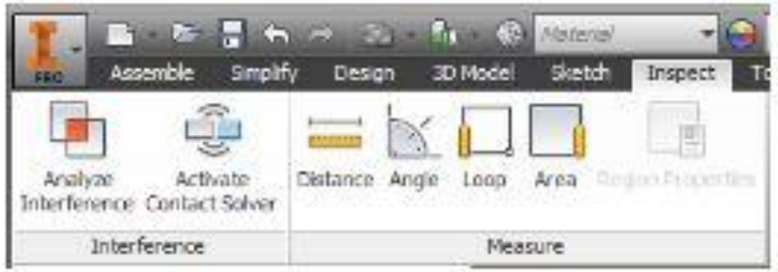
- To use the Measure Tool to it's full extent, you'll need…

PATIENCE

- You can measure Distance, Angle, Loop, and Area
- Region Properties will calculate the area of a selected closed sketch
- The Measure Tool is the same for Part and Assembly mode on the Inspect tab> Measure panel



Part File



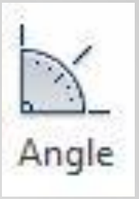
Assembly File

Measure Tool – Distance or Length

- Select the geometry, results are shown in the dialogue box
- To take a new measurement, select Restart in the drop-down list
- How to:
  - Measure Edge Length – Select the edge
  - Measure Diameter – Hover (here's the "patience" issue) the cursor over the cylindrical face or edge until you see the diameter line, then click
  - Measure Radius – Same process as Diameter
  - Measure Position of a point from origin – Select the point, the position of the point relative to the coordinate system axes is displayed
  - Measure Distance between points – Select the 1st point, then the 2nd point, the deltas are displayed with respect to the 1st point
  - Measure Distance between edges – Select the two edges
  - Measure Distance between an edge and center of a circular edge – Select the edge and place the cursor on the circular edge until a "green dot" displays at the center of the circle, click to select the center
  - Measure to a Midpoint – Hover the cursor over the 1st midpoint of a linear or arc entity, select. Select 2nd point or midpoint

Measure Tool – Angle

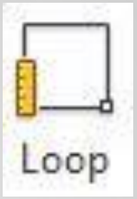
- Select the geometry, the result is displayed in the dialogue box



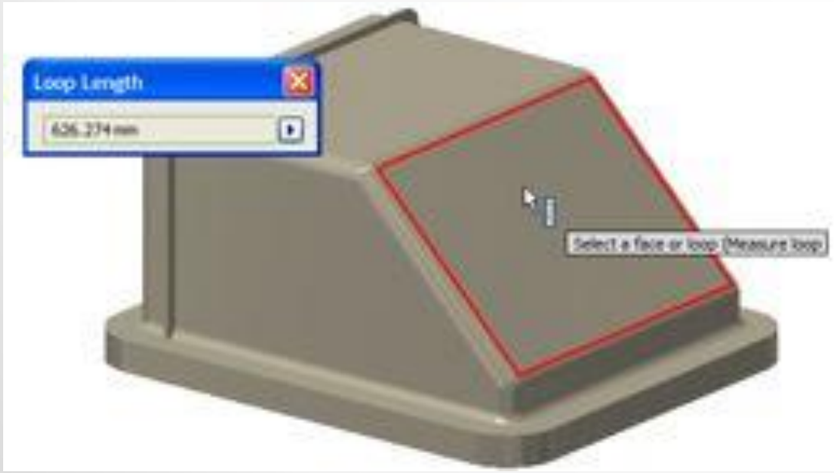
  - To measure the angle between two lines or faces, select the two lines or faces.
  - To measure the angle created by three points, select the first two points that define a line and then select the third point that defines the angle.
  - To measure the distance to a midpoint of a segment, hover over the midpoint until a yellow dot displays and then select it
  - To take a new measurement, select Restart in the drop-down list

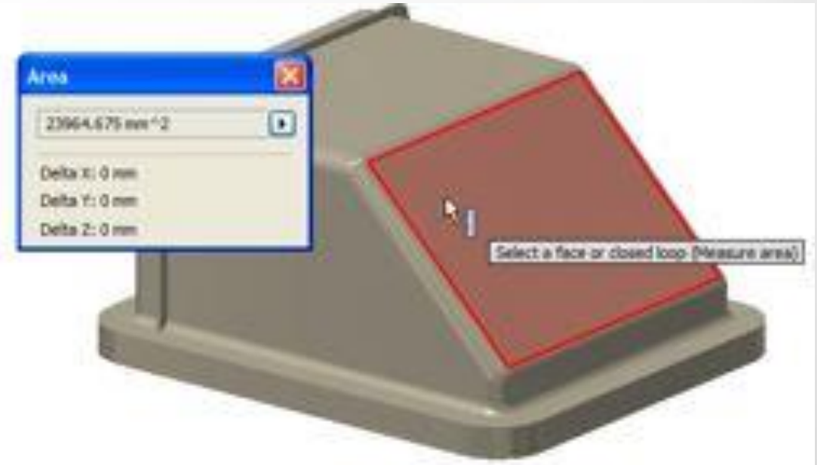
Measure Tool – Loop & Area

- Loop – Select the face or a loop, value is displayed in the dialogue box
  - To take a new measurement, select Restart from the drop-down list



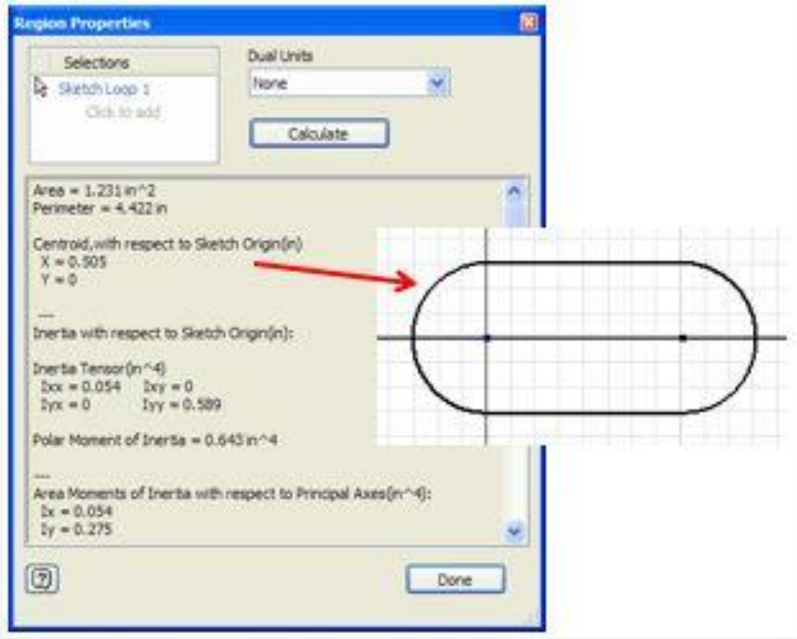
- Area – Select a face or a closed loop, value is displayed in the dialogue box
  - To take a new measurement, select Restart from the drop-down list





Measure Tool – Region Properties

- Calculates properties for a closed sketched area
- Must be in a sketch
- Select a section and click the “Calculate” button
- Calculations are completed based on the sketch coordinate system



Section #3 – Customizing

Customizing Inventor

- Use the Application Options/Document Settings dialogue box to adjust a few settings that either can help load times and navigation to personal preference
  - Grid Lines
  - Auto-projecting part origin & edges
  - Place and ground first component in an assembly
  - Increase or decrease the Undo file size
  - Show My Home (or not) *[if you notice lag, clear the preview files]*
  - Change the color and image of your background as well as the icons
  - Reverse the Zoom behavior (to be like AutoCAD)
  - Drawing File Type (as of 2015, Inventor defaults to .DWG as opposed to .IDW)
  - Enabling dimension edit on creation (big time saver)
  - Change the units of the file (inch to mm or vice versa)
  - Change the colors of the Sheet, Sheet Outline, Highlight and Selection
  - Change the 3D Sketch Auto-Bend radius

Customizing Inventor

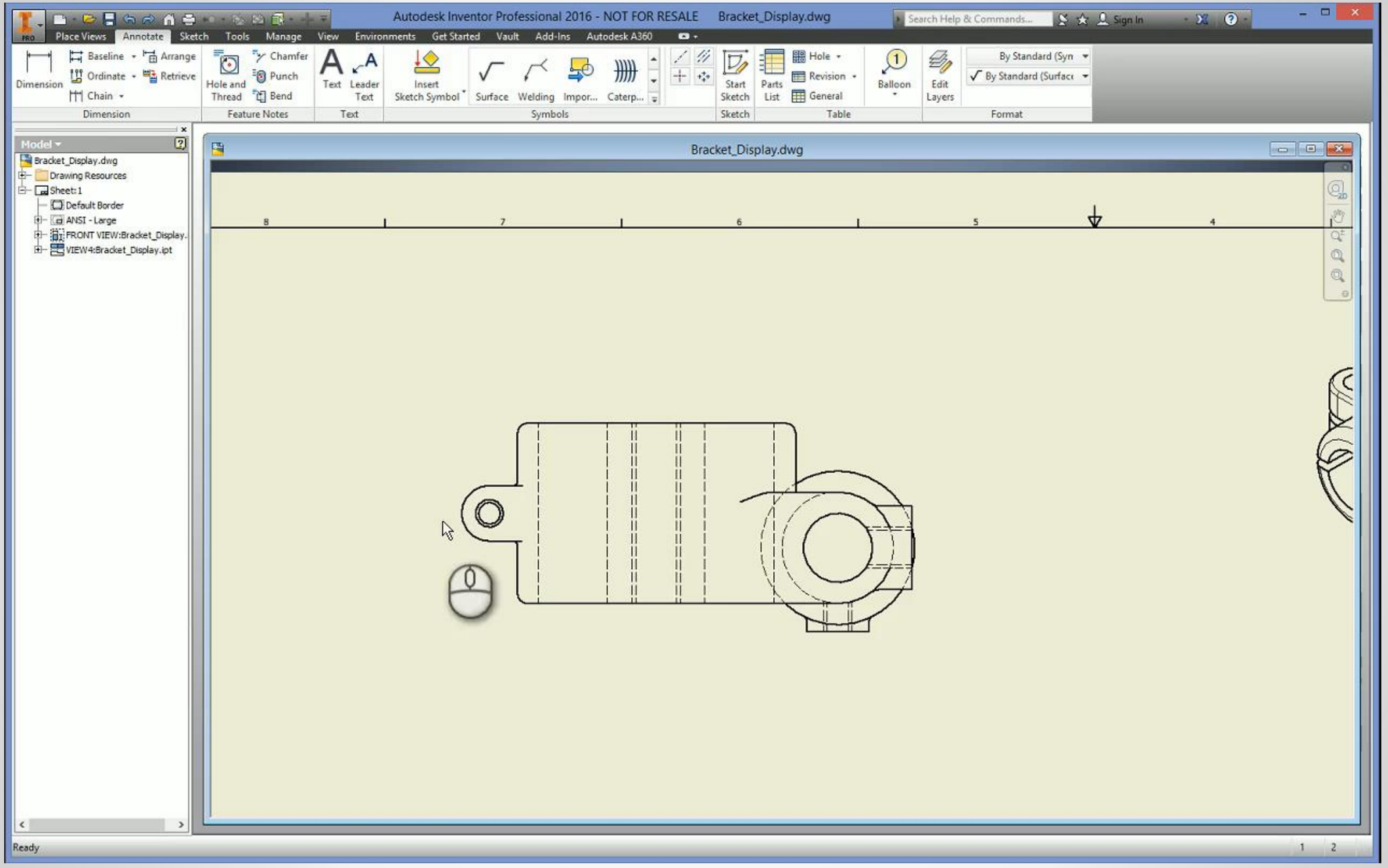
- Use the Tools> Options> Customize
  - Select the Ribbon Tab allowing you to add a User Commands panel to any of the tabs in the Ribbon
  - Select the Keyboard tab allowing you to set command aliases for keyboard shortcuts
  - Select Marking Menu allowing you to customize the RMB Marking Menu or Heads-up Display

- Use the Tools> Application Options General tab to control command prompts
  - Set Dynamic Prompts
  - Show Command alias input dialogue
  - Show Autocomplete for alias commands

Section #4 – Symbols

Symbols

- Standard Symbols
  - Using symbols eliminates any risk of users inserting slightly different variations of the same symbol or one that is just plain wrong per industry or in-house standards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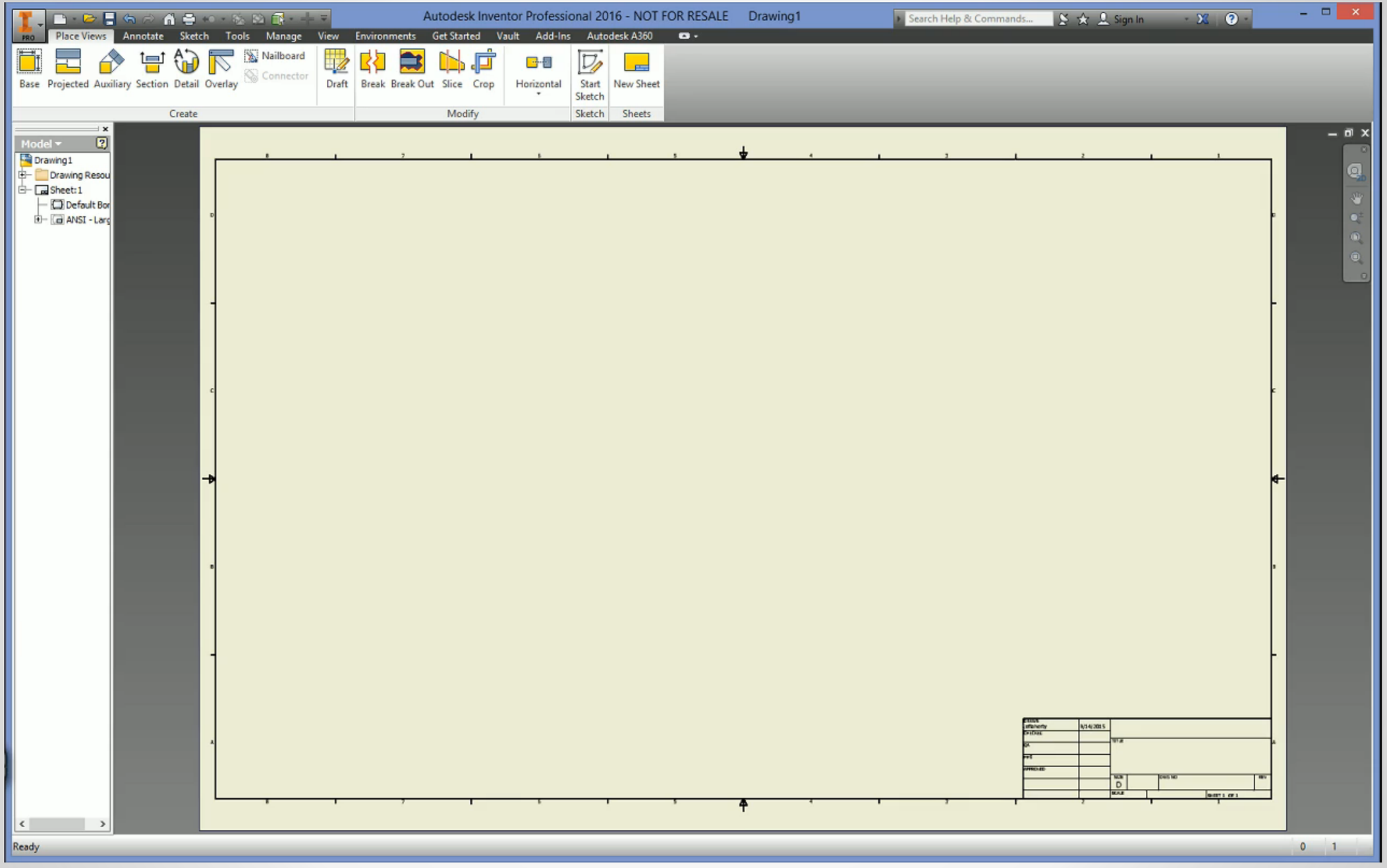
Symbols

- Custom Sketched Symbols
  - You can create a Custom Sketched Symbol per your detailing needs
  - Creating your symbols in a template file will provide consistency and prevents them from being inadvertently deleted and allows access to all users of that template
  - A symbol should be any sketched graphical item you find yourself using over and over again
  - Symbols can include text as well as prompted text
  - Besides Geometry and Text, Symbols can also include:
    - iProperties
    - Imported Bitmap Images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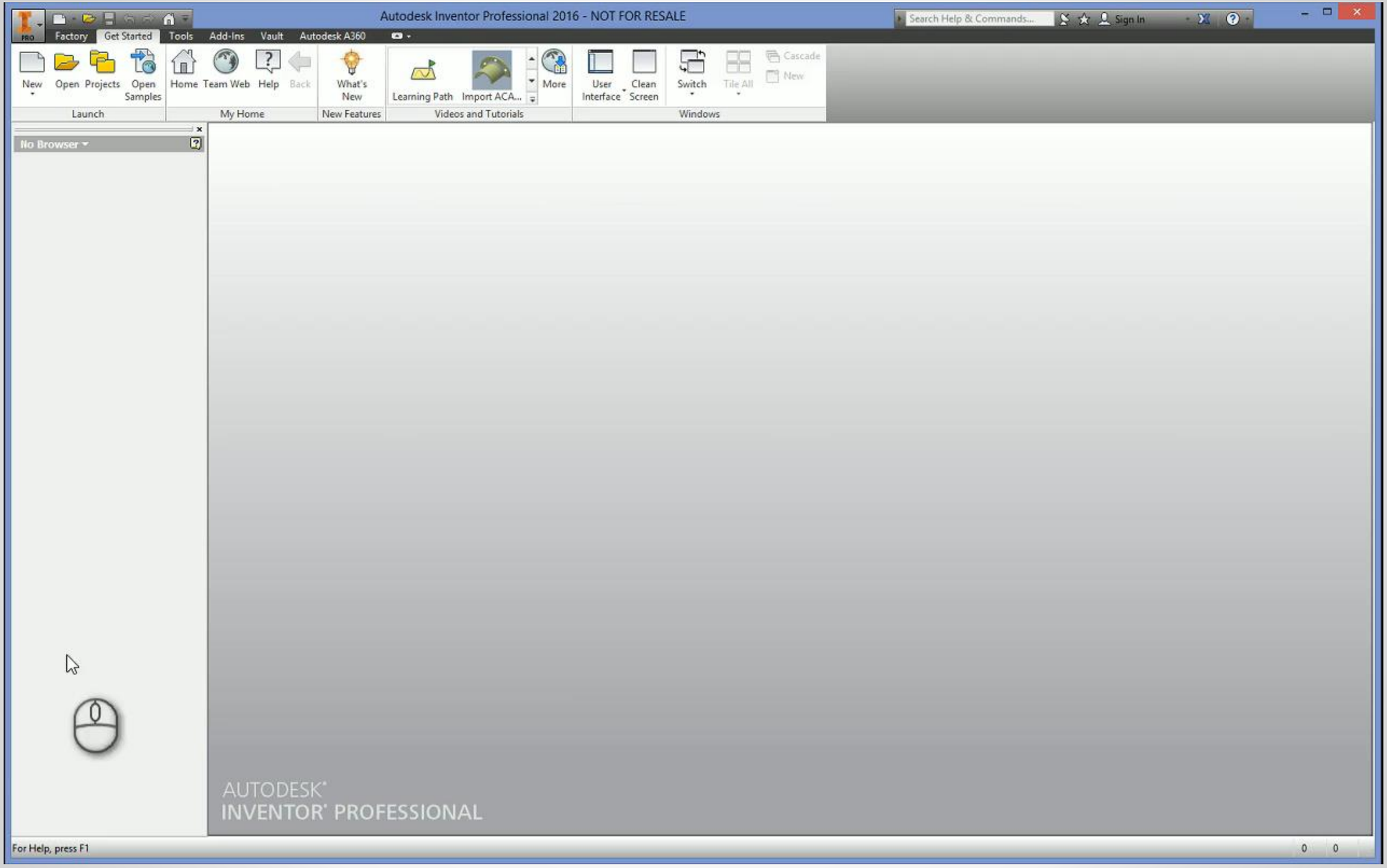
Section #5 – Styles & Standards

Styles & Standards

- Dimensioning – Zeros
  - ASME Y14.5M-1994 says...
    - 1.6.1(a) Where the dimension is less than one millimeter, a zero ***precedes*** the decimal marker *(0,12 mm or 0.12 mm)*
    - 1.6.1(b) Where the dimension is a whole number, ***neither*** the decimal marker nor a zero is shown *(12 not: 12.0 or 12,0)*
    - 1.6.2(a) A zero is ***not used*** before the decimal marker for values of less than one inch *(.12” not: 0.12”)*
    - 1.6.2(b) A dimension is expressed to the ***same number*** of decimal places as its tolerance *(1.23 ± .43 not: 1.23 ± .4 or 1.23 ± .432)*

Styles & Standards

- Set your required dimensioning style in your drawing template
  - Doing so will offer consistency throughout all your drawing files
  - Once set in this manner, you can forget about them until the need comes to add or make a required change
- Different standards can be set for each template
- Correct the Chamfer label text. (We'll cover this in just a bit)
- Educate your users to be aware of the proper way to select your in-house templates
  - Aka – Don't use the "pull-down" menus
  - Let me explain why not…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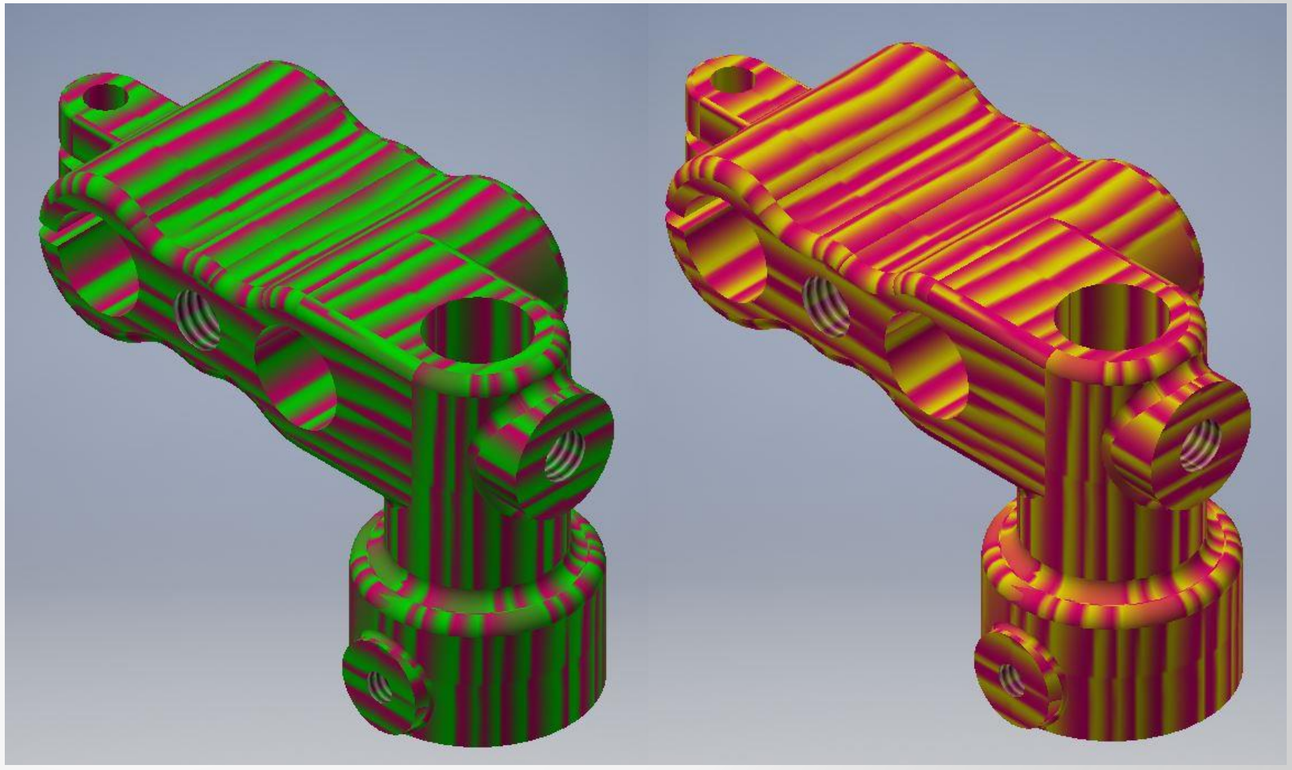
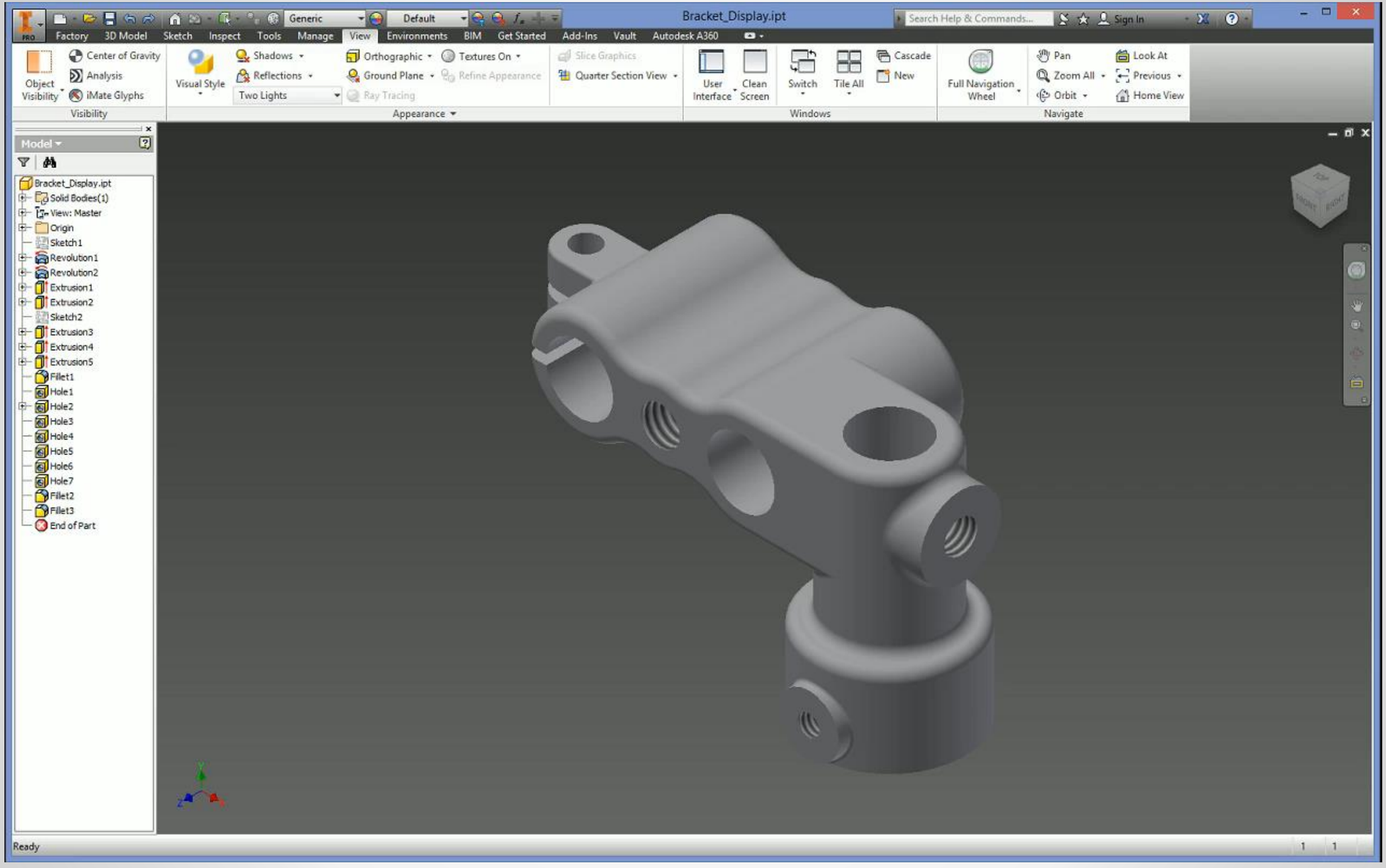
Styles & Standards (Tip)

- Do you have users (or you, yourself) that forget to change the material of the part file from Generic to what it is supposed to be?

- Change the Generic material's Default Appearance to some retina burning color combo

Note: *First step is to make sure you have or use the Visual Style of "Realistic" active. (this "trick" only works in Realistic mode for 2016)*

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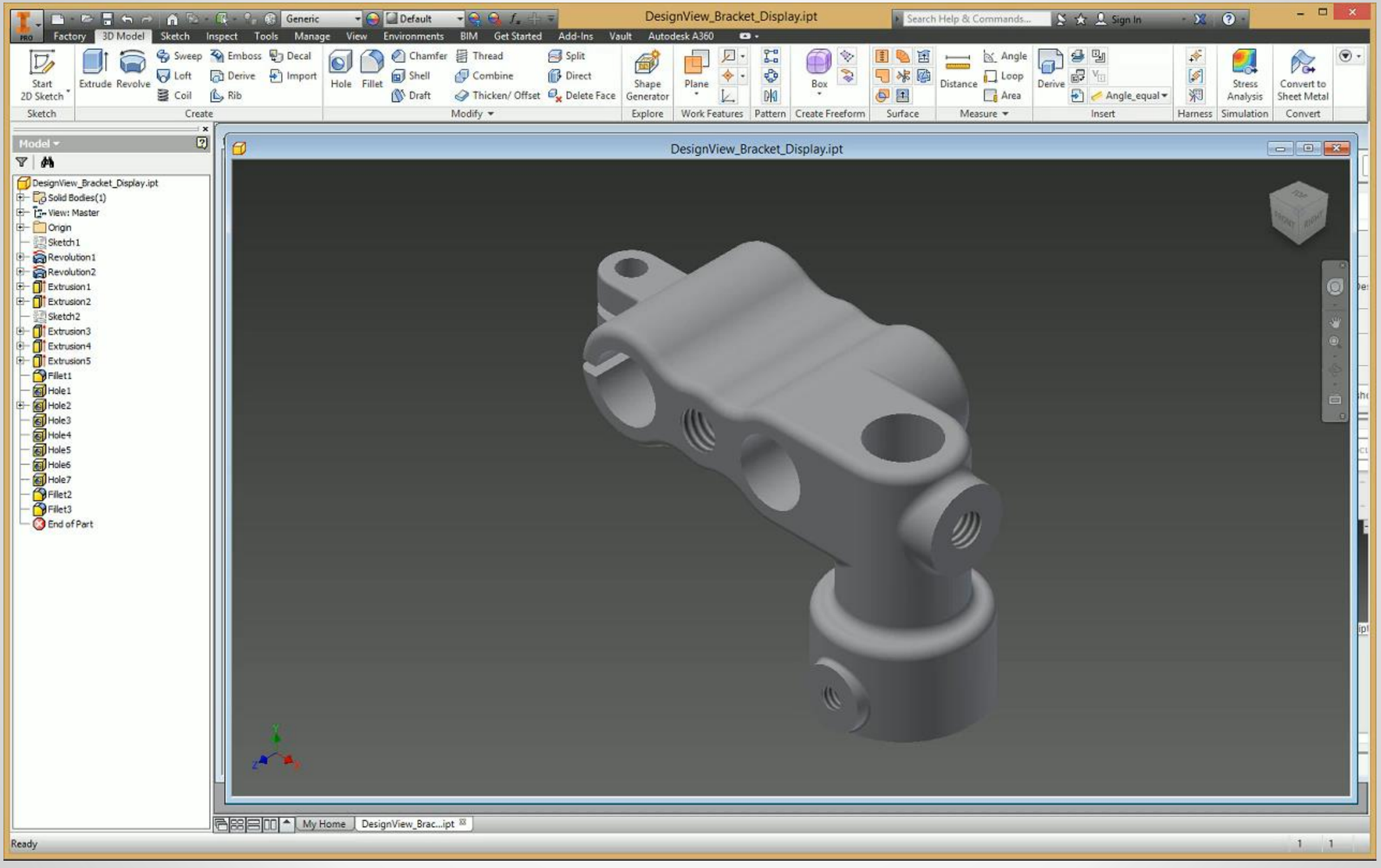
Extras - Administrator Notes

Administrator Notes

- ALWAYS, store the out-of-the-box templates to a safe location and back them up.
  - Copy them to create your own, in-house templates. This way, if and when you make a mistake, you'll always have the originals to fall back on. Trust me on this, I get numerous requests for the original template files because this practice was not followed. Back up your in-house, customized templates as well once you've finished creating and any time you update them
- Verify/set your drawing types to DWG instead of IDW, you'll gain interoperability with AutoCad, Acad Mechanical & Acad LT, Acad Blocks in 2D &3D as well as business systems that utilize the DWG format.
- Use the iProperties fields to help populate your title blocks and Parts Lists
- If you notice your files (part/assembly/drawing, etc.) acting flakey, run them thru the migration process in the Task Scheduler (Demo)
  - Select the options of:
    - Total Rebuild
    - Purge Old Versions
    - Compact Model History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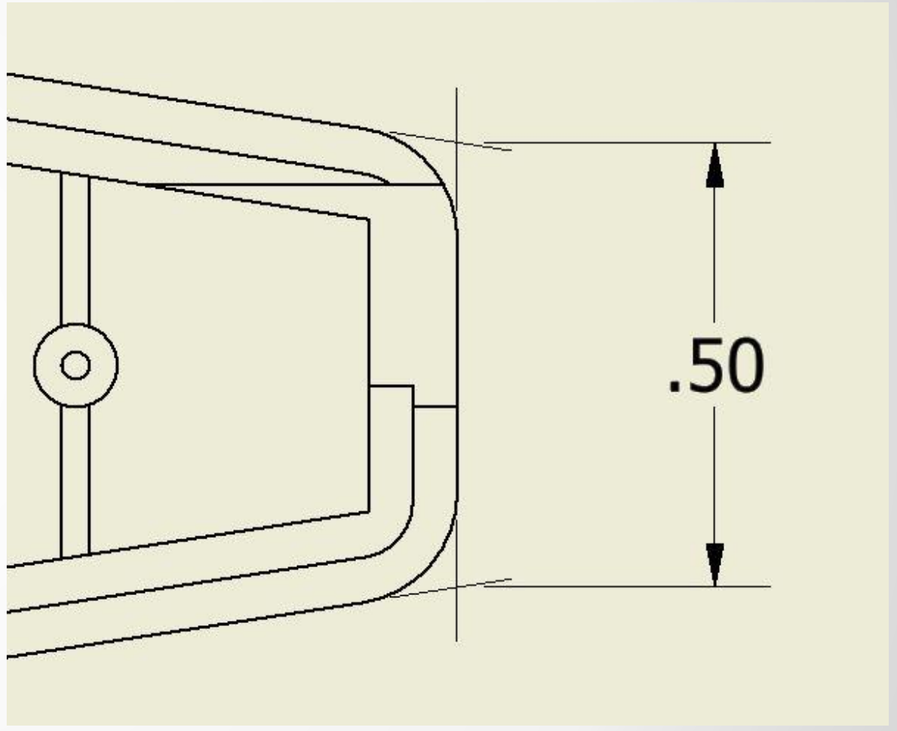
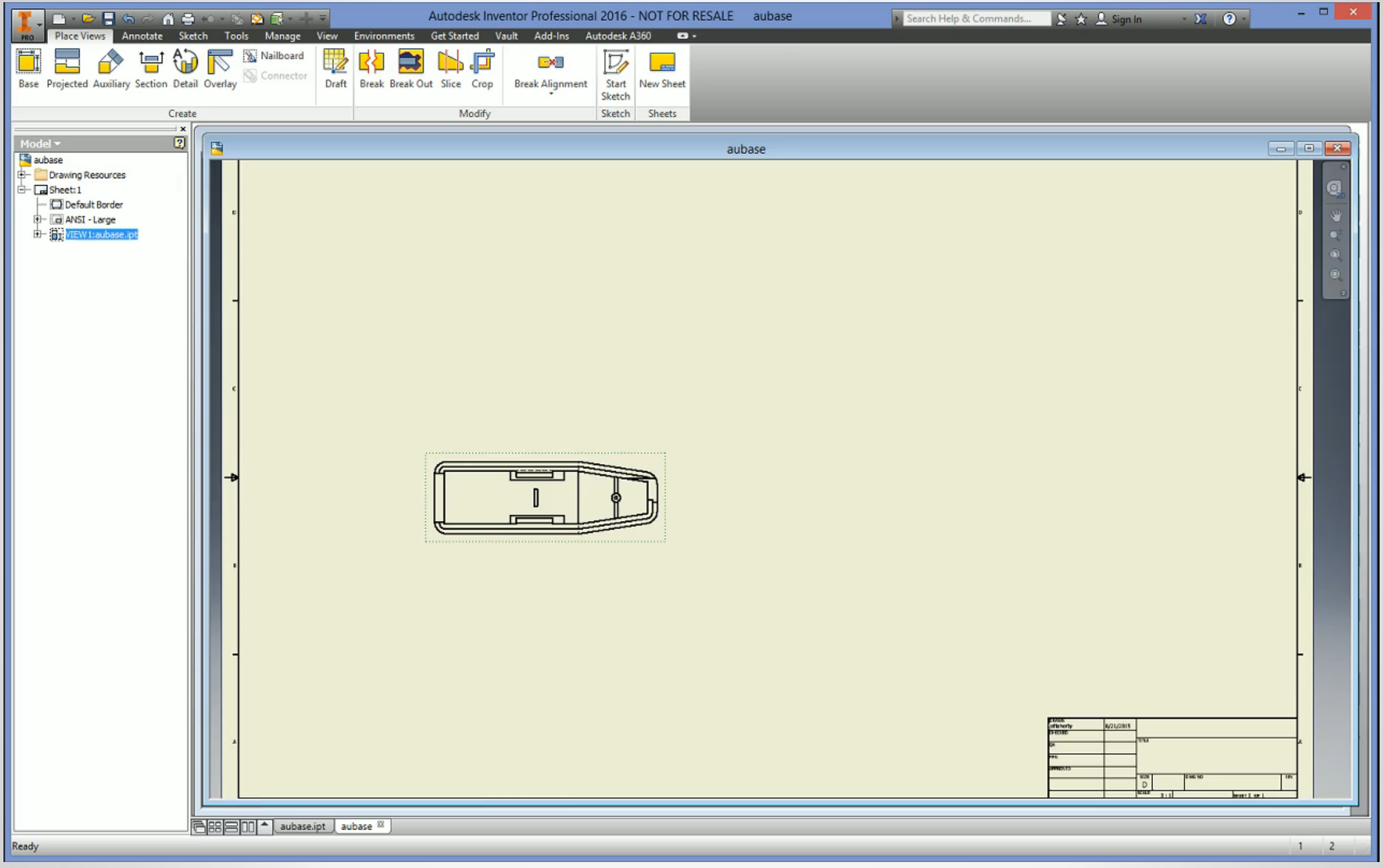
Section #6 – Dimensioning/Text/Notes

Dimensions

- Intersection Dimensions

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Autodesk A360
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Projected
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Overlay
Nailboard
Connector
Draft
Break
Break Out
Slice
Crop
Break Alignment
Start Sketch
New Sheet
Create
Modify
Sketch
Sheets
Model
aubase
Drawing Resources
Sheet:1
Default Border
ANSI - Large
VIEW 1:aubase.ipt
aubase
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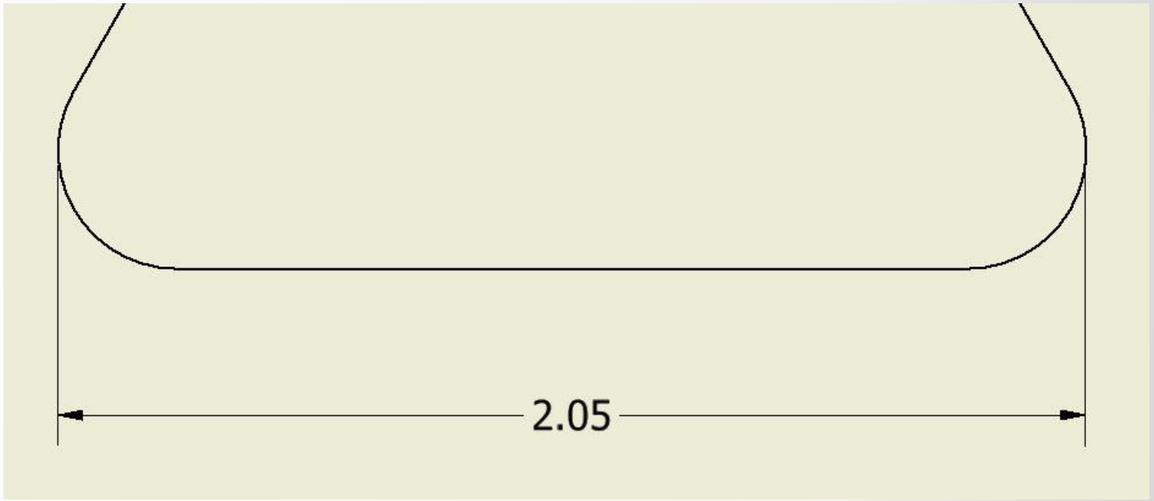
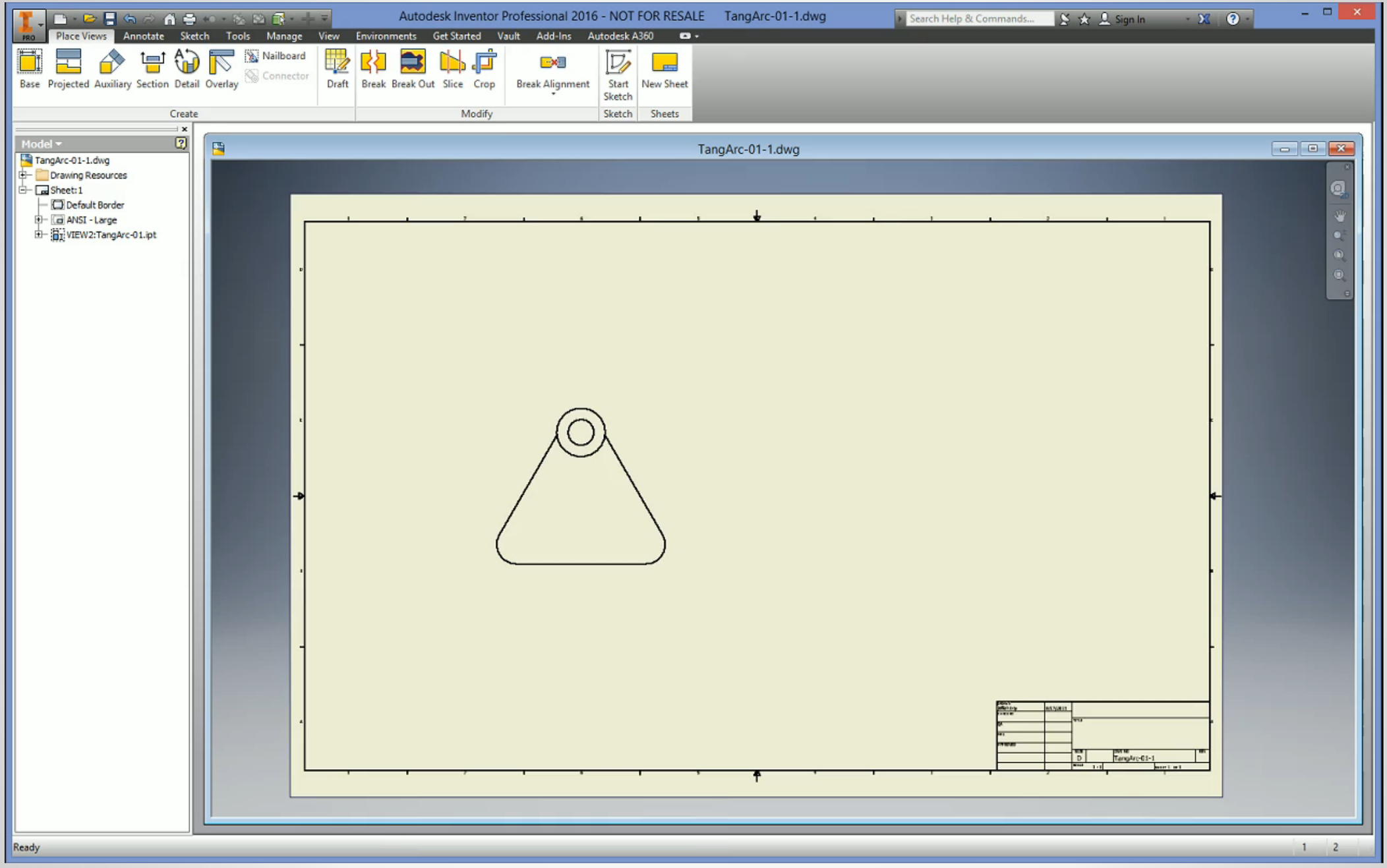
Ready

Dimensions

- Tangent Arc Dimensions




Autodesk Inventor Professional 2016 - NOT FOR RESALE
TangArc-01-1.dwg
Search Help & Commands...
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Drawing Resources
Sheet:1
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ANSI - Large
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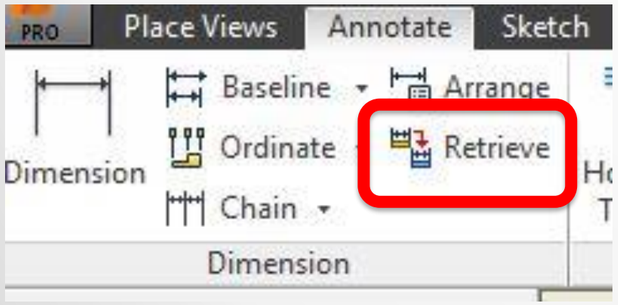
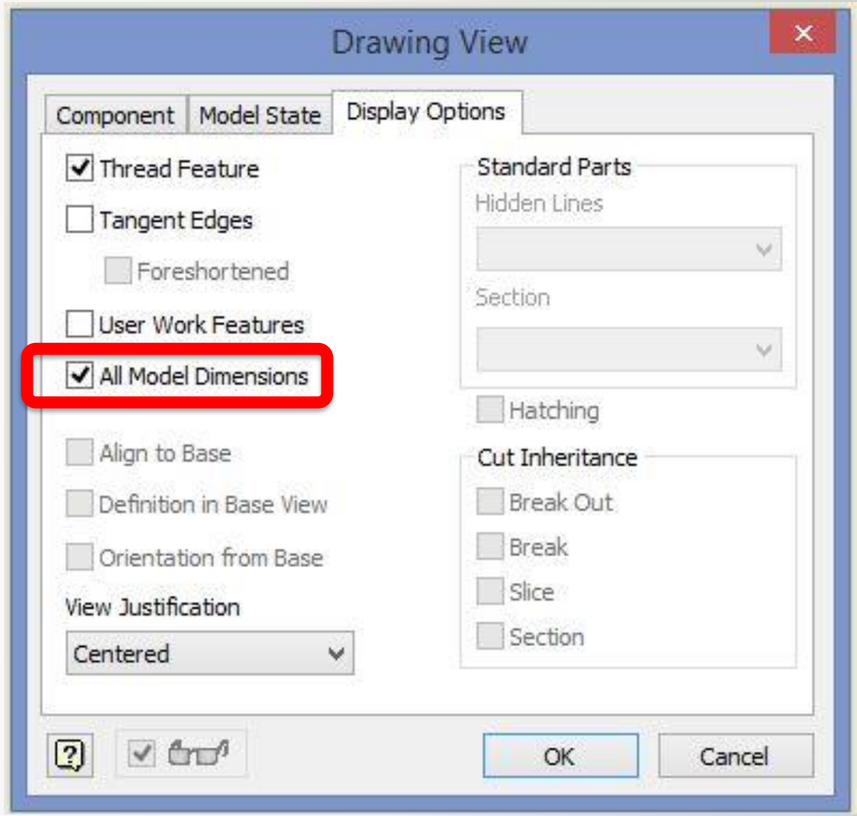
Ready

Dimensions

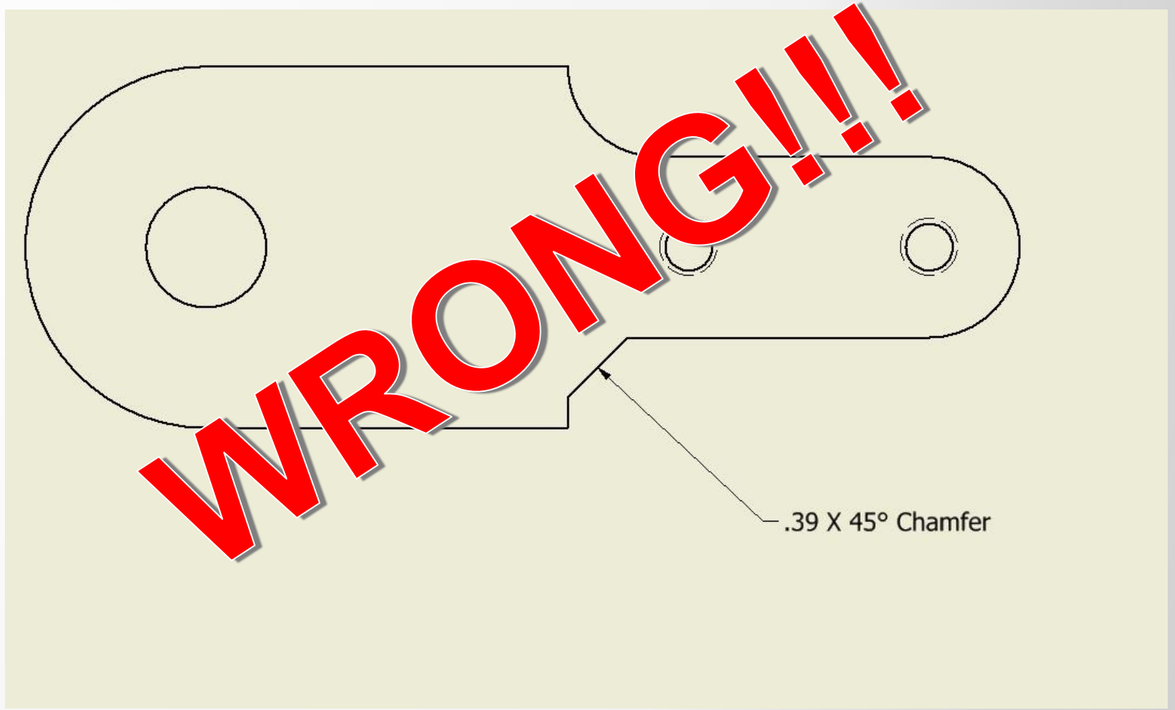
- Model Dimensions
  - @ View Creation on the Display Options tab (Note that this brings in all model dims)
  - After the fact or to bring in specific dimensions:
    - Ribbon Menu
    - RMB Menu

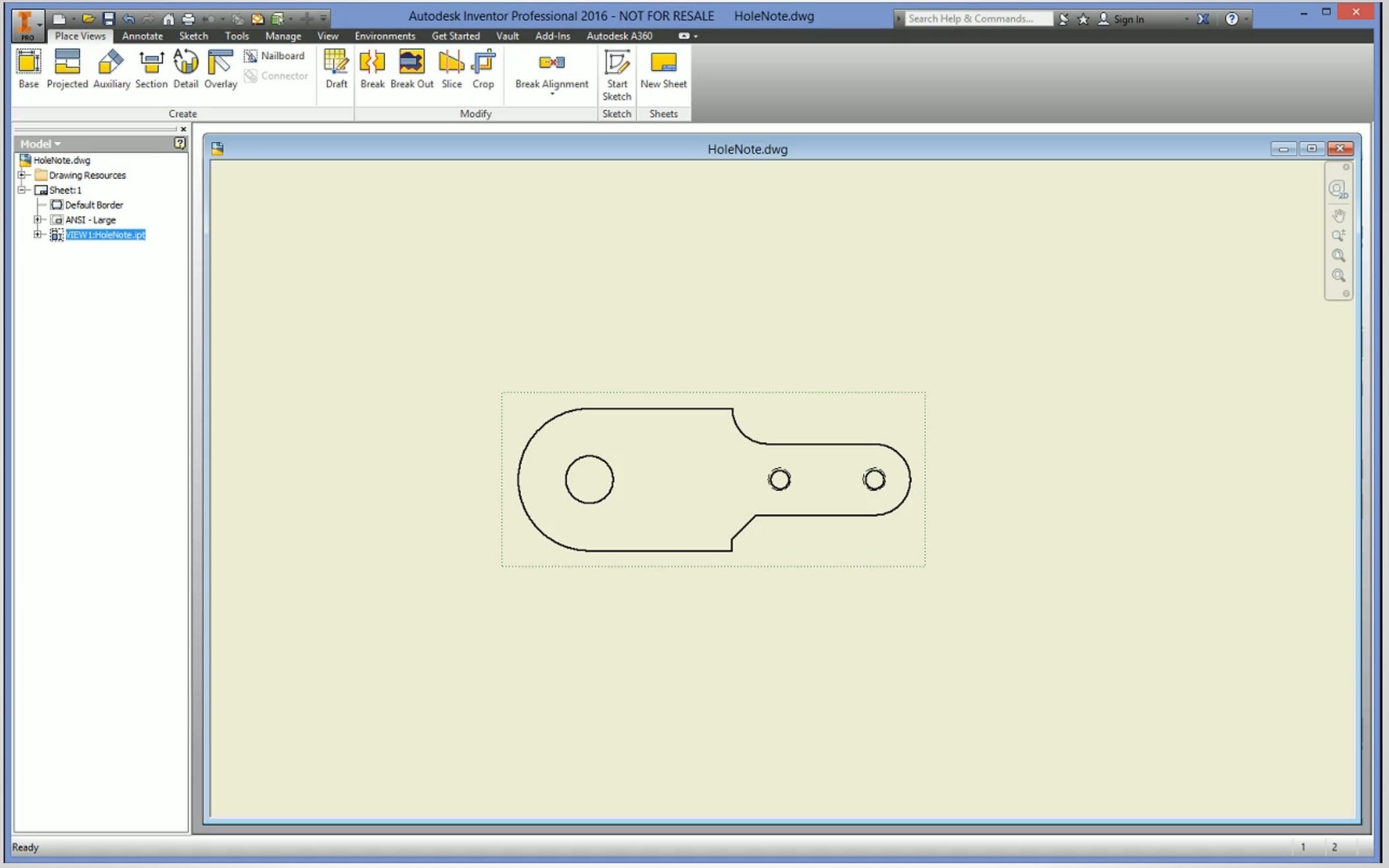




Centerlines, Hole, Thread & Chamfer Notes

- Automatic Centerlines
- Hole & Thread Notes
- Chamfer Note

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HoleNote.dwg
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Sheet:1
Default Border
ANSI - Large
VIEW 1:HoleNote.ipt
HoleNote.dwg
Ready
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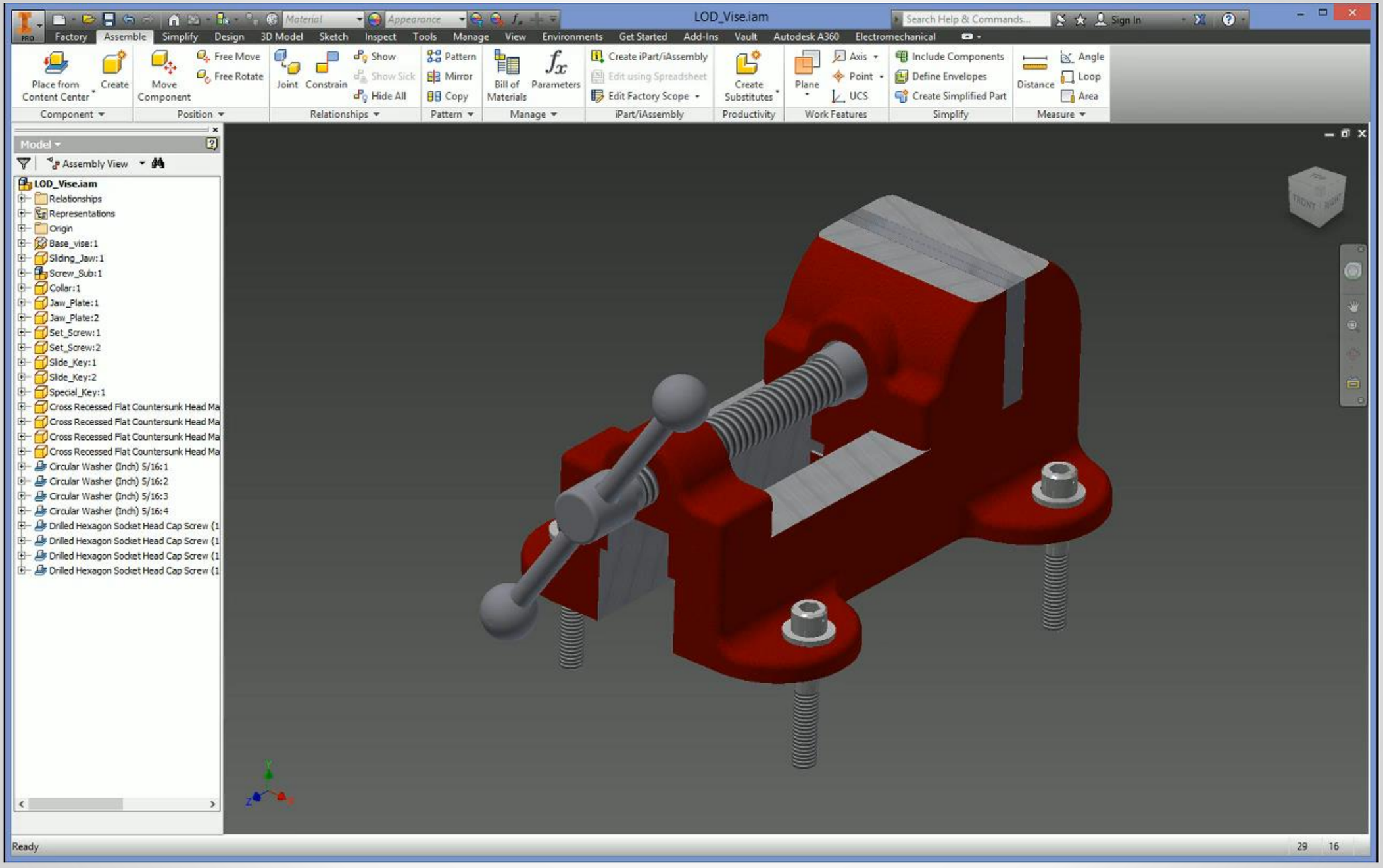
2

Section #7 – Modeling Techniques for Drawings

Modeling for Drawings

- Level of Detail
  - The Level of Detail options can greatly reduce the time and effort needed to create your assembly detail drawings
  - LOD's can show numerous variations of the assembly all in one database
  - Each LOD can be selected upon view creation
  - LOD's can also greatly reduce load time and save a ton of disk space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Assemble
Simplify
Design
3D Model
Sketch
Inspect
Tools
Manage
View
Environments
Get Started
Add-Ins
Vault
Autodesk A360
Electromechanical
Place from Content Center
Create
Move Component
Free Move
Free Rotate
Joint
Constrain
Show
Show Sick
Hide All
Pattern
Mirror
Copy
Bill of Materials
Parameters
Create iPart/iAssembly
Edit using Spreadsheet
Edit Factory Scope
Create Substitutes
Plane
Axis
Point
UCS
Include Components
Define Envelopes
Create Simplified Part
Distance
Angle
Loop
Area
Component
Position
Relationships
Pattern
Manage
iPart/iAssembly
Productivity
Work Features
Simplify
Measure
Model
Assembly View
LOD_Vise.iam
Relationships
Representations
Origin
Base_vise:1
Sliding_Jaw:1
Screw_Sub:1
Collar:1
Jaw_Plate:1
Jaw_Plate:2
Set_Screw:1
Set_Screw:2
Slide_Key:1
Slide_Key:2
Special_Key:1
Cross Recessed Flat Countersunk Head Ma
Cross Recessed Flat Countersunk Head Ma
Cross Recessed Flat Countersunk Head Ma
Cross Recessed Flat Countersunk Head Ma
Circular Washer (Inch) 5/16:1
Circular Washer (Inch) 5/16:2
Circular Washer (Inch) 5/16:3
Circular Washer (Inch) 5/16:4
Drilled Hexagon Socket Head Cap Screw (1
Drilled Hexagon Socket Head Cap Screw (1
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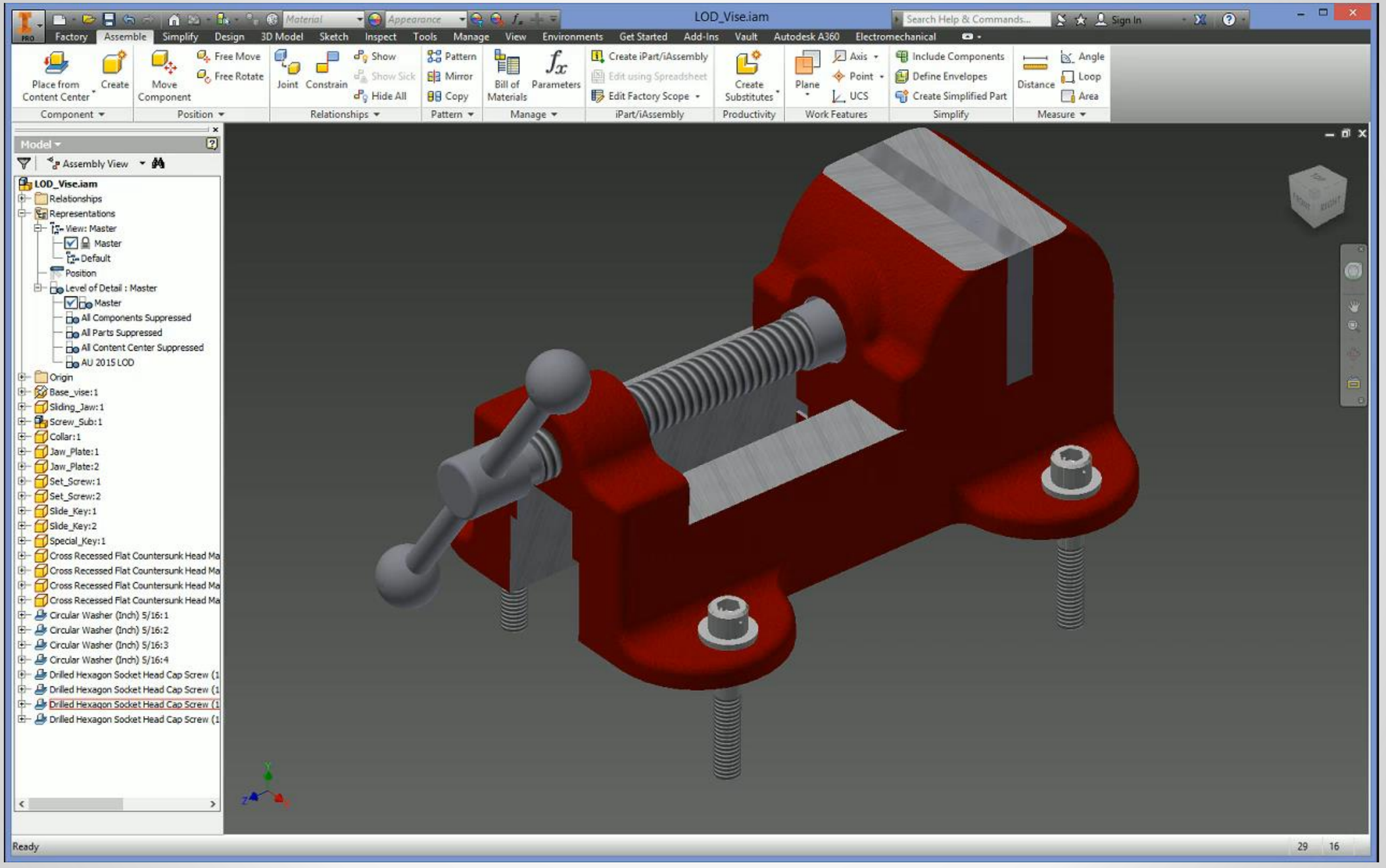
Ready
29
16

Modeling for Drawings

- Design Views
  - Design Views can provide you with a much easier and faster way to detail your assembly files
  - They can be used to show a specific orientation or state of the assembly, color options, etc.

- Hopefully you attended my earlier AU class, CADZilla!!! Where I went into detail on how to tame large and complicated files *(if not, see me after class)*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Add-Ins
Vault
Autodesk A360
Electromechanical
Place from Content Center
Create
Move Component
Free Move
Free Rotate
Joint
Constrain
Show
Show Sick
Hide All
Pattern
Mirror
Copy
Bill of Materials
Parameters
Create iPart/iAssembly
Edit using Spreadsheet
Edit Factory Scope
Create Substitutes
Plane
Axis
Point
UCS
Include Components
Define Envelopes
Create Simplified Part
Distance
Angle
Loop
Area
Component
Position
Relationships
Pattern
Manage
iPart/iAssembly
Productivity
Work Features
Simplify
Measure
Model
Assembly View
LOD_Vise.iam
Relationships
Representations
View: Master
Master
Default
Position
Level of Detail : Master
Master
All Components Suppressed
All Parts Suppressed
All Content Center Suppressed
AU 2015 LOD
Origin
Base_vise:1
Sliding_Jaw:1
Screw_Sub:1
Collar:1
Jaw_Plate:1
Jaw_Plate:2
Set_Screw:1
Set_Screw:2
Slide_Key:1
Slide_Key:2
Special_Key:1
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Ready
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Conclusion

- I can’t stress it enough, standardize and automate as many items as you can, especially when it comes to your in-house templates. Setting the standards in your templates makes sure everyone’s following the same “rules” and at the same time, the more commonly forgotten items are automated. Setting your templates up to populate specific fields with iProperties from part and assembly files either eliminates or minimizes user error.
- Spread the knowledge to your co-workers that weren’t able to attend this class. Getting everyone on board with the processes you use, will eliminate confusion and will increase productivity.
- Drive home the idea of not only using but adhering to standards, whether it be ANSI, ISO, etc. Most in-house standards are based on one of these industry standards. Following a recognized industry standard eliminates confusion, minimizes cost and scrap as well as protecting you and your company on legal issues if it comes to that. You might get some pushback, mainly from those that simply may not understand the benefits, explain it to them and hold your ground. *The sum is greater than the parts.*

Conclusion

- ***Questions?***

Thank You for coming!

- Be sure to fill out the class survey
  - *Please be kind as this is my first crack at this and I'd like to be back*
- Come up and get my business card & drop yours off

More Questions? Visit the AU Answer Bar

- Seek answers to all of your technical product questions by visiting the **Answer Bar**.
- Open daily 8am-10am and Noon-6pm and located just outside of Hall C on Level 2.
- Staffed by Autodesk developers, QA, & support engineers ready to help you through your most challenging technical questions.

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