

Get Your Welders and Designers Connected

(20-Plus Tips on Using Welds in Inventor)

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Learning Objectives

- Why should you model your welds?
- Basics of the Weld Environment
- Tips for modelling welds - even the **tricky** ones
- Deliver your weld data to the production floor

Description

Does your manufacturing team consistently know where your design should have welds placed in your weldments? Does your product get welded differently each time? Does your welder have issues interpreting your weld symbols? Why not add welds into your models to alleviate these issues? If you have Inventor software, you have the ability to add welds into your assemblies. Get all the tips and tricks from Unverferth Manufacturing and learn how the firm has added welds into its Inventor models to show exactly where the weld beads should be placed by manufacturing. No more guessing and incorrect interpretation when it comes to welding parts together. Communicate design intent more precisely with the manufacturing team using fewer words and Inventor software's weldment environment.

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Notes:

- Any time you see a  . This indicates a wish on the Inventor Ideas portal, so please follow the links in the Appendix and give yourself, and us a Vote to help get these wishes in the product.
- If accessing this document through the AU 2019 Mobile App or website, be sure to check for newer versions.
- The YouTube links below will not go live until the morning of Tuesday Nov. 19th 2019.

Speakers

Mike Hilvers [\(\[mikeh@unverferth.com\]\(mailto:mikeh@unverferth.com\)\)](mailto:mikeh@unverferth.com)



Design Engineer / CAD Mgr. / Standards Mgr. for an Agricultural / Farm Equipment Manufacturer for the past 24 years. We started with AutoCAD, grew up with Mechanical Desktop taking it thru its full lifespan. The latest evolution, in conjunction with Autodesk, transitioned us to grabbing hold of Inventor and Vault in their infancy before they went public, and maturing into the processes we currently operate under. I'm a Graduate of The Ohio State University with B.S. in Agricultural Engineering. On the side I'm a very fortunate dad and husband. On the side, which is also my hobby, is being the 5th Generation to Farm the ground we live on with my wife and kids, making them the 6th generation to farm the land that dates back to the Presidency of Abe Lincoln.

Steve Hilvers [\(\[steveh@unverferth.com\]\(mailto:steveh@unverferth.com\)\)](mailto:steveh@unverferth.com)

Over 30+ years of machinery design. I cut my teeth on a drafting board way back in the day, then 2D AutoCAD, and started 3D parametric with Mechanical Desktop, Inventor, and Vault Professional. Involved with every aspect of a products life from writing rules for product configurator to shipping the final product. When I'm not working at Unverferth Mfg., my friends around home have put their trust in me to be a Township Trustee. So, I help maintain roads, plow snow, and help manage the Township's business. In what time I have left, I hone my woodworking skills, build and tinker with about anything and be a dad.



If your company uses weld symbols on a 2D flat print, has welders in the shop or contracted, you more than likely have had the welder stop welding to decipher a print. That person then gets the design team involved to ask the intention of the weld call outs. If they are pausing to decipher the intent of the print, they're probably not welding your assembly as intended, and your company is sending inconsistent products out the door with a higher cost of manufacturing.

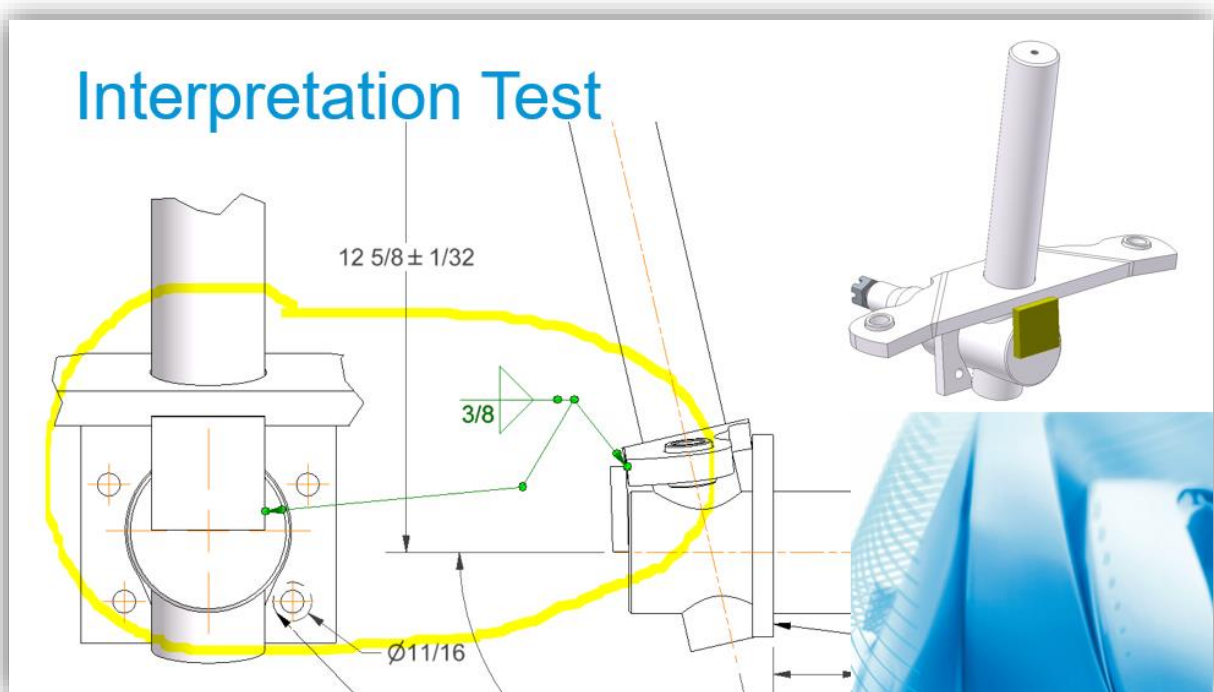
If the welders are asking the design group questions, which most organizations hope they are.... the welder and designer are not performing their job as efficiently as they could have been. They are there to weld and design products. Sometimes the designer even has to review the intent of their own weld symbols that they created. In essence, they are the only interpreter of the welds symbols on a 2D print.

Objective 1: Why should you model your welds?

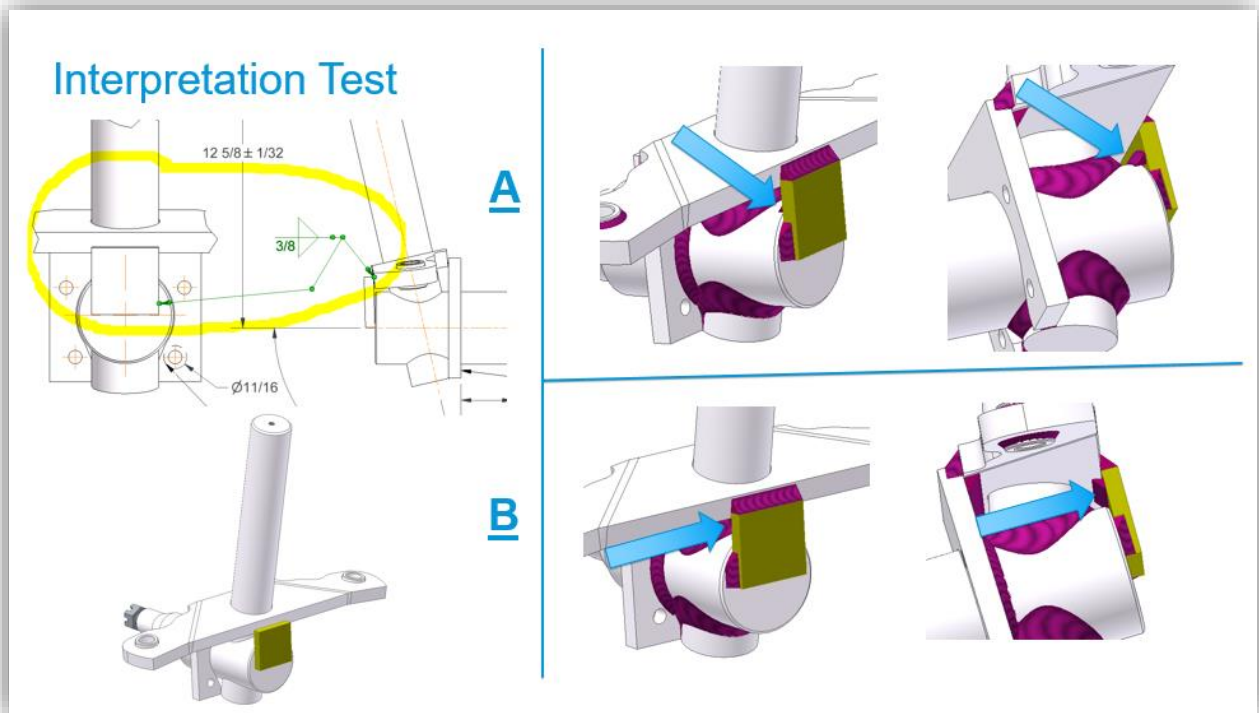
- Effectively communicate welds to the shop floor with no potential of misinterpretation
- Get your Welders and Designers connected without speaking a word
- Have workers doing what they're hired to Do – Weld & Design

Let's take a test?

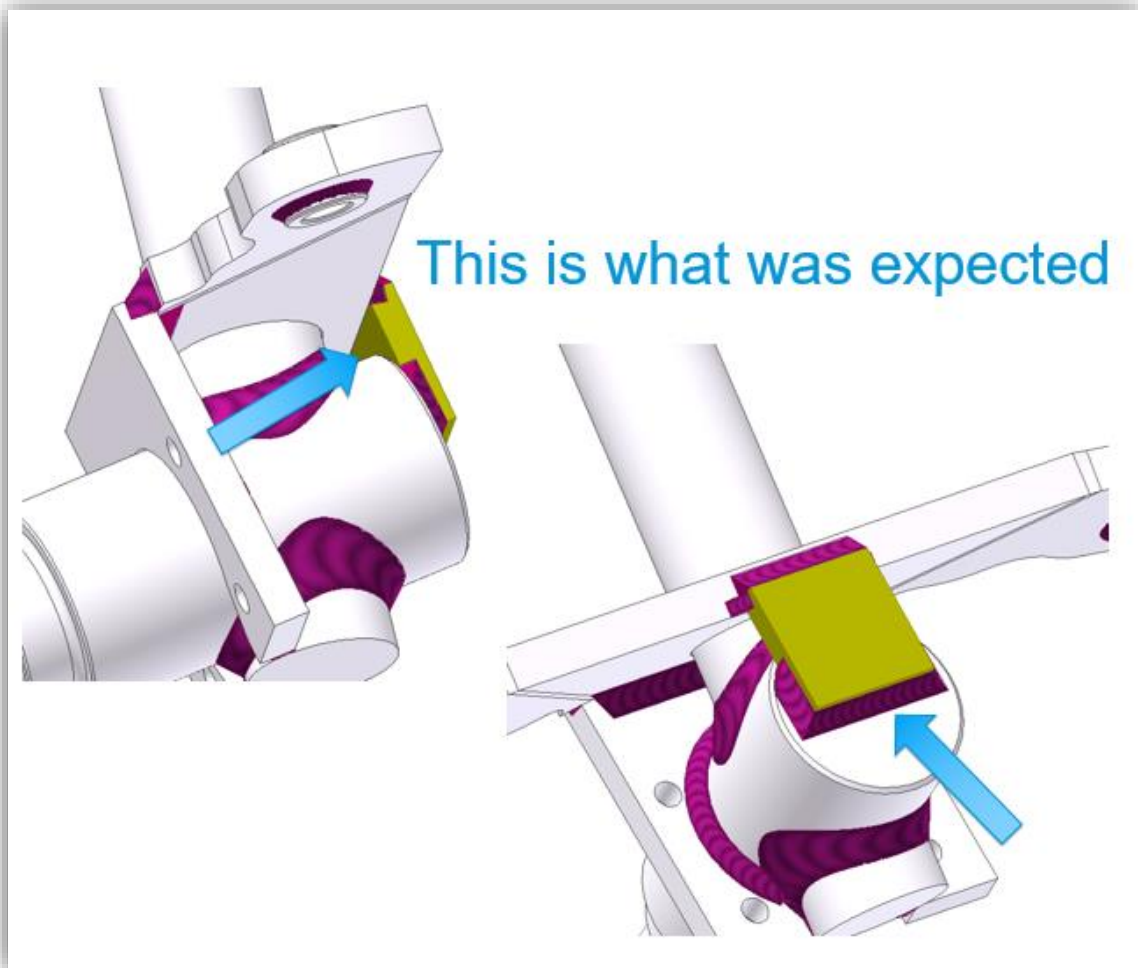
What is your interpretation of the weld highlighted on this print?



Do you Guess A or B?



You're wrong – This is what was intended.



Our interpretation of that weld symbol is that the Designer who created the weld symbol was wrong. But with the 3D modeled weld, all interpretation is gone.

What's in it for you and your Company?

Really it's quite simple and straight forward. If we can effectively communicate welds to the shop floor we'll spend less time interpreting and communicating welds which gains us:

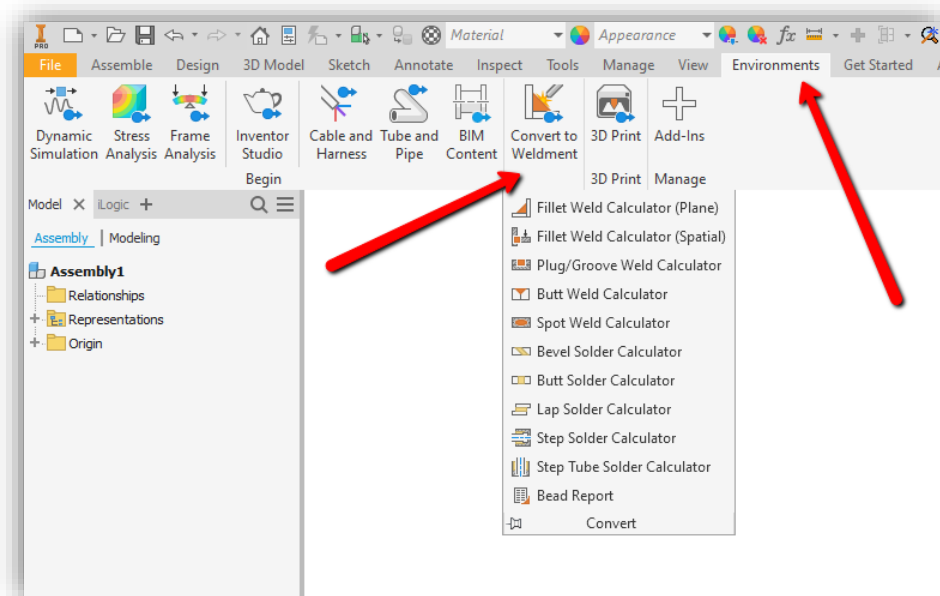
- More Time Welding
- More Time Designing
- Consistent products out the door

Objective 2: Basics of the Weld Environment

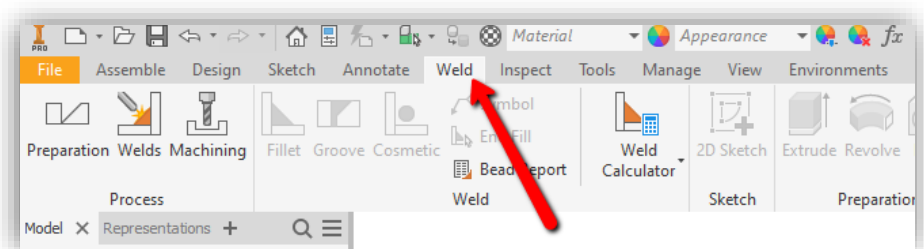
Enter the Weld Environment

https://youtu.be/c7Q_1lm4XrA

To model welds and create a weldment the file must be an IAM. Like most inventor Environments you click the “Environments” ribbon and then select the “Convert to Weldment” icon.

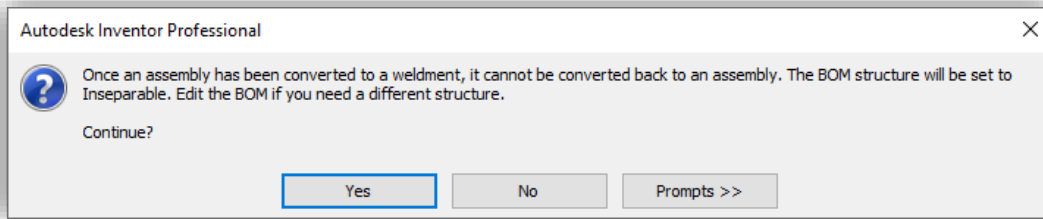


Once you enter the weld environment for a given file a “Welds” Ribbon is created. You do NOT have to use the “Environment Ribbon” again for that file.



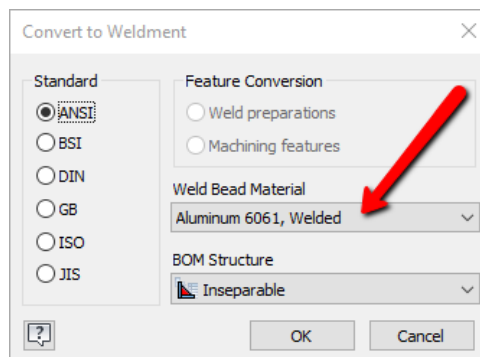
When you enter the weld environment you’ll be prompted with 2 windows:

1st one tells you that once a file is converted to a weldment you cannot go back to a regular iam.



- The only negative that we've found is that you can NOT have Positional Representations with a weldment. So if you must have a POS REP in an IAM, then the weld environment is not for you
- If you create a POS REP in an IAM that is already converted to a weldment, you will not be able to get back in the weld environment until you delete the POS REPs.
- If you can't click the "Convert to Weldment" because it's greyed out – that most likely means you have a POS REP in the IAM.

2nd is the one where you pick a Standard, Bead Material and BOM structure or BOM occurrence.



- Note the material does affect the Physical Properties like Weight.
- We almost always set the BOM Structure as Normal. The only time it really matters is if you set it to Purchased, Phantom or Reference as the components that go into this weldment will NOT be included in the Bill of Material (BOM). This acts just like a non Weldment IAM if you go under Tools>Document Settings>Bill of Materials Tab.

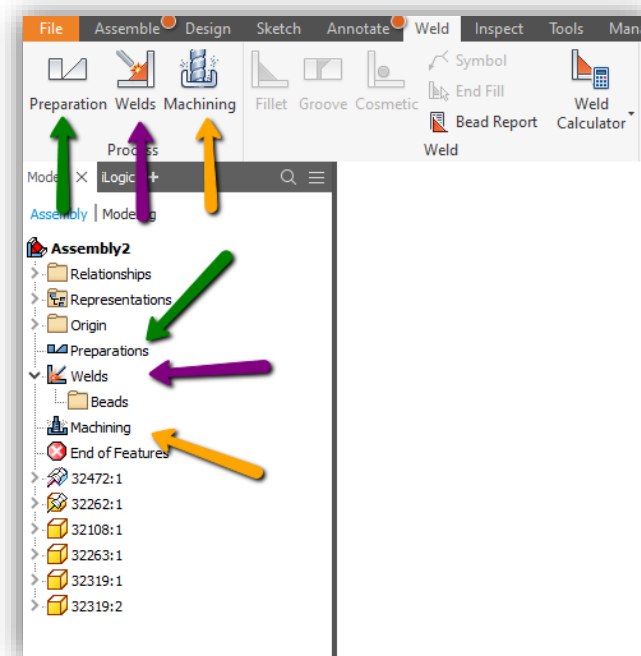
Weld Templates:

https://youtu.be/c7Q_1Im4XrA?t=296

Weld.iam templates are included with the install of Inventor which can be preset with the Material and BOM Structure as shown above. We don't use them because when we start a new project we typically design top down or middle out and we never know where the weldment level will be until the design matures. This is a wish I have for the Weld Environment to be able to preset Material and the BOM Structure within a standard IAM.

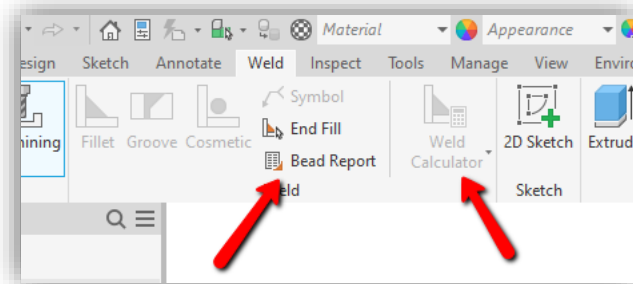
Weld Processes:

https://youtu.be/c7Q_IIm4XrA?t=600



- **Preparation** – Typically are thought of as features that are needed to happen at this level of the weldment such as grind a chamfer on some edges before welding to get good weld penetration. We don't use preparations in this capacity as we manufacture all sub parts to final spec, so when a part comes into the weldment it is ready to weld.
 - But we do use preparations to help us model welds. For instance if we need to do a plug or butt weld, we'll cut a groove in the parts at the iam level and then do a groove weld to fill it. We'll discuss this later.
- **Welds** – Obviously the Weld tab is where we create the weld beads and symbols.
- **Machining** - is for post welding processes or machining. For example if we create an axle and weld a plate to the end of a rod and we want the 2 parts to be perfectly square we'll weld and the face off the hub/plate to square them up. Same thing can be done with threads. Sometimes we need to machine to eliminate any warping from welding to get a true face. As of 2020 you can also add internal and external threads to a weldment/iam.

Design Tools



- **Bead report**

https://youtu.be/c7Q_IIm4XrA?t=498

Generates an Excel sheet of all the beads in a weldment that can be exported. It's convenient to determine how much weld is in a model and can be used to order weld wire or put a cost on the weld process. Note these are not linked and will not updated once exported. You'll have to Export any time revisions take place.

- You can also check the Include All Subassemblies so you don't have to run a report for each level to get a top level weldment bead report.

- **Weld Calculator**

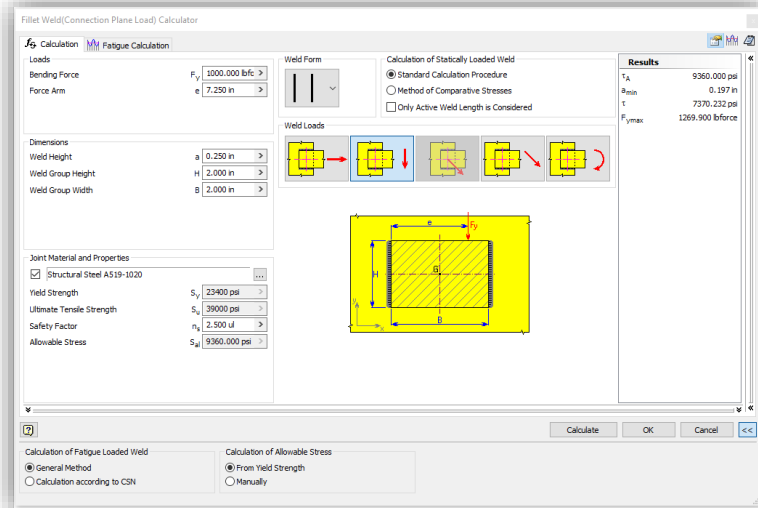
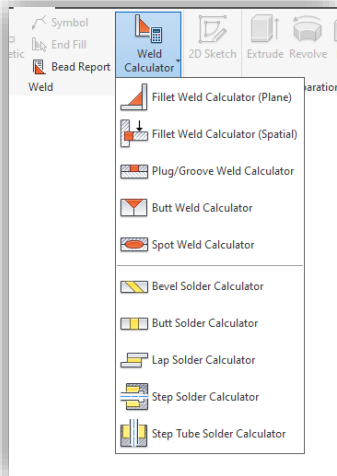
https://youtu.be/c7Q_IIm4XrA?t=393

Is a nice tool help determine how much weld is required on a design. It's more for the simplistic welds and calculating 1 weld at a time. It's a design tool and would be quite valuable for a person new to weld design.

- If the Weld Calculator is not available you are in one of the Processes like Preparations, Welds and Machining and you need to hit the Return Button at the right side of the Weld Ribbon to get out of the process.



- There are a lot of options and Calculators included that is more than we can cover in this class.



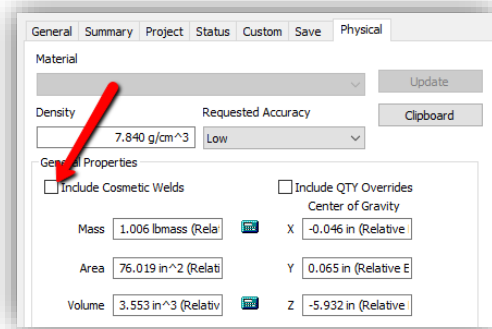
Objective 3: Tips for modeling welds:

Even the tricky ones

Because Video are so much more dense with information and thus more efficient to learn new material in a shorter period of time we've created a series of short videos that will be on YouTube channel: "AU2019 & UM" So here we go:

- **Fillet Welds** – File (*B - UT-2205.iam*)
<https://youtu.be/xHTEaZxy4vA?t=9>
 - Must manually switch selection sets
 - **Ctrl** Pick to De-Select
 - Chain Edges for auto select of tangent edge
 - Selection order can change outcome
 - Weld Contour or caps
 - In Accurate Leg length 🤖
- **Multiple Partial welds in 1 weld feature** - File (*C - 7778.iam*)
<https://youtu.be/xHTEaZxy4vA?t=448>
 - Inventor treats the weld as 1 continuous welds when you make multiple selections.
 - Must play with the Pitch to get the weld centered on each plate.
- **Groove Welds** - File (*D - Groove Welds.iam*)
<https://youtu.be/xHTEaZxy4vA?t=655>
 - Face 1 which will be the profile
 - Face 2 is ending surfaces
 - Pick a direction to extrude along or Radial for Round to Round.
- **Cosmetic Welds** - File (*E - 78431.iam*)
<https://youtu.be/xHTEaZxy4vA?t=861>
 - Pick edge where want Cosmetic Weld
 - Edge will highlight – will not be an actual weld bead
 - Do enter the cross section of the weld as that will influence Mass and the Physical properties

- You must check iProperties Physical tab box to include the cosmetic welds.



- **Preparations – Plug Welds** - File (E - 78431.iam)
<https://youtu.be/xHTEaZxy4vA?t=947>
 - Can use to prepare parts before welding.
 - We use preparations to create holes for plug welds or spot welds
- **Model a weld as an IPT** - File (F-1234.iam)
<https://youtu.be/xHTEaZxy4vA?t=1044>
 - For a Butt Weld with Crown – File (F-1234.iam)
 - For welding rounds to rounds - File (E-78431.iam)
 - You may not be able to Model all welds accurately so we let the 3D model show where and the 3D Weld Symbols and Annotations to give the details on the size.
- **Wrap Welds around end of parts** - File (G-4444.iam)
<https://youtu.be/xHTEaZxy4vA?t=1160>
 - When making your selection sets for a fillet weld you can pick the ends of weld beads/features
- **View Representations on the Model** - File (G-4444.iam & G-4444.idw)
<https://youtu.be/RRZwOgG5zrw>
 - **Default** – Is the Final Machine Color for consuming the next level up.
 - Show welds
 - Do NOT show 3D annotation or symbols
 - **Welds** – Show Welds and Annotations
 - Use Annotation Contrast colors to really make the welds Pop.
 - We use 2 custom colors w/ parts white and welds Pink
 - This combination prints well in black and white.
 - **No Welds**
 - No Welds and No Annotations
 - Used on IDW for dimensions – less clutter.
 - Can save Video crunching power also on large assemblies.

- **IDW View Representations** - File (G-4444.iam & G-4444.idw)
<https://youtu.be/RRZwOgG5zrw?t=147>
 - Edit view and Model State Tab must be used to turn welds and preparations on.
 - Can be set in the IDW template.
 - We don't use the Model State tab to control individual weld features.
 - Since we're creating View Reps for clarity on the IDW we also use them to control Weld feature visibility.
 - We try to show the welds, symbols and annotations on an ISO view(s) using the Annotation contrast colors or the View Rep "Welds". We also shade those views to get the welds to "POP" on the print.
 - We use the View Rep "No Welds" on the ortho views to make dimensioning clearer on the prints by eliminating the weld clutter.
- **Model Weld Symbols/Annotation** - File (G-4444.iam)
<https://youtu.be/RRZwOgG5zrw?t=313>
 - Symbols can be linked to the Weld but **MUST** be created at time of weld.
 - Linked Weld Symbol will only populate Decimal Value 🤖
 - So we don't link them. We create but do not link.
 - When editing beads you must edit both the weld and the Annotation.
 - If can't model accurately – model what fits and use Weld Symbol to call out size.
 - Use annotation to call out NO WELD BEYOND THIS FACE.....
- **IDW Weld Symbols/Annotation** - File (G-4444.idw)
<https://youtu.be/RRZwOgG5zrw?t=565>
 - Right click on view and Get Weld Symbols
 - If you chose not to use the "Get" feature and manually annotate the welds
 - We recommend to still annotate the 3D model
 - For the future when 3D annotation becomes the norm.
 - For reference when manually applying the symbols.
 - Can quickly flip to the 3D model and reference what is already done.
- **Stages of Weld currently can't be done** - File (H - 32267.IDW) 🤖
 - Weld these parts and then add this part and weld.....
 - Welds currently are all or nothing - you can't control visibility of each weld
- **Find in Browser of individual of Beads** - File (H - 32267.iam)
https://youtu.be/PKnZXeJv_7c?t=64
 - Use selection filter of "Select Feature"
 - Typically you will have to hover or select other
 - It is very sensitive
 - When in Browser & Weld Features where you pick can yield different results.
 - Welds act like 1 part file or 1 feature in most scenarios.

- **Stitch welds on a Round** - File ((I -87264.iam)
https://youtu.be/PKnZXeJv_7c?t=126 
 - Start offset must be done from Preparation Geometry or a split face.
 - Create a prep slot or groove and becomes start point of weld.
 - Slot or groove must be longer than the leg length of weld.
 - “From – To” does NOT work 
 - Requires perpendicular faces that intersect the weld path - no planes

General Tid Bits on the Weld Environment:

- You can **NOT** pattern or array Welds 
- You can **NOT** Control Visibility of individual welds – All or Nothing 
- You can suppress individual welds, but can **NOT** tie suppression to a View Rep. 
- Funky welds may have to be done with combination of welds.
- Don't expect 100% accurate welds – Give visual of where weld should be – let 3D symbol specify the size
- Welds do Bog down Models
- Save welds until Design is Mature
- Suppress welds when tweaking design, constrains or replacing parts
- If you have weld errors - Turn welds on 1 at a time to fix if necessary.

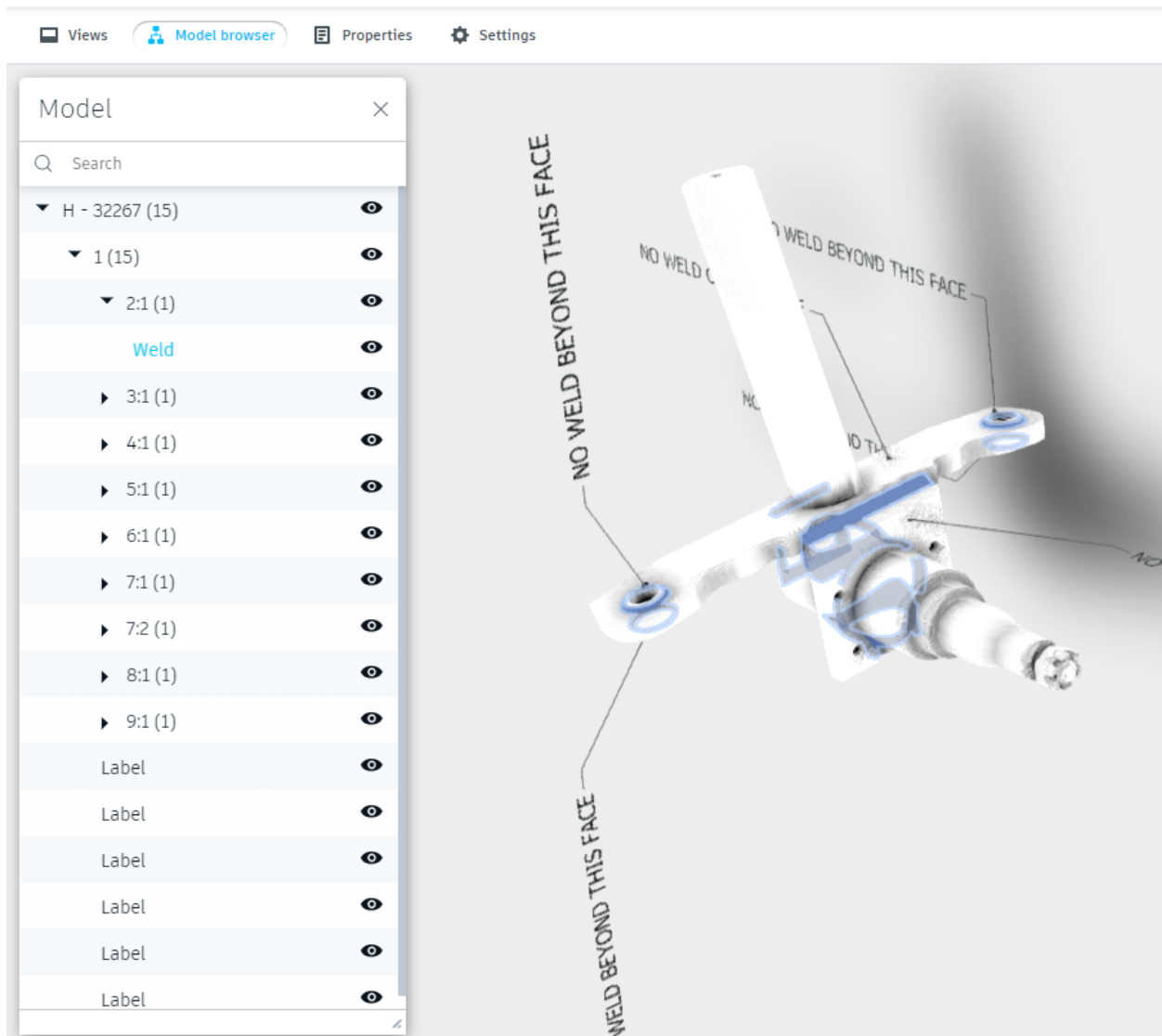
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Shared Views (Upload to Cloud via Inventor and Vault Pro)

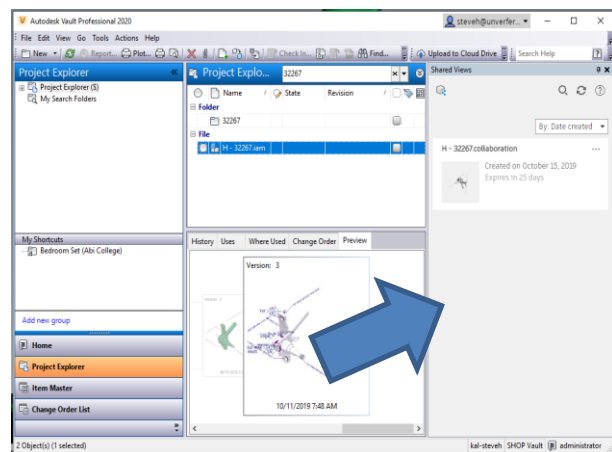
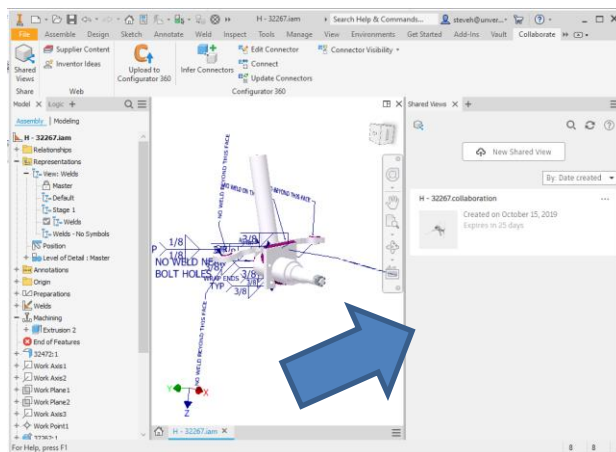
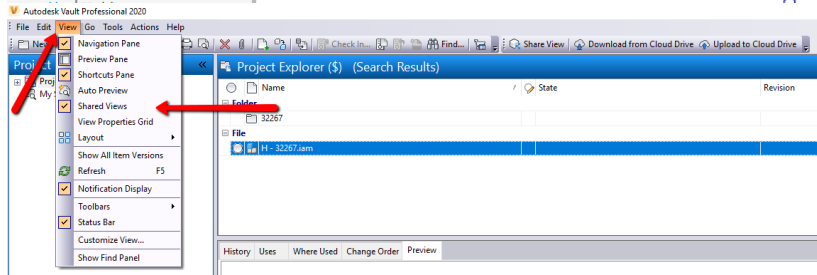
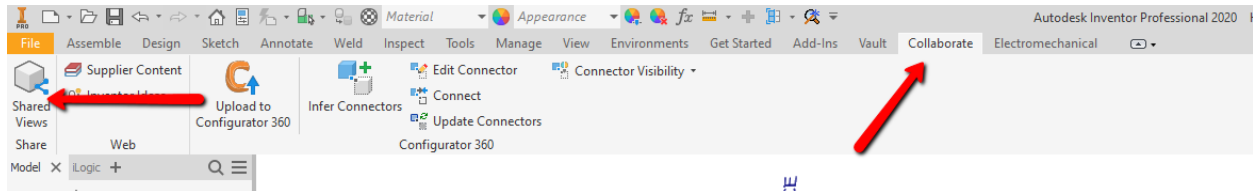
<https://autode.sk/33xpKsB> (Link to Shared View unless it's expired)

Allows you to easily upload an assembled to the cloud using the Colaborate/Shared Views built directly into Inventor/Vault. But, biggest short coming is that the weld annotation show up on models, but **weld symbols do not**.

 AUTODESK VIEWER > H - 32267.collaboration



How do you access Shared Views?



- Weld annotations do not show up.

- Uses the LMV (Large Model Viewer) technology from Autodesk.
- Available in 2018 and after
- For Inventor available through "Collaborate" Toolbar.
- For Vault Pro, available through "View" drop down...

Pro's

- Any device anywhere anytime, just need access to internet
- 30 day expiration on file unless extended, don't have to worry about deleting old data off.

- Just need a URL to view file, no Autodesk Account required
- URL is used to view file, files aren't actually emailed to consumer like dwf or pdf
- Single file uploaded, even on assemblies
- View Rep active at time of Share View is created is what you will get.
- Quick and Easy
- Only share what you want to share
- Better graphics...fast on all devices

Con's

- 30 day expiration, once it's gone it's gone. Can't retain history
- Can Share Weldts environment from Vault, but can't from Inventor.

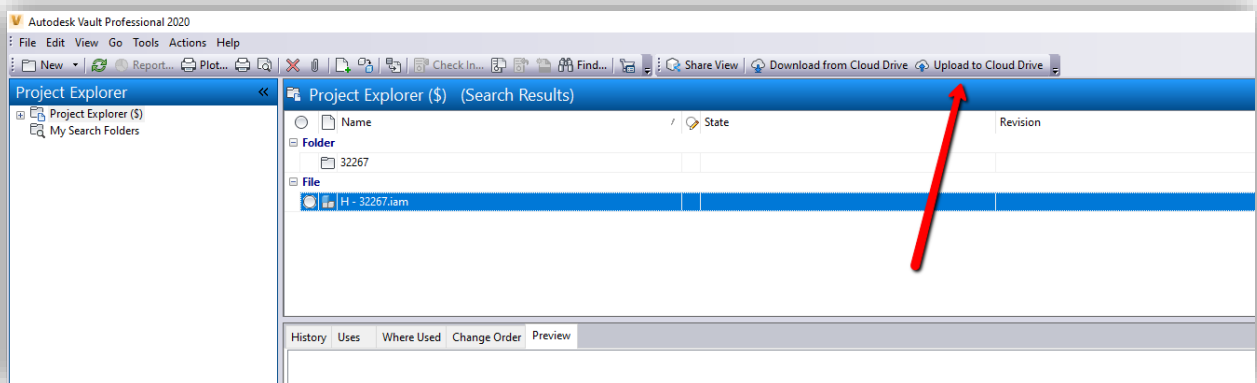
<https://knowledge.autodesk.com/support/inventor/troubleshooting/caas/sfdcarticles/sfdcarticles/Shared-View-cannot-be-created-from-a-Weldment-Assembly-in-Inventor.html>

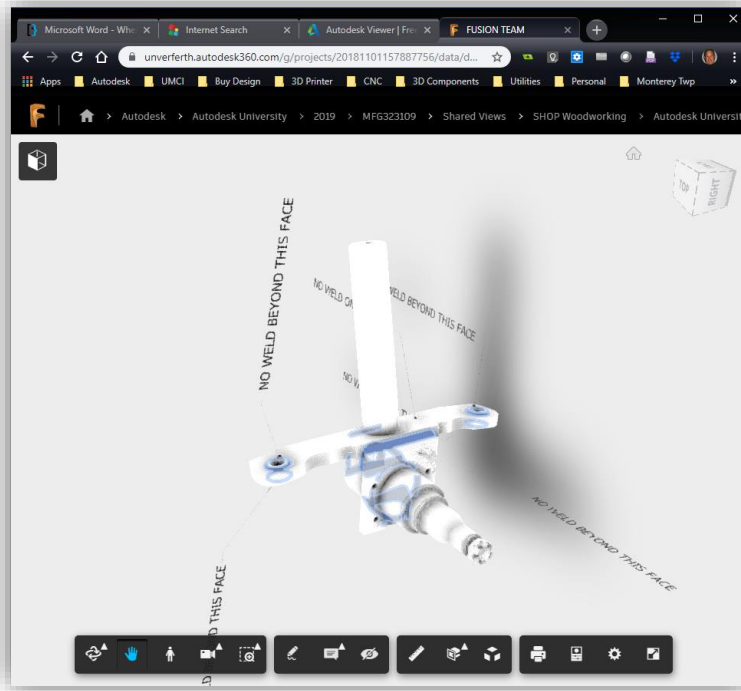
- Weld Annotations don't show up.

Fusion Team (Upload to Cloud)

Fusion team allows you to upload an assembly to the cloud. For Vault Pro users, there is a mechanism which allows a user to upload directly to Fusion Team ("Upload to Cloud Drive"), which is very similar to Shared Views. Whereas, for Inventor, you have to upload thru the Autodesk Connect App or manually upload files to Fusion Team. With either, short coming is that weld annotations show up on model, but **weld symbols are not displayed**.

<https://a360.co/31wYOL>





Pro's

- Any device, anytime, anywhere as long as you have access to internet
- Single URL to share
- Can upload directly from Vault (similar to Shared Views)
- Inventor uses Autodesk Connect or manual upload to FT.
- No expiration
- To view, just need URL. Doesn't require an Autodesk Account.

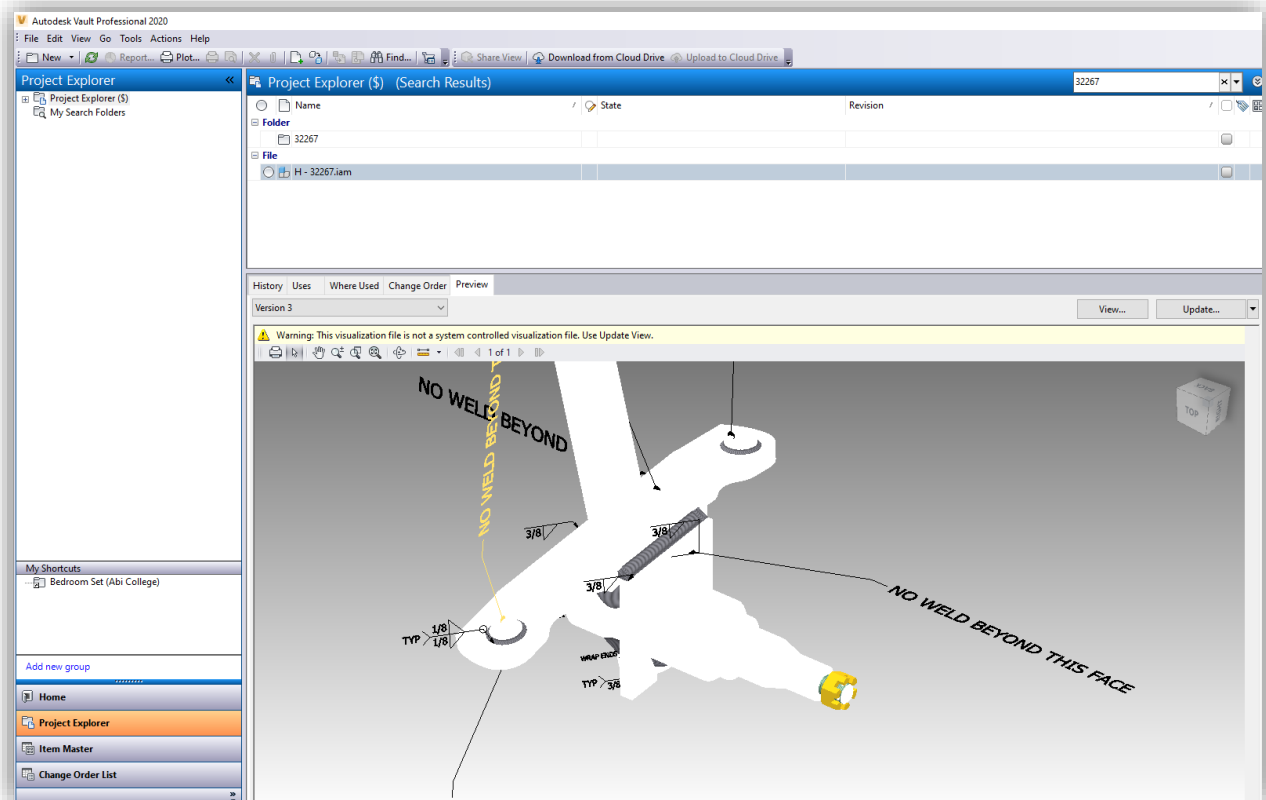
Con's

- **Weld Annotations show up on model, but weld symbols do not.**
- Using native Inventor files, all files are uploaded for an assembly. Unless you upload a dwf manually.
- Inventor does not have a "Upload to Cloud" feature like Vault.
- Weld MBD annotations show up, but Weld Symbols do not.

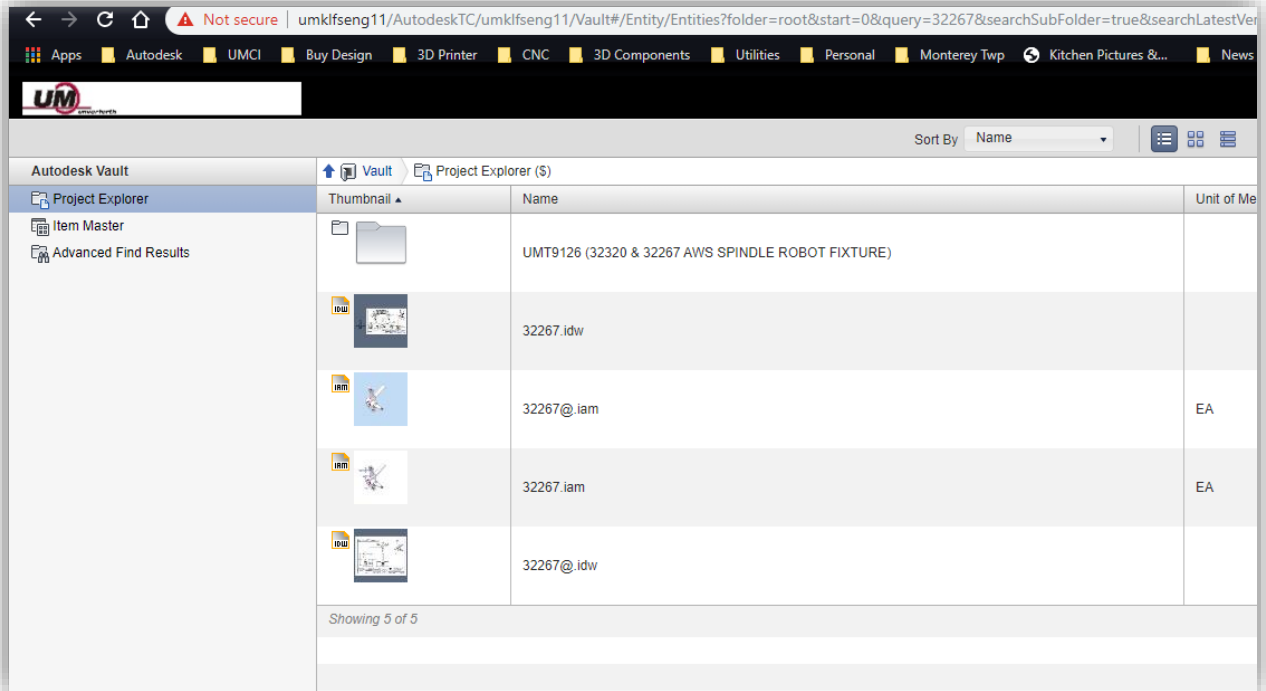
Vault Thick/Thin Client (Accesses Vault database directly)

The clients for Vault allow you to access the models directly from the Vault database. No extra steps to publish model.

Thick Client



Thin Client



Pro's

- Files are ready to use after file is checked into Vault, no need to upload to cloud.
- **Weld symbols show up.**
- More secure because files are within firewall

Con's

- Must be run on Windows based system (iOS, Android base systems not an option)
- Design Review is the viewing app....no longer updated.
- Larger models require larger computer
- Graphics are less than favorable

Forge (Hybrid of Vault and Cloud)

With Forge becoming more powerful and with more expertise being available, this is becoming a viable option. Upon release of an item (or file based release), Forge would basically publish the file to the cloud where you would access via the LMV.

We are currently exploring this option with Autodesk and our reseller.

Would provide us with a good viewer with any device, secure because Vault is triggering the visibility.

Only question is the visibility of weld symbols on the model.



Home Brewed App (Uses Vault and Design Review to view files)

App basically acts like the thin client by more user friendly and less noise. The app allows us to look at the model attached to the item and view it in Design Review. Item must be in Released or Obsolete state in order to view the model.

The image is a composite of three screenshots from a software application, likely a CAD or PDM system.

The top screenshot shows the 'Ohio' printer selection and a list of items. A red arrow points to the '3D' button in the 'Item' column. The 'Item' column also contains a table with columns 'Item' and 'Error'.

The middle screenshot shows a 3D model of a mechanical part, likely a spindle, with red arrows pointing to specific features. The model is labeled '32267' and '32267-001'.

The bottom screenshot shows a detailed technical drawing of the mechanical part, including a parts list and various dimensions. The parts list includes:

ITEM	PIN	QTY	DESCRIPTION
1	32108	1	PLATE - BRAKE FLANGE
2	32262	1	SHAFT - SPINDLE 2"
3	32063	1	SPINDLE
4	32319	2	TUBE BUSHING
5	32472	1	RADIUS ARM
6	32325	1	PLATE BAR
7	8280-016	1	SLOTTED NUT 3/4 UNF G2

The drawing also includes various dimensions and notes, such as 'NOTE: STAMP "D" THIS AREA WITH 1/8" PUNCH' and 'NOTE: LIMIT WELD IN THIS AREA, 1/8" MAX. (1/8" MAX. IN ALL AREAS)'. The drawing is labeled 'SECTION A-A' and 'REAR SPINDLE WELDT'.

Pro's

- Vault manages access to model.
- All within firewall....Secure
- **Weld symbols show up**
- Much more user friendly when compared to Thin Client

Con's

- Must be run on Windows based system (iOS / Android base systems not an option)
- Design Review is the viewing app....no longer updated.
- Larger models require larger computer
- Graphics are less than favorable

In Closing

- Get parts welded as intended
- Spend less time interpreting and communicating welds
- Spend More time Welding
- Spend More time Designing
- Build consistent products out the door
- So get modeling those welds and...

Get Your Welders and Designers Connected

Got An Idea? Share it!

- Share your idea directly with the Autodesk Development Team
- Community can support ideas to surface the most relevant



V AUTODESK® **VAULT**
<https://autode.sk/vaultidea>

I AUTODESK® **INVENTOR**®
<https://autode.sk/inventoridea>



There's 20 + Tips

There's also

15 Lovely Weld Wishes

Awaiting your Vote on the Inventor Ideas

Get involved with the Inventor & Vault product teams!

Help shape and define the future of the Inventor & Vault products!

Our Feedback Communities serve as our portal into our engagement program. Get access to insider builds through Alpha/Beta. Be included in exclusive online sharing sessions. Stay current on product roadmaps. Interact directly with members of the product teams.

Your thumb print on product direction is just a click away!



<https://autode.sk/VaultFeedback>



<https://autode.sk/InventorFeedback>

Appendix 1 - Weld Environment Wishes with Hyperlinks

1. Convert to Weldment Material and BOM structure: <https://forums.autodesk.com/t5/inventor-ideas/convert-to-weldment-environment-default-material-amp-bom/idi-p/6337316>
2. Can't have POS REPS: <https://forums.autodesk.com/t5/inventor-ideas/position-representations-in-weldments/idc-p/9112564#M36858>
3. Inaccurate Leg Lengths: <https://forums.autodesk.com/t5/inventor-ideas/inventor-weld-leg-length-inaccurate/idi-p/9112646>
4. Spot/Butt Weld: <https://forums.autodesk.com/t5/inventor-ideas/spot-weld-in-inventor-weldments/idc-p/9112380#M36853>
5. Round to Round Weld: <https://forums.autodesk.com/t5/inventor-ideas/weld-type-for-round-to-round-parts-in-the-inventor-weldment/idi-p/9112852>
6. Must create linked weld symbol at time of weld creation. Can't add symbol later: <https://forums.autodesk.com/t5/inventor-ideas/fractional-weld-sizes/idi-p/4301520>
7. Fractional Weld Symbols: <https://forums.autodesk.com/t5/inventor-ideas/fractional-weld-sizes/idi-p/4301520>
8. Stages of Weld: <https://forums.autodesk.com/t5/inventor-ideas/welds/idi-p/4831353>
9. Round Part "Start-End" offset must be from geometry not work features or Origin planes.
10. Round part Fillet welds "From-To" doesn't work without geometry: <https://forums.autodesk.com/t5/inventor-ideas/inventor-fillet-weld-quot-from-to-quot-on-round-part-requires/idi-p/9112890>
11. Pattern Welds: <https://forums.autodesk.com/t5/inventor-ideas/pattern-welds-feature/idi-p/7985064>
12. No Individual Weld Visibility control: <https://forums.autodesk.com/t5/inventor-ideas/weld-visibility-in-weldment-assemblies/idi-p/4508109>
13. Can't Tie Individual weld suppression to View Rep: <https://forums.autodesk.com/t5/inventor-ideas/inventor-fillet-weld-quot-start-length-quot-on-round-requires/idi-p/9112899>

14. Shared Views – Inventor: Weldments can't be shared from Inventor:
<https://forums.autodesk.com/t5/inventor-ideas/shared-view-for-weld-environment-can-t-do-weld-annotations-must/idi-p/9113066>
15. Shared Views – Vault: Weld Symbols don't show up when shared from Vault
16. Fusion Team – Weld Symbols don't show up

