

Structuring Data in ACAD Mechanical

Grouping geometry as components is not a new concept to AutoCAD Mechanical. Prior to the introduction of Mechanical Structure, there were two popular ways of organizing geometry: layer groups and blocks.

Layer groups

Allow for grouping of geometry based on layers. Suppose you are creating a shaft component for a gear box. You can create a layer group named “MainGearShaft” and place all geometry that belongs to the shaft on that layer group. You can then treat the shaft as a single entity and perform operations, such as applying visibility overrides, on it.

Blocks

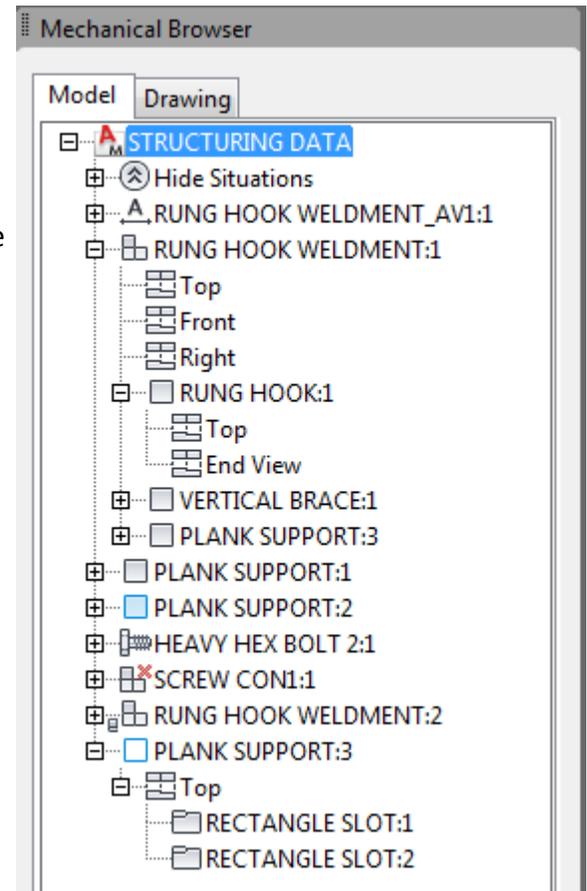
Allow for defining a block with the geometry that represents a component and later adding a part reference to it. When a component must be inserted to a drawing, insert the corresponding block. The BOM updates automatically and you save time because the geometry is reused.

Structure

Mechanical structure is primarily a way to group lines, arcs, and circles (geometry) as parts, and later, group parts as assemblies. In mechanical structure jargon, parts and assemblies are commonly referred to as components.

Starting Structure

In order to use Mechanical Structure the interface needs to be set to the Structure workspace and the **STRUCT** status bar toggle needs to be enabled. If a user tries to turn off STRUCT when using Structure, an error message will appear that it cannot be disabled while Structure content exists.



Why Structure?

Three words...reduce, reuse, and recycle. With Mechanical Structure you can reduce the amount of time spent refining multiple instances of the same part in your design for simple or complex changes in geometry. You can reuse your components over and over again recycling them into new assemblies with the ease of use of the Structure Catalog and no-nonsense X-refs.

Design A may have 50% reusable parts to be recycled into Design B and their entire definition can be referenced into that new design with little effort. Once placed into that new design the link can also be localized to sever ties to the first design when a new part is desired from an existing one as to not inadvertently update an already completed design with iterative data.

Structure helps you focus your mind on the parts and assemblies instead of the lines and arcs that normally clutter geometry making your design components overlap and harder to modify or visualize.

Structure Lingo

 **Component** – A component is essentially any part or assembly of parts within the structure

 **Component View** – A component view is a definition of the geometry that makes up a standard multi-view of the component. These have presets as well as the option to name the views whatever the user desires.

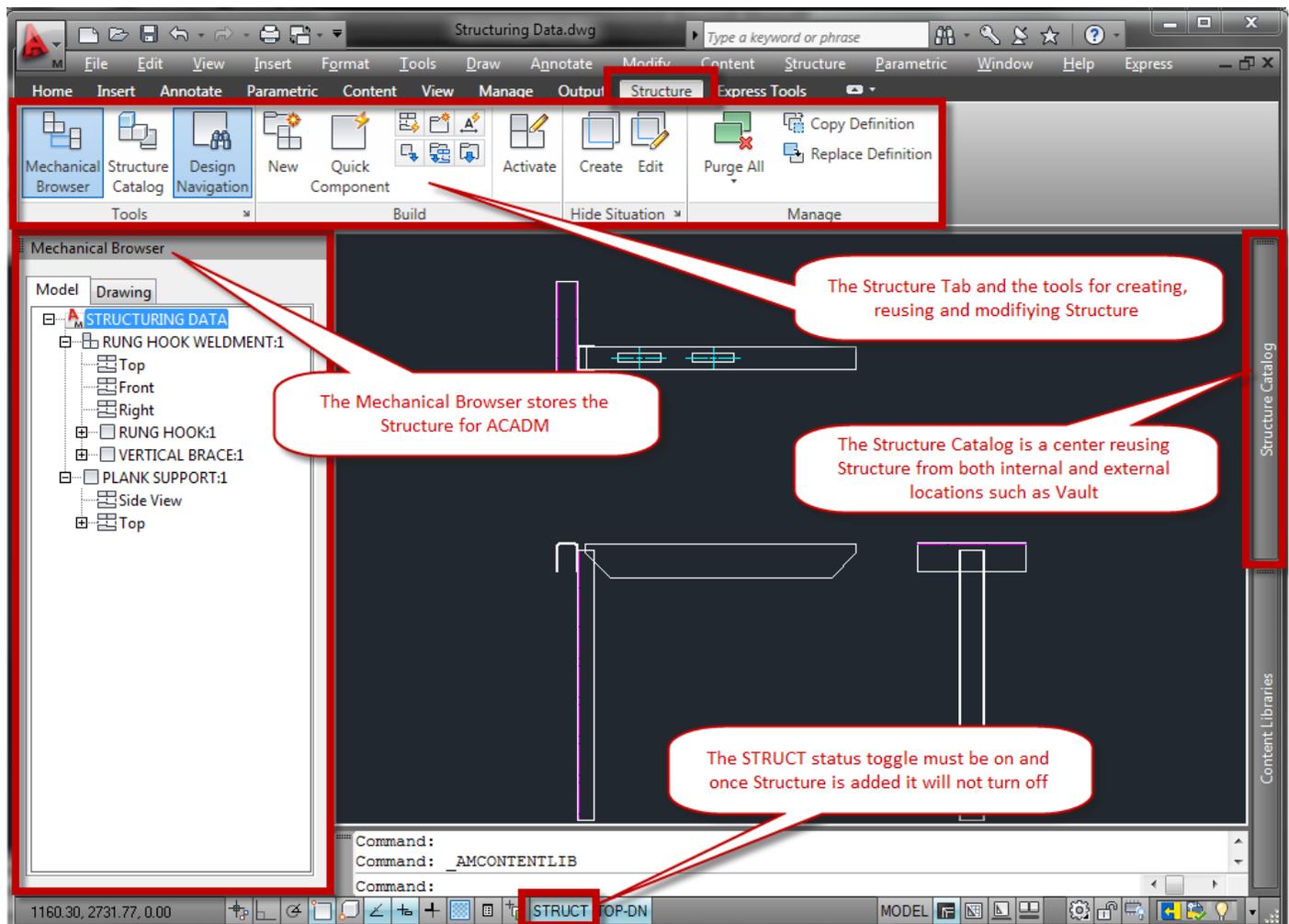
X-Ref (External Reference) – When you attach a drawing file as an X-ref, you link that referenced drawing to the current drawing. Any changes to the referenced drawing are displayed in the current drawing when it is opened or reloaded. A drawing file can be attached as an X-ref to multiple drawings at the same time. Conversely, multiple drawings can be attached as referenced drawings to a single drawing.

Localize – Detaching an X-Ref so that it no longer references another file and geometry is stored locally.

Externalize – Creating a new separate file based on existing local data and automatically attaching as an X-Ref.

Creating a Part Component

With the Structure Workspace active and the STRUCT status bar icon turned on you can now utilize the Structure Tab of the Ribbon that is added in this workspace. Make sure the Mechanical Browser is also turned on. This will be a palette similar to the Properties palette that can be anchored or docked on either side of the screen.



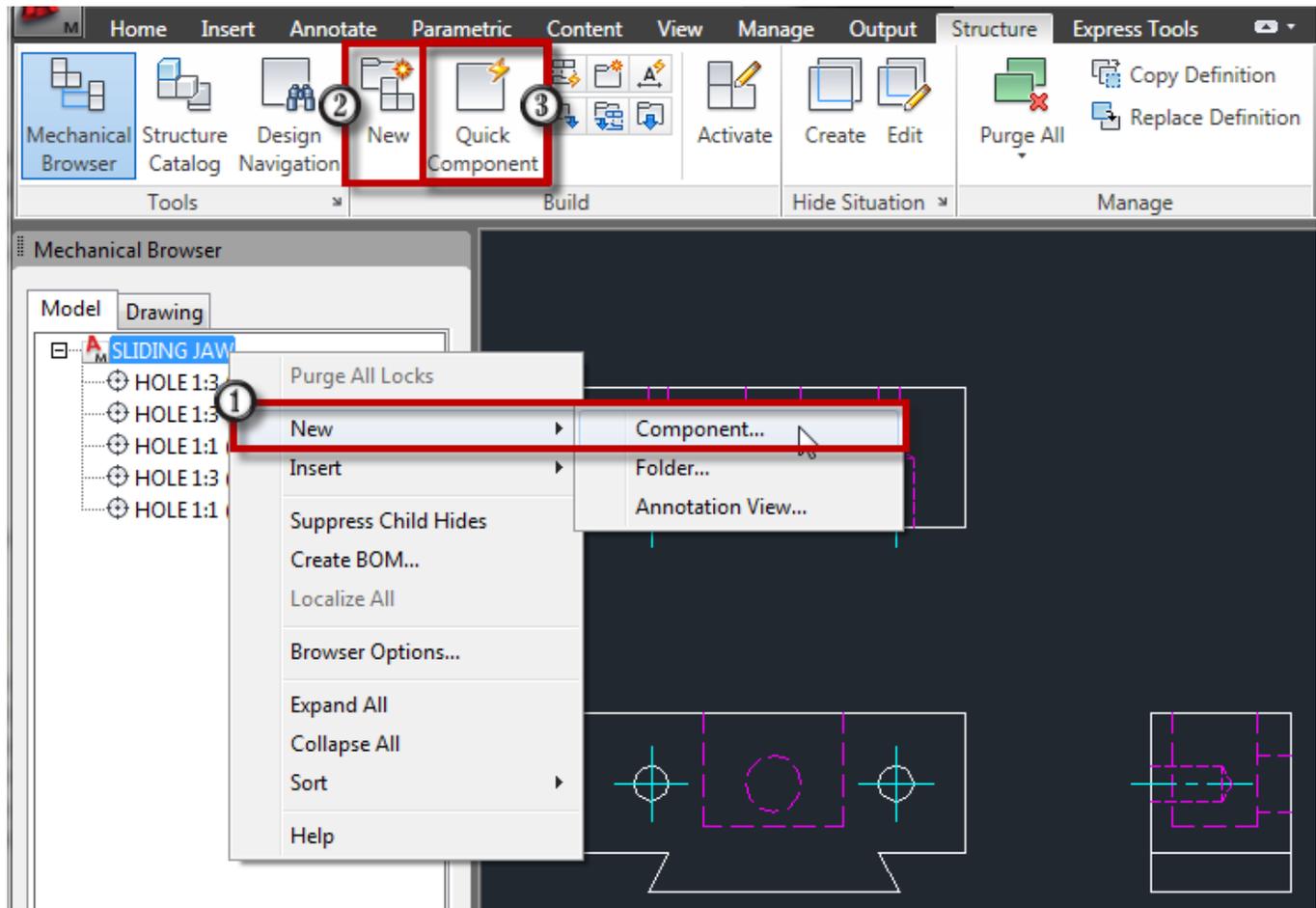
Top-Down vs. Bottom-Up Creation

These are two ways to create Structure in AutoCAD Mechanical. One way is to start with Assemblies and then add parts to the assemblies and then create geometry for the parts and is commonly known as a Top-Down. The other method starts with geometry to create the part components and then structure them into their assemblies either in that file or in a separate file and is known as a Bottom-Up. The best method will depend on a couple factors such as how you wish to document your designs and how much legacy data must be reused. Structure can also be created automatically with the use of the Standard Content in Mechanical such as Screw Templates and Shaft Generators.

Creating Part Components (Bottom-Up)

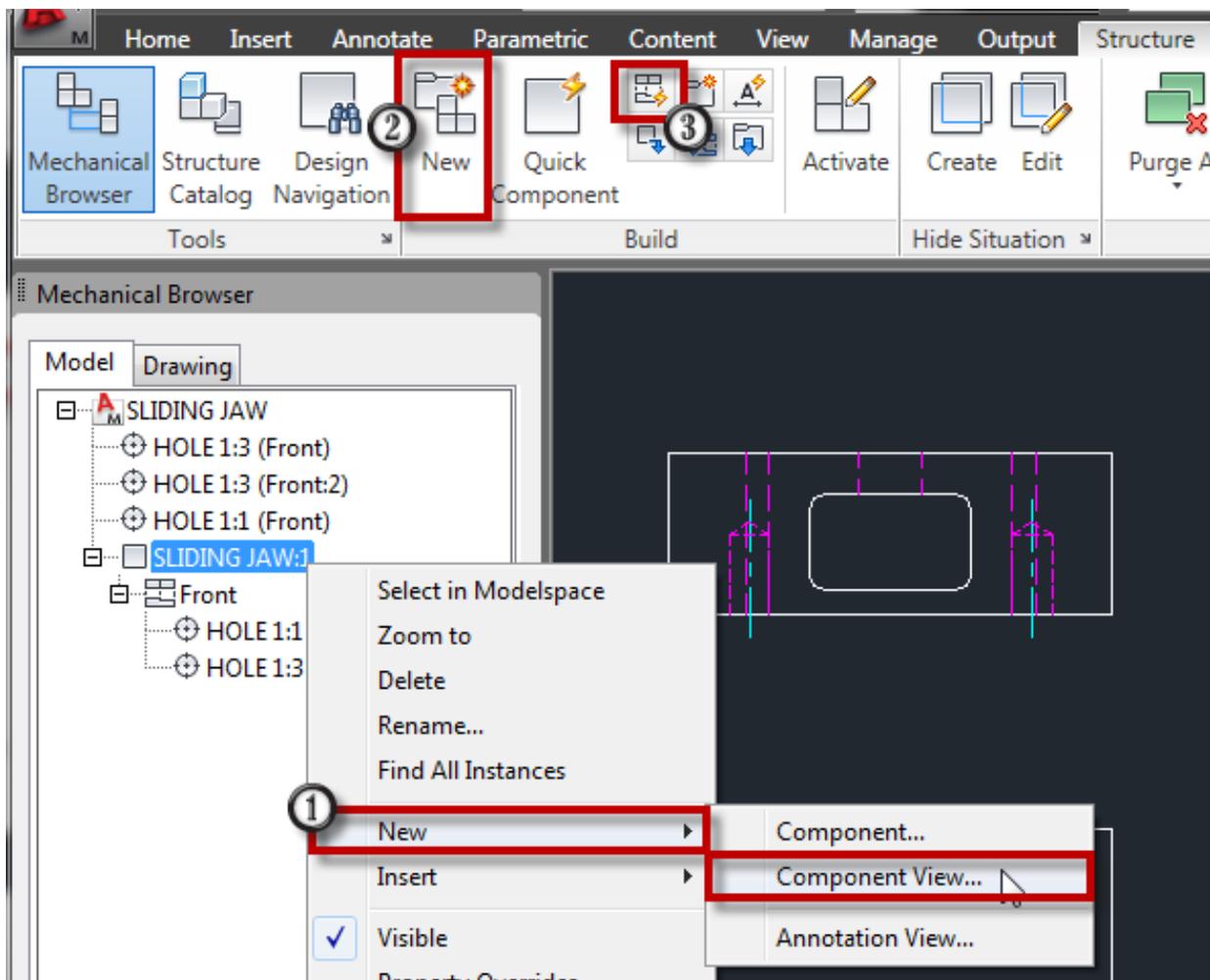
This method is the most common when individual part drawings already exist and simply need structure added to them. Drawings such as standard parts or commonly used legacy data are the most common sources for these in Mechanical projects. Here we have a three view part with Mechanical Structure enabled ready to create the component. Notice there are already some features listed in the Mechanical Browser because they were created with the Content tab of the Ribbon. There are three ways to create a new component in the browser...

1. Right Mouse Button on the file name and select New → Component – This will prompt you for all the pertinent information in the view creation process such as precise base point and view name.
2. The **AMSNEW** Button – This launch a dialog box with all the same information as the previous method
3. The **AMSCREATE** Button – This will very quickly just add a Component with default naming and 0,0 base point to the browser without any user input and is commonly done to create structure first and then geometry second whereas the first two methods commonly would have geometry present.



When using structure, it is important to add additional views to the component that are necessary for two different reasons. The first reason is simply for what is needed to document the design in a standard layout. The other reason would be for any type of view that is required to show specific location or placement in an assembly. The latter requirement is usually satisfied by the first need. Adding additional views can be done three different ways as well.

1. Right mouse button on the new component in the browser and select New → Component View – this method is identical to the method to create the component
2. The **AMSNEW** command – this method is the same as before as well, but different options need selected in the dialog box to create a component view of an existing part
3. The **AMSCREATE** command – this is the same command as before but Mechanical tells the command to look for a parent reference first before asking for the geometry. This comes in handy when the Mechanical Browser gets especially large in big assemblies.



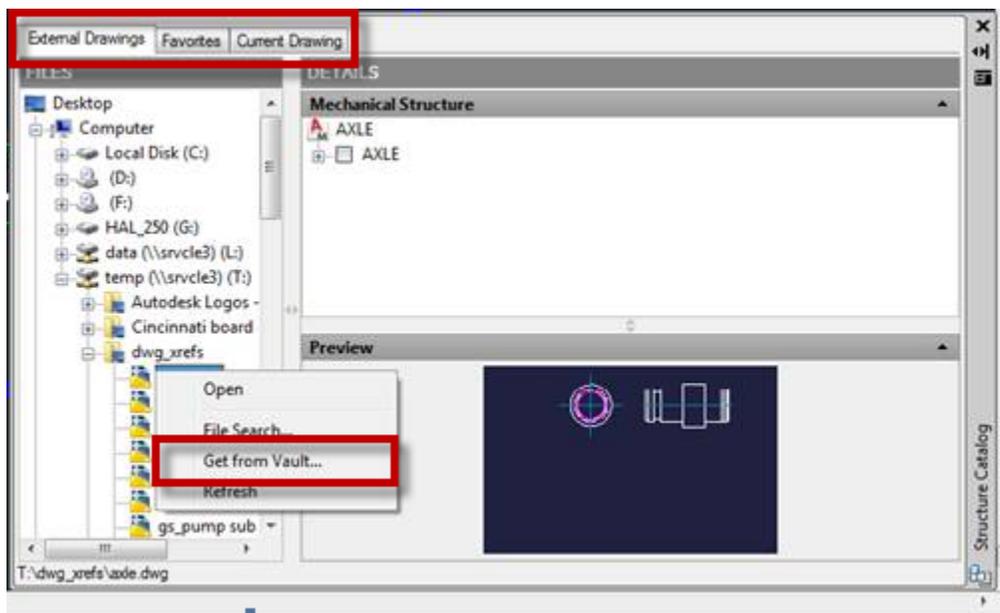
Overall, the structure part is very easy to make changes to. Simply activate the view by double clicking in the browser or using object selection cycling and perform standard modifications or additions of geometry. The Automatic Construction Line tools of AutoCAD Mechanical also aid in this design process to make sure the geometry is updating across views accordingly. If layer groups or block references were used instead, it would be very prone to errors and slow modifications due to the nature of blocks and layers toggling.

Creating an Assembly Component

In AutoCAD Mechanical there are many ways to document and organize drawing data which gives the software a great deal of flexibility especially when it comes to structured components. Commonly, parts are detailed individually and then assembled into another drawing and documented for placement and location there. In other cases designs are intended to be all inclusive of a single file. AutoCAD Mechanical can accommodate both of these methods as well as a combination of the two based on the design and documentation need.

Organize Parts into Assemblies and Sub-Assemblies

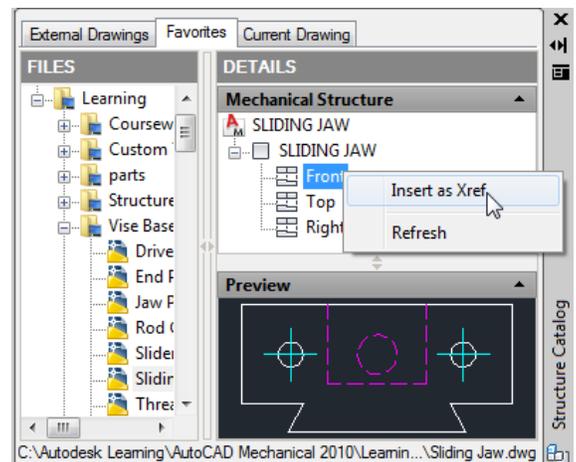
Not only does working with Structure make object selection and representation much easier it also simplifies the reuse of data. This is accomplished with the **Structure Catalog Palette**. Data can be added to a drawing by browsing files stored in the Autodesk Vault, company file server, favorite directories such as libraries, and even the current drawing to add more instances of components. If external data is to be used it is recommended to use the Autodesk Vault. Autodesk Vault maintains link references for renamed files whereas in a large group setting, X-refs can become messy and links can be invalidated by improper movement.



Referencing in Parts from another drawing

Whether from a library of standard parts, a design working directory, or a vaulted file store, the Structure Catalog can locate files and then browse or insert their Mechanical Structure. In this example the, the Sliding Jaw part file is searched for and then inserted to the Vise assembly.

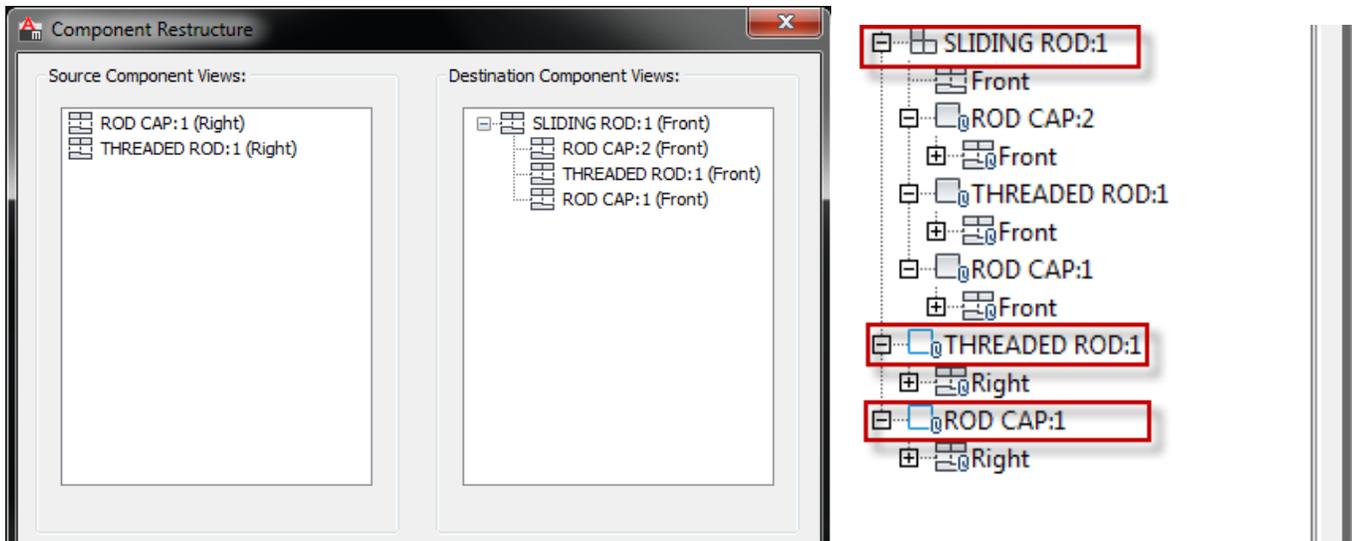
To insert, simply find a view such as the Front view shown at right and then either right mouse click and choose Insert as X-ref. You can also simply drag and drop the view into canvas to start placing it. The Component name will automatically populate the Mechanical Browser.



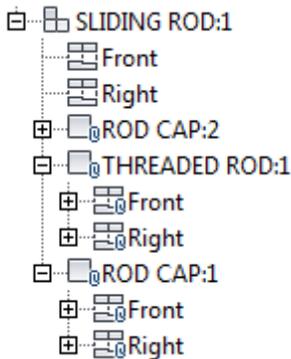
Creating the Assembly Structure

In a drawing where multiple components have been placed by either documentation method (single DWG or individual detail DWGs) the demotion process can be used to create assemblies. This is done by manipulating the Assembly browser with dragging and dropping of the existing part components. Follow one of the above mentioned Part creation methods first to get initial name and views for the new component. If you are using the Mechanical Browser or AMSNEW to create the component you will be able to select the geometry right away for view placement.

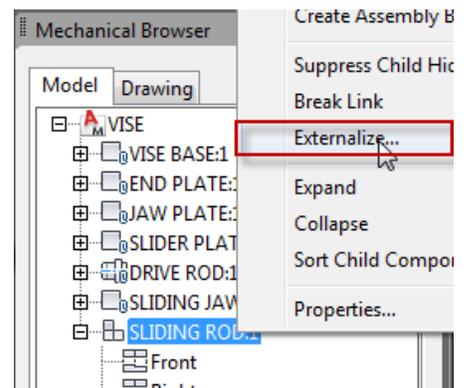
When already defined structure is selected for geometry of this component the views of the component being selected will have to be reference to a new view of the assembly of that geometry. In the example below, the new Component Sliding Rod is being created with geometry from Threaded Rod:1 and Rod Cap:1 and Rod Cap:2. The views on the left are currently not linked to the Sliding Rod:1 (Front) view. This will leave them orphaned in the browser and their icon will state they are Ghost components until resolved. Notice below that the Sliding Rod Component is now an Assembly Component and the Threaded Rod and Rod Cap are lighter in color stating their Ghost status.



To resolve the Ghost status of the two remaining components, simply create another Component View of the new Sliding Rod assembly and select the remaining geometry on the screen for those components. This will resolve the browser and relocates the remaining views into the assembly component views. Notice that currently this Sliding Rod:1 is part of the current drawing file. The other components were referenced in and are designated as such with the small paperclip icon located next to the component and view icons.



Depending on how documentation is to be organized it may be beneficial to keep this file stored separately somewhere else so it can be documented as a Subassembly instead of the entire assembly. To save this out as its own file, right click on the assembly and choose Externalize. This will create a new file and link the external references accordingly. The assembly icon will also change to reflect this.

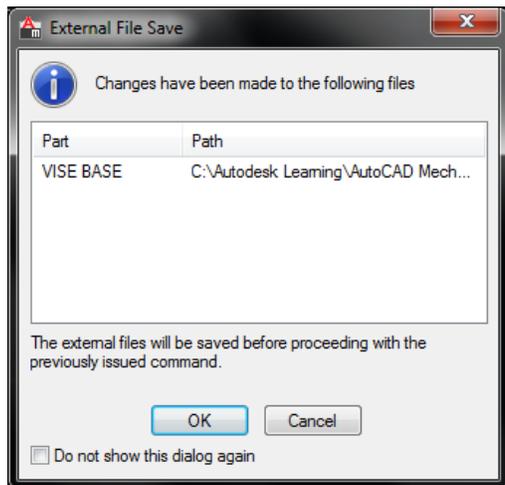


Creating a Part Component in an Assembly (Top Down)

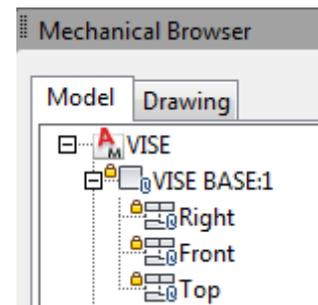
To create a Part Component in context of the assembly, simply follow the same methods as before for creating a part, but instead performing the geometry creation inside the assembly file. This method is very powerful for creating geometry while being able to see how it fits with the rest of the design. After this has been designed, use the Externalize option again to create a separate file that can be used elsewhere, detailed separately, or placed into a file store for explicit tracking for that part.

Modifying Geometry and Iterations

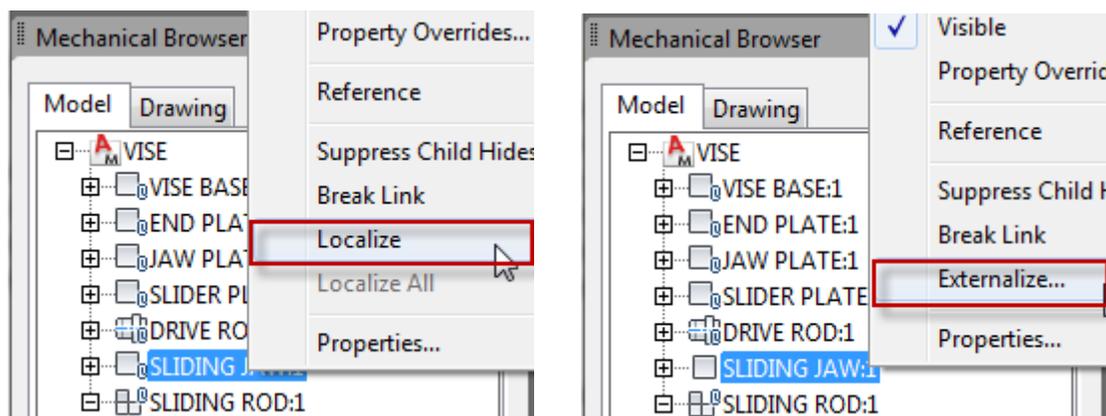
When modifying geometry of a new inside an assembly file that contains external reference links to the part files, modifications can take place right inside the assembly file and force the external reference file to update based on those design changes. Before changing anything, it is important to know where that file is used in other designs. A change inside one assembly to an externally reference part will update it in all assemblies the next time they are opened. Autodesk Vault has searching and indexing tools to check where a file may be used in other designs and gives the designer a very quick understanding of where and how that external reference is used throughout the company's drawing data.



When changes are made a Lock Icon will appear on the component in the structure stating that locks are in place based on edits made in context of the assembly. Purging the locks will prevent users from performing any Undo operations past this point.



After checking your designs if found that the current change is to be for the current design only and updates to existing data is not advisable, a **Localize** of the X-ref needs to take place. AutoCAD Mechanical will then ask if you want to detach the X-ref files if no more references are in the current drawing. From here, changes can be made safely to the Component and then **Externalize** again for new file iteration.



Conclusion

The most significant advantage of mechanical structure is reuse. Just like in the case of blocks, once a component (part or subassembly) is defined, you can insert the component as many times as required. Using the structure catalog, you can explore the components in other drawings and quickly insert them in to your drawing. You can insert them as copies or as X-Refs. Components in your existing drawings instantly become reusable.

If you insert a component as an X-Ref, the component is associative. When the component is modified in one drawing, the change is reflected in all drawings that use the component. To make iterations, localize and then externalize the data for a new traceable file.

The mechanical browser ensures that the structure of a drawing is easily understood by anyone. With the mechanical browser you can select an entire assembly or a view of a part. This can significantly speed up the drafting process, since mechanical structure is designed for the explicit purpose of organizing a drawing. The features go beyond visibility enhancements (offered by layer groups), reuse of geometry, and automatic BOM updates (offered by blocks).

For example, the mechanical browser shows the hierarchical organization of components so you can quickly identify structure components, as well as use right-click menu options for component-related operations. Similarly, you can use the selection modes to click objects in model space and select parts and assemblies, instead of lines, arcs, and circles.

Overall, this method of organization will lead to fewer errors in designs, less rework, easier iterations of designs, and more stable use of external references as well as stay more focused on the manufacturing hierarchy and documentation structure.

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Mark is an Application Engineer with IMAGINiT Technologies, specializing in manufacturing environments. He has implemented the Autodesk Manufacturing products with several industries including the blow/injection molding, automotive, and custom machinery markets. Inventor has been a profound augmentation in his abilities allowing him to bring 3D digital prototyping to the forefront of the industries with which he has interacted. He has extensive experience and a comprehensive understanding of the technical, practical business and human dimensions of implementation. He is an effective and skillful communicator, consulting with his clients to help achieve their business objectives. Mark is an ATC certified instructor and he provides training, support, and implementation. Mark is PSE and ATC certified in AutoCAD®, AutoCAD® Mechanical, AutoCAD® Electrical, Autodesk® Data Management, and Autodesk® Inventor™.



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