



Walt Disney World Swan and Dolphin Resort
Orlando, Florida

Building Bridges Between AutoCAD® Mechanical and Autodesk Inventor®

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and Colleen Klein (Assistant); Kevin Smedley (Assistant)

MA22-1L Do you have a mix of 2D and 3D users? Want them to work together as one big happy family -- on the same team, with the same data? Learn how your AutoCAD Mechanical users can link to Inventor designs. This 90-minute lab allows you to experience first-hand the Inventor Link in AutoCAD Mechanical. We will work with a real-life example of documenting a part and an assembly created in Inventor. You will then change the design and update your Mechanical drawings. See for yourself how everyone can work together.

About the Speaker:

Christiaan is an accomplished instructor and an Autodesk Inventor Certified Expert with 16 years of experience as a specialist in mechanical design primarily using Autodesk products. He combines his expertise, knowledge, and patience to teach innovative courses in AutoCAD, and the Autodesk Inventor Professional Series products. Christiaan has also incorporated a company, Sellier Enterprises, Inc., that specializes in mechanical design for industrial automated equipment, including the high speed machine-automated packaging industry.

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Imagine that you are an automatic paper towel dispenser manufacturer, responsible for manufacturing eighty percent of your product. The predicament encountered is that the knowledge base of your drafting staff is experienced in AutoCAD Mechanical. Your mechanical design staff uses Inventor. This lab presents a step-by-step procedure in working between AutoCAD Mechanical and Inventor. We will accomplish this through a series of tasks. They are as follows:

- **Task 1** Firing up the software and creating an Inventor Link for a part between Mechanical and Inventor
- **Task 2** Creating views and Detailing the linked part
- **Task 3** Modifying the linked part in Inventor and addressing the modifications in Mechanical
- **Task 4** Creating an Inventor Link for an assembly between Mechanical and Inventor
- **Task 5** Creating views, a parts list and ballooning the linked assembly

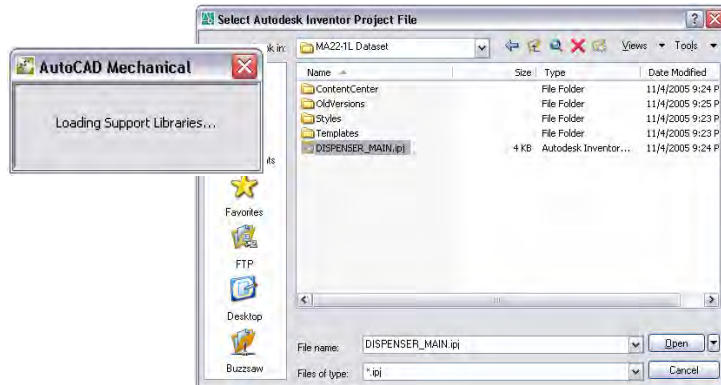
Getting back to the challenge at hand, fresh out of the design department we have a new gear cover and tension roller that's been modified to streamline the design. We need to detail the new gear cover and tension roller sub-assembly. First, let's open AutoCAD Mechanical.

Task 1 Firing up the software and creating an Inventor Link for a part between Mechanical and Inventor

From the desktop, double click on:



Next, let's set the search Inventor's search paths. This is done so that we can have easy access to the Inventor files we need to detail. From the command line, type **amivproject**. Mechanical will load the support libraries, this might take a minute. You will see the following message and then the **Select Autodesk Inventor Project File** dialog box.



Next, select the **DISPENSER_MAIN.ipj**.

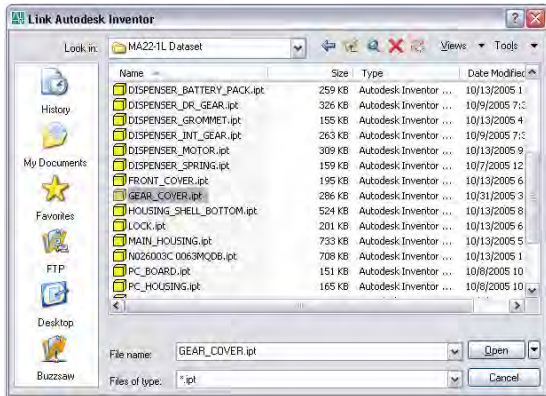
Next, jump back to AutoCAD Mechanical and Open a **New Inventor Link** by using the **New** toolbar fly-out or the **File** pull-down menu.



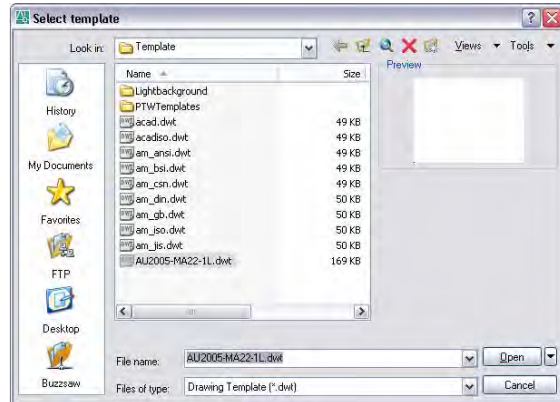
Choose an appropriate AutoCAD Mechanical template, in our case it will be the **AU2005-MA22-1L.dwt** template.

Next, let's choose first the **GEAR_COVER.ipt**, which is found in our project file under **C:\.....\MA22-1L**.

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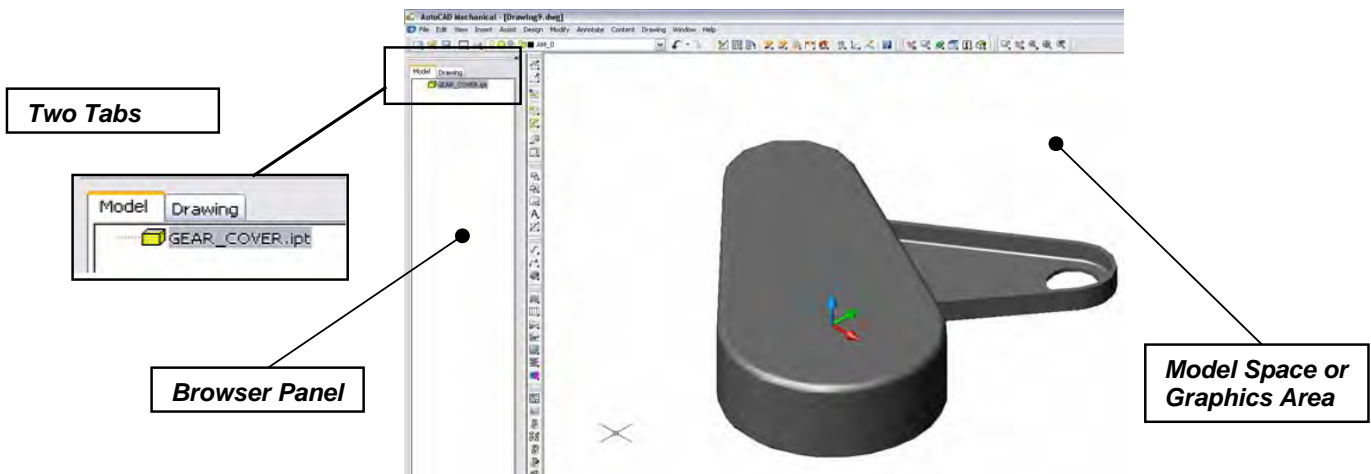


Link Autodesk Inventor Dialog



Template Location

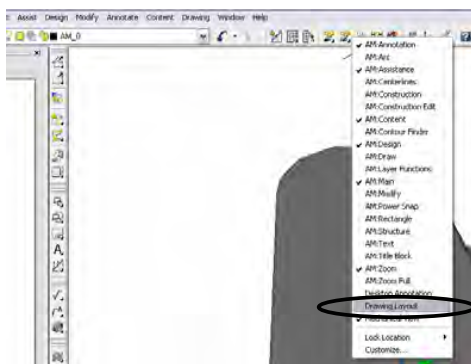
Our screen should now look like the following image. Great, now that we have our linked relationship set up, let's detail our part. The first thing we need to do is get into the drawing environment. Notice the two tabs on Mechanical's **Browser Panel** the **Model** and **Drawing** tab. These tabs are used to toggle between the drawing and model environment of our file. We now need to toggle into our drawing environment so click on the **Drawing** tab.



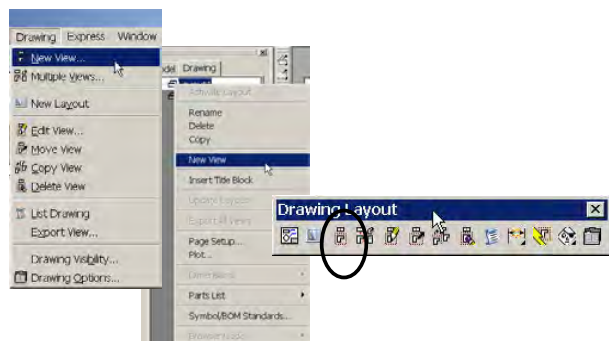
Excellent, Task 1 is now complete let's move on to our next task, Task 2.

Task 2 Creating views and Detailing the linked part

Now that we are in the drawing environment, let's place some views. First, let's activate our **Drawing Layout toolbar** and dock it. Right click on any toolbar and select **Drawing Layout**. Next, let's place a view by either using the **Drawing pull-down menu**, or right-click in the **Model Space area** and select **New View**.

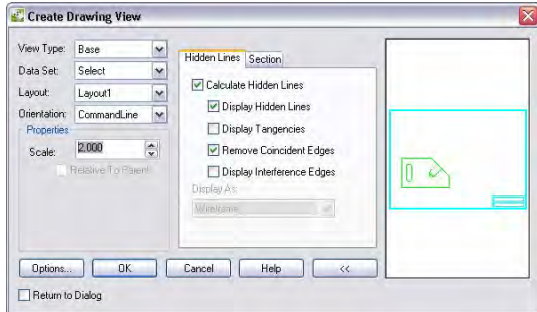


Activating the Layout Toolbar

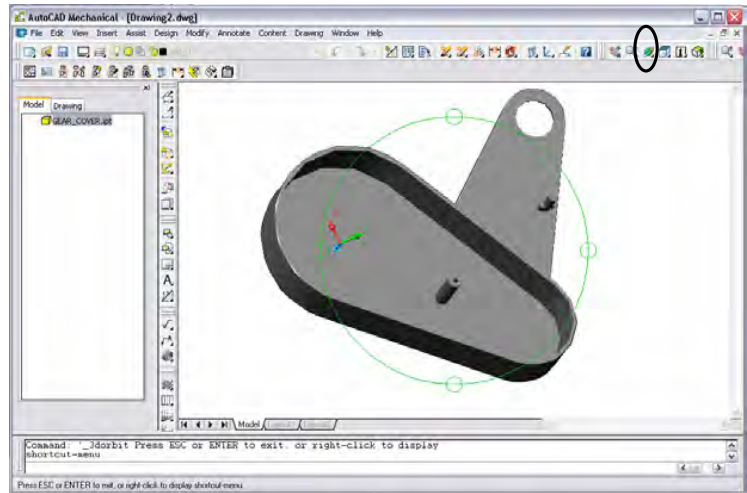


Activating the New View Command

Once we're in the **Create Drawing View** dialog, accept the default inputs except, change the scale to **2.00** and select **OK**. This toggles us back to the model space area. Once we are in model space, let's rotate the part so we see the inside face as shown in the following image. Let's select the **3D Orbit** button, found on the **Mechanical View** toolbar and rotate the model into the orientation as shown below.



Create Drawing View Dialog



Rotate the View with 3D Orbit

Once we're in the model space area, select the inside face as shown for the **planar face**. If the wrong face is selected, toggle through the selections with your right mouse button until the correct face is selected then hit **<Enter>**. Next, select the bottom edge of the part as the **straight edge**. The **axis orientation** should be such that the **Y axis** is pointing up as shown below, again, if this isn't the case then right mouse button to toggle through the options until this situation is created.



Planar Face



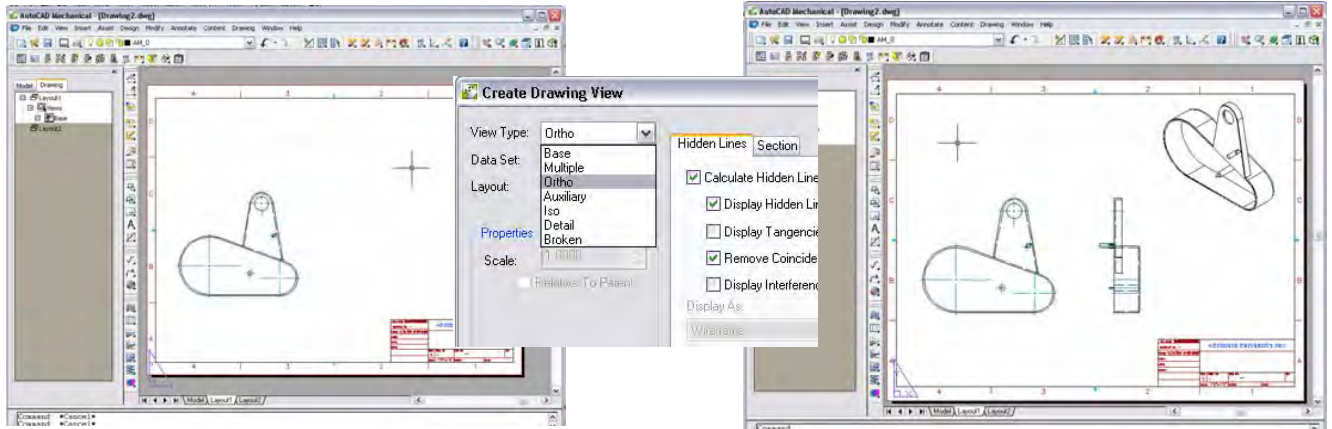
Straight Edge



Axis Orientation

Once the information for the command line is satisfied, Mechanical toggles us back to the drawing environment. At this point we will select where we want our base view, let's place in the **Zone B4** of our drawing. Let's now create an **Ortho View** and an **Iso View**. Once again, we will create a new view, but this time we will select **Ortho** as our **View Type**. Select the parent view in our case it's our **Base View** and place it to the right, as shown in the following figure. Notice that, since this is an orthographic view, we are restricted to only placing the view in an orthographic relationship to the base view. Next, we will create an isometric view in the same fashion, select **Iso** as the **View Type**, select the Base view as the parent and place it in the upper right corner of our drawing as shown in the following image. It's interesting to note how our browser is populated and labeled with the views we've created.

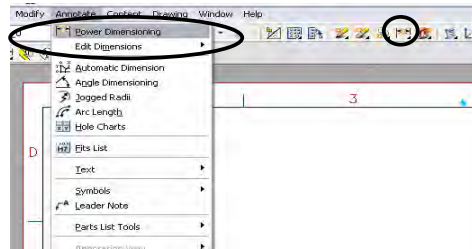
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Base View

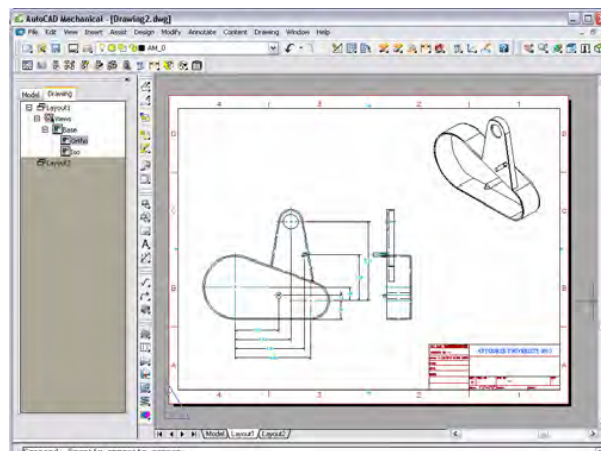
Ortho and Iso Views

Let's save our work at this point. Once our work is saved, let's begin detailing the part. We will first establish some locating dimensions for our radii and diameters. We will use the **Power Dimensioning** tool to do this. This can be found on our **Main toolbar** at the top of the screen. It can also be activated from our **Annotate pull-down menu**.



Activating the Power Dimensioning Command

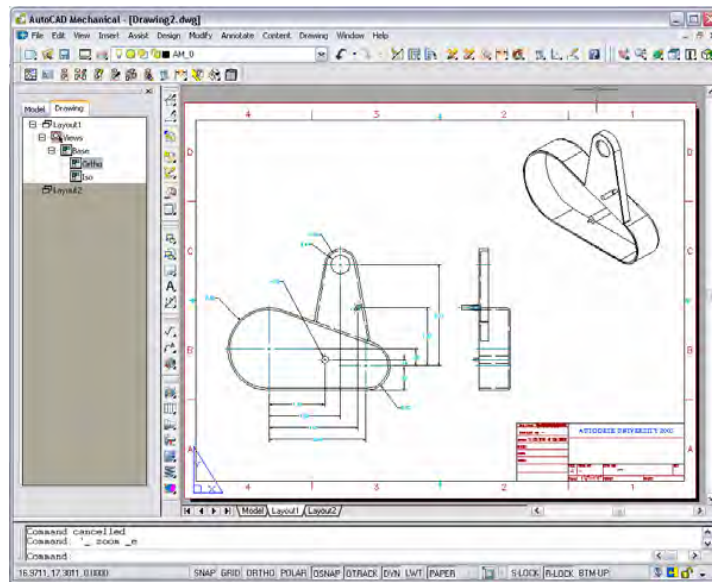
Select the endpoint of the centerline to locate our first point of the radius then select the endpoint of the centerline for the diameter of the boss and place the dimension as shown in the following figure. The next dimension is from the endpoint of the first radius to the endpoint of the hole. Before the dimension is placed, let's hover our cursor over the first dimension and place it below when it turns red. This ensures proper spacing of your dimensions, this is one of the reasons we use power dimension. Another reason is when a dimension is no longer required, if a power erase is used, then the dimensions will realign automatically. Try it! Our drawing should now look like the following figure.



****Your drawing should now look like this figure****

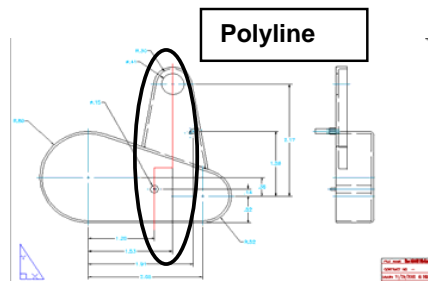
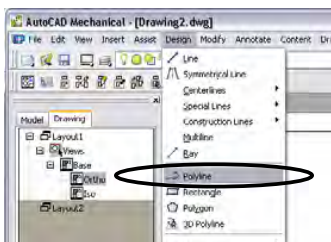
It looks as though that our ortho view is a bit too close to our base view so let's move it. This can be done by right clicking the **Ortho** view in either the browser or the view itself and then selecting **Move** and sliding the view a little to the right. Did you notice that the ortho view only moves along the X-axis? This is because Mechanical enforces drafting standards, which helps with keeping drawings uniform and its one less thing to worry about when you placing views. At this point, saving our work would probably be a good idea.

Now let's go ahead and dimension the size of the radii and diameters of the part. We will again use our **Power Dimensioning** tool to accomplish this, so let's activate the tool and hit **<Enter>** on our keyboard to access the circular dimensioning of the tool. Select the large radius and place the dimension. Dimension the remaining radii and diameters. Your drawing should look like the following figure.



****Your drawing should now look like this figure****

We can now detail the boss, but we really can't see the entire height, it's partially obstructed. So, let's create a section view so that we can see the entire boss. The first thing we need to do is to establish a cutting plane. We can do this by using a **polyline** to sketch it out. To activate the **polyline**, click on the **Design** pull-down menu and sketch the line shown below inside the elliptical circle. Our cutting plane is now established.

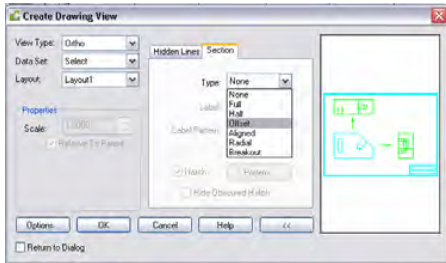


Activating the Polyline Command

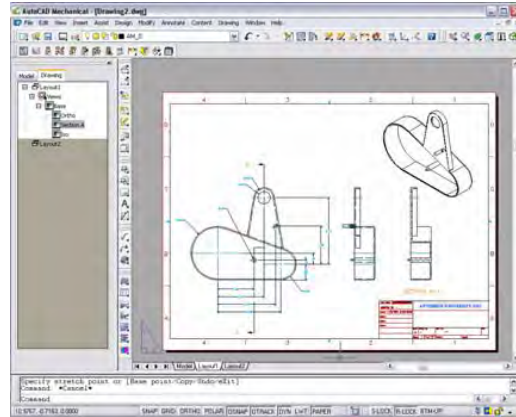
Polyline Sketch

Now let's create a section view. Right click on either the **Base view** in the **Browser** or the view itself and select **New View**. Once in the **Create Drawing View** dialog box, select **Ortho** as the **View Type** and click on the **Section** tab. In the **Section** tab select **Offset** as the **Type**. Next, specify the location of the section view, in our case, to the right of our ortho view. Next, select the polyline as our cut line. Our Section A-A is now established. Our drawing should look like the following figure. Task 2 is now complete. Let's now save the part, from the **File** pull-down by selecting **Save**.

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Activating Offset Section View Dialog

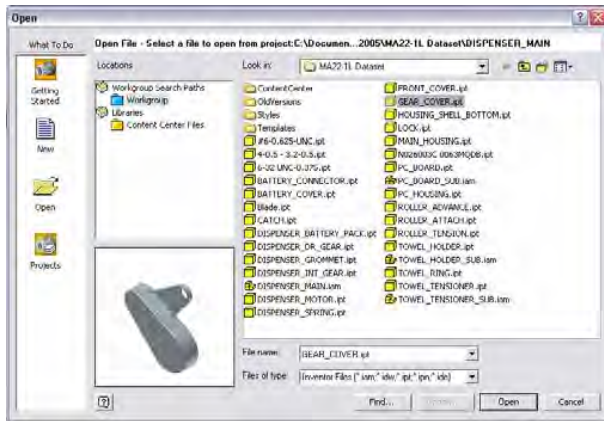


Drawing with New Section View A-A

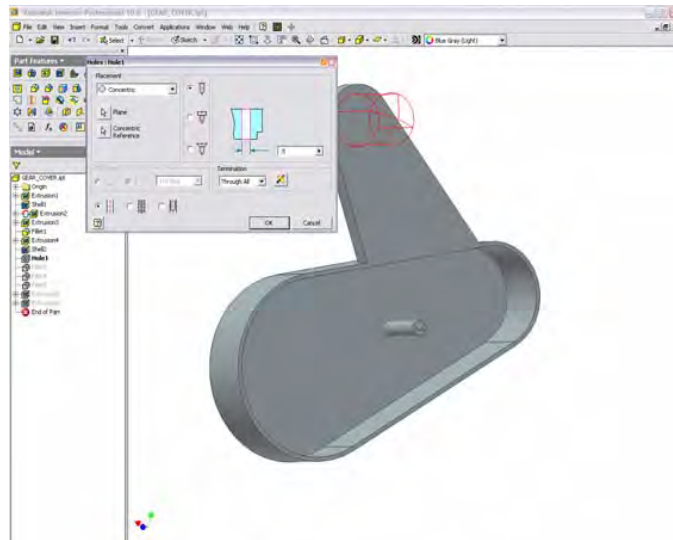
Task 3 Modifying the linked part in Inventor and addressing the modifications in Mechanical

We've just gotten word from our engineering department that a change needs to be made to the part, the diameter of the hole will change to **.50**. So let's fire up Inventor and make the change and see how it affects our drawing. Let's go to our windows explorer under **C:...../MA22-1L** and double click on the **DISPENSER_MAIN.ipj**. This launches a session of Inventor in the correct project.

We need to open the **GEAR_COVER.ipt**. Once we're in the part, we will right click on **Hole 1** in the **browser** and select **Edit Feature**. Once we're in the **Holes** dialog, we will change the value from **.42** to **.50** and then select **OK**. Again, let's save the part, from the **File** pull-down by selecting **Save**.



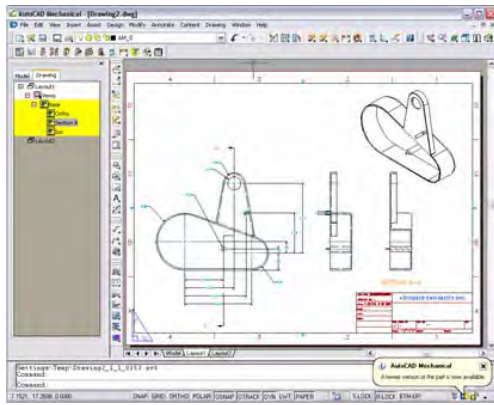
Selecting the GEAR_COVER.ipt



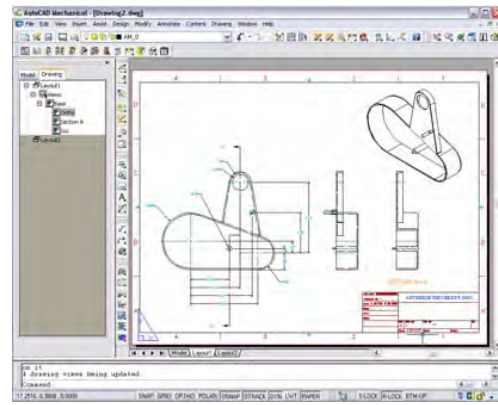
Changing the Hole Value

Toggle back to Mechanical. Did you notice that your browser now has a yellow highlight? (A red browser highlight would indicate that our .ipt document is missing). This is a visual clue that indicates our Inventor Link needs to be updated. We can update our link by simply right clicking in the **Browser** and selecting **Update**. Notice that our hole grew to the correct size and our browser is back to normal. Let's save our work, close the current drawing so that we can create a new link and detail our sub-assembly. Did you notice that our link between the **.dwg** file and the **.ipt** file is fully associative? That's the power of the Inventor Link! Task 3 is now complete.

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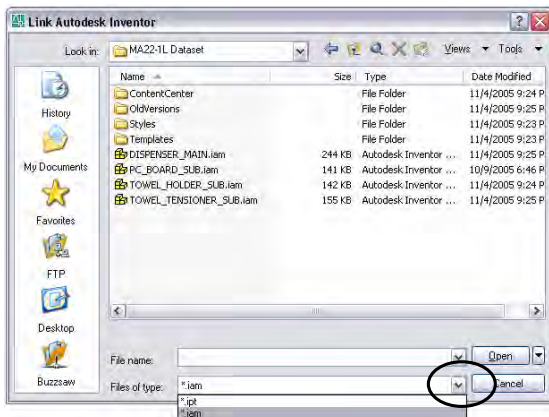
Before



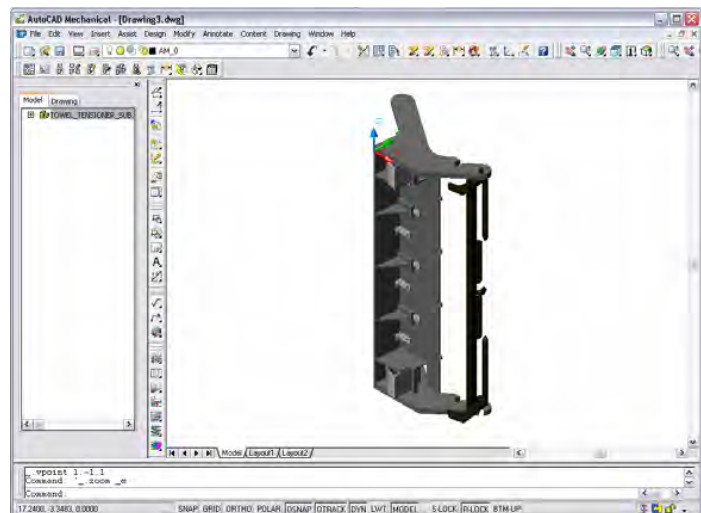
After

Task 4 Creating an Inventor Link for an assembly between Mechanical and Inventor

Let's start this task by creating our link to the assembly, in our case sub-assembly. We do this in the same fashion that we did for the part file. Use the **File** pull-down menu and select **Inventor Link**. Next, we will select the same template as before **AU2005-MA22-1L.dwt**. For our file, this time we will select an assembly .iam file for the file type. This can be accomplished by selecting the down arrow in the file type selection of our dialog box. Next select the file **TOWEL_TENSIONER.iam**, be patient, this could take up to a minute or so to process. As a side note, Mechanical will not link Inventor weldments or sheet metal parts, yet. Task 4 is now complete.



Assembly File Selection

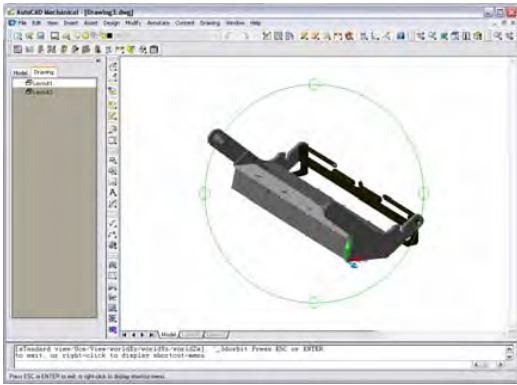


Your screen should now look like this figure

Task 5 Creating views, a parts list and ballooning the linked assembly

We are now ready to create some views for our sub-assembly. Let's click on the **Drawing** tab to enter the drawing environment. Once we're in the drawing environment, we are ready to create our first view. Access the **New View** tool by right clicking in the **Graphics Area** and selecting **New View**. We use a similar process as we did previously to create our **Base View**. We will accept the defaults in the **Create Drawing View** dialog and Mechanical takes us over to model space. We will select the front face of our tensioner sub-assembly; the problem is that we can't see it in its current orientation. So, still within the View command, again, select the **3D Orbit** button and rotate the model as shown below. Next, let's select the front face as shown in the next figure for our **Planar Face**. Next, we will select the bottom edge as of the tensioner as the **Straight Edge** as shown.

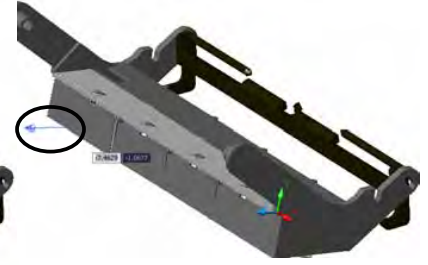
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3D Orbit

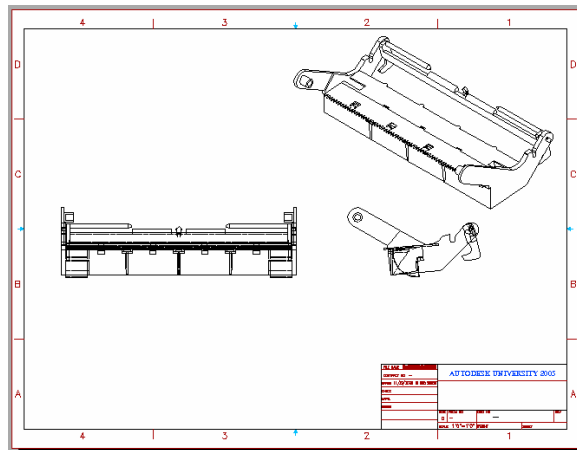


Planar Face



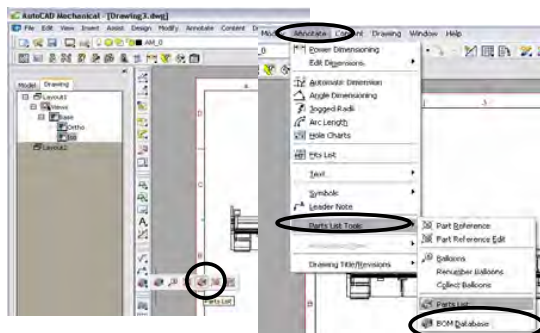
Straight Edge

Let's place the center of the view in **Zone C3** of the drawing as shown. The view could take up to a minute to process. Create a side view and an Isometric view as we did previously for our part. If you get stuck, refer back to pages 3 and 4 to refresh your memory. Your drawing should now look like the following figure. Let's save our work.



****Your drawing should now look like this figure****

Let's finish our sub-assembly drawing by accessing and modifying the **BOM Database**. Let's also create a parts list and call out our parts on our sub-assembly with balloons. First let's access our **BOM Database**. This is the first button on our **AM:Annotation** toolbar fly-out next to our browser. It can also be activated from our **Annotate** pull-down menu under **Parts List Tools** then **BOM Database**. Once in the **BOM Database**, we need to select the plus sign to expand our **TOWEL_TENSIONER** sub-assembly. This is done so that we can display our parts list properly. Let's **OK** out of the dialog and move on to our Parts List placement.

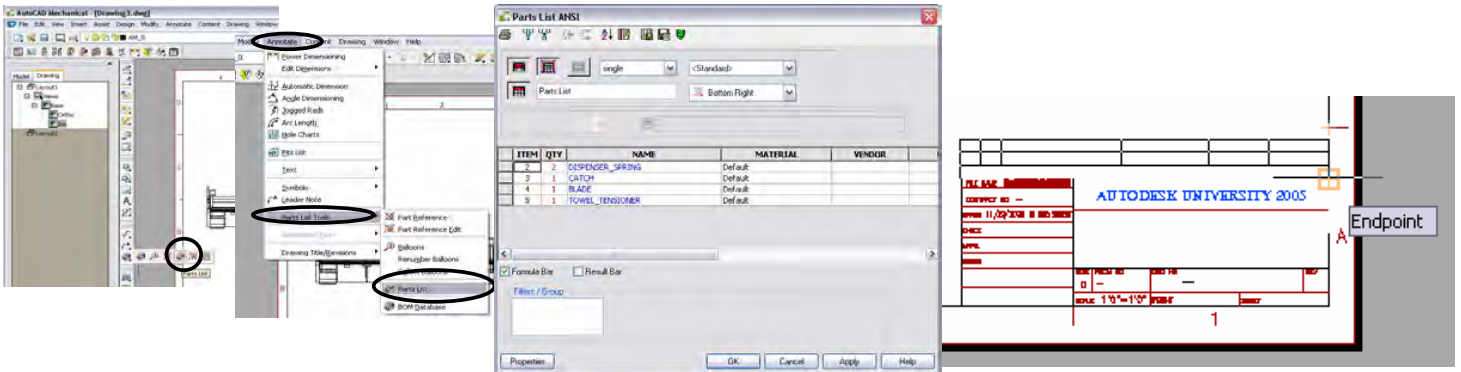


Activating the BOM Database



Expanding the BOM

The Parts List can be accessed from the fourth button on our **AM:Annotation** toolbar fly-out next to our browser or it can also be activated from our **Annotate** pull-down menu under **Parts List Tools**. The Parts List dialog looks much like the BOM Database dialog. Let's OK out of that and place the parts list as shown in the following figure.

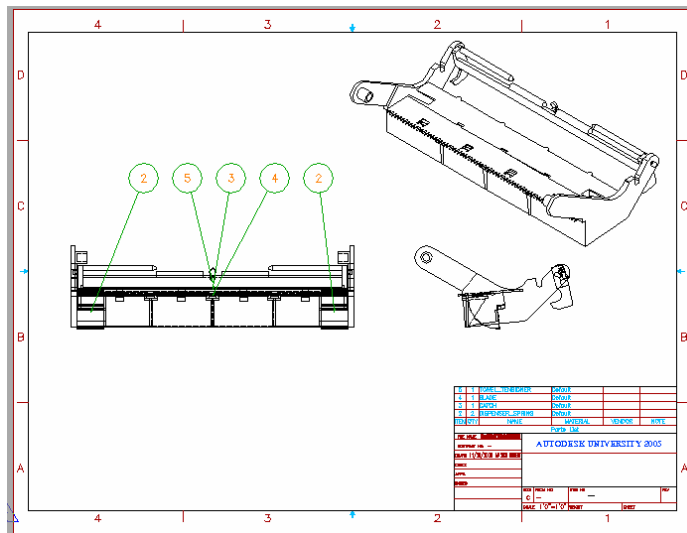


Activating the Parts List

Parts List Dialog

Parts List Placement

Let's go ahead and balloon our sub-assembly. The balloon command can be accessed from the same location as the BOM Database and Parts List. Once the Balloon command is activated, type "a" in the **command line** for the **AutoAll** option. Next, let's select each part reference on our base view and place the balloons as shown in the following figure. Part references are icons with a red circle and an X.

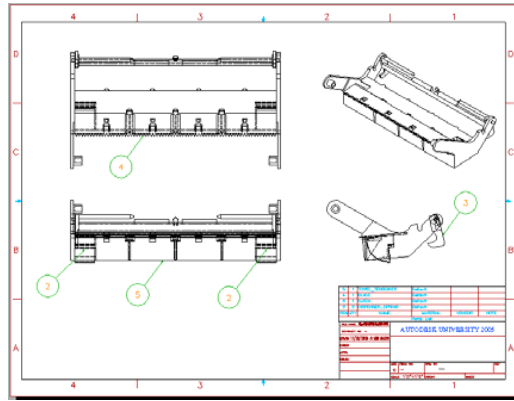


Part Reference

****Your drawing should now look like this figure****

It looks as though items 3 and 4 can be better represented in different views. First, we will need to add a top view. So let's create an **Ortho** view from the **Base** view. Next, we can move our **Base** view slightly down. Next, let's **power erase** items 3, 4 and 5 and re-balloon the items as shown below. We can call out our parts by activating the balloon command and simply click on the part to be called out. Finally, let's bump the scale down on our **Iso** view. This can be done with a right click on our iso view and change the scale to **.75**. Our drawing should look like the image below. Congratulations, we have completed our tasks and the engineering department is happy!

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****Your drawing should now look like this figure****

A quick checklist to ensure success with the Inventor Link

1. Make sure Mechanical is set to the desired Inventor project using the command `amivproject`.
2. Make sure the proper company standard template is used to enforce company standard consistency and time savings.
3. Keep your sub-assemblies small and manageable.
4. Do not use weldments or sheet metal parts; they don't link to Mechanical, yet.
5. Browser highlight color coding:
 - a. RED – document is missing the .ipt. (Inventor Part File Extension)
 - b. YELLOW – the Inventor links needs an update.
6. Make all model changes in Inventor since model changes to an .ipt are not allowed in Mechanical
7. The Inventor Link extracts Parametric Dimensions, iProperties and Thread and Hole annotation information.
8. If the .ipt or .iam (Inventor Assembly File Extension) does not contain nominal dimensions, parametric dimensions will not be imported. Since parametric dimensions are obtained from the .ipt or .iam., modifications to such dimensions are not allowed.
9. Following are some possible reasons for an Inventor Link file not to load properly
 - a. The Autodesk Inventor Companion library files are not installed, missing or out-of-date. The library files can go out of date if you install an older version of Mechanical Desktop or Autodesk Inventor after this version was installed. In either case, reinstall this version of AutoCAD Mechanical / Mechanical Desktop, with Autodesk Inventor Companion support enabled.
 - b. An incompatible version of Autodesk Inventor is currently running on your computer. In such a case, close Autodesk Inventor and reload the companion file.
 - c. You have two versions of Autodesk Inventor installed on your computer and the last run version is not compatible with this version of Autodesk Inventor Companion. In such a case, start the latest version of Autodesk Inventor and reload the companion file.
 - d. The Autodesk Inventor Companion part color is the same as the background color of model space. Change model space background color and the loaded part will be visible.
 - e. The chart on the following page is a handy chart for your reference found in Mechanical's help for when problems are encountered in your linking process.

We hope you have found this lab useful and that hope it contributes to your ongoing success!

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AutoCAD Mechanical's Inventor Link Message Chart

Message	Why are you seeing this message?	What you can do
IVL001	Feature data from derived components is not available in AutoCAD Mechanical/Mechanical Desktop. As a result, AMNOTE cannot pick up feature data automatically.	Check with the designer of the Autodesk Inventor Document for feature data.
IVL002	You have selected a template that has options specific to AutoCAD Mechanical mechanical structure or MDT components turned on.	Use a template that does not have these incompatible options turned on.
IVL003	Only one session of Autodesk Inventor can run at a given time. The version that is currently running is not compatible with the version of AutoCAD Mechanical/Mechanical Desktop you are running.	Close Autodesk Inventor. If the message persists, use task manager to check for sessions that have not terminated completely and end them.
IVL004	The file you are trying to connect to may not be a valid Autodesk Inventor document.	Use Autodesk Inventor to verify the status of the file.
IVL005	AutoCAD Mechanical/Mechanical Desktop has no equivalent for MAX tolerances. Dimensions set to MAX tolerance will be treated as dimensions with no tolerance.	Contact the designer of the Autodesk Inventor document for tolerance values. If appropriate, you may override tolerance values from AutoCAD Mechanical/Mechanical Desktop.
IVL006	AutoCAD Mechanical/Mechanical Desktop has no equivalent for MIN tolerances. Dimensions set to MIN tolerance will be treated as dimensions with no tolerance.	Contact the designer of the Autodesk Inventor document for tolerance values. If appropriate, you may override tolerance values from AutoCAD Mechanical/Mechanical Desktop.
IVL007	Some parts do not contain nominal dimensions and they may not be drawn to scale.	Contact the designer of the Autodesk Inventor document to identify the parts containing non-nominal dimensions. If appropriate, request for a file with nominal dimensions.
IVL008	All Autodesk Inventor documents must be closed when the active project file is changed.	If Autodesk Inventor is currently running, close the application. If an Autodesk Inventor linked drawing is open, close that as well. If the message persists, use task manager to check for sessions that have not terminated completely and end them.
IVL009	Autodesk Inventor link support files may not exist or may have been overwritten by a subsequent installation.	Install a compatible version of Autodesk Inventor or install Autodesk Inventor link support files.
IVL010	The Autodesk Inventor link does not support assembly features. The document you attempted to link to contained assembly features and hence could not be opened.	
IVL011	The Autodesk Inventor link is unable to find the part files for the assembly in the last known location. AutoCAD Mechanical/Mechanical Desktop can locate part files using the project (.ipj) file corresponding to the assembly file.	Use the AMIVPROJECT command to set the correct .ipj file or copy all the inventor files to the folder containing the linked drawing.
IVL012	The unit type used for the standard/template you chose to create the linked drawing is different to the unit type used to create the Autodesk Inventor document.	Create a new linked drawing using a standard with the correct unit type.
IVL013	One of the file you are linking to contains virtual components. The Autodesk Inventor link does not support virtual components.	Click OK and proceed. An Autodesk Inventor linked drawing will be created, but the drawing will not contain any virtual components.

