CHAPTER 2

Creating 2D Drawings from 3D Data

When you've never used Autodesk Inventor, it is very easy to focus entirely on the three-dimensional (3D) design tools, but many people still need to produce two-dimensional (2D) production drawings.

Chances are, if you're reading this book, you have created detail drawings and will likely need to create them in the future. Creating detail drawings from the solid (or surface) model is so easy it can often be thought of as fun.

To create a new drawing in Inventor, it is not necessary to have any 3D file open. In fact, if your system resources are limited, it is best not to have the file open.

- **Drawing views of a part**
- **Editing views**
- **Adding detail to drawing views**
- **Dimensioning**

Drawing Views of a Part

When you've created a drawing view, it must accurately reflect the model geometry, and if there is a change to the geometry, it needs to be reflected in all drawing views. This is just standard drafting practice, and the way Inventor works as well, but you may notice things happen a bit more quickly than on the drawing board.

This section introduces the types of drawing views and then proceeds through a series of exercises creating and working with views.

Types of Drawing Views

Certification Objective

Any type of drawing view that you've ever created on the drafting board or in a 2D CAD system can be created with Inventor. It's just a lot faster in Inventor because you're generating views of the object(s) rather than generating a lot of separate pieces of geometry to represent the same edge or feature.

The Base View If you think about how you currently do drawing layouts, you are already working with a hierarchy, whether it's conscious or not. You have a view that others are positioned around. In Inventor, this is referred to as the *base view*, and there can be as many as you need or as many components as you like, all on one page.

Projected Views Once a base view is placed, you can immediately place standard orthographic projections and isometric views from that base view. Any change that is made to the scale, position, or contents of the base view will be reflected in the projected view by default.

Section Views These are placed much like projected views but with a section line defining what geometry will be generated. The section line can be multisegmented or include arcs.

Auxiliary View A specialized view requires the selection of an existing drawing view edge to project around to display the true shape of a face.

Detail View This is another type of view that is hard to make in a 2D system, especially if the geometry may change. For this view, you can use round or rectangular boundaries with smooth or rough edges for the view. The detail view can also be created with any scale.

A section view can be made to any depth or even defined outside a part to create a custom view.

Starting a New Drawing

To have a view, you must first have a drawing. Beginning a new drawing is just like creating any other file in Inventor; click a template and get to work.

1. On the Quick Access toolbar, click the New icon.
2. Make sure that `2012 Essentials.ipj` is listed as the active project file in the dialog box.
3. In the New File dialog box, select the Metric tab (see Figure 2.1).
4. Locate and double-click the `ANSI (mm).idw` template file.

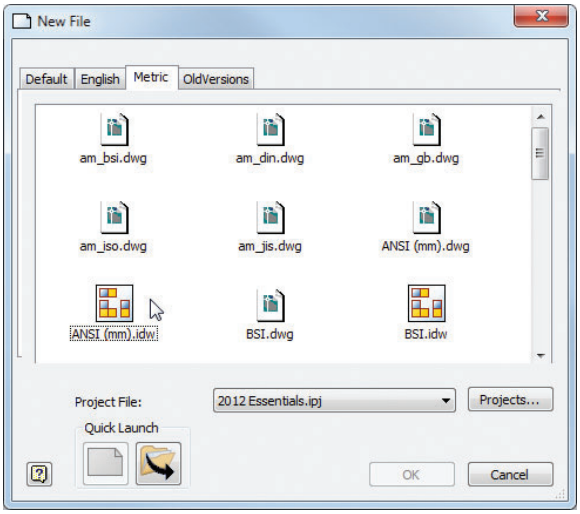


FIGURE 2.1 Templates are organized by model units.

This will open a new drawing page in the Design window and update the Ribbon to present the Place Views tab and its tools.

Placing the Base and Projected Views

Now you will begin the process of creating your drawing. For this exercise, you will use the marking menu tools.

Certification Objective

1. Right-click in the Design window, and move up to select the Base View tool; or, right-click and drag up quickly to activate the tool.

2. In the dialog, click the icon to select an existing file at the end of the File drop-down.
3. Using the same Open dialog you used in Chapter 1, use the parts shortcut in the upper left; then open the Parts\`Chapter 2` folder, and select the `30 Degree Adapter` part file.

   If you move your cursor over the drawing page, a preview will appear, showing how the component will be sized and positioned on the page.
4. In the upper right, click a few different options in the Orientation field and see the effect each one has on the preview. Make sure you end up back at the Front view.

When you have 3D files open, they will automatically appear in the File drop-down list.

5. In the lower left of the dialog, set the view scale to .7 and see the effect on the preview.

   At the bottom of many dialogs is an arrow icon. Clicking this will cause the dialog to roll up to just its title bar. If a dialog is in your way, use this option to make it easier to see your screen. Moving your cursor near the dialog will expand it to its full size, and reselecting the arrow will hold the dialog open.

6. Position the view similar to Figure 2.2, and click the mouse to place the drawing view.

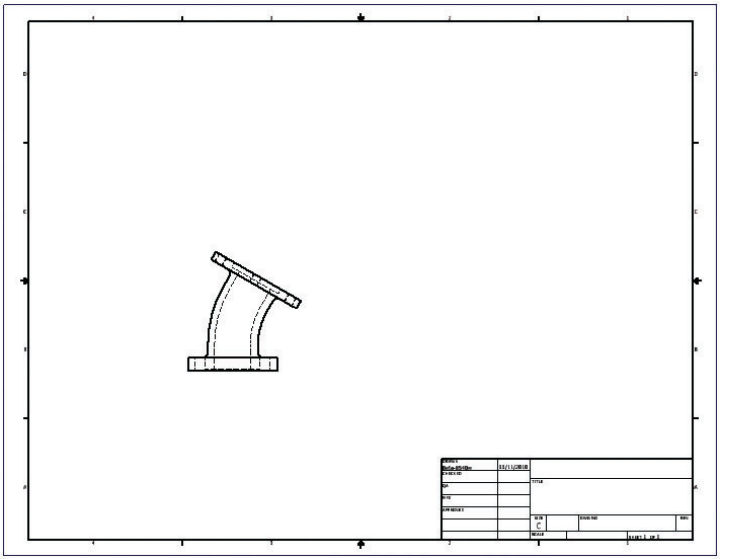


FIGURE 2.2 Placing the base view of the adapter

7. Now move your cursor down and to the right and to the upper-right corner, and click the mouse to place the views that are shown in Figure 2.3. You must right-click and select Create from the context menu to generate the views.

The ability to quickly place the base view and the projected views in one step is a great way to quickly see how the drawing will lay out.

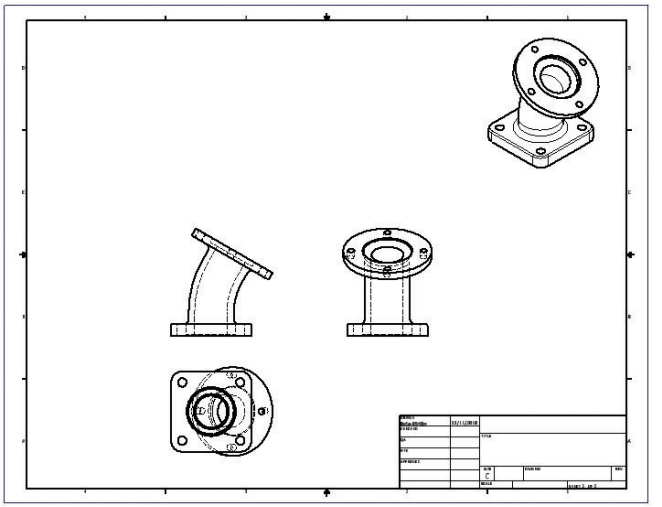


FIGURE 2.3 Locating the bottom, side, and isometric views

Placing a Section View

Certification Objective

Section views are very important for detailing machine parts and assemblies. In this exercise, you will see how easy they are to create:

1. Open `c2-02.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Start the Section tool from the Create panel on the Place Views tab.
3. Click the projected view to the right of the base view to choose it as the parent of the section view.
4. As you move over and around the parent view, you will see dotted lines that show alignment to view geometry. When an inference line appears from the center of the top face (see Figure 2.4), click to start creating a section line.

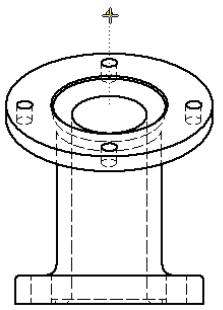


FIGURE 2.4 Inference lines will allow the section to be connected to view geometry.

5. Drag the line down through the part to define the section line, and click a second point to start creating the section view. See Figure 2.5. Right-click and select Continue from the context menu to finish defining the section line.

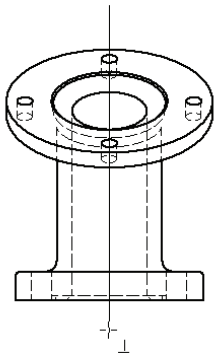


FIGURE 2.5 Include some overlap of the view in the section line length.

6. The Section View dialog will appear, allowing you to change the scale and view identifier. Move your mouse to the right to see a preview of the view. See Figure 2.6.

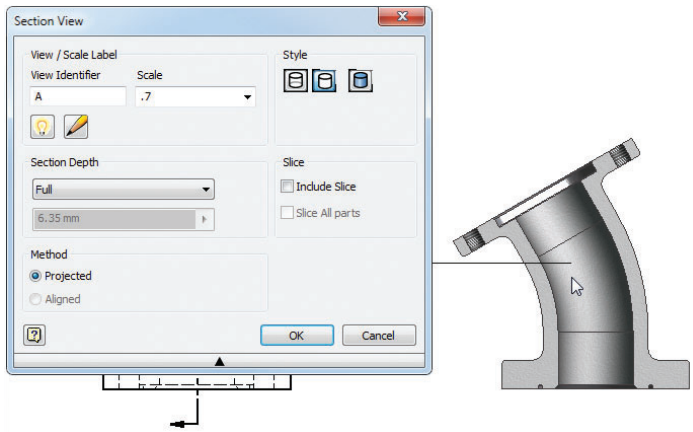


FIGURE 2.6 Placing the section view

7. Click the location for the view to generate the view. Figure 2.7 shows the result.

It is possible to define a section view of nearly any existing drawing view. Auxiliary views are very similar to section views but are defined by model geometry.

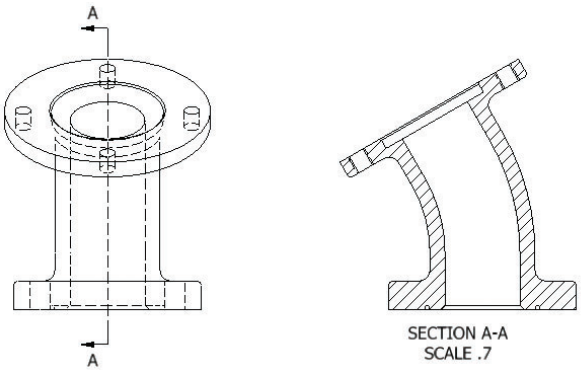


FIGURE 2.7 The finished section view

Creating an Auxiliary View

Auxiliary views can be a challenge to define accurately with a 2D CAD system. In this case, you want to more easily detail the round face of the part, which does not align with a standard orthographic projection.

1. Open `c2-03.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Move your cursor near the base view, and when it highlights, right-click to bring up the marking menu and context menu with other tools.
3. Select Create View ➢ Auxiliary View from the context menu.
4. After the Auxiliary View dialog appears, click the angled edge on the top of the base view, as shown in Figure 2.8.

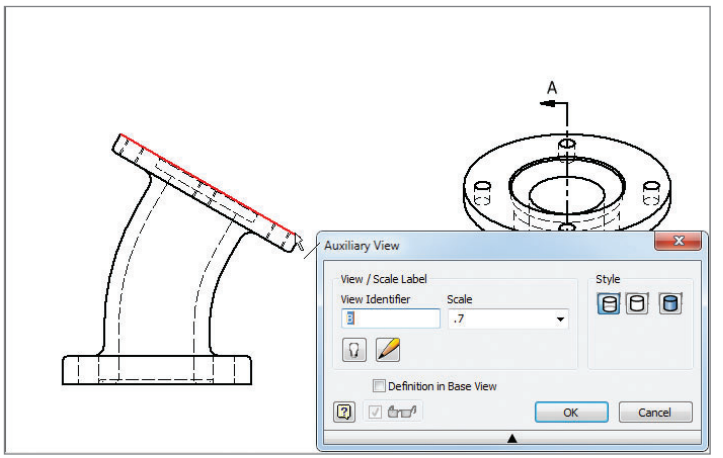


FIGURE 2.8 Selecting an edge for the projection reference

5. Click to place your view, as shown in Figure 2.9.

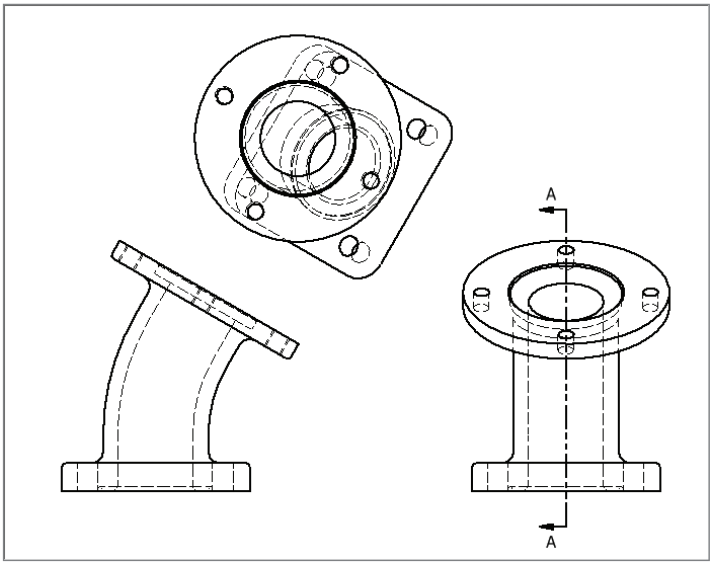


FIGURE 2.9 An easy auxiliary view

Creating a Detail View

The auxiliary view offers more clarity, but the section view offers an opportunity to see things otherwise obscured in an orthographic view. To do so, you'll create a detail view to add clarity:

1. Open `c2-04.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Start the Detail tool from the Ribbon or from the context menu.
3. Click the section view as the parent, which will open the Detail View dialog.
4. Set Scale to 2 and Fence Shape to Rectangular; then click near the O-ring groove, and drag the fence, as shown in Figure 2.10.
5. Drag the new view to any open spot on the page, and click to place it.

Now that you have a collection of views, you should make the drawing look better as a whole. By moving, rotating, or even changing the appearance of the views, you can make a big difference.

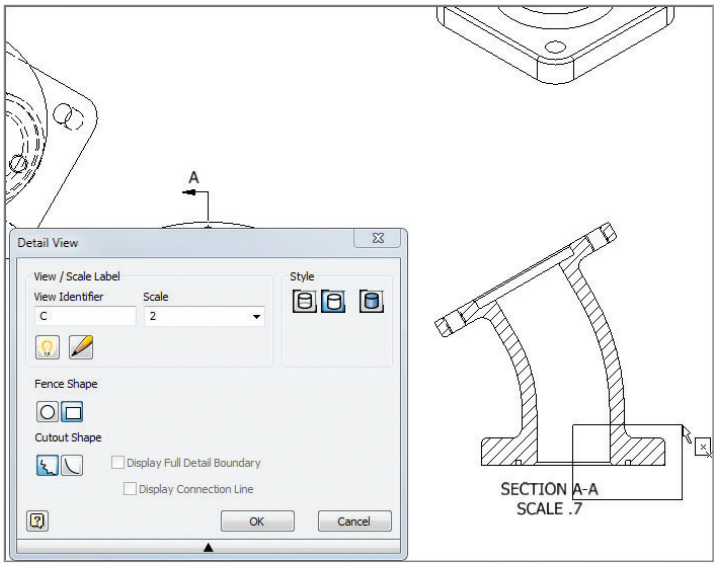


FIGURE 2.10 Setting up the focus of the detail view

Editing Views

The detail drawing is an important communication tool, so it is equally important to make it as clear as possible. In this section, you will focus on tools that will help you put the finishing touches on your drawings.

View Alignment

When views are created from other views, they inherit the appearance and scale of the parent view. Projected, auxiliary, and section views also inherit alignment from the parent view. Using this alignment and sometimes breaking the alignment make it simple to reorganize the drawing.

The simplest way to add clarity to a drawing is to shuffle the position of the drawing views. Doing this is a simple drag-and-drop operation.

1. Open `c2-05.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Click and drag the base view up and to the left. In Figure 2.11, you can see how the views projected from it and their children keep the alignment of orthographic projection.

You can even move drawing views to the opposite side from where they were created.

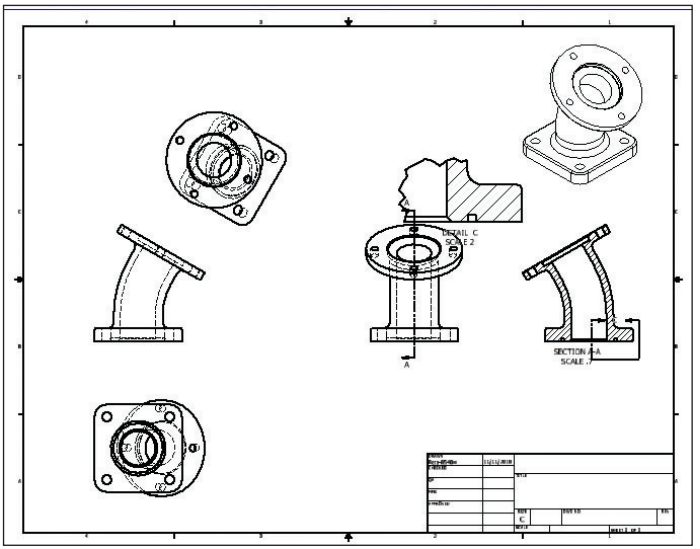


FIGURE 2.11 Moving a view does not break its proper alignment with others.

3. Move the right side view and the section views to give more space to them. Notice that they remain in alignment with the base view but do not change its position.
4. Move the detail view to the left of the title block, as shown in Figure 2.12.

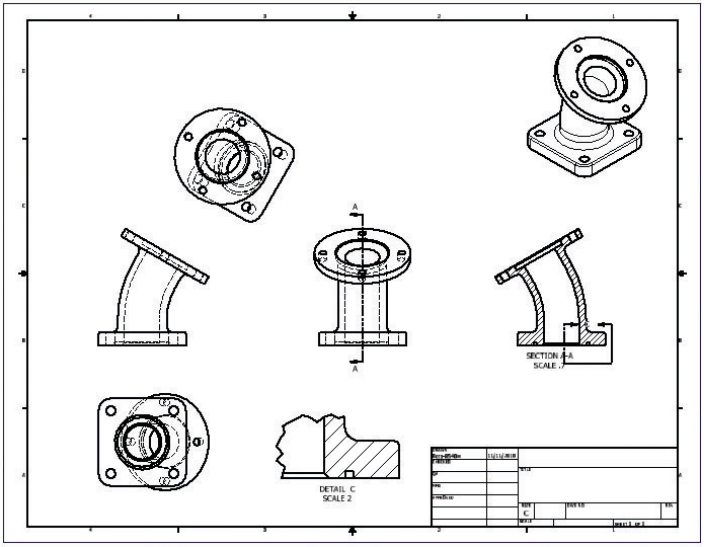


FIGURE 2.12 The detail view is free to be moved anywhere.

Moving views goes a long way toward making things look better. Sometimes, you may not want to maintain view alignment, or there may be times where it impedes the detailing of geometry.

Changing Alignment

It is possible to edit a view and break its alignment to a parent view using simple right-click options or using the tools in the Modify panel. You can also add alignment between views using tools from the same menus.

For example, to change the location of the auxiliary view, you need to break its alignment with the base view. To make it easier to dimension, you may want to rotate it as well. The Rotate tool will do both steps for you.

1. Open `c2-06.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Click the auxiliary view, right-click, and select Rotate from the context menu.
3. In the Rotate View dialog, keep the rotation method By Edge, but set the alignment to Vertical and check that the direction is set to clockwise.
4. Click the straight edge on the right side of the view per Figure 2.13.

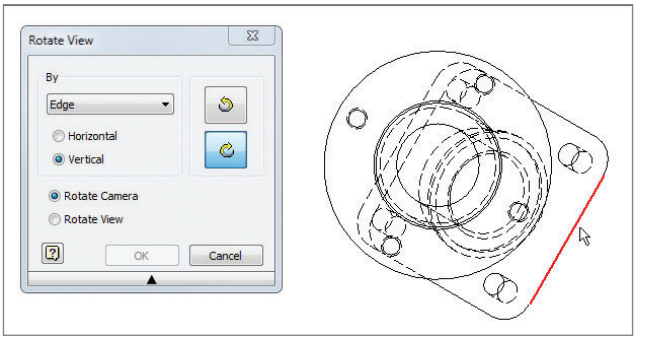


FIGURE 2.13 Rotating the drawing view can make detailing easier.

5. Click OK after the part has been rotated.

Now that the view has been rotated, you can detail it. But you may also want to reestablish an alignment with the base view to help others understand the geometry.

This exercise shows an uncommon and specialized drawing practice. It is built around the classic drafting idea of making it easier for people to recognize the geometry.

1. Open `c2-07.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. In the Modify panel, expand the options under Break Alignment, and select Vertical.
3. Click the rotated auxiliary view and then the base view. See Figure 2.14 for the results.

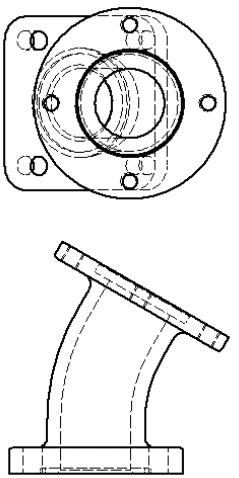


FIGURE 2.14 Views can be aligned to other views that were not originally their parent.

Now that you think you have enough room between the views and they are aligned, you will make some of them easier to interpret.

View Appearance

Certification Objective

Changing how a view looks or is scaled can have a dramatic effect on how easy it is to understand.

You can use a number of techniques, including removing hidden lines, adding shading, changing scale, or turning the visibility of selected entities off altogether. Any or a combination of these can be done without changing the accuracy of the view.

A quick way to add clarity is to remove hidden lines when they're not needed to understand the geometry of the component. In this exercise, you'll also add shading.

1. Open `c2-08.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Double-click in the auxiliary view (upper left) to open the Drawing View dialog.

3. Deselect the Style From Base check box.

4. To the left of the Style From Base icon, click the Hidden Line Removed icon, and click OK to close the dialog and apply the change.
5. Double-click the isometric view to edit it.

6. Click the Shaded icon.
7. Select the Display Options tab in the dialog, and then remove the check mark from Tangent Edges.
8. Change the scale of the view to .5.
9. Click OK to update the drawing. See Figure 2.15.

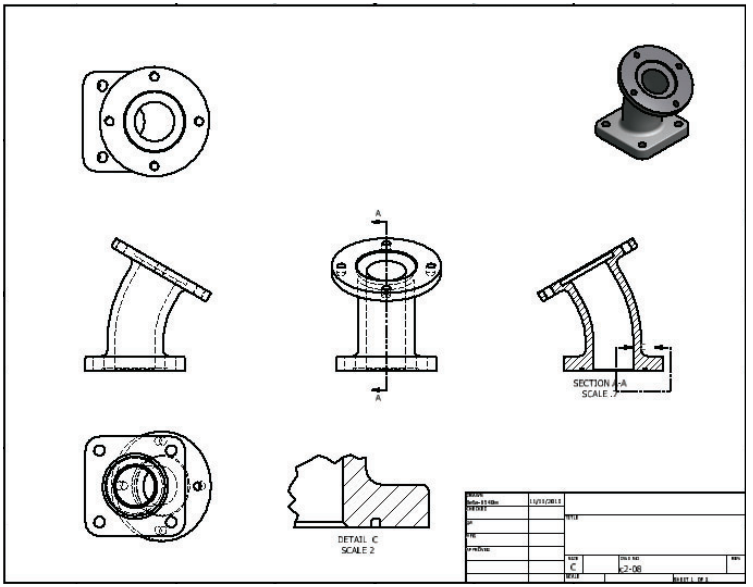


FIGURE 2.15 Updated drawing views

Removing unneeded hidden lines and adding shading can make the drawing easier to understand without adding more views. You could continue to make adjustments to the views for hours. Let's instead turn our attention to detailing the views you have.

Adding Detail to Drawing Views

There are many things that you add to a drawing to make it complete. Much of the work is tedious but necessary. Reducing the time it takes to add all of the detail is one of the unsung strengths of Inventor. In the next several pages, you will see some great examples of the productivity tools for detailing and dimensioning.

Most of the tools you will be using are located on the Annotate tab. This is one time where Inventor doesn't automatically change the active tab, because everyone uses the detailing tools differently.

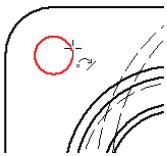
Center Marks and Centerlines

You often need to add geometry to the drawing views to assist in locating dimensions. The next several exercises will walk you through the most commonly used tools for this purpose, center marks and centerlines.

Center marks make it easy to locate the center of a hole or radius.

1. Open `c2-09.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Click the Annotate tab in the Ribbon to change your active tab and see the Annotation tools.
3. Zoom in on the bottom view in the lower left of the drawing.

4. Click the Center Mark tool on the Symbols panel of the Annotate tab.
5. Move your cursor near the edge of one of the holes. When it highlights, click your mouse to place a center mark.



6. Repeat to place center marks on all four holes, as shown in Figure 2.16.

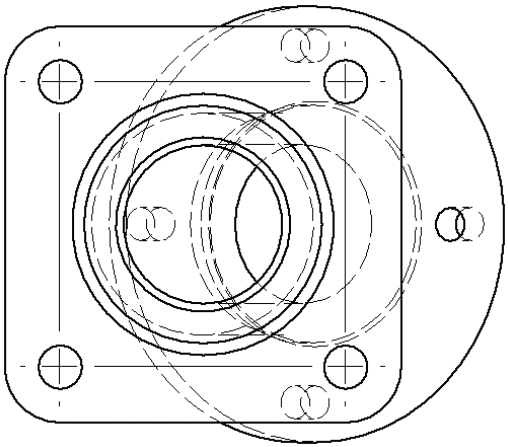
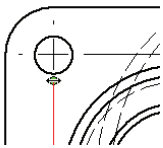


FIGURE 2.16 Placing center marks on holes by simply picking them

7. To connect the center marks, drag the center mark extensions toward each other.



Now that you've added and edited center marks for a basic rectangular hole pattern, you can now move on to bolt circles.

Center marks and centerlines have many uses in Inventor. The ability to easily make them and drag-edit the extents of these marks helps the user to save time cleaning up the drawing. They can also be used to show linear alignments between holes or even bolt circles.

1. Open `c2-10.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Zoom in on the rotated auxiliary view in the upper-left corner.
3. Select the Centerline tool from the Symbols panel of the Annotation tab.

You can also easily create a series of centerlines in an arc based on a center.

4. Begin by selecting the hole at 12 o'clock of the round face; then click the holes at 3 o'clock, 6 o'clock, and 9 o'clock, as shown in Figure 2.17.

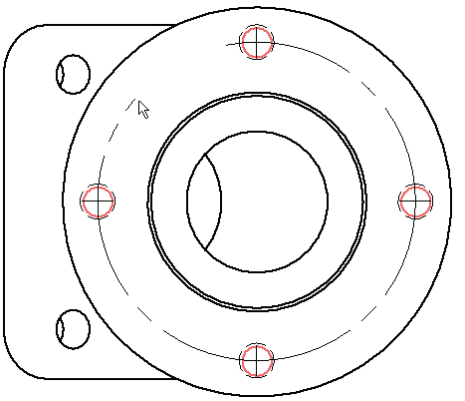


FIGURE 2.17 As you place the center marks for the bolt circle, the centerline previews.

5. Finish the Bolt circle by clicking the 12 o'clock hole again; then right-click and select Create from the context menu. See Figure 2.18.

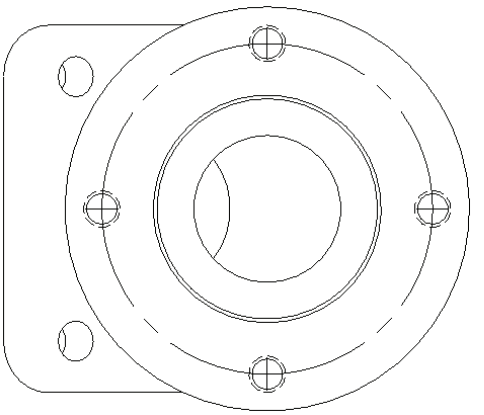
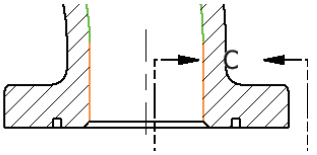


FIGURE 2.18 Generating a bolt circle with the Centerline tool

6. Press the Esc key to leave the Centerline tool.

You can use the Centerline Bisector tool on straight or curved segments to find the midline between them. Straight segments do not need to be parallel.

1. Open `c2-11.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Zoom in on the section view on the right side of the drawing.
3. Select the Centerline Bisector tool on the Symbols panel of the Annotation tab.
4. Click the Orange lines in the lower left and lower right of the open portion of the part to place a straight segment.



5. Click the green arcs on the left and right to place a curved segment.
6. Place another straight segment by picking the blue angled line at the top left and top right.
7. You can also extend the straight segments beyond their original size for clarity. See Figure 2.19.

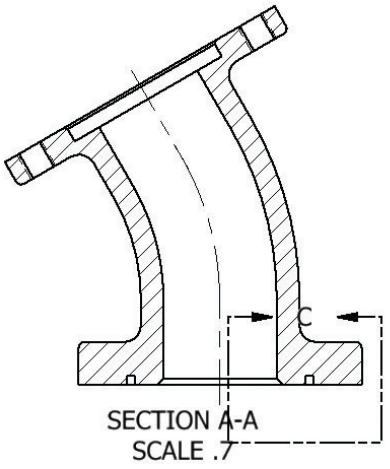


FIGURE 2.19 Placing a complex centerline bisector

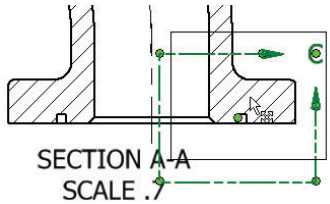
While you are looking at the section view, you should improve its appearance.

Editing a Detail View Placement and Callout

Previously, you moved views around to improve the layout of the drawing. Callouts, section lines, and detail boundaries can also be edited to improve their clarity.

1. Open `c2-12.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Zoom in on the section view.
3. Select the letter *C* in the detail view boundary, and drag it to where the upper-right corner is now.

   In addition to relocating the callout, you can change the letter by editing the text to read whatever you like.
4. Click the boundary. When the corners and center highlight, drag the center upward and to the right a short distance.



5. Select the *SECTION A-A* text, and drag it below the view so it is legible. Figure 2.20 shows the result.

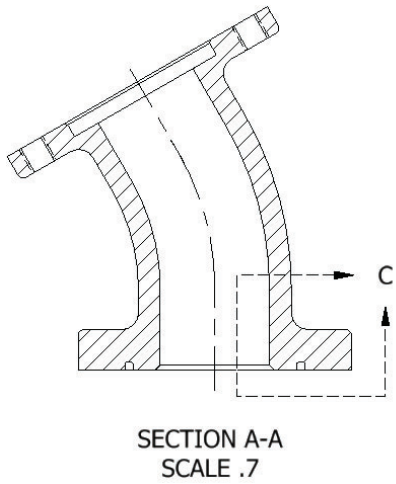


FIGURE 2.20 Relocate the section view label for a better drawing view.

Now that the views have been updated with new or repositioned elements, you should turn your attention to dimensioning.

Dimensioning

If you've used traditional 2D CAD tools in the past for drafting, then the tools Inventor uses to apply dimensions should seem familiar but different at the same time. Rather than offer the user a broad array of dimensioning tools, Inventor uses intelligence that changes the type of dimension based on the selected geometry. Add the ability to change many options while you are placing the dimension, and you are sure to appreciate the power available to you.

The General Dimension Tool

To place individual dimensions in the drawing, the General Dimension tool will give you most anything you will need. For horizontal, vertical, aligned, radial, and diameter dimensions, you can just use this tool.

1. Open `c2-13.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Zoom in on the bottom view in the lower left of the drawing.

3. From the Dimension panel of the Annotate tab, select the Dimension tool.
4. Click the center marks in the lower- and upper-right corners (Figure 2.21).

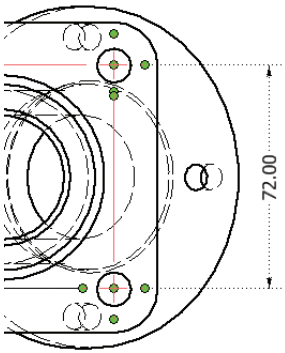


FIGURE 2.21 Click the center marks or other geometry to place dimensions.

5. Move your cursor to the right. You will see the dimension change to dotted lines when you have good spacing away from the geometry.
6. Click to place the dimension.
7. When the Edit Dimension dialog appears, deselect the Edit Dimension When Created check box, and click OK to close the dialog.
8. The General Dimension tool is still running. Click the radius in the lower-right corner, and notice that you are now creating a radial dimension. This dimension will snap at common angles.
9. Click to place the new dimension (Figure 2.22).

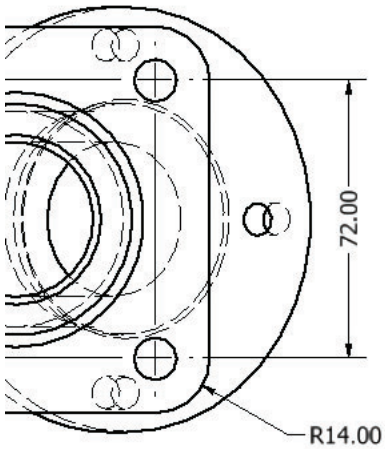


FIGURE 2.22 Adding a radial dimension

10. Pan up to the base view.
11. Click the upper edge of the base of the part and then the lower edge of the angle face. Notice that the dimension now turns to an angular dimension.
12. Place the new dimension to the side, as shown in Figure 2.23.
13. Now select the top edge of the angle portion.
14. Place the linear diameter above the drawing view (Figure 2.24).

As you see, the General Dimension tool is incredibly flexible.

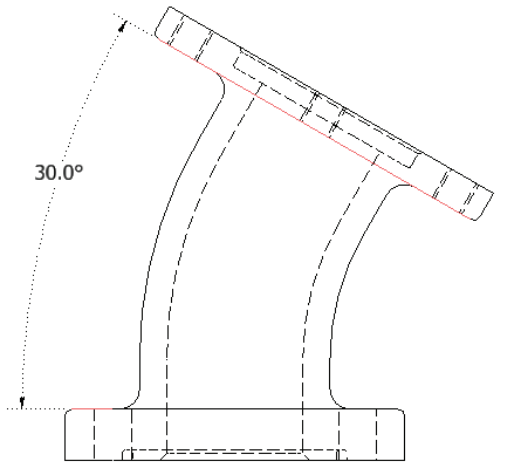


FIGURE 2.23 Even angular dimensions can be placed with the General Dimension tool.

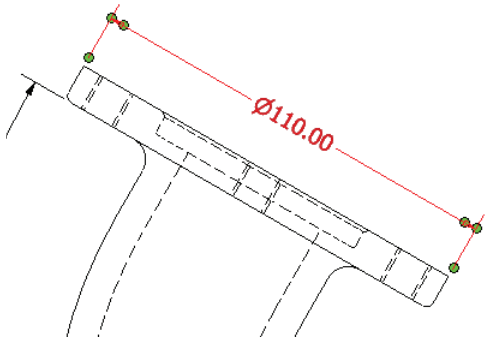


FIGURE 2.24 The geometry selected determines the type of dimension placed.

The Baseline and Baseline Set Dimension Tools

When you need to place several dimensions from the same datum, you can use the baseline dimensioning tools, Baseline and Baseline Set. The difference between the two is that Baseline Set keeps the placed dimensions' styles and precision linked along with other variables.

1. Open `c2-14.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.

2. Zoom in on the bottom view in the lower left of the drawing.
3. From the Dimension panel of the Annotate tab, expand the Baseline tool, and select Baseline Set.

   The tool will start immediately and wait for you to select the geometry.
4. Click the left edge of the square face, the top-left hole, the top-right hole, and the right edge.
5. Right-click and select Continue from the context menu to stop selecting geometry and begin placing dimensions.
6. The preview of the dimensions will appear. Move them above the view.
7. Click to place the dimensions, and then right-click and select Create from the context menu to finish. See Figure 2.25.

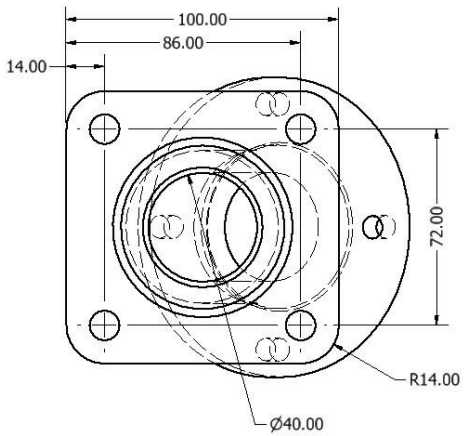


FIGURE 2.25 Placing multiple dimensions with the Baseline Dimension tool

The Chain and Chain Set Dimension Tools

Used commonly in architecture and when part cost is more important than high precision, the Chain dimensioning tools work very much like the Baseline tools but place dimensions in a series rather than from a common datum.

1. Open `c2-15.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.

2. Zoom in on the bottom view in the lower left of the drawing.
3. From the Dimension panel of the Annotate tab, select the Chain Dimension tool.

   The tool will start immediately and wait for you to select the geometry.
4. Click the top edge of the square face, the top-left hole, the bottom-left hole, and the bottom edge.
5. Right-click and select Continue from the context menu to stop selecting geometry and begin placing dimensions.
6. The preview of the dimensions will appear. Move them to the left of the view.
7. Click to place the dimensions; then right-click and select Create from the context menu to finish. See Figure 2.26.

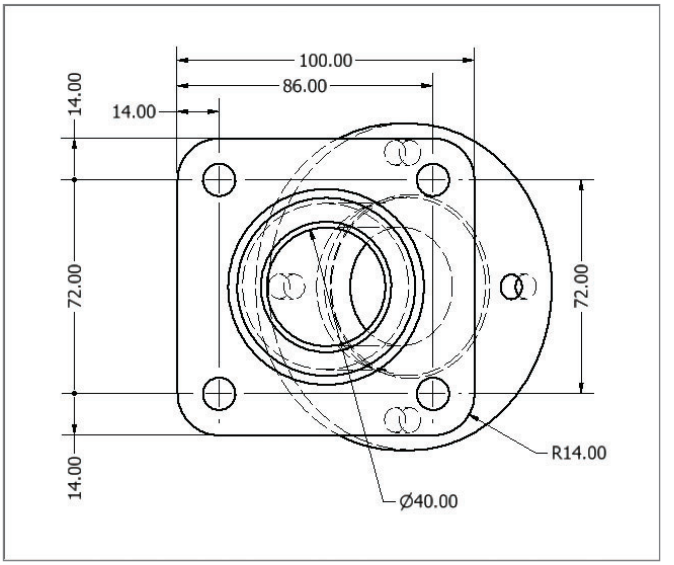


FIGURE 2.26 Chain dimensioning is much easier with a specific tool.

The Ordinate and Ordinate Set Dimension Tools

The Ordinate dimensioning tools work much like the Baseline tools but place the dimension value at the end of the extension line, as shown in Figure 2.27.

The difference between the two is that the Ordinate Dimension tool prompts for the placement of a datum point in the view. The Ordinate Set dimension tool does not, and it therefore has a workflow that is essentially the same as that of the Baseline Dimension tool.

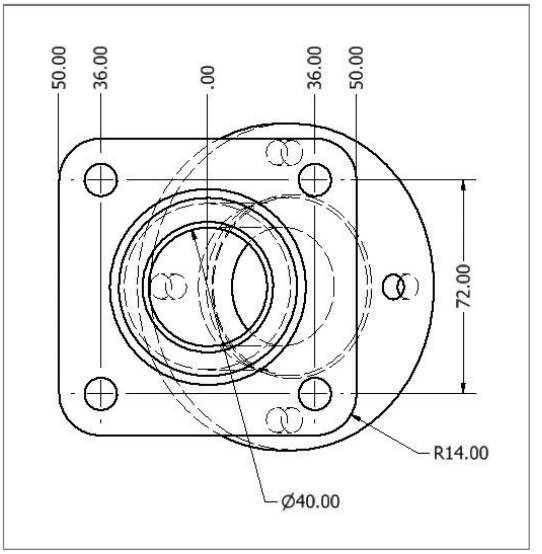


FIGURE 2.27 Ordinate dimensions work with the same workflow as chain dimensions.

Editing Dimensions

When you placed the baseline dimension set, it was placed almost as a unit. The dimensions share many properties. You can still change these properties, and if need be, you can make a dimension independent.

1. Open `c2-16.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Zoom in on the bottom view in the lower left of the drawing.
3. Click the 72.00 dimension on the left.

   Once you've selected the dimension, two drop-downs will become active in the Format panel on the Annotate tab.
4. On the lower drop-down, change the Dimension style to Default—mm [in] (ANSI).

5. Right-click the 14.00 dimension, and select Detach Member from the context menu.
6. Double-click the 14.00 dimension.
7. When the Edit Dimension dialog opens, select the Precision And Tolerance tab.
8. In the Precision field on the right, use the Primary Unit drop-down, and change the value to 1.1. Click OK. See Figure 2.28.

You can also delete a member and use the Arrange tool to bring the dimensions back in order.

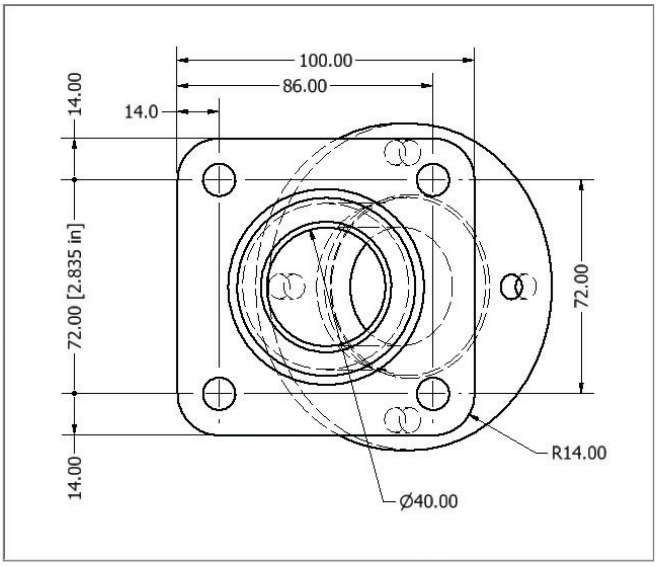


FIGURE 2.28 The dimensions can still be edited after being placed.

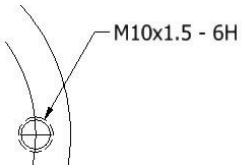
The Edit Dimension dialog offers a lot of options that I didn't cover in the exercise, including tolerancing, fits, and even the ability to define an inspection dimension.

The Hole and Thread Notes Tool

Inventor's part modeling tools give you the ability to define holes by thread and their clearances for fasteners based on standards. To document these features, use the Hole and Thread Notes tool, which extracts the information directly from the feature.

1. Open `c2-17.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.

2. Zoom in on the rotated auxiliary view in the upper left of the drawing.
3. Find and start the Hole and Thread tool in the Feature Notes panel of the Annotation tab.
4. Click the hole at 3 o'clock, and place the dimension above and to the right of the hole.



5. Press the Esc key to end the tool.

   Now, you will edit the annotation to add detail.
6. Double-click the Hole note.
7. In the Edit Hole Note window that appears (Figure 2.29), press the Home key or click the cursor before the text <THDCD>.

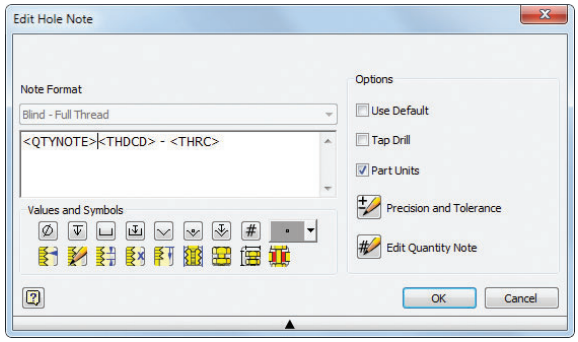


FIGURE 2.29 The Edit Hole Note window

8. Click the Quantity Note icon to add the ability to display the number of like holes in the hole note. Then click OK. See Figure 2.30.

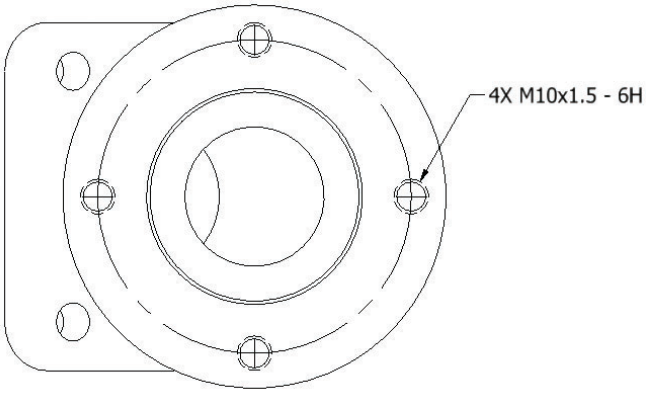


FIGURE 2.30 The Hole note can also display the number of holes.

Retrieving Model Dimensions

The parametric dimensions that are used in sketches to construct the parts can also be used to detail the part.

1. Open `c2-18.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Zoom in on the rotated auxiliary view in the upper left of the drawing.
3. Right-click in the drawing view, and select Retrieve Dimensions from the context menu; or, select the Retrieve tool from the Dimension panel of the Annotate tab.
4. Once the Retrieve Dimensions dialog is open, select one of the tapped holes in the drawing view.

   The drill diameter of the holes and the diameter of the bolt circle will be displayed. When Inventor needs more information from the user, an icon will appear in the dialog with a red arrow. Often, the button will be depressed, and selection can be made immediately. Sometimes, you will need to select the button to begin picking entities.

5. Click the icon to select dimensions, and click the 90mm diameter of the bolt circle.

6. Click OK to close the dialog and include the dimension in the drawing view, as shown in Figure 2.31.

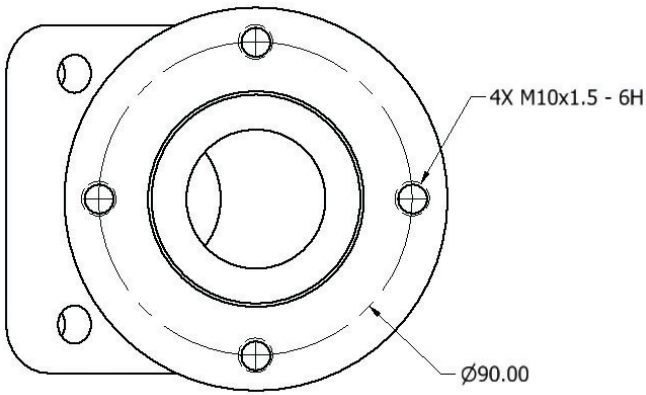


FIGURE 2.31 Reusing model dimensions in the drawing view

Associativity

The ability to create drawing views and annotate them should be clear by this point. What I haven't done is demonstrated the advantage of creating 2D parts from 3D geometry. When you edit a 3D part, any change is reflected in the 2D drawing.

This capability is key for productivity and really demonstrates the final value of creating 2D drawings from 3D parts. The technique used in this exercise is not the most common way of making the changes, but it does show a technique that is available.

1. Open `c2-19.idw` from the `Drawings\Chapter 2` folder, making certain that the `2012 Essentials` project file is active.
2. Zoom in on the rotated auxiliary view in the upper left of the drawing.
3. Right-click the 90mm diameter dimension. Because it is a model dimension, it will offer a special option.
4. Select Edit Model Dimension from the context menu.
5. In the Edit Dimension dialog, change the value to **85mm**, and click the green check mark to make the change; close the dialog. See Figure 2.32 for the results.

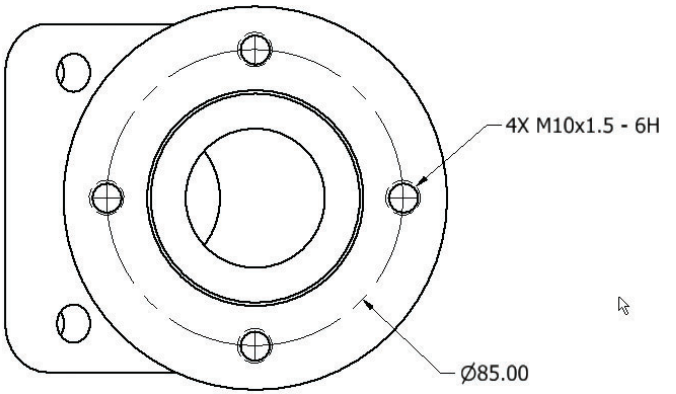


FIGURE 2.32 Any change to the geometry of the model is reflected in the drawing views.

The drawing view that you were focused on is not the only view that was changed. All views of this part on all pages in any drawing file will be updated based on this change.

The Essentials and Beyond

There are countless ways to approach creating your detail drawings, and Inventor maintains Autodesk's legacy of flexibility but adds a level of simplicity that's not possible when creating 2D drawings from scratch.

Additional Exercises

- Experiment with all the tools on the Place Views tab.
- Use DWG templates to create drawings that can be read using AutoCAD.
- Work with individual dimensions and dimension sets to see what method you might typically want to use.
- Try breaking the alignment of the section view to see that it still maintains its definition.