Producing 2D Documentation from 3D Models in AutoCAD®
David Cohn – 4D Technologies

Modeling in 3D is fine, but eventually you need to produce 2D drawings. In this class, you will learn about tools in AutoCAD that let you quickly take 3D models and use various tools, such as drawing views and section planes, to represent your models in 2D to create orthographic, section, and detail views. Discover which methods let you quickly update your 2D views when the 3D model changes and learn how to add dimensions and notes to create completed two-dimensional documentation. We will look at each tool and consider its pros and cons to develop best practices suited to your particular needs. Don’t draw it twice—leverage the power of 3D to create your 2D drawings.

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

- Create base views and projected views
- Use section planes to create sections and elevations
- Jog sections to cut through the most interesting parts of a model
- Add dimensions and notes to document 3D models in 2D layouts

About the Speaker
David has more than 25 years of hands-on experience with AutoCAD® and 12 years with Revit® as a user, developer, author, and consultant. He is the technical publishing manager with 4D Technologies/CADLearning, a contributing editor to Desktop Engineering magazine, the former publisher and editor-in-chief of CADCAMNet and Engineering Automation Report, the former senior editor of CADalyst magazine, and the author of more than a dozen books about AutoCAD. A licensed architect, David was also one of the earliest AutoCAD third-party software developers, creating numerous AutoCAD add-on programs. As an industry consultant, David has worked with many companies, including Autodesk. He has taught college-level AutoCAD courses and is always a popular presenter at both Autodesk University and AUGI® CAD Camps.

david.cohn@cadlearning.com
www.cadlearning.com
People have been creating two-dimensional drawings using AutoCAD for many years. And while it’s often easier to actually model real-world objects in 3D, eventually you will probably need to create two-dimensional drawings of those three-dimensional objects. You don’t have to manually create all of those 2D drawings. AutoCAD provides tools that can create those views automatically directly from the 3D objects. These tools include:

- Drawing Views
- Section Plane
- Live Section
- Add Jog
- Flatshot

There are also three related tools—Solid View, Solid Drawing, and Solid Profile—that can be used to create two-dimensional representations of 3D objects and place them in a paper space layout in a way that mimics conventional 2D drafting. We will look at this legacy method as well, but it lacks many of the advantages of the newer tools in AutoCAD.

Creating Drawing Views

The Drawing Views tools, which are new in AutoCAD 2012, let you create standard drawing views from 3D models. Those models can be created in AutoCAD or Inventor or imported from other CAD applications, such as CATIA, NX, Pro/E, SolidWorks, and so on. For our purposes, we’ll focus on models that have been created in AutoCAD, but once a model has been imported from some other CAD application, the process of creating drawing views is basically the same.

You can use the Drawing Views tools to create standard drawing views, such as top, front, and side views, as well as various isometric views. These views are created as drawing view objects, which is a new type of object in AutoCAD 2012. These drawing views remain associative to the model from which they were created. Although you cannot select the view geometry to modify it, if you make any changes to the 3D model, the view geometry automatically updates.

The Drawing Views tools are located on the Annotate ribbon. The first step is to place a base view. In order to do this, you must first switch to a blank paper space layout. If you
attempt to use the command while working in model space, AutoCAD responds, “This command is not supported in model space.”

When you create a drawing view, the program draws the view geometry on pre-determined layers, depending on what the geometry represents. For example, visible geometry is created on a layer called Visible while hidden geometry is created on a layer called Hidden.

The first view you place is called a base view. Base views are drawing views that are derived directly from the 3D model. When you start the Base View tool, the ribbon changes to the Drawing View Creation contextual ribbon, and in the command window, you can see that there are a number of options. For example, you can control the orientation of the view (top, front, side, and so on), the view style (such as wireframe or shaded), and the scale.

AutoCAD will automatically set an appropriate scale so that the view you’re creating will fit onto the layout, but you can change the scale if you wish. You can also always change the scale of the view, or any of its other settings, after it has been placed onto the sheet, even if you have added other views and annotations.

Once you have adjusted the view settings, you simply click to place the base view. As soon as you do, AutoCAD displays a shortcut menu with the same set of options. To finish placing the base view, click the Exit option, or simply press ENTER.

AutoCAD then immediately prompts you to specify the location of a projected view. For example, if your base view is a front view (which is the default), after placing that view, if you move your cursor above the base view, AutoCAD will create a top view. If you move the cursor to the right of the base view, AutoCAD will create a side view.

After placing the projected view, AutoCAD will again prompt you to specify the location of another projected view. If you prefer, you can press ENTER to exit from the command and then come back later to place other projected views.

When placing projected views, AutoCAD does not display the Drawing View Creation contextual ribbon because projected views are child views of the parent view. That means that, by default,
the projected view automatically takes on the properties of the base view. So it will have the same scale and view style as the base view. After placing the view, you can always change those settings later.

If you move the cursor at an angle from the base view, AutoCAD will create an appropriate isometric view. The isometric projection depends on where you move the cursor.

When you’re finished placing projected views, you must press ENTER to exit from the command. As soon as the command ends, the views all update to reflect the view style, which in this case was wireframe with hidden edges.

Notice that you do not see any viewports. That’s because the views are actually created as a new type of object called a drawing view. If you move the cursor over a view, a tooltip appears that displays information about the view type, alignment, style, and scale.

**Editing Drawing Views**

After you’ve created and placed drawing views, you can edit those views to change their view style, scale, object visibility, and view options, or to move the view. But you cannot change the geometry from within the view. The view geometry can only be changed by modifying the 3D objects on which the view is based.

To edit a drawing view, you can select the Edit View tool from the Drawing Views panel of the Annotate ribbon. AutoCAD prompts you to select a view. Click to select the view you want to modify. Alternately, you can select the view, right-click, and then choose Edit View from the shortcut menu. Or, you can also simply double-click on the drawing view you want to edit.

The ribbon changes to the Drawing View Editor contextual ribbon and AutoCAD also displays a shortcut menu. You can then use the tools in the ribbon or the menu to edit the drawing view. For example, you can expand the View Style drop-down and choose a different style. If you are editing the parent view, all child views will update to reflect your changes. But if you are editing a child view, the settings are initially based on its parent view, but you can change the setting to any other available style.
The tools in the Object Visibility drop-down enable you to control whether the view includes interference edges, tangent edges, and so on.

If you click the View Options button, AutoCAD displays the View Options dialog. You can then use the tools in this dialog to control whether the view remains fixed in the same location even if the geometry in the view subsequently changes, or remains centered, in which case the view grows or shrinks about its center if the geometry within the view changes. Typically, you only use the Centered option for views placed at the center of a layout.

Notice also that if the drawing view is based on 3D models created using Autodesk Inventor, you can also control the Line Style and Hidden Line Calculation.

You can also move the view, by using the Move command or by simply clicking and dragging the square blue grip at the center of the view. When you move a view, any related views automatically move as well. For example, if you were to move the parent view (the front view) towards the top of the screen, the side view would move up so that it remains aligned with it. Similarly, if you were to move the front view toward the right, the top view would move as well.

The grip at the center of the view is a multi-functional grip. If you hover the cursor over the grip, an extended tooltip appears, containing options to rotate the view,
break its alignment with other views, or repair the alignment. In general, it’s not considered good drafting practice to break the alignment between views.

Setting the Drafting Standards for Drawing Views

When you create drawing views, those views are based on the current drafting standards in effect. Those settings impact all new drawing views you create, but any changes you make have no effect on any existing drawing views.

Before creating drawing views, you can set the drafting standards on which you want those views to be based. To do so, expand the Drawing Views panel in the Annotate ribbon and click the small arrow in the lower-right corner. AutoCAD displays the Drafting Standard dialog.

In the Projection Type area, you can set the projection angle for drawing views. The projection angle defines where projected views are placed. For example, if the active projection type is First Angle, top views are placed below the parent view. With the Third Angle, top views are placed above the parent view.

In the Thread Style area, you can set the appearance of thread edges in the drawing. The ISO standard uses a partial circle to indicate a thread edge while the ANSI standard uses a full circle. Note that this setting only applies to drawing views based on Autodesk Inventor models.

In the Shading/Preview area, you can choose the shaded preview quality and preview type. For example, in the Shaded View Quality drop-down, you can specify the resolution of the shaded preview. While higher resolutions provide better quality previews, resolutions above 150 dpi may not be achievable for larger models. When the specified resolution is not achievable, the program automatically drops the quality to an achievable level.

In the Preview Type drop-down, you can choose whether AutoCAD displays a shaded view when you’re creating a projected view or just shows a bounding box. Note that the settings in the Shading/Preview area only affect the preview. Once the view has been placed, its appearance is based on the actual drawing view settings.

Once you have finished adjusting these settings, click OK to close the Drafting Standard dialog.
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Updating Drawing Views
While you work on a drawing containing drawing views, AutoCAD periodically checks whether the referenced 3D source model has changed. If the source changes, a balloon message displays next to the drawing view icon in the status bar in the lower-right corner of the application window. Red markers also appear at the corners of any drawing views that are no longer up to date.

The balloon includes a link. If you click this link, all of the out-of-date drawing views are immediately updated to reflect the changes made to the 3D model.

If instead, you close the balloon without updating the views, the red markers remain in the corners of the out-of-date views and you can see an alert symbol on the drawing views icon on the status bar. In that case, you can update the views manually, by either right-clicking on the drawing views icon on the status bar, or by expanding the Drawing Views panel on the Annotate ribbon.

Note that you can update either an individual view or all views. For example, if you choose Update View, the program prompts you to select the view you want to update. Click to select a view. Although you can only select views by clicking on them, you can select more than one view. Once the views you want to update are selected, press ENTER. The views you selected are immediately updated. AutoCAD may display the notification balloon again, and you will still see red markers in the corners of other views that are still out-of-date.

If you use the view update tool and choose Update All Views, any views that are still out-of-date are immediately updated to reflect the changes made to the 3D model.

Adding Annotations to Drawing Views
Because drawing views are a special type of object that only exists in a paper space layout (and only displays 3D geometry), when you add annotations such as text and dimensions, you must add them to the paper space layout.

When you dimension model space objects in paper space, you use a function referred to as trans-spatial dimensioning. While this function has been available in AutoCAD for many releases, it is a bit problematic. If you have added dimensions and the model subsequently changes, any affected dimensions will most likely become disassociated from the model and will likely need to be re-associated.

When annotations associated to a view lose their associativity due to changes in the model, AutoCAD displays an alert. While this alert gives you the option of trying to re-associate annotations, this typically does not succeed. In that case, AutoCAD displays a second alert, informing you that one or more annotations are no longer associative and offers to run the DIMREASSOCIATE command. When you run this command, you will be able to manually re-associate the trans-spatial dimension with the geometry in the drawing view.
Because of the problems associated with trans-spatial dimensioning, you should wait until your model is not likely to change before adding annotations to drawing views. Of course, this may not always be possible, so you will likely have to re-associate dimensions when changes occur.

**Best Practices**

While drawing views provide a quick way to create 2D documentation based on 3D models, they currently have several limitations:

- As we’ve already seen, annotating drawing views can be somewhat problematic. Trans-spatial dimensioning results in dimensions that can easily become disassociated from the objects they annotate. In that case, you’ll have to re-associate those dimensions.

- Drawing views can only be used to create standard orthographic elevations, standard isometric views, and some auxiliary views. Drawing views cannot be used to create cross-sections or non-standard isometric views. To create these types of views, you’ll need to use other tools, such as Section Planes and Flatshot.

- When you create drawing views, AutoCAD includes all of the 3D geometry. Therefore, if your model contains multiple parts, you will either have to model each part in a separate AutoCAD drawing or use other tools to create your 2D documentation. Alternately, you could create each 3D part on its own layer and turn selective layers on and off when creating specific drawing views. But when you change layer settings, AutoCAD will warn that the model has changed and offer to update views. The only way to keep drawing views from updating to show additional parts is to not update those views. But this can be problematic if other aspects of the model have changed.
Working With Section Planes

The Section Plane tool (SECTIONPLANE command) lets you create one or more section objects and place them wherever needed within a 3D model. These planes can be moved through a model composed of 3D solids, surfaces, meshes, and regions to obtain different views.

The section plane contains a section line that stores the section object properties. You can create multiple section objects to store different properties.

Once you've created the section plane, you can turn on live sectioning to see the inner details. You can even move the section plane through the 3D model to see various portions of the cross section in real time. And once you've got section planes positioned where you want them, you can use the section planes to create 2D blocks of the section and elevation views and place those views into a drawing to create two-dimensional documentation.

If you subsequently make changes to the 3D model, you can easily update the blocks created based on those section planes to quickly update the two-dimensional views to reflect those changes.

In order to create a cross section, you must first create a section plane, which you can do by using the Section Plane tool in the Section panel of the Home ribbon (when working in the 3D Modeling workspace). Note that these tools are also available in the Section panel on the Solid and Mesh ribbons.

The section plane is created on the current layer, so you may first want to create a new layer on which the section plane will be created. That way you can control its visibility separately from the other objects in the drawing.

When you start the SECTIONPLANE command, AutoCAD prompts you to locate the section plane. You can select a face or any other point on the object to locate the section line. There are also options that enable you to align a section plane to one of the standard orthographic views, or you can draw the section plane.

Once the section plane has been created, you can then move it as you would any other object in the drawing. One of the best ways to move the section plane is to select it and then use the 3D Move Gizmo.

Even after the section plane has been positioned, it isn’t actually cutting anything yet. To have the section plane show the cross section, you need to turn on live sectioning. You can do this one of two ways. If you click the Live Section tool, available in the Section panel, AutoCAD prompts you to select a section object. As soon as you select the section plane, AutoCAD uses that plane to display a cross section through the model. You can also
select the section plane, right-click, and choose Activate Live Sectioning from the shortcut menu.

Live sectioning is a toggle. When turned on, the section plane cuts through the model, displaying it as a cross section. When turned off, the section plane remains visible but the model is no longer shown in cross section.

If the section is looking in the wrong direction, you can flip it by clicking the small arrow, the Direction grip, located near its midpoint grip.

**Adding Jogs to a Section Plane**

When you first create a section plane, it is a single plane that follows the path of the section line. But sometimes the 3D models you create are not symmetrical. Sometimes you may want to jog the section so that it cuts through the most interesting portions of the 3D model. This is quite easy to do using the Add Jog tool.

You can add a jog to a section plane by using the Add Jog tool (SECTIONPLANEJOG command), available in the Section panel, or by right-clicking the section plane and choosing Add Jog to Section from the shortcut menu.

After starting the command and selecting the section plane, AutoCAD prompts you to specify a point on the section line to add the jog. It is often helpful to change the orientation of the view so that you can better see where you are positioning the jog. Click on the section plane where you want to add the jog. The Nearest object snap is temporarily turned on to help you place the jog. The jog is immediately created perpendicular to the selected section.

Note that after adding the jog, if it isn’t positioned where you want it, you can reposition the jog by dragging the section object grips. You can also use the Add Jog tool again to add additional jogs to the section plane.

You can also choose Show Cut-Away Geometry from the shortcut menu to display the portion of the 3D model on the opposite side of the section plane.

**Changing the Section Object States and Using Grips to Modify Section Objects**

When you select a section plane object, several of its grips become visible, including a Menu grip and the Direction grip. If you click the Menu grip, you can change the display of the section object to one of three possible display states:
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- **Section Plane.** The section line and transparent section plane indicator are displayed. The cutting plane extends infinitely in all directions.
- **Section Boundary.** A 2D box shows the XY-extents of the cutting plane. The cutting plane along the Z-axis extends infinitely.
- **Section Volume.** A 3D box shows the extents of the cutting plane in all directions.

Depending on the section plane display state, various additional grips also become visible when you select the section object. You can then use these grips to move and resize the section object to adjust the location, length, width, and height of the cutting area.

- **Base grip.** Acts as the basepoint for moving, scaling, and rotating the section object. It always displays adjacent to the Menu grip.
- **Second grip.** Rotates the section object around the base grip.
- **Menu grip.** Displays a menu of section object states, which control the display of visual information about the cutting planes.
- **Direction grip.** Controls the viewing direction of the 2D section. To reverse the viewing direction of the section plane, click the Direction grip.
- **Arrow grip.** (Section boundary and Volume states only.) Modifies the section object by modifying the shape and position of the section plane. Only orthogonal movements in the direction of the arrow are permitted.
- **Segment end grips.** (Section Boundary and Volume states only.) Stretches the vertices of the section plane. You cannot move segment end grips so that segments intersect. Segment end grips are displayed at the endpoints of jogged segments.

Note that unlike other types of grips in AutoCAD, you can select only one section grip at a time. You cannot press the SHIFT key to select multiple section grips.

**Creating 2D Sections and Elevations**

Once you have created and positioned section planes, you can save the representation of the cross-sectional area where the section object intersects the 3D model as a block. You can then use that block to create two-dimensional documentation.
To create a block from a section object, use the Create Block tool (SECTIONPLANETOBLOCK command), which can be found in the Section panel. You can also select a section object, right-click, and then choose Generate 2D/3D Section from the shortcut menu.

When you use this tool, AutoCAD displays a Generate Section/Elevation dialog. Note that the dialog box may initially display in a reduced state. You can click the arrow in the lower-left corner to expand the dialog.

If you start the command from the ribbon and no section object has been selected, the dialog box displays an alert and you must click the Select Section Plane button and select the section plane you want to use to create the block before proceeding.

Once you've selected a section plane, the other controls in the lower portion of the dialog become available.

**2D/3D** These controls let you choose to create a two-dimensional section/elevation or a 3D section.

**Source Geometry** These controls let you choose to include all of the 3D objects in the drawing (including objects in xrefs and blocks) or manually select just those objects you want to include.

**Destination** These controls let you choose to insert the section/elevation as a new block, replace an existing block, or export the geometry to a separate drawing file.

**Section Settings** This button displays the Section Settings dialog.

To create a two-dimensional cross section and insert it as a block in the current drawing:

1. Open the Generate Section/Elevation dialog.
2. Select the section plane from which you want to create the section.
3. In the 2D/3D area, choose 2D Section/Elevation.
4. Under Source Geometry, choose Include All Objects.
5. Under Destination, choose Insert As New Block.
6. Click the Create button.
7. In the drawing, specify an insertion point for the block and press ENTER to accept the default values for scaling and rotation angle.

An unnamed block is inserted consisting of the 2D cross-section geometry. The block is inserted on the current layer and aligned with the current UCS.

**Tip:** When using this tool, it's often useful to first split the viewport so that you can see a plan view in one viewport and a 3D view in the other. When AutoCAD prompts you to select the insertion point for the block, click in the plan view viewport to make it active and then finish inserting the block.
If you subsequently make changes to the 3D model, you can repeat this process, but instead of inserting a new block, you can choose Replace Existing Block and then select the block you previously created. AutoCAD immediately updates the block to reflect the changes you made to the 3D model.

Creating 2D Elevations
You can use this same method to create two-dimensional elevations, but instead of creating section planes that cut through the model, you can position section planes away from the 3D objects. The object is then projected onto the section plane, creating an elevation view.

To create an elevation of the right side of a 3D object:

1. Start the Section Plane tool.
2. Right-click and choose the Orthographic option.
3. Right-click and choose the Right orthographic alignment.
   AutoCAD immediately creates a new section plane and places it within the 3D model.
4. Select the section plane and use the 3D Move Gizmo to pull it along the X-axis until the plane is outside the volume of the 3D model.

Once the new section plane has been created, you can use the tools in the Generate Section/Elevation dialog to create a new two-dimensional block that shows the 3D model projected onto the section plane and place that block into your drawing. Again, if you subsequently make changes to the 3D model, you can use the tools in the Generate Section/Elevation dialog to replace the block with an updated version.

To create additional elevations, simply position additional section planes using the other orthographic alignments. You can create as many section planes as necessary. Remember that each section plane is created on the current layer. You may want to create additional layers to control the visibility of the additional section planes you create.

As you insert each additional elevation view block, remember that you can use the Basepoint option during block insertion to align each block in its proper relationship. This will make it easier
to organize the orthographic views when you create viewports and place them on a paper space layout.

Controlling Section Settings
When you create or update sections or elevations based on a section plane, you can control various aspects of those views using the Section Settings dialog.

To display the Section Settings dialog, click the arrow in the lower-right corner of the Section panel (SECTIONPLANESETTINGS command). You can also display this dialog by clicking the Section Settings button in the Generate Section/Elevation dialog or by selecting a section plane and then right-clicking and choosing Live Section Settings from the shortcut menu.

The Section Settings dialog contains display settings for creating 2D and 3D sections and for live sectioning. Note that if a section plane was not already selected when you open the Section Settings dialog, you must select one (by clicking the Select Section Plane button) before you can adjust any of the settings.

The section settings are stored separately for each section plane. The radio buttons in the upper portion of the dialog determine which settings types are displayed in the properties list.

- **2D Section/Elevation Block Creation Settings**
  Determines how a 2D section or elevation from a 3D object is displayed when generated.

- **3D Section Block Creation Settings**
  Determines how a 3D object is displayed when generated.

- **Live Section Settings**
  Determines how sectioned objects are displayed in the drawing when live sectioning is turned on.

- **Activate Live Section** (Available only when the Live Section Settings option is selected.)
  Turns on live sectioning for the selected section object.
The controls in the properties list in the lower portion of the dialog set the properties to be applied to the new section block. This list is divided into three, four, or five sections, depending on the type of section setting selected.

**Intersection Boundary** Sets the appearance of line segments that outline the intersection surface of the section object plane.

**Intersection Fill** Sets the optional fill that is displayed inside the boundary area of the cut surface where the section object intersects the 3D object.

**Background Lines** *(2D and 3D section blocks only.)* Controls the display of background lines.

**Curve Tangency Lines** *(2D section blocks only.)* Controls the inclusion of curved lines that are tangent to the section plane.

**Cut-Away Geometry** Sets the properties for the cut-away objects.

Controls in each section let you determine whether the particular type of objects are displayed, and if displayed, their color, layer, linetype, linetype scale, lineweight, and so on.

For example, when generating a cross section, you can select the hatch pattern applied to the sectioned geometry and control the pattern’s various settings, such as hatch scale and hatch angle. The layer controls enable you to select an existing layer or you can select “LayerByObject” to have AutoCAD automatically create new layers and split the selected block component onto a separate layer.

By default, the settings only apply to the current section object. If you select the Apply Settings to All Section Objects check box, however, the settings will apply to all section objects in the drawing.

You can click the Reset button to return all settings in the dialog box to their default settings.

When you’re done making changes, click OK to apply the changes to the section object. Changes will not affect existing blocks until you use the Replace Existing Block tool in the Generate Section/Elevation dialog to update the block.

**Tip:** After you change section settings to produce the desired results for different types of views, you can drag-and-drop a section object onto a tool palette to save those settings as a new palette tool, so you can easily create a new section object using those same settings in the future.

**Adding Annotations**

When working with blocks created using section planes, you can add dimensions and annotations in model space. In that case, you should use annotative objects, so that the annotations scale properly so that their size adjusts to the scale of the paper space viewport.

In spite of the fact that model space dimensions are more firmly attached to the objects they annotate than when using trans-spatial dimensioning, annotations will still likely become disassociated from the geometry in the view blocks if you ever update the blocks and will need to be re-associated. But this task is often easier when working in model space.

Since fixing lots of annotations can be quite tedious, you should still wait until your model is not likely to change before adding annotations to drawing views.
Best Practices
Because there is currently no way to create cross-sections using drawing views, the Section Plane tools are the method to use to create cross-section views. They also offer some advantages when generating elevation views as well.

- Many users find it easier to add annotations in model space.
- Section planes can be used to create non-standard views that cannot be created using drawing views.
- When your drawing contains multiple 3D parts, you can model each part on a separate layer and easily control the creation of view blocks of selected parts.
- You can manually edit view blocks to make changes (something not possible when using drawing views), although those edits would be lost if your 3D model changed and you subsequently updated the blocks.

Tip: You can associate section objects with named views and cameras in the View Manager. When you subsequently activate a named view or camera that has an associated section object, live sectioning is turned on for that section object, letting you quickly restore the desired settings.

Creating 2D Representations of a 3D Object
You can use the Flatshot tool (FLATSHOT command) to create a two-dimensional representation of a 3D object. When you use this tool, AutoCAD projects the edges of all 3D solids, surfaces, and meshes in the drawing onto a plane parallel to the current viewing plane. The 2D representation of these edges can then be inserted as a block on the XY plane of the current UCS or saved as a separate drawing.

The process is like taking a photograph of the entire 3D model and then laying the photograph flat. This feature is useful for creating technical illustrations, and can also be used to create two-dimensional elevations.

The flatshot process works only in model space. Start by setting up the view you want, including orthographic, perspective, or parallel views. All 3D objects in the model space viewport are captured. Therefore, you need to be sure to place the objects you do not want captured onto layers that are turned off or frozen before capturing the geometry.
Once the model is set up the way you want it to be captured, select the Flatshot tool, located in the Section panel on the Home, Solid, or Mesh ribbons. AutoCAD displays the Flatshot dialog. This dialog lets you control various aspects of the creation of the 2D representation.

**Destination**  These controls let you choose to insert the flattened representation as a new block, replace an existing block, or export the geometry to a separate drawing file.

**Foreground Lines**  These controls determine the color and linetype for lines that are not obscured in the flattened view.

**Obscured Lines**  These controls determine whether obscured lines are displayed in the flattened view, and if so, the color and linetype used to display those obscured lines.

**Include Tangential Edges**  When selected, silhouette edges for curved surfaces will be included in the flattened 2D representation.

### Using Flatshot to Create 2D Elevations

While the Flatshot tool is most useful for creating two-dimensional representations of 3D models shown in a perspective projection to depict the three-dimensional geometry for creating technical illustrations, if you first reorient the model to display a parallel projection elevation view, this tool can also be used to create two-dimensional elevation views.

To create a 2D elevation using Flatshot:

1. Use the ViewCube to reorient the model to the desired top, front, or side orthographic view.
2. Change the projection mode to a Parallel projection.
3. Start the Flatshot tool.
4. Under Destination, choose Insert As New Block.
5. Adjust the settings for foreground and obscured lines and determine whether to include tangential edges.
6. Click Create.
7. In the drawing, specify an insertion point for the block and press ENTER to accept the default values for scaling and rotation angle.

**Tip:** When using this tool, it’s often useful to first split the viewport so that you can see a plan view in one viewport and a 3D view in the other. When AutoCAD prompts you to select the insertion point for the block, click in the plan view viewport to make it active and then finish inserting the block.

An unnamed block is inserted consisting of the flattened geometry. The block is inserted on the current layer. Because the block is aligned with the current UCS, it may appear strange when first inserted. However, if you reorient the drawing to a plan view of the current UCS, the elevation will appear correct.

**Tip:** Create the elevations on the WCS and then use commands such as MOVE to arrange the elevations into the proper positions for a typical set of orthographic views.
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As is true with views created using section planes, if you subsequently make changes to the 3D model, you can replace the existing blocks to reflect the changes you made to the 3D model.

Adding Annotations
When working with blocks created using the Flatshot command, you can add dimensions and annotations in model space. Doing so has the same advantages and disadvantages as when working with section planes.

Best Practices
Because there is currently no way to create non-standard isometric views using drawing views, the Flatshot tool is the preferred method for creating such views. The Flatshot tool also has some advantages when creating standard orthographic views:

- It does not require you to first create section planes.
- It does a better job of correctly rendering obscured internal geometry than view blocks created using section planes.

But the Flatshot tool has a significant disadvantage. If you need to update a block previously created using Flatshot, you must first reorient your model back to the view that was current when you initially used Flatshot, or the resulting block will change.

Using Solid View, Solid Draw, and Solid Profile Tools
You can also use three related tools to automate the steps needed to create 2D representations of solids that mimic conventional 2D drafting. These three tools—Solid View, Solid Drawing, and Solid Profile—are found by expanding the Modeling panel on the Home ribbon tab (when working in the 3D Modeling workspace).

These tools are meant to be used in the order in which they appear in the Modeling panel and you need to follow the proper sequence of steps in order to get the desired results.

Once you’ve created your 3D model, you can use these tools to create 2D paper space layouts containing the desired views. The following command sequence will be used to create the classroom example:
Command: **SOLVIEW**
Enter an option [Ucs/Ortho/Auxiliary/Section): **U**
Enter an option [Named/World/?/Current]<Current>: **W**
Enter view scale <1.0000>: **.5**
Specify view center: *(locate view center near upper-left area of the paper space layout)*
Specify view center <specify viewport>: **ENTER**
Specify first corner of viewport: *(locate one corner of the viewport)*
Specify opposite corner of viewport: *(locate the opposite corner of the viewport)*
Enter view name: **TOP**
Enter an option [Ucs/Ortho/Auxiliary/Section): **O**
Specify side of viewport to project: *(click the bottom edge of the TOP viewport)*
Specify view center: *(locate view center below the TOP viewport)*
Specify view center <specify viewport>: **ENTER**
Specify first corner of viewport: *(locate one corner of the viewport)*
Specify opposite corner of viewport: *(locate the opposite corner of the viewport)*
Enter view name: **FRONT**
Enter an option [Ucs/Ortho/Auxiliary/Section): **O**
Specify side of viewport to project: *(click the right edge of the FRONT viewport)*
Specify view center: *(locate view center to the right of the FRONT viewport)*
Specify view center <specify viewport>: **ENTER**
Specify first corner of viewport: *(locate one corner of the viewport)*
Specify opposite corner of viewport: *(locate the opposite corner of the viewport)*
Enter view name: **SIDE**
Enter an option [Ucs/Ortho/Auxiliary/Section): **S**
Specify first point of cutting plane: *(snap to the left midpoint in TOP view)*
Specify second point of cutting plane: *(snap to the right midpoint in TOP view)*
Specify side to view from: *(click below cutting plane defined in TOP view)*
Enter view scale <0.5000>: **ENTER**
Specify view center: *(locate view center below the FRONT viewport)*
Specify view center <specify viewport>: **ENTER**
Specify first corner of viewport: *(locate one corner of the viewport)*
Specify opposite corner of viewport: *(locate the opposite corner of the viewport)*
Enter view name: **SECTION**
Enter an option [Ucs/Ortho/Auxiliary/Section): **ENTER**

At this point, the views should look something like the illustration below. The views are all arranged, but the viewports still display the actual 3D model using the current visual style. These will need to be converted into two-dimensional versions, using the Solid Drawing tool.
The Solid Drawing tool converts the 3D objects displayed in the viewports created using the Solid View tool into two-dimensional representations of those objects. When you start the command, AutoCAD prompts you to select the viewports to draw and prompts you to select objects. Select the four viewports and then press ENTER. AutoCAD immediately processes the viewports, converting the 3D model into 2D lines. The results should look similar to the illustration below.

If you look in the Layer Properties Manager palette, you will see that AutoCAD has created new layer names based on the names of the various views created using the Solid View tool. Then, for each viewport, AutoCAD has created a layer for dimensions, a layer for hidden lines, and a layer for visible lines. For the section view, it has also created a layer for the hatch patterns applied to the cross section. You can then change the color, linetype, and linewidth for each of these layers, as necessary.

You can also use the Solid Profile tool to create a paper space viewport containing a two-dimensional profile of a three-dimensional solid. The profile is based on the current viewpoint and can display hidden lines on a separate layer so that you can control their color and linetype.

Before you can use the Solid Profile tool, you need to create a new viewport and orient the three-dimensional model the way you want it to be displayed in the resulting profile drawing.

Once the viewport has been created and the model is sized and oriented in that viewport the way you want it to appear in the resulting profile view, make sure the viewport is the current view and then click the Solid Profile tool (SOLPROF command). AutoCAD prompts you to select objects. Click to select the 3D model and then press ENTER. AutoCAD then asks three questions (the answers used in our class example are shown):

Display hidden profile lines on separate layers? [Yes/No] <Y>: Y
Project profile lines onto a plane? [Yes/No] <Y>: Y
Delete tangential edges? [Yes/No] <Y>: Y
When the command is finished, it doesn’t appear that anything has happened. But if you again look in the Layer Properties Manager palette, you will see that AutoCAD has created two new layers. The layer name beginning with the letters PH contains the hidden profile lines; the layer name beginning with the letters PV contains the visible profile lines. If you freeze the layers on which the original model was created, you can clearly see the new 2D profile.

**Adding Annotations**

Since all of the views that are created are paper space viewports and have layers pre-defined for adding dimensions and notes, you can add dimensions and annotations in model space in each viewport. In that case, you should use annotative objects so that the annotations scale properly to the scale of the viewport.

Since the views are created in paper space, you can also dimension in paper space. But because the 2D views are unrelated to the 3D model, the dimensions values must be scaled to match the scale of the view. Otherwise, the dimensions will not be accurate.

And since each view contains 2D geometry that is not associated with the 3D model, if your model changes, there is no easy way to update the views and annotations. For that reason, you should not add annotations until your model is no longer likely to change.

**Best Practices**

You can use the Solid View, Solid Drawing, and Solid Profile tools to create various 2D orthographic views of the 3D model, all nicely arranged on a layout, to create your finished documentation. But this tool only creates two-dimensional views that are no longer related to the original 3D model. If you modify the model after you’ve created these views, you’ll have to create all of the views over again.

Using Drawing Views, Section Planes, or even Flatshot enable you to create drawing views that remain associated to the original 3D model. This enables you to more easily update those views if you subsequently make changes to the 3D model.

In addition, the Solid View, Solid Drawing, and Solid Profile tools are legacy tools that require much more user intervention.

**Conclusion**

It truly is easier to create real-world objects as three-dimensional models. Doing so greatly reduces the risk of errors. Then, when you need to create two-dimensional drawings of those three-dimensional objects, you can use the tools described in this class to create those drawings automatically directly from the 3D objects.
But each of the tools we’ve discussed has certain advantages and disadvantages. When creating 2D documentation from 3D models, you should use the tool that best meets your needs and best fits your workflow.

Note that the drawings used in this class will be available online at:

www.dscohn.com and on the AU website

For comprehensive video-based instruction for Autodesk software, visit:

www.cadlearning.com