From Crisp Design to Clean Documentation – An Introduction to Fusion 360 Drawings

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Learning Objectives

- Explore the full feature set that Fusion 360 Drawings offer, including the user interface, commands, and version management
- Learn about the various access points, shortcuts, and customizations in the drawings to work efficiently
- Learn how to create and edit production-quality drawings in Fusion 360 with a toolset tailored to meet manufacturing standards
- Learn how to export or share the drawings with individuals or the public using Autodesk 360’s collaboration tools

Description

This class is centered on the documentation component of Fusion 360 software application, Fusion Drawings. Learn about the user interface, commands, and functionality that the drawings offer. Gain a clear understanding of some key concepts regarding how the drawing relates to the 3D model to achieve easy customization and minimized workflows. Learn how to create and edit production-quality drawings in Fusion 360 software, rapidly.

Your AU Experts

Krithika Sundararajan is a Software Quality Assurance Engineer at Autodesk, Inc., with a Bachelor’s degree in Computer Engineering. She has experience with AutoCAD, AutoCAD web and Fusion 360 software applications, and she’s currently working on the Drawings component of Fusion 360 software. She is very passionate about the product and creating value for its users.
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Setting Up Fusion 360

Installation
Fusion 360 is available for Windows and Mac operating systems. The latest version can be downloaded from: http://www.autodesk.com/products/fusion-360/try-buy

The application is also available for download on the Mac App Store. Updates are delivered every 6-8 weeks and will be downloaded automatically.

Licensing
Users are required to sign into Fusion 360 using their Autodesk 360 accounts and will receive a 30-day trial of the product. Subscriptions are offered on a monthly or annual basis and the license provides access to all features in Fusion 360. After the 30-day trial you will still have access to your designs if you choose not to subscribe.

Students, Educators, Start-ups and enthusiasts also receive an extended free license to Fusion 360. For more information, visit the Autodesk Learning site at: http://fusion360.autodesk.com/learning/ (Get Started > Installation and Subscription)

Accessing Fusion 360 Drawings

What is Fusion 360?
Fusion 360 is a cloud-based 3D CAD, CAM, and CAE platform for product development. It combines industrial and mechanical design, simulation, collaboration, and machining in a single package. The tools in Fusion 360 enable fast and easy exploration of design ideas with an integrated concept-to-production toolset.

What is Fusion 360 Drawings?
Mechanical drawings can be created as a derived document of a Fusion 360 model. When we create a drawing from a model, the system generates a 2D projection of the components selected. Built on AutoCAD’s drawing engine, Fusion 360’s Drawings module has a distinctive toolset designed to meet manufacturing standards.

It Begins with the Model
To begin creating drawings in Fusion 360, we must have a model of the part / assembly we wish to document. Designs can be created in the Modelling workspace. Users can also import models from over 50 file formats.

Importing a Model into Fusion 360
1. Open the Data Panel. Create a project or navigate into an existing project.
2. Click on the Upload button and choose the desired file(s). Once the upload is complete, thumbnails for the file(s) will appear on the data panel. The file format is f3d.
3. Double click on the thumbnail to open the design.
Creating the Drawing from the Design

Drawings can be created from the design or animation keyframes (more information on creating drawings from animation keyframes is provided under Exploded Views in the Advanced Features section on Page 11). Drawings may be created from the full assembly or components. Below are some of the methods to create drawings from the design.

**From the File Menu**
1. With the model of interest as the active design, select New Drawing > From Design, from the File Menu.
2. In the Create Drawing command dialog that appears, Full Assembly is checked by default. This can be unchecked and individual parts can be selected from the model for documentation.
3. Other configuration options such as Drawing Standard, Annotation Units and Sheet Size are also available in the Create Drawing command dialog. Fusion 360 Drawings currently support ASME and ISO standards.

**From the Browser Tree**
1. Select one or more components from the browser tree.
2. Right click > from the contextual menu, select Create New Drawing.
Creating Drawings

The Drawings Workspace
The below screenshot illustrates the various UI elements within the Drawings environment.

Similar to other workspaces in the application, contextual marking menus can be accessed with a Right Click on the canvas. Dynamic prompts are also displayed when a command is active, to prompt the user for the next input.

Drawings Set Up
Configuration settings can be accessed both within a drawing and prior to drawing creation.

User Preferences
A set of global preferences for the user can be accessed from the Preferences dialog. Drawings specific settings are found in the Drawings tab of the Preferences dialog. These values will be used as defaults when creating new drawings.

Document Settings
Annotation and Sheet Settings for the active drawing can be accessing from the Document Settings bar. Annotation Settings affect existing and newly created annotations in the drawing.
Creating Views
When a drawing is created, the Base View command is launched which is a 2D projection of the components chosen in the model.

Creating a Base View
1. Launch the Base View command from the Views panel.
2. In the command dialog that appears, the properties of the view can be changed.
3. Place the Base View at the desired location and commit.

Creating Projections, Section and Detail Views
Orthographic and isometric projections can be created using the Projected View command. To create a section view:

1. Launch Section View command from the Views panel.
2. Select the view to be sectioned.
3. Create the section line. When all segments in the line have been placed, hit Enter or choose Continue from the marking menu.
4. Place the Section View at the desired location and commit.

Creating Base Views with Custom Orientations
When creating a Base View, a custom orientation can be applied. This can be done as follows.
1. In the modeling environment, rotate the model to the desired orientation.
2. In the Browser Tree, select Named Views > right click > New Named View. The custom orientation is saved with the specified name.
3. Save the design and update the drawing (choose Get Latest from the Reference menu in the QAT).
4. Launch Base View command. This time, the newly saved orientation appears in the Orientation dropdown in the command dialog.

![Image](image.png)

**FIGURE 6: CREATING NAMED VIEWS FOR CUSTOM ORIENTATION**

**Creating Annotations**

Fusion 360 Drawings support Dimensions, Leader, Text, Centerline and Center Mark, among other annotations. The various annotations can be accessed from the Annotation panel in the toolbar. Sample workflows for creating some annotations are shown below.

**Creating Dimensions**

With the Dimension command, objects, points, edges, existing dimensions, or two points can be selected. The most appropriate dimension type is created. When existing dimensions are selected, more dimensions can be chained to the selected dimension.

**Creating a Linear Dimension**

1. Launch the Linear Dimension command.
2. Select two points on a view to dimension the length. Or press spacebar to select an edge of the view. Place the dimension.
3. Once the dimension has been created, the command repeats itself. We can continue creating more linear dimensions or cancel to end the command.

**Creating a Baseline Dimension**

1. Launch the Baseline Dimension command.
2. Select an edge of an existing linear / aligned dimension.
3. With that edge as the base, more linear / aligned dimensions can be created by simply specifying the second point for the dimension.
4. We can continue creating more dimensions or cancel to end the command.

**Creating Text**

1. Launch the Text command.
2. Create a bounding box for the text.
3. Enter text into the In Place Text Editor and click outside, on the canvas, to commit.
Undo / Redo Commands

Undo and Redo options are available in the Quick Access Toolbar.

Saving Drawings

To save a drawing, click on the Save button from the Quick Access Toolbar. After the first save, a thumbnail for the drawing will appear on the data panel. Related Data option from the File menu (or the Data Panel) will now display a link to the Model and vice versa. Subsequent saves of the drawing will create new versions of the drawing.

Model Associativity

Updating Drawings to the Latest Model Version

Changes made to the model can be applied to related drawings with a single update command.

1. Create a drawing > Place some views and annotations
2. Switch to the design and make some geometry changes.
3. Save the design and switch back to the drawing.
4. The Reference menu in the QAT displays a warning about changes to the model. At this point, the drawing references the older version of the design.
5. To consume the changes to the model, select Get Latest from the Reference menu. All views update to reflect the latest geometry. Annotations update to display the new values.

![Figure 7: Updating Drawings](image)

Disassociated Annotations

Drastic changes to the model geometry or changes in the hierarchy of the components may result in annotations that are no longer associated correctly to views. In such cases, a yellow notification badge appears on the disassociated annotation. Clicking on this badge will display a contextual marking menu. We can choose to Delete the annotation or Re-associate it to new points on the view.

![Figure 8: Marking Menu for Disassociated Annotations](image)
Editing Drawings

Double Click to Edit
Double clicking on entities in the drawing (Views, Dimensions, Hatch, Titleblock, etc.) provides options to edit the entity. Below are some examples.

Editing Views
1. Double click an existing view. This brings up the View Properties dialog.
2. Properties such as scale, shading and edge visibility can be modified from the dialog.

Editing Dimensions
1. Double click an existing dimension. This brings up the Dimension Properties dialog.
2. Properties such as tolerance, alternate units, representation and inspection frame can be modified from the dialog. Precision, tolerance and alternate units are applied as overrides on the dimension.
3. To clear property overrides from a dimension, select the dimension > right click > choose Restore Defaults from the contextual overflow menu.

Editing Text
Text content in Leader, Text and Dimensions can be edited by double clicking on the entity. This pops up the In Place Text Editor. After editing the text, clicking outside, on the canvas, will commit the operation.
Move and Delete commands
The Move and Delete commands are available in the Modify panel.

**Moving Entities**
1. To move entities in the drawing, launch the Move command.
2. Select the entities to be moved by clicking on them one by one or making a cross selection or window selection.
3. When the selection is complete, hit Enter or choose Continue from the marking menu.
4. Specify the displacement by picking two points.

**Selecting Multiple Entities**
Operations such as Move, Delete, Restore Defaults work with an existing selection set. To create the selection set, we can use cross selection or window selection. Entities can also be added to / deleted from the selection set individually using the Shift key in combination with mouse click.

Advanced Features

Exploded Views
Drawings can be created from animation keyframes. In such drawings, the base view is an exploded view and will reference a specific time on the timeline of a storyboard.

*Creating a Drawing with Exploded Views*
1. Create or open a design with multiple components.
2. Switch to the Animation workspace.
3. From the Browser tree, right click on Components > choose Auto Explode: All Levels. Other Transformation options are available in the Transform panel on the toolbar.
4. File Menu > New Drawing > From Animation. The resulting drawing’s base view is a 2D representation of the transformations in the animation keyframe.

**Working with Exploded Views**

We can work with exploded views as we would with views created from the design. Projections, sections and detail views can be created from exploded views and annotations can be added to them. Changes made to the animation keyframe are reflected in the drawing.

**Bill of Materials**

Balloons and Parts List can be created from the BOM panel in the toolbar.

**Creating a Parts List**

1. From the BOM panel, select Parts List. A preview appears. When the cursor moves into the lower half of the canvas, the Parts List flips.
2. Specify a point on the drawing to place the Parts List.
3. Part Number, Description and Material columns reflect the properties applied to the design. To change these values, switch to the modelling workspace > select the component from the Browser Tree > right click and choose Properties (to change Part Number or Description) or Physical Material (to change the material applied) > Save the Design and update the drawing.
4. The Qty column indicates the number of copies of a component in the Design.

**Creating a Balloon**
1. From the BOM panel, select Balloon.
2. Select an edge on the view and place the Balloon. The Balloon remains associated with the drawing view.
3. Options to renumber or align balloons are available in the BOM panel.

<table>
<thead>
<tr>
<th>Item</th>
<th>Qty</th>
<th>PartNumber</th>
<th>Description</th>
<th>Material</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>Body</td>
<td>Steel</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>Arm</td>
<td>Steel</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>Axls</td>
<td>3 mm Copper</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>Head</td>
<td>Steel</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>Arm_MF</td>
<td>Steel</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>Axls_MF</td>
<td>3 mm Copper</td>
<td></td>
</tr>
</tbody>
</table>

**FIGURE 14: BILL OF MATERIAL**

**Geometric Dimensioning & Tolerancing (GD&T) Symbols**
Fusion 360 Drawings support GD&T symbols Datum ID and Feature Control Frame. They can be added from the Symbol panel in the toolbar. The dialogs for these commands also include a Symbol Palette that provides access to special characters, defined by the ASME and ISO standards (ASME Y14.5-1994 and ISO 1101:2012). Surface attachment option is also available for ISO which can be accessed from the contextual marking menu, after picking the entity to create the GD&T symbol on.

As with other entities, existing symbols can be edited through double click.

**FIGURE 15: GD&T SYMBOLS**
Sharing Drawings

Exporting Drawings
Drawings can be exported to PDF or DWG (AutoCAD 2013) formats using the options in the Output panel.

Collaboration
Fusion 360’s tools allow for easy collaboration and rapid iteration. Below are a couple of methods to share files.

Invite Project Members
We can control who has access to the design and other data within projects. To invite other collaborators to access a project:
1. Click People tab in the top portion of the Data Panel
2. Enter email addresses
3. Click Invite

More information on this can be found on the Fusion 360 Learning site: http://fusion360.autodesk.com/learning/ (Manage > Invite or Remove Project Members)

Sharing Files
To share individual files:
1. In the Data Panel, select the thumbnail of the document to be shared
2. Right click and select Share Public Link.

This provides various options for sharing the document. More information on this can be found on the Fusion 360 Learning site: http://fusion360.autodesk.com/learning/ (Manage > Share Documents)

Version Management

Updating Drawings to the Latest Model Version
This has been discussed earlier, under the section on Model Associativity (Page 9). This change can be undone.

Drawing Version History
Every Save operation adds a new version to the file. To view older versions of a file, click on the ‘i’ button on the file thumbnail in the Data Panel. We can open older versions or promote them as the latest.
Useful Resources

Learning Site
Information on the various features in Fusion 360 can be found on the Learning site or Autodesk Knowledge Network:

http://fusion360.autodesk.com/learning/
https://knowledge.autodesk.com/support/fusion-360

Fusion 360 Forum
To view trending discussions about the product, the roadmap, ask or answer questions, please visit the forum: http://forums.autodesk.com/t5/fusion-360/ct-p/1234