

MFG468049

Speed up drawing creation through the power of templates in Fusion 360

Stewart Sabadell
Chief Product Owner, Fusion 360 Drawings
Autodesk, Inc.

Andrew de Leon
Senior Principal UX Designer, Fusion 360 Drawings
Autodesk, Inc.

Learning Objectives

- Create a new custom title block designed for your business, brand, and industry needs.
- Create a template with document and sheet settings designed for your company and your projects.
- Create a "smart" template that can automatically generate sheets, views, parts lists, and bend tables.
- Learn the benefits of using templates versus preferences

Description

Drawing templates, when used to create drawings in Fusion 360, can both accelerate the time it takes to complete your mechanical engineering documentation, and ensure your drawings consistently represent your business and project standards. We will start by creating a new title block, with fields that can be automatically populated with information from the design, project, and your profile. We will add this to a new template, then walk through other options for customizing the template for your specific office and business needs. We will then show the power of "placeholder" elements in your template, and demonstrate how they can help remove much of the tedium of creating a new drawing "from scratch".

Speaker(s)



Stew Sabadell has spent his career working with Autodesk technology. For a number of years he was developing add-on applications for AutoCAD, then joined Autodesk. Within the year he was working on DWG import technology for 3ds max. After a number of releases of max, he joined the AutoCAD organization and contributed to five major releases. He joined the Fusion 360 Drawings team in 2014 as development manager, and has been in the Chief Product Owner role for a little over a year.



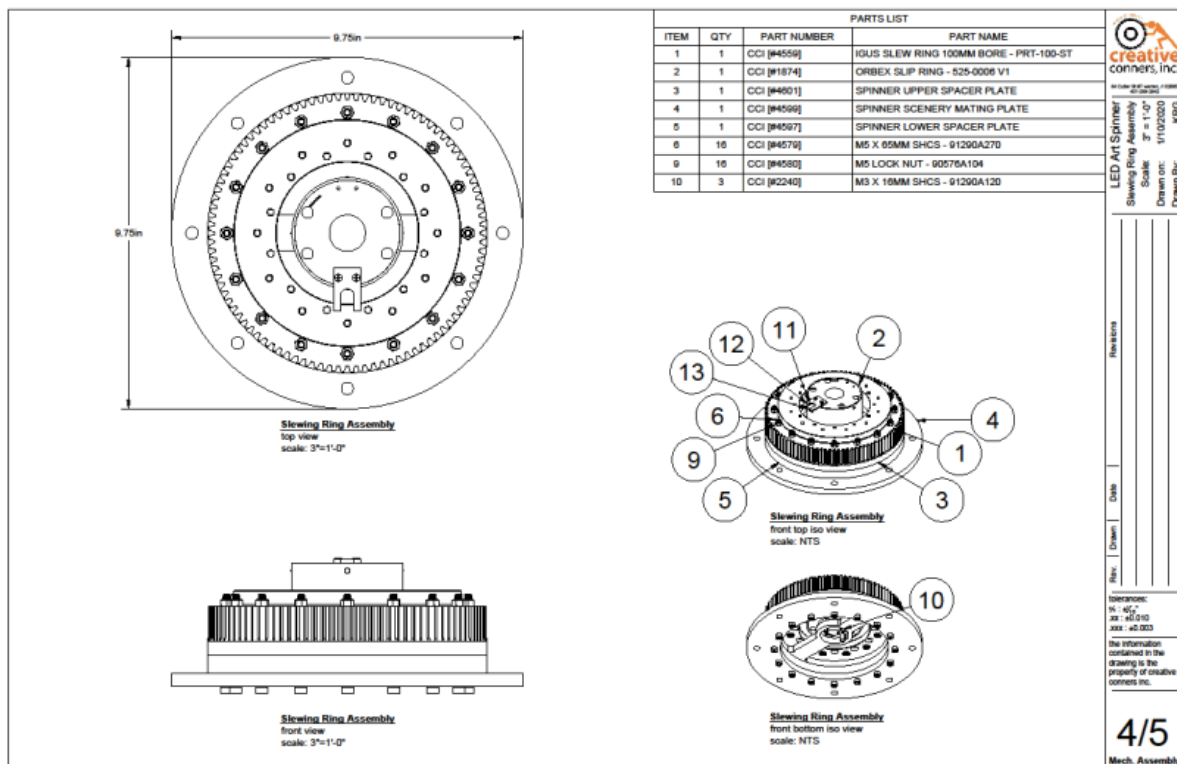
Andrew de Leon is a Senior Principal User Experience Designer at Autodesk, Inc., with 20 years' experience in the manufacturing industry and 15 years in user experience design. He has experience with AutoCAD, AutoCAD Mechanical, Inventor, and Fusion 360. He's passionate about manufacturing and design, and if he goes missing he's usually in the shed tinkering.

Introduction

Creating engineering drawings of your parts and assemblies is tedious but absolutely necessary. It is how you ensure your product is manufactured correctly, whether in-house or contracted out. Our goal with the Fusion Drawings workspace is to help you create accurate, on-standard drawings as quickly as possible.

How you start off - when you create a new drawing - can have a big impact on that process. We're going to show you how to streamline that process and get your new drawings set up so you can focus on documenting your designs.

The general idea is, when you create a new drawing, you want it set up to your company preferences and industry requirements. Fonts, sheet sizes, and graphical conventions for your symbols. For example, ISO or ASME, leading and trailing zeros, units, level of precision, and so on. You should not waste time re-customizing every new drawing.



COPYRIGHT © 2020 CREATIVE CONNERS, INC. USED WITH PERMISSION.

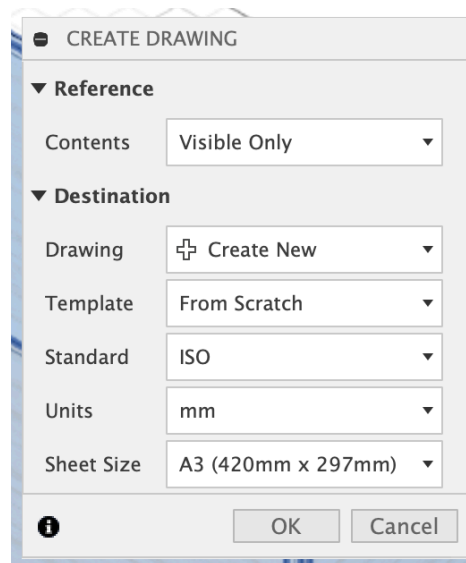
At the end of this class you will have the tools you need so that when you create any new drawing you can get directly to work, documenting your assembly and components for manufacturing.

We will first look at Preferences, then Templates, Title Blocks, and finally something we call "Smart" Templates, which go way beyond things like text height to get you well on your way to a complete document set.

Preferences

Drawings Preferences in Fusion can be confusing. The most important thing to know about Drawing Preferences is that these settings are only used for **new drawing creation**. They have **no impact on existing drawings**. This is different as compared to the rest of Fusion's preferences.

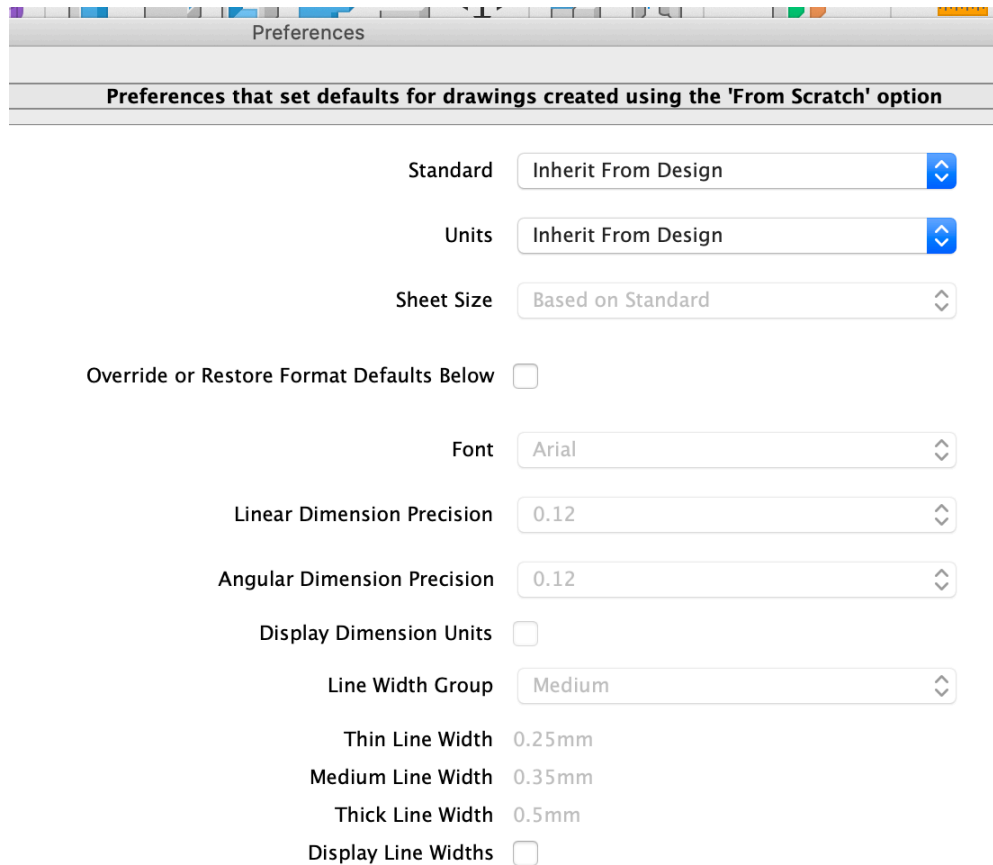
In addition, they only apply when you do not use a template. This is called "From Scratch" in the Create Drawing dialog. We will explore templates a bit later.



THE CREATE DRAWING DIALOG

These preferences persist through editing sessions and even updates to Fusion. They are stored with each user profile. They are even machine independent. Log in anywhere and you have your preferences.

Let's quickly review what Preferences will let you control. We'll only discuss settings that have some non-obvious behaviors.



Preferences

Preferences that set defaults for drawings created using the 'From Scratch' option

Standard Inherit From Design

Units Inherit From Design

Sheet Size Based on Standard

Override or Restore Format Defaults Below ☐

Font Arial

Linear Dimension Precision 0.12

Angular Dimension Precision 0.12

Display Dimension Units ☐

Line Width Group Medium

Thin Line Width 0.25mm

Medium Line Width 0.35mm

Thick Line Width 0.5mm

Display Line Widths ☐

DRAWING PREFERENCES

Standard

This option lets you choose between the ISO and ASME industry drafting standards. This is one of two preferences you cannot change once you have created your drawing. Once a drawing has been created, the standard is fixed. You can, however, choose which standard you want to use every time you create a drawing.

Inherit from Design

When you create a drawing, if your design units are metric it will default to ISO; if design units are imperial it will default to ASME.

ISO or ASME

You can instead set this explicitly to the one of the currently supported standards, and when you do this, a few more settings are exposed that give you even greater control:

- Sheet Size (the control is enabled)
- Projection Angle
- Text Height
- Display Trailing Zero
- Display Leading Zero

Preferences

Preferences that set defaults for drawings created using the 'From Scratch' option

Standard

ISO

Units

Inherit From Design

Sheet Size

A3 (420mm x 297mm)

Override or Restore Format Defaults Below

☒

Projection Angle

First Angle

Text Height

3.5mm

Font

Arial

Linear Dimension Precision

0.12

Angular Dimension Precision

0.12

Display Dimension Units

☐

Display Trailing Zeros

☐

Display Leading Zeros

☒

Line Width Group

Medium

Thin Line Width

0.25mm

Medium Line Width

0.35mm

Thick Line Width

0.5mm

Display Line Widths

☐

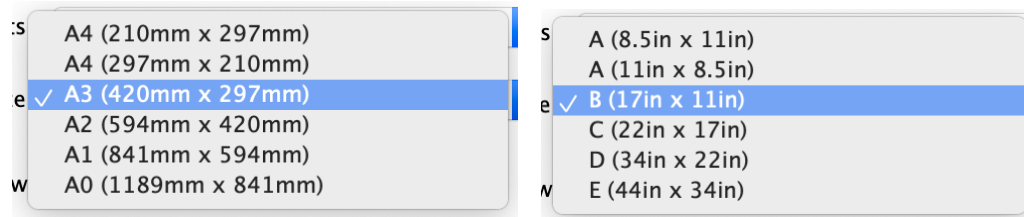
PREFERENCES, WITH ADDITIONAL OPTIONS DISPLAYED

Units

Pretty self-explanatory; if you set this to either mm or inch it will override the units you used to create your design or part. And when you use inch units you have access to fractional inches; see Dimension Format.

Sheet Size

The options here are based on the standard you choose. ISO sheets are metric sheet sizes, and ASME sheets are imperial sheet sizes. This option is only enabled if you choose a standard in preferences.



ISO AND ASME SHEET SIZES

Note that since ISO sheet sizes are in mm, the “paper” units are mm. With ASME sheet sizes, they are in inches. This plays a role when defining a new title block. We will explore that later.

Override or Restore Format Defaults toggle

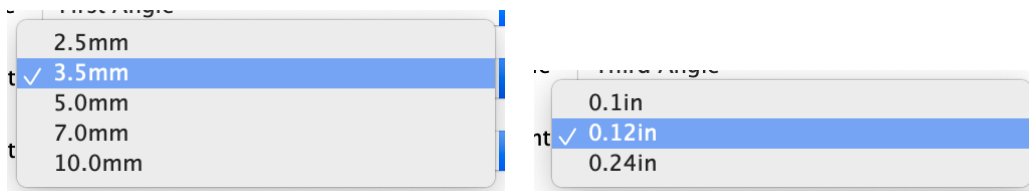
When you toggle this on you can override other defaults; when you toggle it back off, it will restore the default settings for the currently selected standard. The goal here is to help keep your drawings “on standard” for maximum industry compatibility.

Projection Angle

This option is only displayed if you choose a standard in preferences, and this is the only place you can change this if you want to deviate from the drafting standard. ISO calls for “first angle” projections, and ASME calls for “third angle” projections. Change this preference if you want to override this choice.

Text Height

This option is only displayed if you choose a standard in preferences, because the available text heights are either metric for ISO, or inches for imperial. The available heights are designed to help your drawings conform to industry standards.

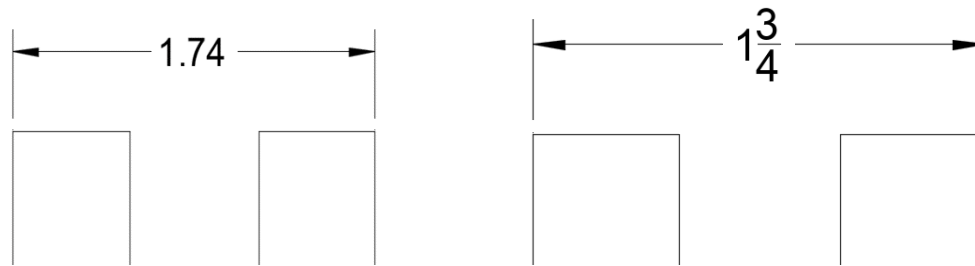


ISO AND ASME TEXT HEIGHTS

The preferences for Font, Linear Precision, and Angular precision are pretty self-explanatory.

Dimension Format

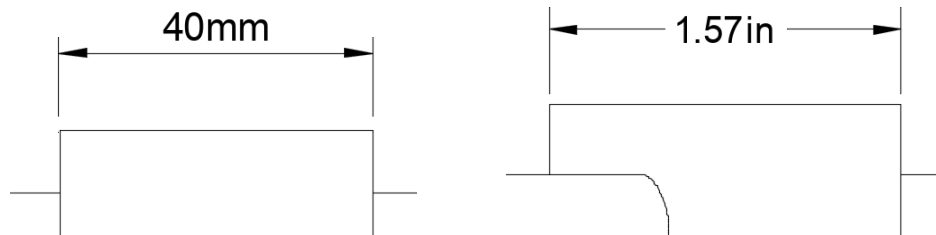
This preference is only displayed when inch units are explicitly chosen in preferences. Here you can choose between decimal inches and fractional inches.



INCH DIMENSIONS IN DECIMAL AND FRACTIONAL

Display Dimension Units toggle

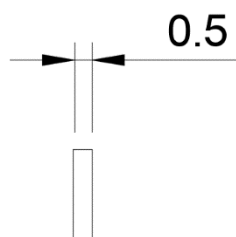
This preference will add unit text to your primary dimension values.



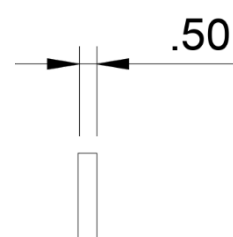
DISPLAYING DIMENSION UNITS IN MM AND INCHES

Display Leading Zero and Display Trailing Zero toggles

These options are only displayed if you choose a standard in preferences.



LEADING ZERO ON, TRAILING ZERO OFF



LEADING ZERO OFF, TRAILING ZERO ON

We will review line widths, a relatively new addition, a bit later.

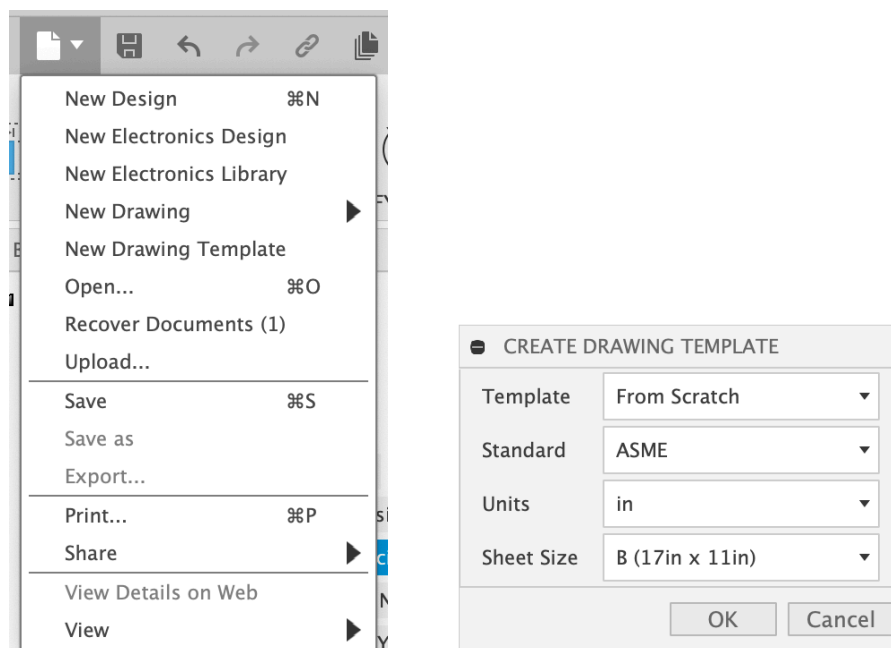
Templates

You can rely on Preferences settings for new drawings, or, you can get even more control over more aspects of your drawings by using customized templates. Templates let you control and customize all of the "settings" of a drawing, including a default sheet size, and then create new drawings using that configuration. Like Preferences, Templates are only used on drawing creation; editing a template does not have an impact on drawings already created from that template.

There are two main ways to create a template: from “scratch”, or from an existing drawing. You get this process started either from the File→New Drawing Template menu option, or using Save As and changing the format.

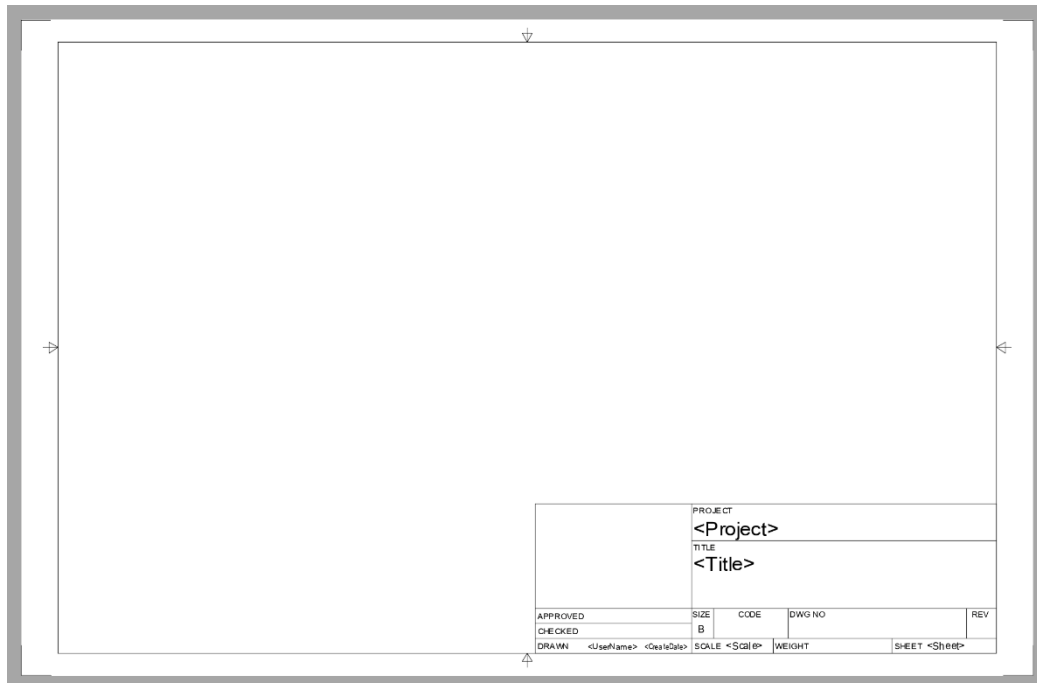
Creating a New Template “From Scratch”

When creating a template “from scratch”, you can control a few basic aspects of your new template. The Sheet Size can be changed later, but the Standard and Units cannot. This process will also use your Preferences, so if you have chosen ISO and Third-Angle Projection (for example), it will use that setting for your new template.



FILE MENU “NEW DRAWING TEMPLATE” AND THE CREATE DRAWING TEMPLATE DIALOG

After the new “from scratch” template opens, the first thing it does is prompt us to place a base view; you may cancel this operation if you want a very simple template. We will talk about “placeholder” views in the topic on Smart Templates later. You do not get this automatic prompt when using the Save As creation approach.

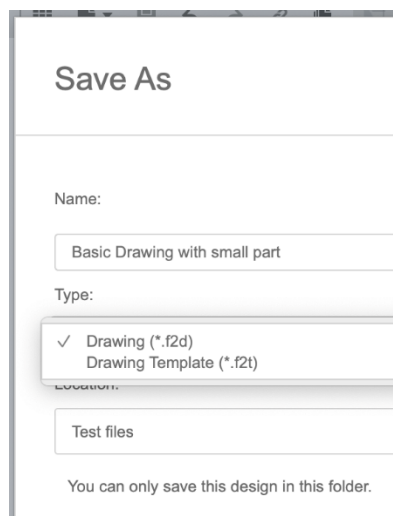


A BRAND NEW ASME DRAWING “FROM SCRATCH”

Creating a New Template from an Existing Drawing

If you have been using Fusion and creating drawings for some time, and are ready to create a template, this can be a time-saver, as the Document and Sheet settings will already be set to your liking.

The Save as command is available on the File menu. When the dialog comes up, change the Type to “Drawing Template (*.f2t)”.



SAVE AS TO “DRAWING TEMPLATE”

Using this process, your new template will “inherit”:

- All Document settings
- All existing sheets, including sheet sizes and sheet names
- Title blocks, both those displayed on each sheet and those available for use within the drawing
- Any non-associative objects, such as text, images, and generic (editable) tables

This means the following elements are not brought into the new template from the source drawing:

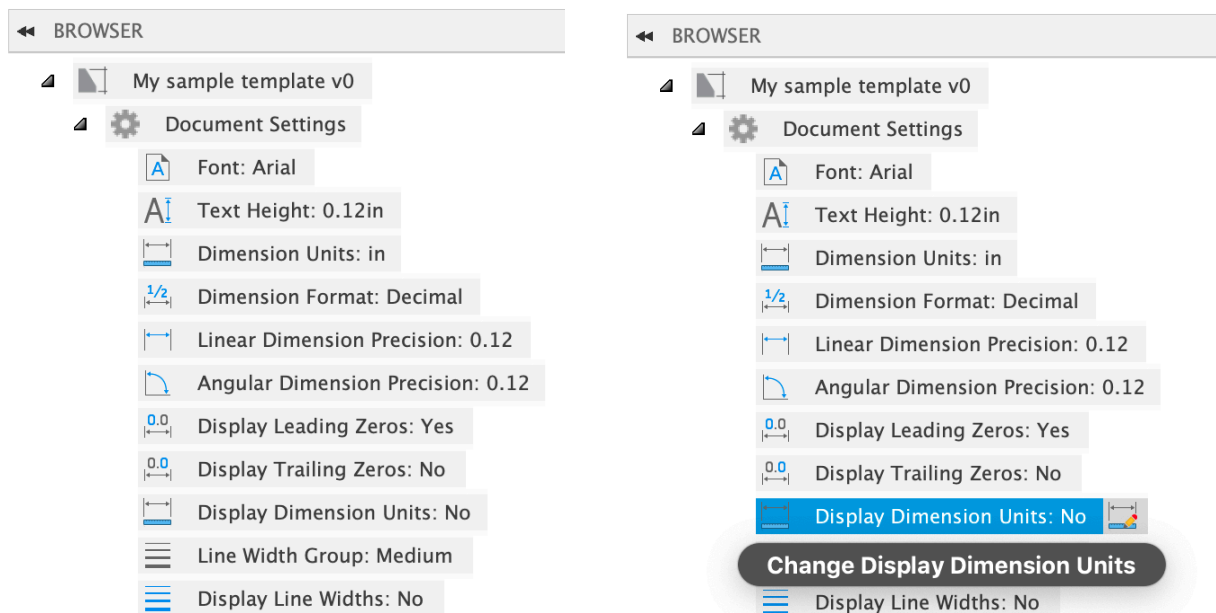
- Model views, including section lines, detail bubbles, and section and detail view labels
- Dimensions, edge extensions, leaders, and balloon notes
- Center marks and centerlines
- Parts lists and bend tables
- Datum ID, Feature Control Frame, and Surface Texture symbols
- Hole & Thread notes
- Drawing sketch objects

Editing Your Template

Once in your template, you will want to customize Document Settings, Sheet Settings, your Title Block, and you can even add a title sheet.

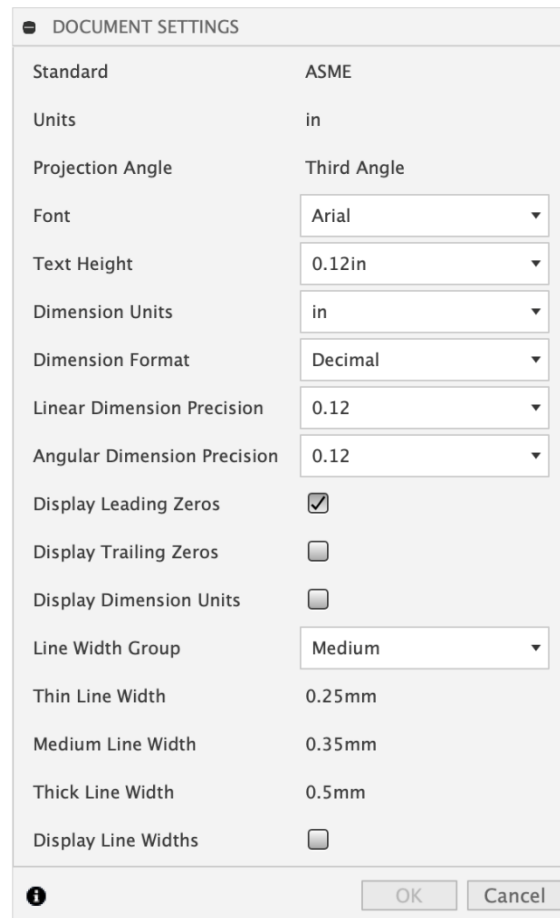
Document Settings

These are accessed in the same way as in a drawing – through the component browser.



DOCUMENT SETTINGS; HOVER OVER ANY ITEM TO DISPLAY THE EDIT BUTTON

The browser displays all of your current settings. If you want to edit any of them, hover over any setting, and an edit icon will appear. Click on the icon and the Document Settings dialog comes up.



DOCUMENT SETTINGS	
Standard	ASME
Units	in
Projection Angle	Third Angle
Font	Arial
Text Height	0.12in
Dimension Units	in
Dimension Format	Decimal
Linear Dimension Precision	0.12
Angular Dimension Precision	0.12
Display Leading Zeros	<input checked="" type="checkbox"/>
Display Trailing Zeros	<input type="checkbox"/>
Display Dimension Units	<input type="checkbox"/>
Line Width Group	Medium
Thin Line Width	0.25mm
Medium Line Width	0.35mm
Thick Line Width	0.5mm
Display Line Widths	<input type="checkbox"/>

EDITING DOCUMENT SETTINGS

The list of settings should look familiar; they are the same settings that we saw in Preferences! You should notice, however, that you cannot change the Standard, Units, or Projection Angle of your template. The same is true for Document Settings in your drawings. You need to make the best choice for your project at the time you create your drawing (or, in this case, template).

Changes to Document Settings affect the entire drawing – all sheets.

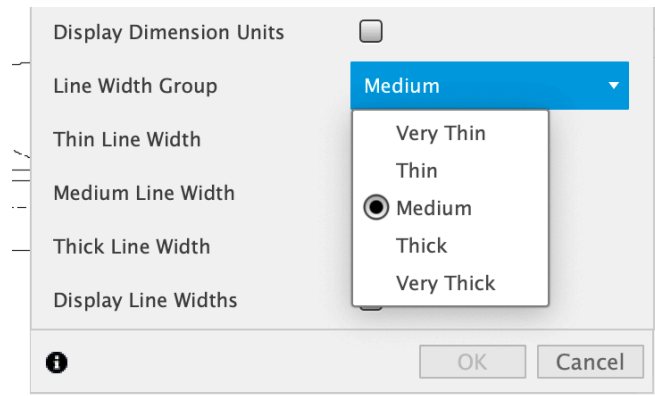
Font

We did not discuss fonts when reviewing preferences, but it is worth a mention here. Changes to the font will change all text in your drawing, with one key exception: title blocks. We will discuss this a bit more when we cover title blocks.

Line Widths

We also did not cover line widths when reviewing preferences. Here's how it works. View geometry and various kinds of annotation all have pre-selected line widths within each

group. That is, outside edges of your view geometry use the Thick width, the default width for your Title Block and Sketch objects is Medium, and all other objects use the Thin width. You can control the actual line “weights” by changing the Line Width Group setting.



CHANGING THE LINE WIDTH GROUP



LINE WIDTHS WITH THE VERY THICK GROUP

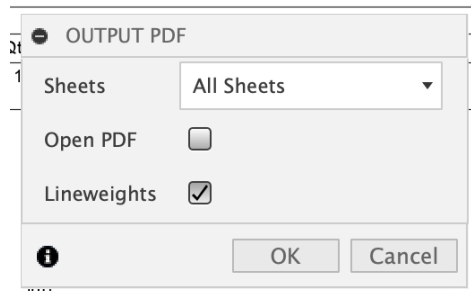


LINE WIDTHS WITH THE VERY THIN GROUP

As with all other Document Settings, these changes impact all sheets in your drawing.

When you have the Display Line Widths toggle turned on, as you zoom in and out, you will see the widths get thicker and thinner on screen. This is a “what you see is what you get” display of line widths. If you have a complicated view on a small sheet, you may find that the geometry details can’t be seen if the widths are too heavy. The same will be true when you print, if you print with line widths. In this case, you can change your Line Width Group to a thinner setting.

If you choose not to display Line Widths when editing your drawing you can still have them in your PDF when you export.



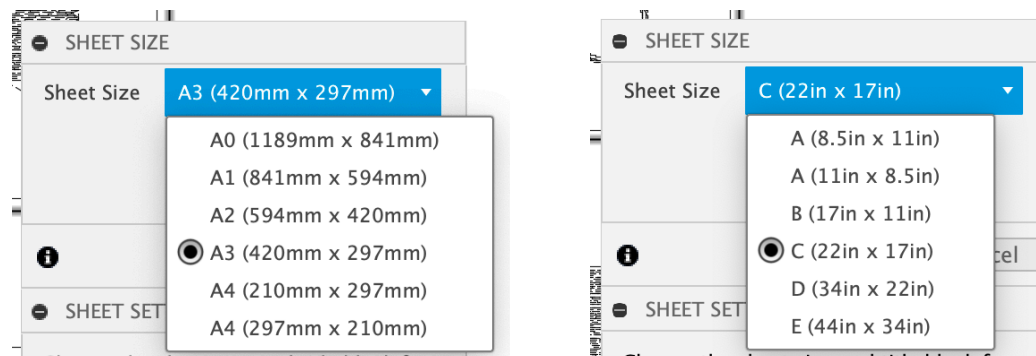
LINEWEIGHTS TOGGLE SEEN ON OUTPUT PDF DIALOG

Sheet settings

As the name suggests these settings apply just to the current sheet in your template (or drawing).

Sheet Size

As mentioned in the Preferences section, sheet sizes are per the chosen standard. A4 through A0 sheet sizes are available for ISO drawings, and A through E sheet sizes for ASME drawings.



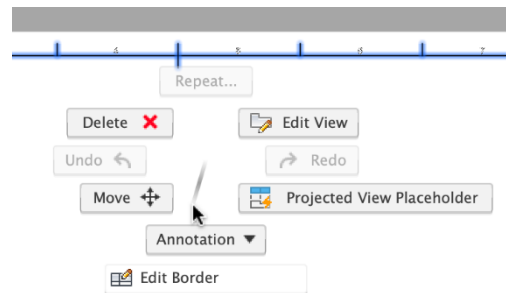
SHEET SIZE DIALOG, SHOWING ISO AND ASME SHEET SIZES

Other Aspects of your Template

You can create and edit title blocks, which we will look at shortly. But what else might you customize about your template

Edit Sheet Border

Each sheet size provided in Fusion 360 includes a predefined border. These borders are based on ISO and ASME standards, however you may edit the border. You can do this by selecting the border, right-click, and choose Edit Border.



ACCESS TO THE EDIT BORDER COMMAND

The environment for editing your border is identical to that seen when editing your title block, which we will look at next. Remember, though, that your edits are specific to the current sheet size. If you want to make a change reflected on all sheet sizes you need to edit each of those borders.

Create a Title Sheet

Your sheet sets may typically include a title sheet for a project. Your template is the best place to add that, if there are common elements you want to use in more than just one drawing.

Create Multiple Sheets

If your typical drawing has a parts list on its own sheet, or different title blocks on different sheets, you can create and name those sheets in your template, and they will be there waiting for you in drawings you create from the template.

Add Typical Blocks of Text

If you frequently have some boilerplate text you include on sheets, you can write and add it in your template, and it will be there in all drawings you create from that template.

Project-specific Information

If you will be creating a number of drawings for a single project, you might create a template just for that project.

Add a Revision Table

You can add a generic "empty" table to your template so that you always have a revision table you can use. It can be customized with the columns you need.

Title Blocks

We've been looking at sheet settings, but haven't really looked closely at title blocks yet. They are a topic all their own! One of the most important and prominent aspects of your sheets is your title block. As you are well aware, every sheet in every drawing nearly always has a title block; it is a fundamental part of your mechanical engineering drawings.

Title blocks carry crucial information about each sheet, and the information contained on that sheet. They can also be your "company signature" for documents you send out of house - your "brand".

As you have seen in our templates, Fusion includes built-in title blocks that conform to ISO and ASME standards, and they might be perfect for you as is.

Dept. Machining	Technical reference Spec 24.7a	Created by Stewart Sabadell 3/13/20	Approved by Andrew De Leon Nov 17 2020
		Document type CAM Dimensions	Document status Pre-release
		Title Plate	DWG No. 14-2297
		Rev. 1	Date of issue Nov 16 2020
		Sheet 1/1	

ISO TITLE BLOCK INCLUDED WITH FUSION

				PROJECT AU 2020 Demonstration			
				TITLE Plate			
APPROVED	Andrew De Leon	11/17/20	SIZE	CODE	DWG NO	REV	
CHECKED	Chris Miller	10/31/20	A	42-a	14-2297	1	
DRAWN	Stewart Sabadell	3/13/20	SCALE	1:2	WEIGHT	1.3 kg	SHEET 1/1

ASME TITLE BLOCK INCLUDED WITH FUSION

However, if you want:

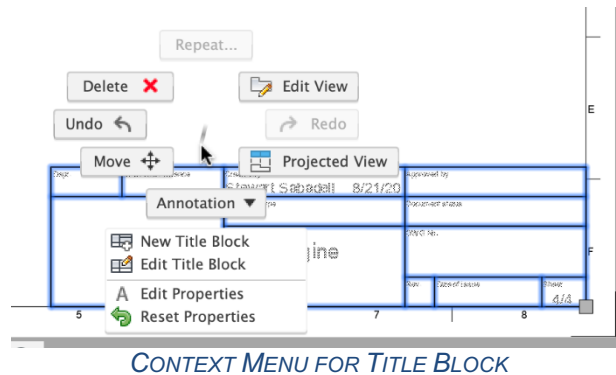
- More information in your title block
- A different layout
- Your corporate logo
- Your company address

Then you will want to build a custom title block.

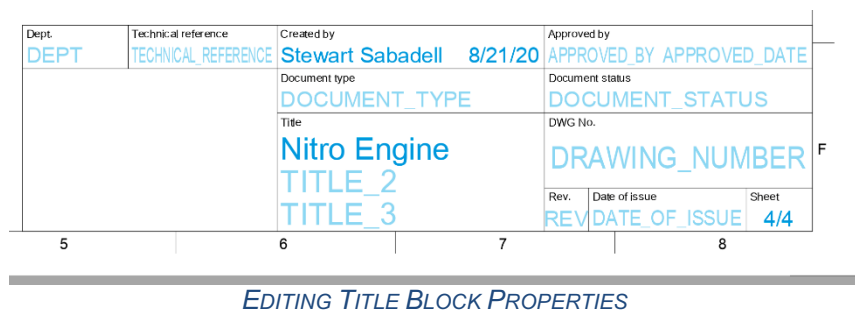
Let's say you want to create your own title block. Before we do that it is important to understand there are two ways to "edit" a title block. You can edit the values that you see in each "cell" in a single title block, on a single sheet. Or, you can edit the "definition" of the title block itself, and see those changes on all of the sheets that contain that title block.

Edit Title Block Properties

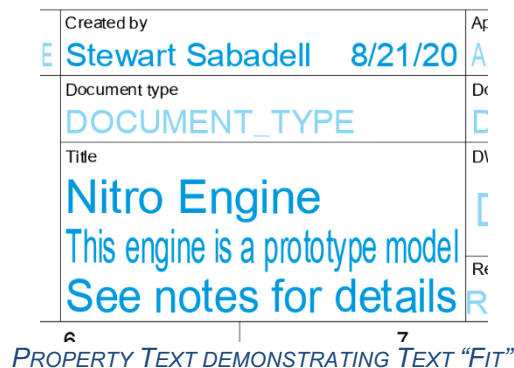
If you want to edit the text values in a title block on just one sheet in your drawing, you can *Edit Properties*. Select the title block, right-click, and choose Edit Properties.



The workspace zooms into your title block, and you now see that some of the text in your title block is light blue, and, you see more text. Lighter blue text are placeholders, showing you what information belongs there. Just click to edit. Darker blue text are properties that already have text; they too can be edited. The changes here only affect this sheet.



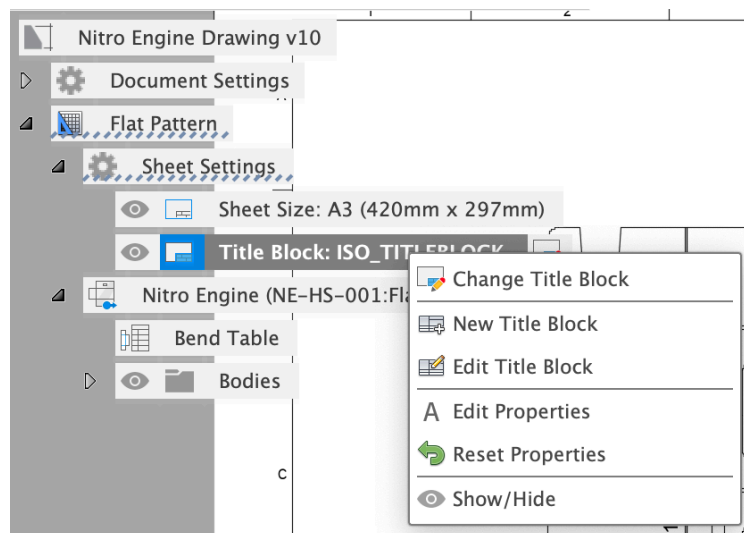
Property text in title blocks has a special behavior that will "squish" your text – make the letters narrow - if the string is longer than will fit in the available space. It may cause some confusion when editing a title block definition, because the "attributes" may overlap, but when you use your title block it can give you a very clean result. You get this behavior with the title blocks provided with Fusion 360, and with your own title blocks.



When you are finished editing the light and dark blue text, click on Finish Properties in the dialog. Now notice that if you switch sheets, the changes you made on one sheet are not reflected on the other sheets.

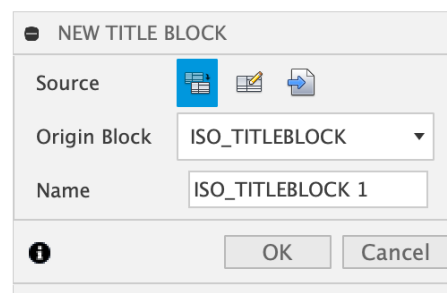
Add and Edit Title Block Definitions

If you want to change the layout, graphics, and/or information contained in a title block, then you want to create a new title block or edit your custom title block. We will start with the title block we have here and make a few modifications. You can create a title block by selecting either the title block in the drawing, or the element in the component browser; right-click and choose New Title Block or Edit Title Block.



CONTEXT MENU FROM COMPONENT BROWSER

The title blocks included with Fusion Drawings can't be edited, but you can start with one, create a new one, and edit that.



CREATING A NEW TITLE BLOCK

You have three options when creating a new title block:

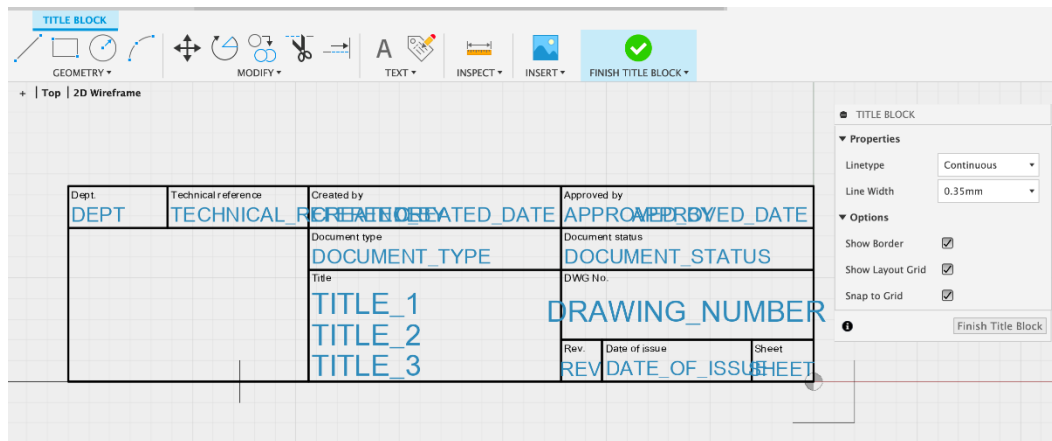
1. You can start with an existing title block, and provide a new name for the copy (which you then edit)
2. You can start with a "clean slate" (from scratch), and create your title block from the beginning
3. You can use an existing AutoCAD DWG file, bring that in, and edit it.

We will look briefly at each of these options.

Creating a Title Block from an Existing Title Block

The simplest path is to say you want to edit the built-in title block. You choose a name for the title block you are creating, which will be a copy of our built-in title block, then say OK to enter title block “edit mode”.

Once you are in “edit mode”, you will be in a different environment where the only thing you are editing is the title block. You can see the border; we display that for reference.



TITLE BLOCK EDITING ENVIRONMENT

Notice the gray background, and that the toolbar has a very different set of tools. We also remove the component browser. In this mode you can add the lines you need for a title block; insert images; add text; and add “attributes”. An Attribute is like a text placeholder. You specify what value it should hold, as well as other properties like text height and placement. When you later use this title block you can enter text into this “placeholder” and it can be unique on each sheet. All other aspects of your title block, including text, will be consistent on all sheets where this is used.

Units in your Title Block Definition

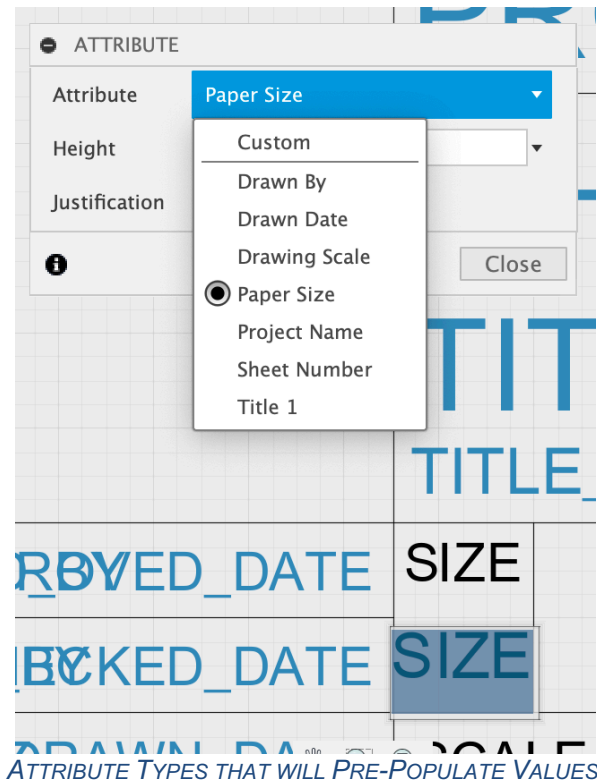
If you are editing a title block in either a drawing or template that is based on the ISO standard, your units in the title block edit environment are mm. Likewise, if your drawing or template is based on ASME, your units are inches.

Creating Geometry for your Title Block

A grid “snap” is turned on by default to help you draw lines and place text. In an ISO drawing, the snap spacing is 1 mm; in an ASME drawing snap spacing is 1/16”. You can turn off both this snapping and the grid if you wish.

Add Text and Attributes

Adding text means it will be the same everywhere this title block is used. Adding attributes gives you a place where you can later easily add text that varies per-sheet. But attributes can also be pre-populated for you when you create a sheet. There are seven attribute “types” that will do this.



The seven types are:

- Drawn By: the person who created the drawing
- Drawn Date: the date when the drawing was created
- Drawing Scale: the scale of the first view that is placed on the sheet
- Paper Size: the letter-code size of the sheet
- Project Name: the name of the project (folder) where your drawing resides
- Sheet Number: displayed as x/y (sheet x of y)
- Title 1: the name of the design you are documenting

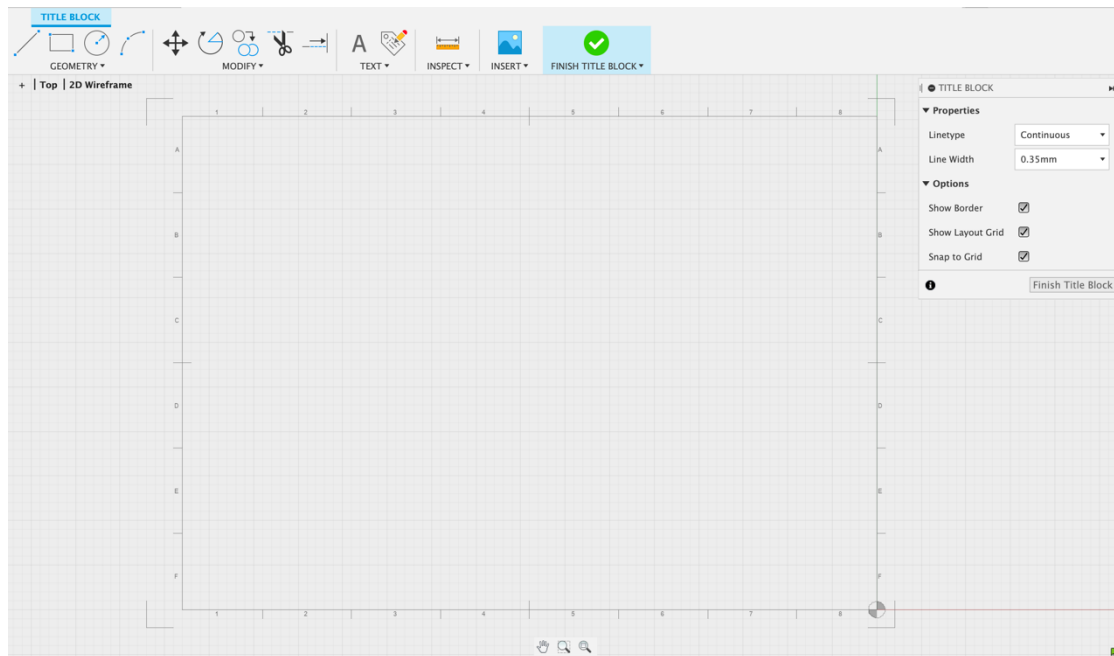
The values that are added for you can be replaced by you in your drawing, when this title block is used; but as a start they can speed up drawing creation by doing some of the work for you. Paper Size and Sheet Number are updated if you change your sheet size, or rearrange the order of your sheets. The other properties are set at the time the sheet is created.

Fonts

Both text and attributes default to the Arial font, but that can be changed when editing your title block. The text and attributes in our built-in title blocks have a special property, such that changes to the Document Settings font do not change the font of this text. However, text and attributes that you add to your title block while in the editor will change with the Document Settings font. You can control this when creating text by choosing a different font (this must be done while directly editing the text), but if this is a concern for attributes, the current solution is to insert a DWG file as your title block. See below.

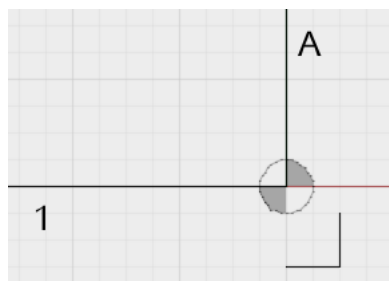
Creating a Title Block from Scratch

If you want to “start fresh” you certainly can. This is what the environment looks like when you first enter it.



TITLE BLOCK EDITING, STARTING FROM SCRATCH

The border is shown to help orient you in the space. The lower right-hand corner of the border is the “anchor” for your title block. That “anchor” is indicated with the half-filled circle icon. This is the point used to be able to correctly place the title block on sheets of different sizes. It is not possible to create a title block that is positioned in the lower left or upper right-hand corners of every sheet size – if you need that, you will need a different title block for each sheet size.



MARKER FOR TITLE BLOCK ANCHOR POINT

Inserting a DWG

The third option for adding a title block to your drawing or template is to import it from its own AutoCAD DWG file. There are a fair number of “rules” you need to follow for this to be successful. Many of these “rules” rely on an understanding of AutoCAD and DWG file terminology.

- The lines, text, and attributes for your title block should be in model space in the drawing.
- Your drawing cannot contain any block definitions.
- The insert point for the drawing must be at the lower right-hand corner.
- You should pay attention to the Layer(s) that your geometry, text, and attributes are on. Ideally place it all on Layer 0; when the title block is inserted into your drawing, the objects will be on the layer used for title blocks.
- For text and attributes, the Text Style you use is important. If you use the Standard text style, and the Document Settings font is changed, it will impact your title block font. Create or choose a different text style to avoid this.

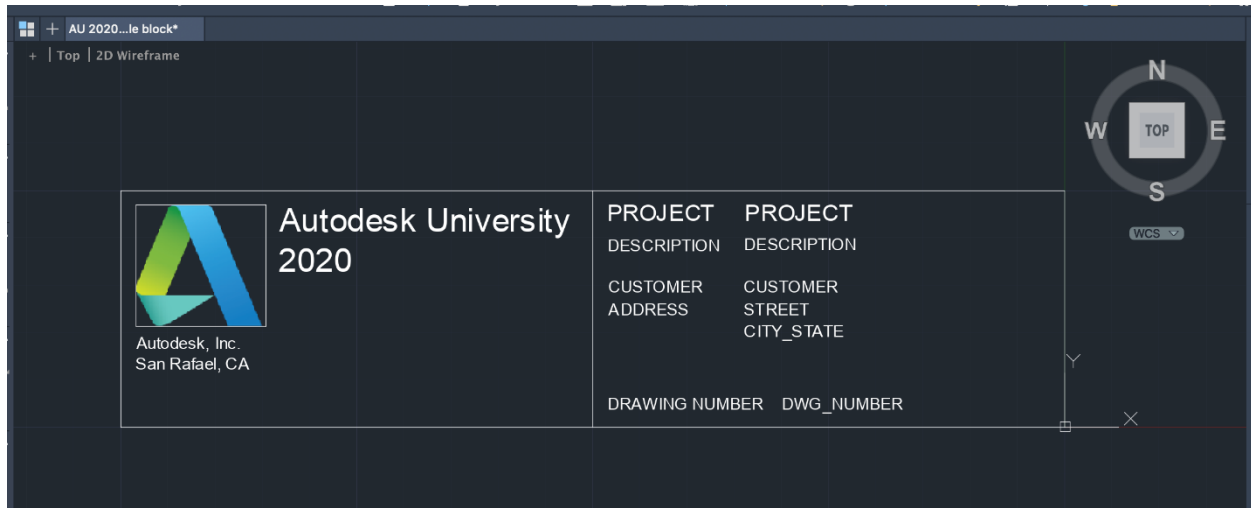
Attributes

If you define attributes, the attribute tag name needs to be chosen with care. The tag names listed below will cause your attribute to be automatically filled in when the title block is used, such as when a new sheet is created. This can be used effectively if that is your goal (value to be filled in for you). However, if you do not want that behavior, use a slightly different attribute tag name.

- TITLE_1
- PROJECT
- DRAWN_BY
- DRAWN_DATE
- CREATED_BY
- CREATED_DATE
- SIZE
- SCALE
- SHEET

Creating your Title Block in AutoCAD

If you follow the “rules” above, your AutoCAD drawing should look something like this before you save.



TITLE BLOCK DRAWING SHOWN IN AUTOCAD

In the image above, note that everything that makes up your title block is in model space. You do not define a block in this drawing – the drawing *is* the block definition. So avoid using BEDIT. Also note the X/Y coordinate system indicator (called UCS in AutoCAD). It is located on the lower right-hand corner of the title block. This is important so that your title block is placed correctly when used on any sheet size.

What might prevent your title block from being inserted? If your drawing contains any named block definitions, other than those used in dimension styles, multi-leader styles, or are used by section and detail views, you will not be able to use this drawing as a title block. You will need to edit the drawing and purge those block definitions.

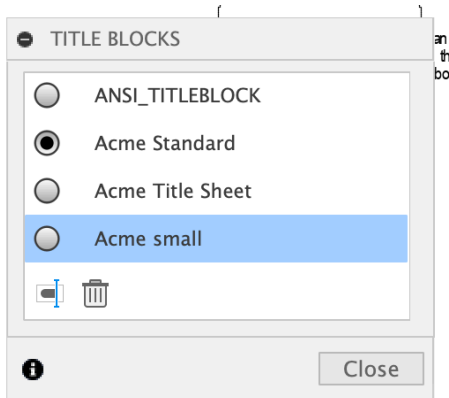
If your drawing contains 3D solids or a geo-location, the information is filtered out. If your drawing contains one or more fields, they are evaluated and converted to text. In any of these cases, you can still use the drawing as a new title block.

All layouts (sheets) in your drawing, and the information on them, is ignored.

Managing Title Blocks

A fairly new feature to Fusion is the ability to rename and delete title blocks. You can have multiple title blocks defined in each drawing and template, but only one used per sheet.

In the component browser, under the Sheet Settings node, hover over the Title Block node, then click on the button that appears. A dialog is displayed. You can choose which title block to display on the current sheet with the radio buttons. Or, if you highlight the row, you can rename and delete the highlighted title block in this drawing.



MANAGE TITLE BLOCKS IN A DRAWING

In the image above, *Acme Standard* is the title block used on the current sheet, and *Acme small* is the title block that will be affected by the rename and delete buttons.

Title Blocks in Templates

We have looked at creating and editing title blocks in a drawing, but if you do this, you then must re-do all that work in every drawing. The real time saver comes when you create and store your company title blocks in your templates. This is really where your title block definitions belong – in your template(s).

When you create or open a template, the interface to edit title block properties and title blocks themselves is the same as in a drawing. There is one key difference, and it has to do with those special attribute types we looked at earlier.

			PROJECT <Project>				A
			TITLE <Title>				
APPROVED			SIZE	CODE	DWG NO		REV
CHECKED			<Size>				
DRAWN	<UserName>	<CreateDate>	SCALE <Scale>		WEIGHT		SHEET <Sheet>
2			1				

ASME TITLE BLOCK IN A TEMPLATE

Dept.	Technical reference	Created by <UserName> <CreateDate>	Approved by
		Document type	Document status
		Title <Title>	
		Rev.	Date of issue

ISO TITLE BLOCK IN A TEMPLATE

Notice that some of the attributes contain text in <> angle brackets. This is to represent the special attributes that will automatically be filled in when you use this template and title block. *You should not edit this text!* If you use Edit Properties, while in the template, and change any of the text in an attribute that has text in <Angle Brackets>, then Drawings assumes you want a specific value used whenever this title block is used, and those attributes will no longer be filled in with the expected values.

Smart Templates

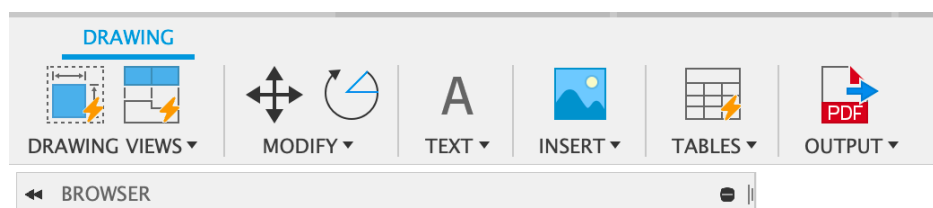
OK, we have templates now that are fully customized for our purposes. New drawings will not need further customization. **But templates can do so much more.**

What does your typical drawing set look like? You likely have a sheet showing the full assembly. You might have an exploded view of your assembly, which may be a good place to include a parts list. For complex assemblies that have "sub-assemblies", you might have sheets for those. And for non-stock parts, you may have individual "component" sheets, one sheet for each component, with detailed dimensions, tolerances, notes, etc.

There is a lot of repetition involved in creating one of those "standard" sheet sets. What if sheet and view placement could be automated?

It can!

When you created a template "from scratch" earlier, it prompted you to place a "placeholder" base view. That is how you make a regular template "smart". You can also directly choose to add placeholder views, from the toolbar you see when editing a template.



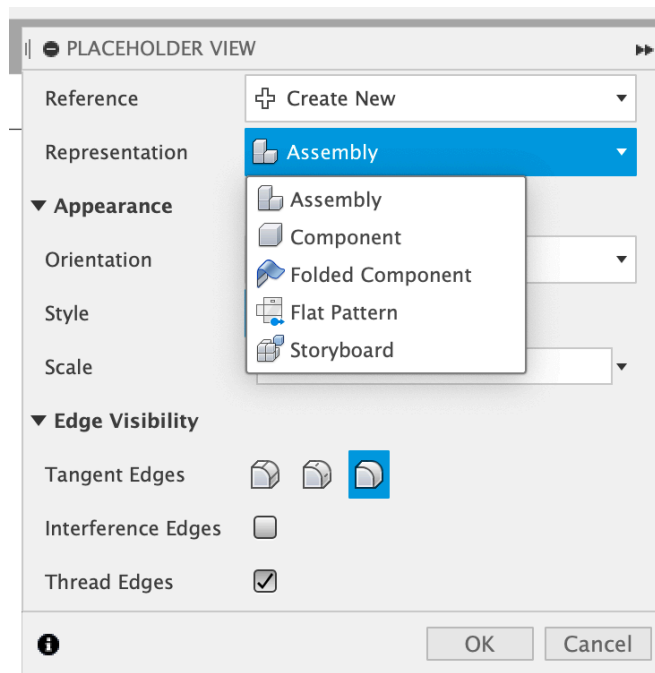
THE TOOLBAR IN TEMPLATE EDIT MODE

The first two buttons on the toolbar are to place views, but those are placeholder views, because this is a template. You can place base and projected placeholder views in your sheet, and when this template is used to create a drawing, they will be replaced by the actual model.

Sheet Types

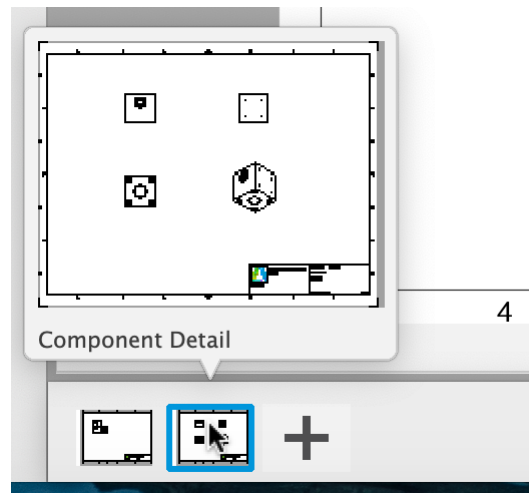
In fact, the views you place determine the sheet type. There are five different sheet types:

- Assembly
- Component
- Sheet Metal Folded Component
- Sheet Metal Flat Pattern
- Storyboard



PLACEHOLDER VIEW DIALOG

You can quickly check the sheet type by looking at the sheet name. Hover over the thumbnail at the bottom of the interface; it displays a larger thumbnail image, and the name of the sheet.



SHEET THUMBNAIL AND NAME IN A SMART TEMPLATE

View Types and Orientation

Each of the sheet types shown above is accessed when using the Base View Placeholder command – they are all “base” views. You can also create Projected View Placeholders from your base views, just as you can normally.

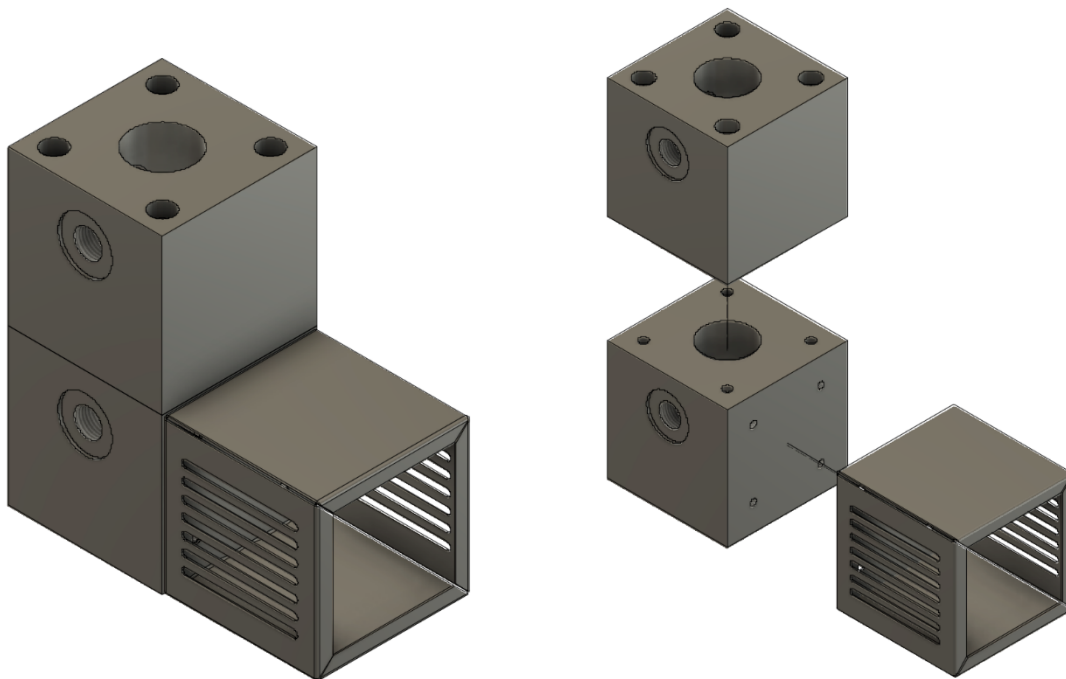
You should pay attention to the orientation of each view to ensure it will match your expectation when this template is used for your model. Do you want the Top, Front, and Right views displayed? Or perhaps all six sides? You decide when you place the placeholder views.

However, if you find, in your resulting drawing, that you do not like the views chosen, you can just change them in the drawing that was created from a smart template, on a per-sheet basis. You can even choose Named Views from your model to replace the chosen base view orientation. This is useful if you have components that are not aligned orthogonally with the rest of your assembly.

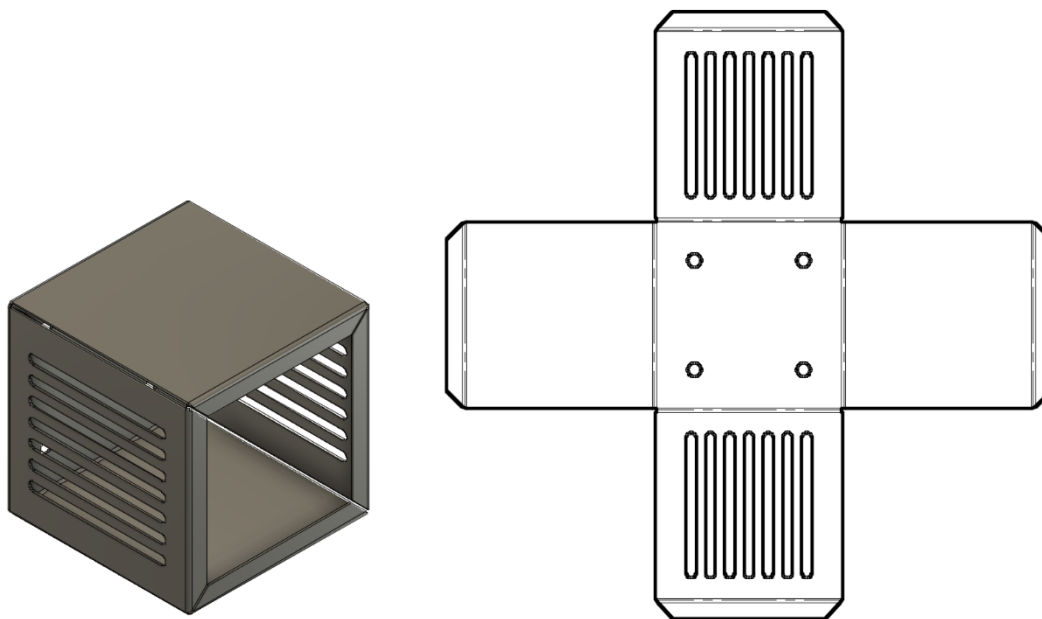
There are two view types that, for obvious reasons, cannot be added as a placeholder to your smart template – section views and detail views. You will have to add those in your drawing, where your section line and detail bubble can reference your model’s geometry.

The Placeholder Model

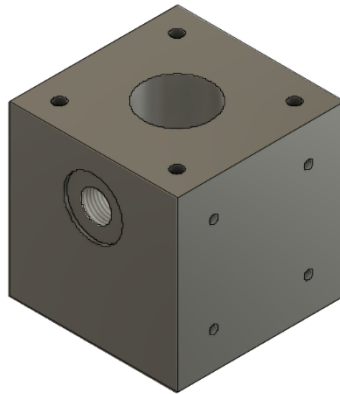
We’ve designed a very generic model for these placeholder views; the specifics of this model are unimportant. But the view orientation is important, as it will be used to place the views of your model, when the template is used to create a drawing. This placeholder view looks like this, in the five different view types. (All but flat pattern shown here in a shaded isometric projection.)



PLACEHOLDER MODEL – ASSEMBLY AND STORYBOARD VIEWS

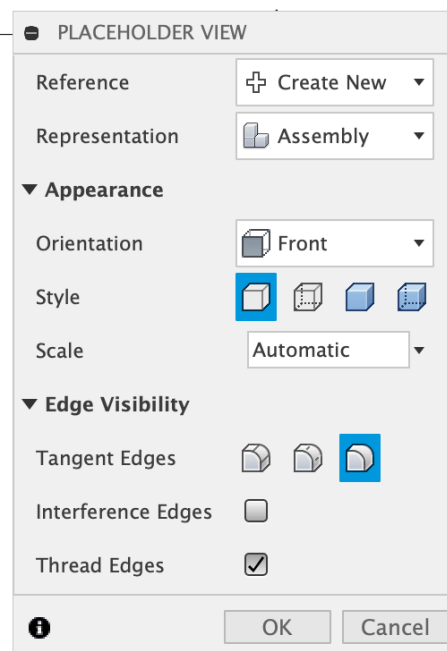


PLACEHOLDER MODEL – FOLDED COMPONENT AND FLAT PATTERN VIEWS



PLACEHOLDER MODEL – SINGLE COMPONENT VIEW

One of the things you might be thinking about: what scale will be used when these placeholder views are replaced with my actual model? As we do now, when placing a base view in a drawing, we look at the extents of the model and make a best guess as to view scale. In the case of a smart template and placeholder views, we call this out with “Automatic” scaling. If you instead choose a specific scale, we will use that scale for your real model, regardless of the model’s size.



PLACEHOLDER VIEW DIALOG

Parts Lists

Views are not the only thing you can add to your smart template that will be “replaced” with the real thing when the template is used to create a drawing. You can also place a parts list. In your

template, the parts list reflects the components of the placeholder model; when it is replaced in your drawing it will reflect the components in *your* model. You can even use one of the new features of parts lists, the All-Level parts list.

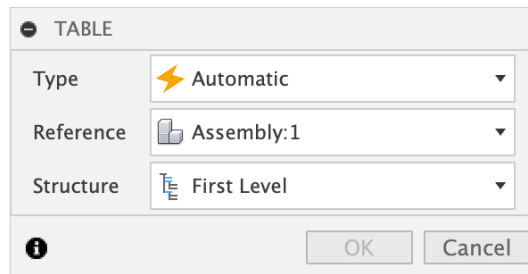
PARTS LIST				
ITEM	QTY	PART NUMBER	DESCRIPTION	MATERIAL
1	1	SUB-ASSEMBLY		STEEL
2	1	COMPONENT3		STEEL

FIRST-LEVEL PARTS LIST AS A PLACEHOLDER

PARTS LIST				
ITEM	QTY	PART NUMBER	DESCRIPTION	MATERIAL
1	1	SUB-ASSEMBLY		STEEL
1.1	1	COMPONENT1		STEEL
1.2	1	COMPONENT2		STEEL
2	1	COMPONENT3		STEEL

ALL-LEVEL PARTS LIST AS A PLACEHOLDER

If you want to place a parts list on its own sheet, you can do this, too. Start the Table command, then check or change the Reference option in the dialog to choose the model reference you want to document with your parts list.

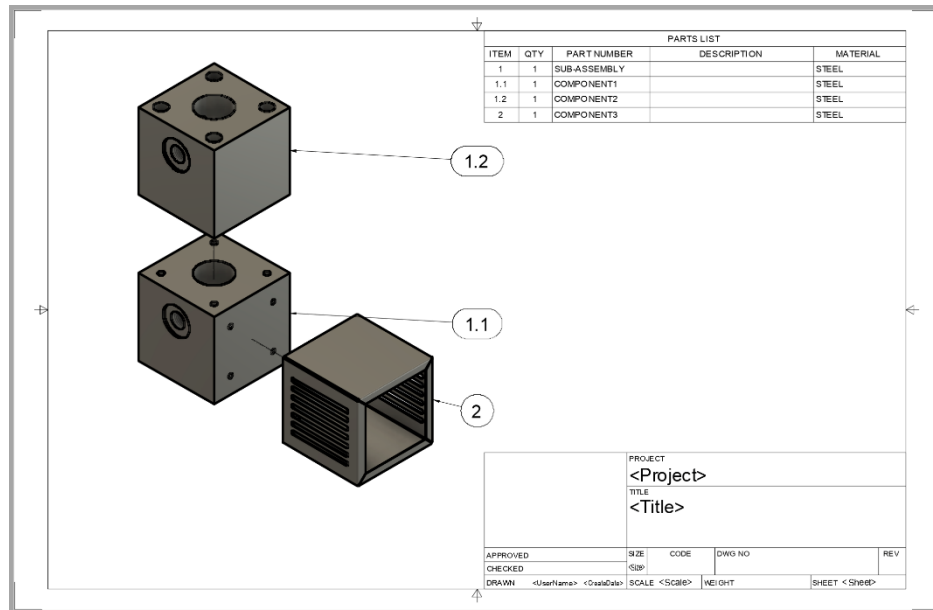


The dialog box is titled "TABLE". It contains three dropdown menus: "Type" set to "Automatic" with a lightning bolt icon, "Reference" set to "Assembly:1" with a document icon, and "Structure" set to "First Level" with a tree icon. At the bottom are "OK" and "Cancel" buttons, and an information icon on the left.

TABLE COMMAND DIALOG

Balloon Notes

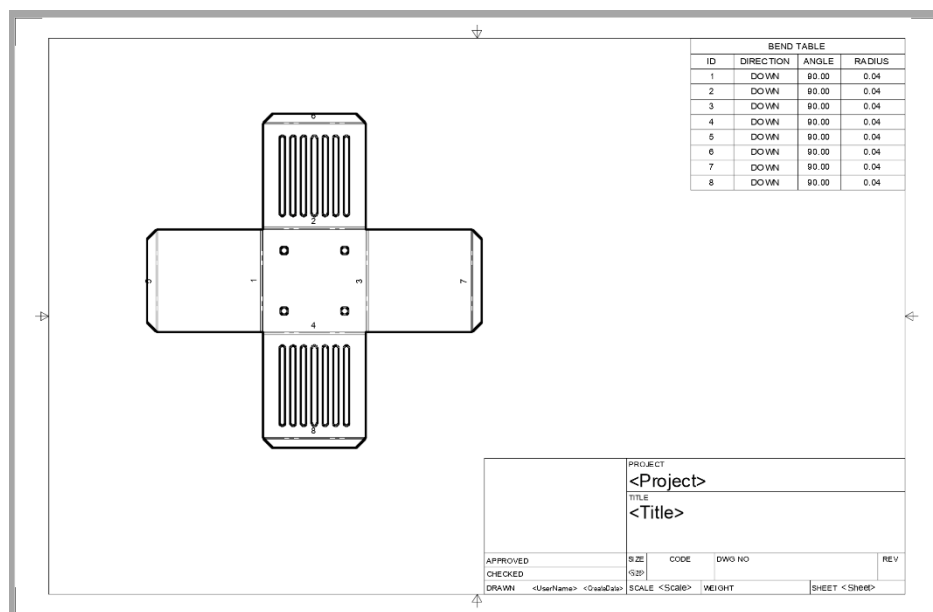
When you place a parts list in your drawing, the referenced view is automatically annotated with balloon notes. The same is true in your smart template, and this translates to automatic balloon notes on *your* model when this template is used to create a drawing.



STORYBOARD SHEET IN A SMART TEMPLATE SHOWING PARTS LIST AND BALLOON NOTES

Bend Table

You saw earlier that we have two view types for sheet metal – Folded Component and Flat Pattern. Well, similar to parts list, you can add a placeholder bend table, and it will be added to your smart template. And, using the method above to choose a Reference, the bend table does not even need to be on the same sheet as your flat pattern (though naturally it is easier cross-reference the two when on the same sheet).



SHEET METAL FLAT PATTERN SHEET IN A SMART TEMPLATE SHOWING BEND TABLE

Smart Template Sheets without Placeholders

Your smart template can include sheets that are either empty, or simply do not contain any placeholder information. These sheets will be included in drawings you create from this template. You may find you want to rearrange sheets after the drawing is created, but that is as simple as a drag and drop.

Smart Template Best Practices

We suggest you consider creating one or more smart templates for each project. You can always start with a company standard template and use Save As – then customize it for the specific project. One of the advantages of doing it this way: you can pre-edit some of the properties in your title blocks, in the template, if they will be the same on all sheets (but different in different projects) and avoid having to edit the title block on every sheet in your final drawing to add this information.

Conclusion

By now you should have a solid understanding of the role that Preferences and Templates can play in configuring your drawings, and how Title Blocks and Smart Templates can get you quickly on your way to completing your critical manufacturing documentation.

Let us know what ideas you have to continue making this process "smarter" and faster.