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**Autodesk Inventor Tutorials**

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# Basic Titleblock Creation
## Latest Revision: 6/14/02

Creating titleblocks in Inventor can be an easy task if you plan ahead.  This tutorial will g over the basics of titleblock creation.  You will want to work in your default template so that it will be available in all of your new details.

First open your template .idw and delete the current Inventor supplied titleblock.  You should be left with a blank sheet with a border.  There are two ways to create a title block.  The first is to sketch the titleblock in Inventor's sketch mode.  The second is to insert an AutoCAD drawing on your titleblock.  We shall cover both in the tutorial.

In the browser bar expand the drawing resources tab and RMB on the titleblocks icon. Select "Define New Titleblock".  This will cause Inventor to enter into sketch mode. Now the decision must be made to insert an AutoCAD titleblock or draw one from scratch.  Let us consider the former first.  Click on the AutoCAD symbol 🔳 in the sketch toolbar.  Find the AutoCAD drawing of interest and highlight it (do not open it at this time), then click on Options.  Figure 1 shows the titleblock we are inserting.  I have found that True Type fonts import better than .shx style fonts.  Of course you can choose to create all of your text in Inventor (which I suggest).  We can delete the imported text later.

On the first screen, set the unit of the file to the correct units (mm, in, feet etc.)  On the second screen you want to make sure that the "Model Space" radio button and "Constrain Endpoints" checkbox are enabled.  Make sure the layers on which your titleblock exists are checked (they usually are by default).  Go to the next screen.  For most applications the default option on the third screen are OK.  If you have problems you can adjust these to suit your needs.  Exit the import wizard and then open the AutoCAD drawing.

| PROJECT NO. | TITLE: | | | |
| CAD FILE NO. | | | | |
| VALIDATED BY: | | | | |
| DATE: | SCALE: | — — | SHEET: | |
| DRAWN BY: | | DWG NO. | | REV: |
| DATE: | | | | |

**Figure 1 - AutoCAD Titleblock**

You should now have the AutoCAD data in sketch format in your IV drawing.  At this point you need to determine if you want to keep the text that was imported or want to use Inventor text (I suggest the latter for continuity's sake).  Select all of the titleblock's lines

(with a window select) and move the lines to a new point in the idw.  The text will be left behind.  You can now delete the text (you must do each one individually.)

**NOTE:  You could have deleted the text in the AutoCAD file to skip this step.  I would recommend this especially if you have a lot of text in your titleblock.**

The other way of creating a titleblock is to sketch it by hand in IV.  The same principles apply.  Instead of inserting the AutoCAD file you would simply begin sketching the lines using the sketch tools.  Don't worry about the position of the titleblock in the .idw (that will be defined later).  Simply draw your titleblock (you can use dimensions to define line positions and sizes).  You can also make the lines colored, alter the thickness etc. to suit you needs.

Regardless of which method you use, you now have an empty titleblock (see Figure 2).  It is now time to place text in the titleblock.

The next steps assume you have setup the desired text formats in the template file.  If you have not done this I would recommend doing so at this time.
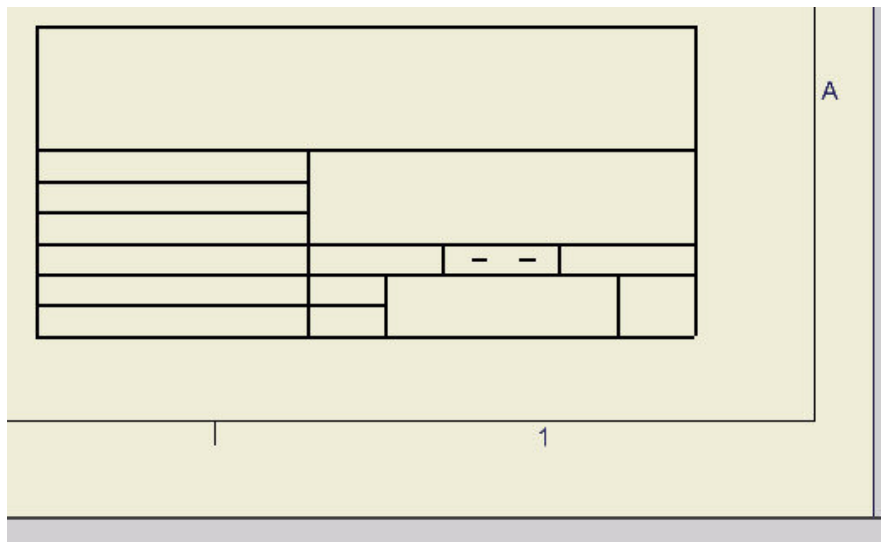


Figure 2 - Empty Titleblock in Inventor

There are two basic types of text in a titleblock.  Static Text that is the same for every titleblock and Property Fields that change for each drawing.

Let's first insert the static text for the titleblock.  Choose the text icon **A** and choose the placement for the first portion of text.  You will be presented with the standard text dialogue box (see Figure 3).  Enter the text desired and format it to your preference.

Exit the dialogue and position the text in the titleblock.  Repeat this for all of the text fields.  Your titleblock will now look something like Figure 4.
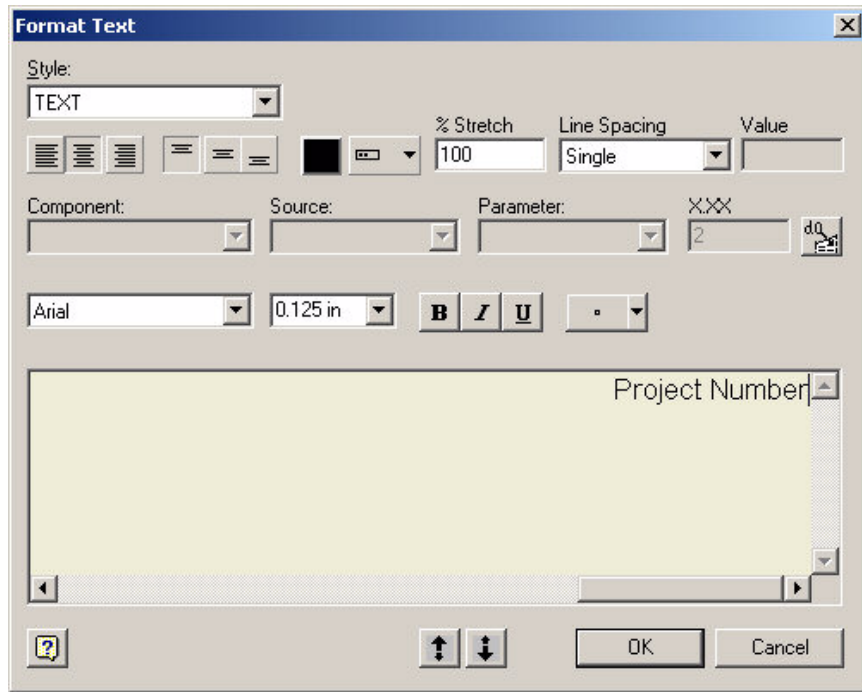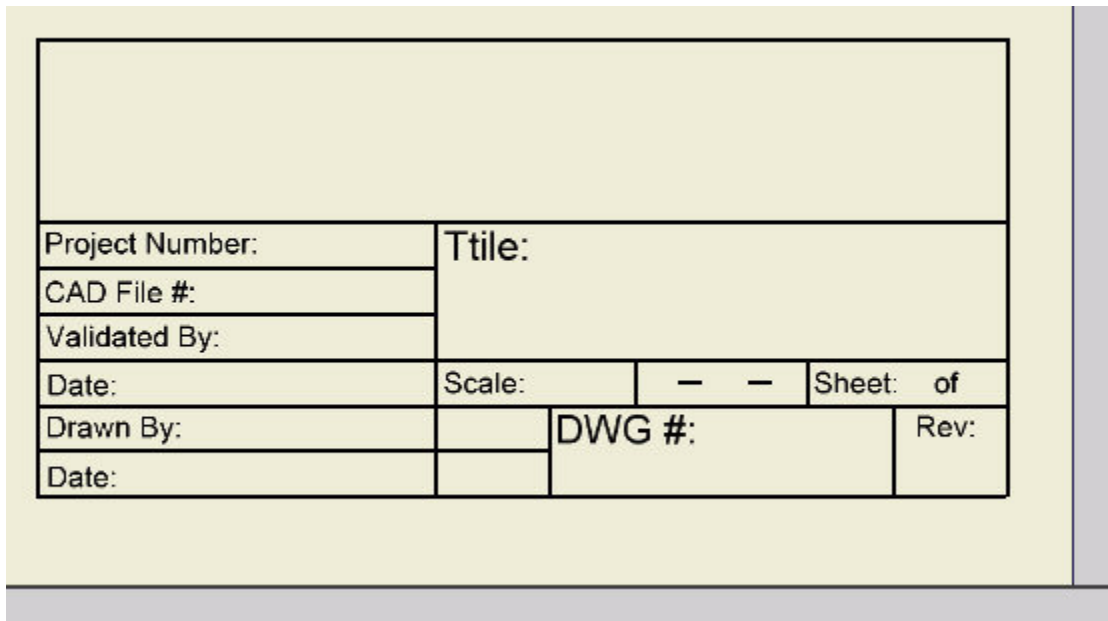
**Figure 3 - Entering Static Text**



**Figure 4 - Titleblock with Static Text**

Now we want to enter in the Property Field text. This is the text that will change for each drawing. Choose the Property Field Icon ![icon] and pick a position for the text. You will be presented with a dialogue box like the one in Figure 5.
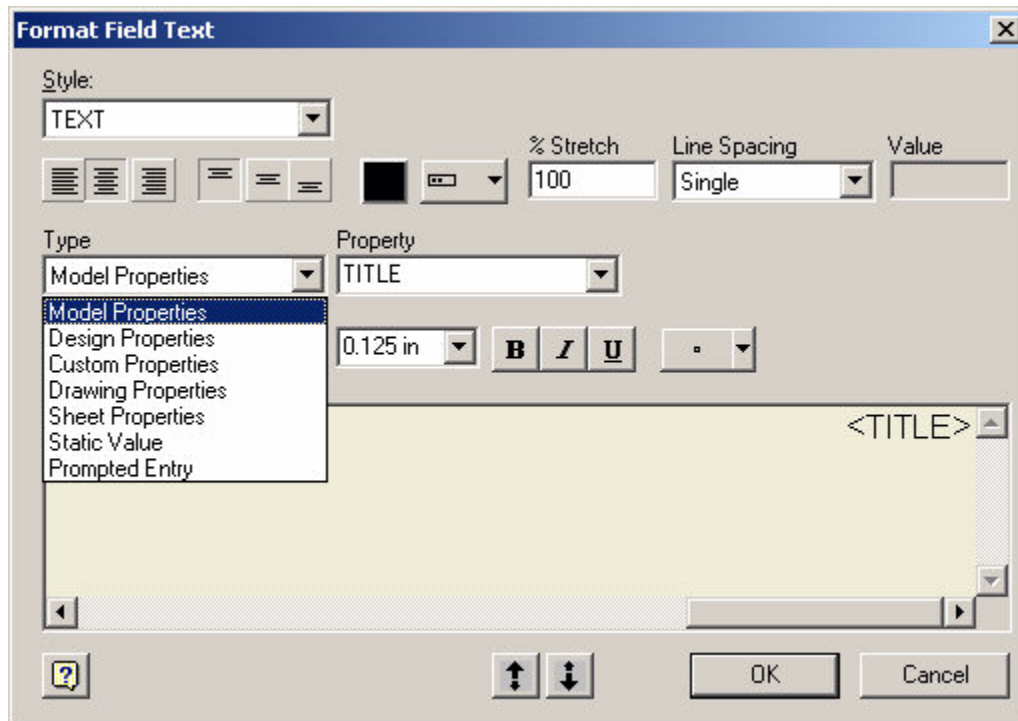
**Figure 5 - Property Field Dialogue Box**

This is very similar to the standard text box except you will notice that there are some extra options.  Under the Type pull down you can specify what type of property you want to use.  Descriptions of these options are listed below:

| Model Property | Properties taken from the referenced model (ipt/iam) |
|---|---|
| Design Property | Properties taken from properties of the .idw itself |
| Custom Property | Custom property fields (from idw only) |
| Drawing Property | Properties of the idw itself (only one option) |
| Sheet Property | Properties that relate to the idw sheet format |
| Static Value | No changing text (same as regular text above) |
| Prompted Entry | Prompts user for an entry upon TB insertion |

You must now make another decision.  Do you want to fill out the properties in the model and have them imported into the titleblock automatically or do you want to specify the properties of the idw and have them appear in the titleblock?  There are valid arguments for each manner however a few things to keep in mind:

1. Custom properties are only those listed in the idw itself.  To get custom properties from the model to the idw please refer to **http://www.mymcad.com/** under Inventor Tips and Tricks #5002

2. The model properties that are inserted into the titleblock are for the first view placed. If you place a second view of a different model, it's values will not be inserted.

Regardless of the method you choose (or a combination of the two) the following steps are the same (just replace model with design or vice versa).

In our example we want to use the model properties. Select Model Properties form the Type pulldown and then Title from the Property pulldown. Be sure to set the justification correctly and then place the text in your titleblock. Repeat the steps for all of the properties required. See Figure 6. Don't worry if the text of the property field is outside the area you want it (e.g. Revision Number) for when filled out the text will only take up as much room as the property value. (e.g. Revision Number will just be "01" or "B")



Figure 6 - Model Property Fields Filled Out

We enter another Property Field but this time choose Sheet Properties from the Type pull down. Choose Sheet Size and place it in the correct spot. Next place the Sheet Properties/Sheet Number in place. We now have to change to Drawing Properties to find the Number of Sheets property and place it. Unfortunately there is no property for Scale so we enter a static text value of NTS in this area. Your titleblock should look something like Figure 7. You will notice that it get rather crowded in some areas (e.g. the sheet area). Many times your property fields will run over other text. You may have to adjust positions after you test the titleblock and see how the property text fits.

Hit the Return icon button to exit sketch mode. You will now have to save your titleblock. Give it a name and the block should appear in the drawing. (see Figure 8). Notice that the sheet properties (sheet size and sheet numbers) are already filled out. This information is based on the .idw so it appears in the block.

**Figure 7 - Property Fields in Place**



**Figure 8 - Titleblock Inserted into Drawing**

If you want the titleblock to appear in one of the other three corners RMB on Sheet1 in the browser and select Edit Sheet. You will be presented with options as to which corners to place the titleblock as well as sheet size and orientation. Set this to how you want your default titleblock to appear and then exit. Save the template file.

Now start a new drawing using this newly created template (use the New command, don't open the template) The titleblock should appear in the .idw but the model properties are not filled out. Now insert a view in the .idw. The properties from the model should propagate to the titleblock and it should now be filled out. See Figure 9.
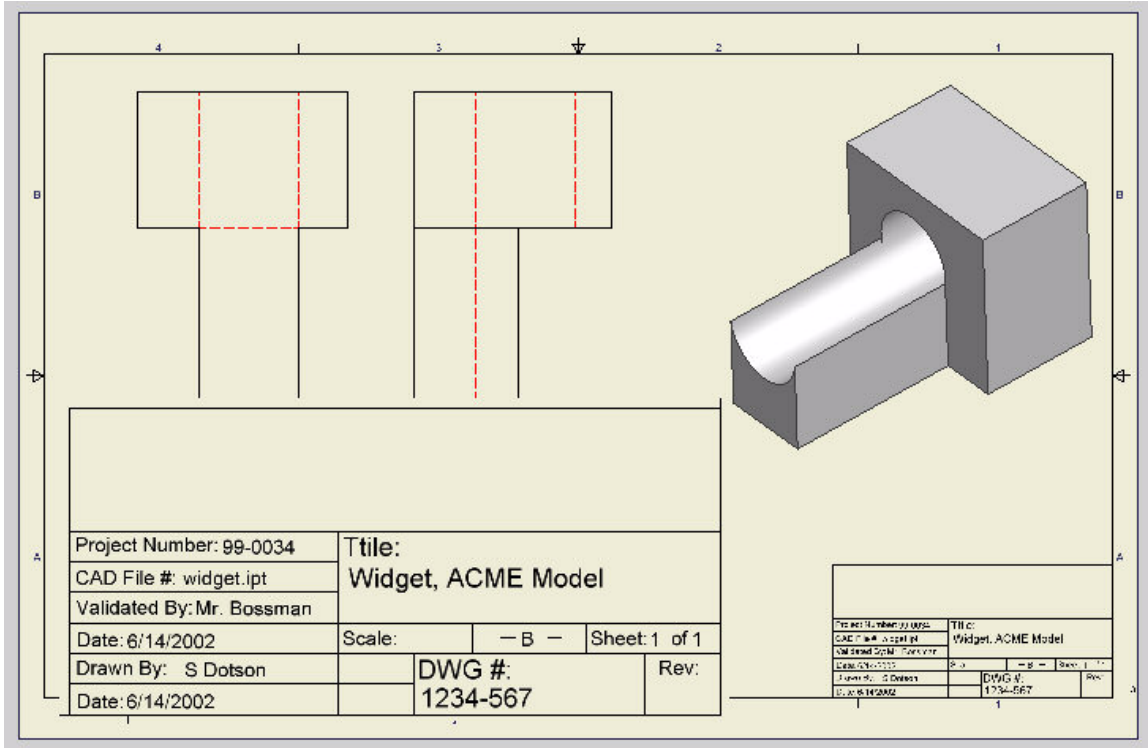


**Figure 9 - Completed Titleblock (Enlarged for Clarity)**

You now have a completed titleblock. You can go back and edit the titleblock if the text or line positions are not satisfactory.

Another frequent question is how to get pictures into titleblocks. This is a long standing issue with .idws Some people can get this to work and others cannot. As of R5.3SP1 it was still not working on my machine (I bring my logos in from AutoCAD files)[1]. You may want to take a quick search of the **newsgroup** to see if any of the suggestions there help you out. Essentially you edit the titleblock and on the command menu choose Insert/Picture. You have the option of linking or embedding the BMP in the titleblock. If you link it, the BMP must reside in your search paths (and if you send the .idw to someone else they will need the BMP). Another way is to copy the BMP from a graphics program (like Paint) and use Edit/Paste Special to insert the BMP. On many machines however as soon as you save the titleblock the image disappears.

Sorry but until this is resolved you are on your own on this one ☺

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[1] A very powerful tool is the fill command in the sketch bar. I've found that I can recreate most logos in sketch mode with a bit of patience.

It will likely take you a few tries to get the titleblock looking like you want it, but keep at it and they can turn out looking very professional.  Figure 10 is an example of one I designed a while back.  It shows that you can indeed cram a lot of info into the titleblock. I have of course blacked out the company's name and logo.

Play around with the options.  You'll discover you can do almost anything you could do in AutoCAD.  Ok, except import a BMP ☺

**Figure 10 - Example Titleblock**

TITLE:

DWG NO.

REV

SIZE B

1 OF 1

PROJECT

CAD FILE #

PART CLASS

DRAWN BY sdotson

DATE 6/14/2002

** UNLESS OTHERWISE NOTED **
DIMENSIONS ARE IN INCHES
THREADS CLASS 2 SURFACES 125
FILLETS±.001 ANGLES±1/2
X.X±.02 X.XX±0.01 X.XXX±0.005
BREAK ALL EDGES 0.005-.01
ALL DIMS APPLY AFTER PLATING
THIRD ANGLE PROJECTION

MATERIAL

FINISH PER

ALT MATERIAL

HARDNESS