

MA234-5: Build the Foundation: Autodesk® Inventor® Drawing Templates

Mark Flayler – IMAGINiT Technologies

MA234-5 The Autodesk Inventor Drawing Manager is used to create critical documentation for the digital prototyping process. After you create a model, you can produce a drawing to document your design. In the drawing, you can place views of a model on one or more drawing sheets. You can then add dimensions and other drawing annotations to document the model. Using a powerful template at the beginning of the documentation process can increase productivity by eliminating redundancy and keeping the team on a set of company-enforced standards. This class is designed for users wanting to get more out of their drawing templates and a better understanding of the resources that go into a potent documentation engine.

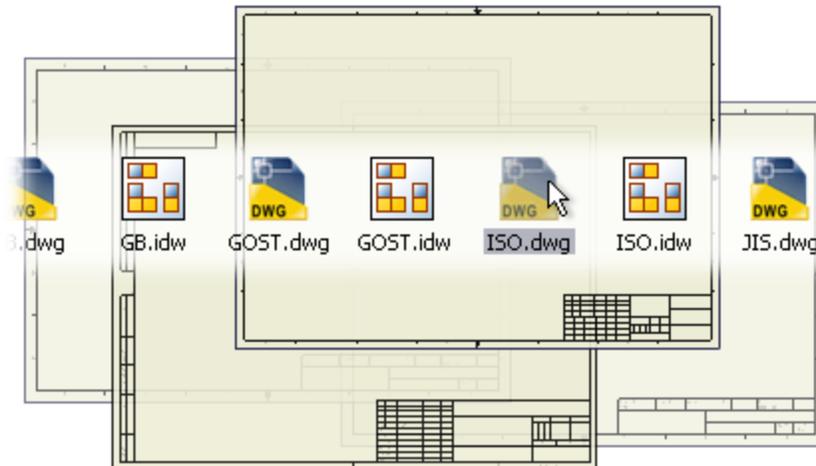
About the Speaker:

Mark has been using Autodesk® products since 1999 in many different manufacturing environments. He has implemented Autodesk products for many diverse industries. Autodesk® Inventor® has profoundly augmented Mark's abilities, allowing him to bring 3D digital prototyping to the forefront of the industries with which he has interacted. Mark has extensive experience and a comprehensive understanding of the technical and practical business and human dimensions of implementation. His expertise has helped his clients to maximize their project's effectiveness and return on investment. He is an effective and skillful communicator, consulting with his clients to help them achieve their business objectives. Mark provides training, support, and implementation on all Autodesk manufacturing solutions.

Blog: <http://blogs.rand.com/manufacturing/>

Introduction

Each new drawing is created from a template. When you install Autodesk Inventor, your selection of a default drafting standard sets the default template that is used to create drawings for modeling files. The creation of a company template to meet standards is critical to any implementation of Autodesk Inventor. A company template can contain custom resources for industry and company documentation procedures. In this class we will look at these creation tools and their not hidden pitfalls and go over some decisions for creating a template that is right for you and your company.



Before we begin a few words on documentation...

As we go through the settings and procedural enlightenment in this course, make sure you document the changes and settings that are taking place for use at your company. You do not have to record every style setting, but make sure you can convey how your template is intended to be used in practice. At the end of your template creation process or during it, make notes and take screen grabs to show users how the data is mapped and how the Drawing Resources are to be used. This will help when the template is being adopted by current and new employees and will be a reminder of when or if you need to revise the drawing templates.



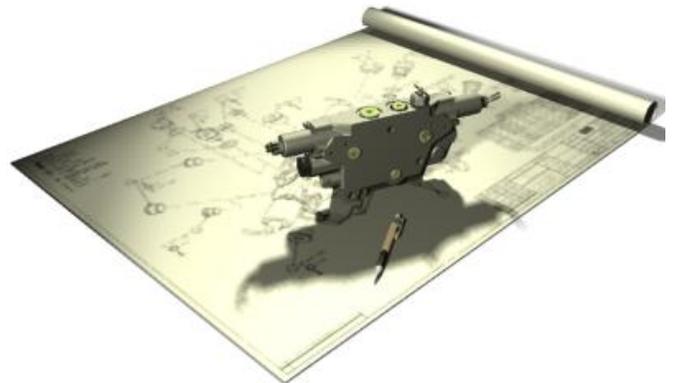
Procedural Consideration

A large part of any template creation process is the incorporation of company principles into the design intent of the file. The following are some questions that you can ask yourself as you create your first Inventor Drawing Template file. These questions may seem mundane to most single users, but sometimes the driving factors of detailing and file tracking are determined by company data management systems, and those systems are not always the friendly neighborhood Vault product. Other reasons for detracting from the norm are industry/company standards that do not follow a specific engineering standard or majority convention.



If you have already been producing drawings for some time you should already know the answers to following questions. However, if you are new to Inventor or trying to fine tune your design documentation, you should take the time to ask yourself some of these sample questions:

- How do you produce the documentation of your designs?
 - One detail print file per modeling file.
 - One detail print file per IAM that would include the part details.
 - Dealer's choice based on the particular assembly.
- Where do you want your design tracking properties to be stored?
 - In the modeling files.
 - In the detail prints.
 - In both.
- How do you handle multiple sheets?
 - As separate detailing files.
 - Inclusive to the single detailing file.
- Where does your control for the styles resolve?
 - In the Administrative Styles Library?
 - In the drawing template?
- Which sheet sizes do you typically use?
 - ANSI, do you only use B and D?
 - ISO, do you only use A4?
 - Of the two above, which size is the most commonly used?
- Do you use Vault or data management software similar to Vault?
 - Yes, we do.
 - No, but we might in the future.
 - Never, data management is overrated.
- What information is to be in the Title Block?
- Which industry standard symbols or notations need to be added?
- What information is to be in the Border?



While I cannot wave a magic wand and create a template and a process that works for everyone, the tools in this course will show you different methods of resolving your questions, which tools to avoid in certain situations, and how to work with Inventor to do something not inherent to its native coding.

Should I Use DWG or IDW?

Beginning in Inventor 2008, Autodesk reintroduced the DWG format to the Inventor kernel after a very long hiatus. This was designed to open many collaborative doors for Inventor alongside a supportive AutoCAD, enabling companies to work internally as well as externally. At that point, for all intents and purposes Autodesk left the IDW format alone for the sake of legacy.



IDW and DWG are identical in Inventor functionality with a few minor differences:

- Inventor DWG files can be opened directly in AutoCAD 2007 and up (previous versions require a free Object Enabler available at Autodesk.com). Once opened in AutoCAD, you can measure, print, or even detail the Inventor geometry.
 - Once opened in AutoCAD, the Inventor views are blocks that cannot be exploded.
 - Jogged and broken views do not measure correctly in AutoCAD due to the parametric nature of the views in Inventor.
- Inventor DWG views are searchable in AutoCAD DesignCenter for factory floor layouts as block views. Therefore, any views created in an Inventor DWG can be reused without exporting and copying.
- Inventor IDWs cannot use AutoCAD created blocks for symbols or notations in Inventor documentation. However, the Inventor DWG can import and place them.
- Inventor DWGs are inherently larger in size due to the advantages above.
- Inventor DWGs can aid in 2D to 3D migration by updating or creating parts/assemblies inside Inventor and then adding them to the existing 2D assembly or layout.
- Use a client's AutoCAD DWG as a starting point to obtain their standards and styles.

Note: Remember that just because you can open an Inventor DWG in AutoCAD, it does not mean you can change the modeling geometry. This is similar to using a mechanical structure in AutoCAD Mechanical. The originating software must make the edits using the original modeling files. This is to prevent improper design tracking when collaborating on designs internally and externally.

If you stay with IDW or use it from the beginning, it is probably for one of the following reasons:

- You have always used IDWs and see no reason to collaborate your data and change your file format.
- You choose to keep your Inventor files completely separate from your AutoCAD drawings due to file format confusion internally or poor tracking policies.

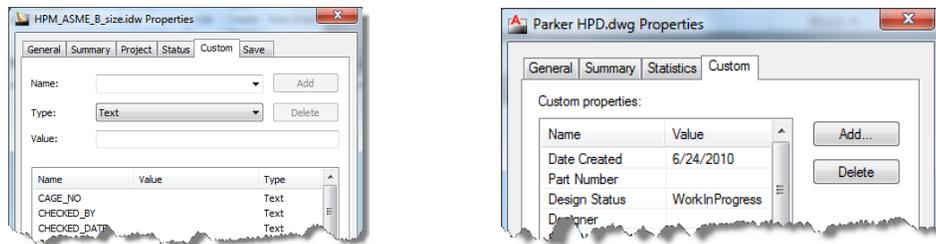
If you fall into one of these categories, I would urge you to reevaluate your decision as IDW files might disappear in the future and more functionality might be added to the DWG format instead. Getting behind the technology early is better than trying to catch up to it later. If needed there is a batch translation tool in the Task Scheduler to turn Inventor IDWs into Inventor DWGs.



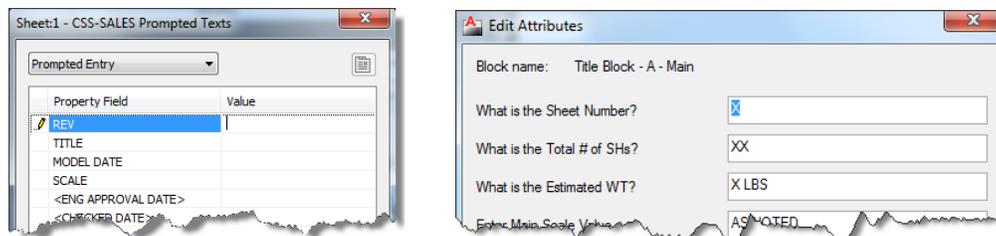
iProperties vs. Prompted Entries

Let's take a deeper look at the differences in iProperties and Prompted Entries. There are pros and cons to both methods of tracking design information but after looking at differences, the segregation of the two input tools becomes clear. If this is not understood correctly, you might find yourself down the wrong path when your business software changes. Let's look at the equivalent tools in AutoCAD.

Inventor iProperties are equivalent to AutoCAD Drawing Properties (DWGPROPS)



Inventor Prompted Entries are equivalent to AutoCAD Block Attributes (ATTDEF)



Based on this precursory knowledge that a lot of transitioning users have about AutoCAD they almost immediately start creating Prompted Entries for the Title Block since it is a similar method to how things are done by the majority of AutoCAD users.

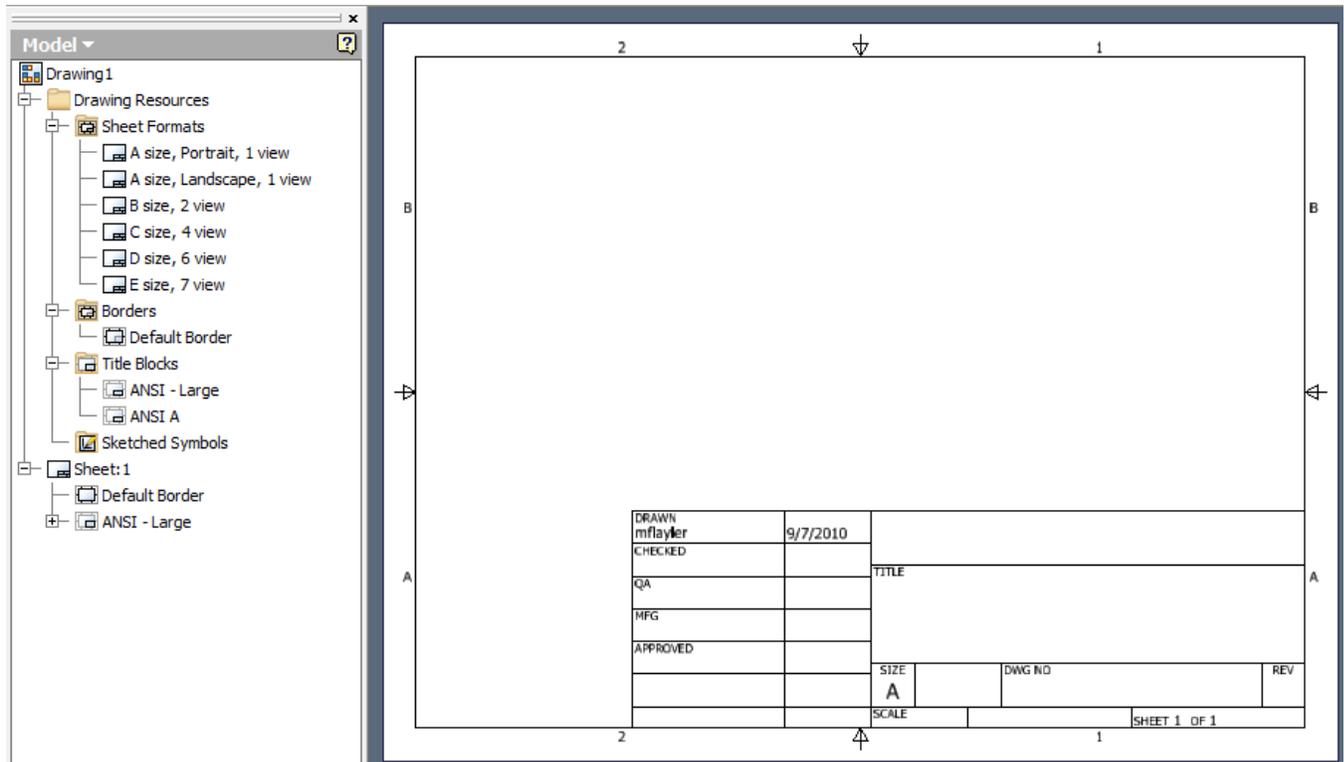
It seems to be a very well kept secret in AutoCAD that DWGPROPS and the use of FIELDS are the best way to populate a Title Block in AutoCAD. When companies find out about this method of Title Block population, it is usually too late as they might already have created a large number of legacy drawings using attributed blocks. This usually becomes apparent when companies start looking at data management solutions. Some of these programs have a VERY hard time working with attributed blocks. Autodesk Vault is perhaps the most skilled at this, but it is not always the system of choice for a large company.

When data management systems index metadata of files they inherently can read the Properties of a file whether it is Word, Adobe, Inventor, or AutoCAD. This makes searching for any data in the data management system very easy to do and greatly speeds up the design process by circumventing the opening of a large amount of files to find the correct one. **In order to have this functionality you must use iProperties in your template.** Prompted Entries will not be searchable.

Moreover they can control the same information across multiple layout sheets so if the information is the same in 7 layouts then the user does not have to change the attributed Title Block 7 times but only change it in the DWGPROPS once. This will work the same way in Inventor across multiple sheets in a drawing as well compared to inputting the same information in prompted entries over and over again.

Anatomy of an Inventor Drawing Template

Let's make sure we are on the same page shall we, or is it the same sheet? In an Inventor drawing, whether it is an IDW or DWG, there is a drawing browser bar on the left side of the screen. The Sheet node in the browser will contain your drawing views, sketches, sheets, and anything you put on your sheet. There is also the Drawing Resources node, which contains the important information for this course. Inside Drawing Resources you will find Sheet Formats, Borders, Title Blocks, and Sketched Symbols. If it is a DWG file you will also see AutoCAD Blocks.



Each Sheet consists of one **Title Block**, one **Border**, and as many **Sketched Symbols** or **AutoCAD Blocks** as needed. These combined resources can then be turned into a **Sheet Format** for quick sheet creation. In the default template included with Inventor, there are six Sheet Formats consisting of the Default Border and one of the two default Title Blocks. I have never met a user of the software that only used these defaults, unless they were in an educational system without proper administrative support.

A lot of people will say, why not just import an AutoCAD block or use an existing AutoCAD DWG for the Inventor template? Why go through this process of creating a unique template? Well, there are good reasons for creating a native Inventor template file, which we will discuss during the class.

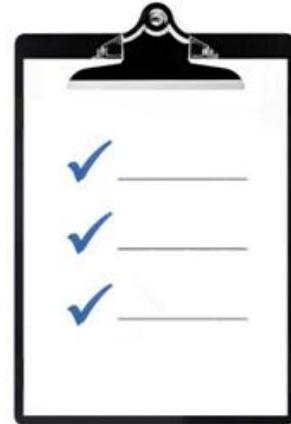
Sometimes, creating an Inventor template from an AutoCAD DWG is acceptable, such as when new client styles and standards are given to a design shop and the shop must adhere to those standards.

On the other hand, some items do not transfer 1 to 1 when imported into Inventor. Creating a native template solves these issues and allows users to experience all of Inventor's Styles and Standards settings.

Creation of Drawing Resources

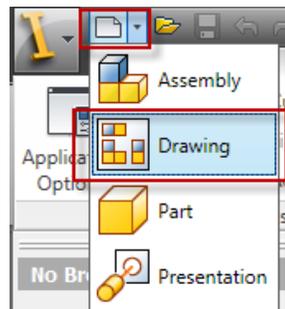
The creation of Drawing Resources, such as Title Blocks, Sheet Formats, Borders, and Sketched Symbols is imperative to the proper setup of a template. Having a uniform look and feel for the layout and usage of these resources is the essence of a company or industry standard. While there is not a specified order in which to create these elements, the following is a recommended workflow.:

- Start a blank drawing
- Set the sheet size.
- Set up local styles
- Create a Border resource.
- Create a Title Block.
- Create any Sketched Symbols or Blocks.
- Create a Sheet Format.
- Double-check your intent.
- Styles Library administration.
- Set your template to default opening settings.
- Save the Template in the template directory.



Start a Blank Drawing

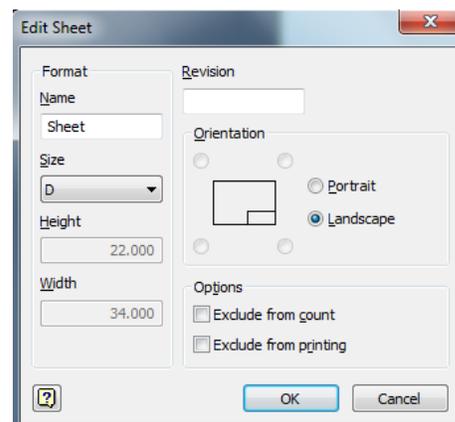
To begin creating a new template from scratch with no drawing resources in it (no Title Blocks, Sheet Formats, Sketched Symbols, Blocks, and only the default Border), hold down CTRL and SHIFT and select the new drawing button in the Quick Access Toolbar.



Set the Sheet Size

Set your default Sheet Size by right-clicking on Sheet: 1. Select a size that is most commonly used by your company (B and D are perhaps the most common).

Exclude from count and **Exclude from Printing** are important if you use Title Pages that contain reference information or other information not necessary to the printable set.

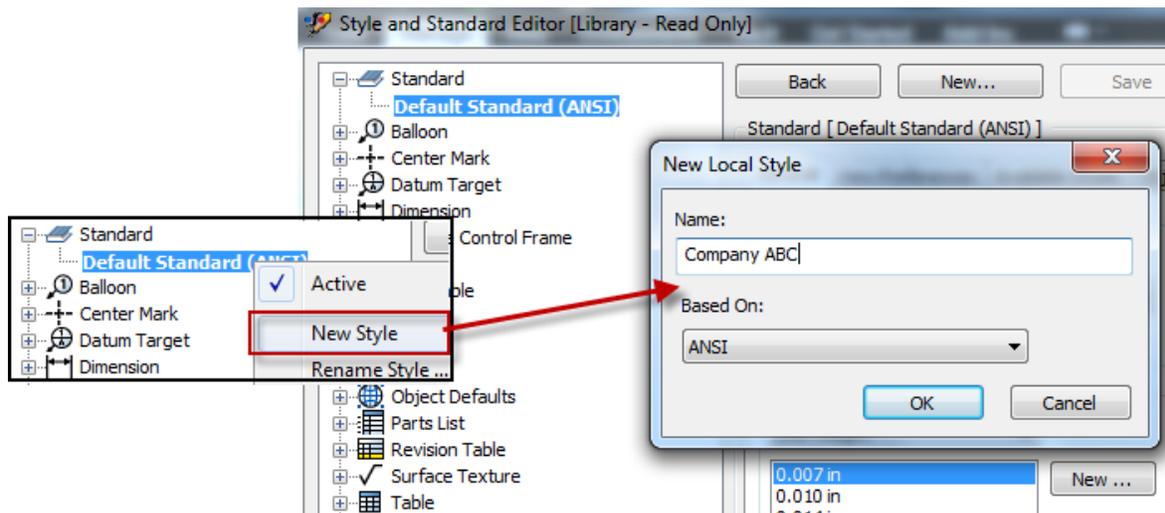


Set Up Local Styles

Before you start creating resources for your template, modify the Styles and Standards for the drawing so that all intended styles are ready to be used when you are creating text, borders, and other resources in the template.

When you create a drawing, it is automatically assigned an active drafting standard that controls the styles used to format dimensions, text, line weights, terminators, and other elements. You can use the default standard or select from another named standard.

Consider creating a Standard for your company so that any changes are easily identified. Take some time to specify the settings that you want to include in your company template.



Create a Border

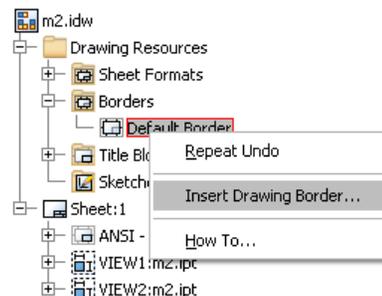
Unlike some systems, Inventor separates the typical drawing title block into a Border, (the shape around your drawing), and the Title Block (the block of information about your drawing). Inventor includes the Default Border, which can be configured to be used for any page size.

Changes to the border on one sheet do not impact the borders contained on other sheets.

The main advantage to using the Default Border, as opposed to a custom border, is that it will automatically resize to suit any page size.

Once the Default Border has been removed from the current sheet, right-click on the Default Border listed below Borders in the Drawing Resources and select **Insert Drawing Border...**

When inserting a new Default border, you can specify the number of zones (both horizontally and vertically) and change how the zones are labeled (Alphabetical, Numeric, or None).

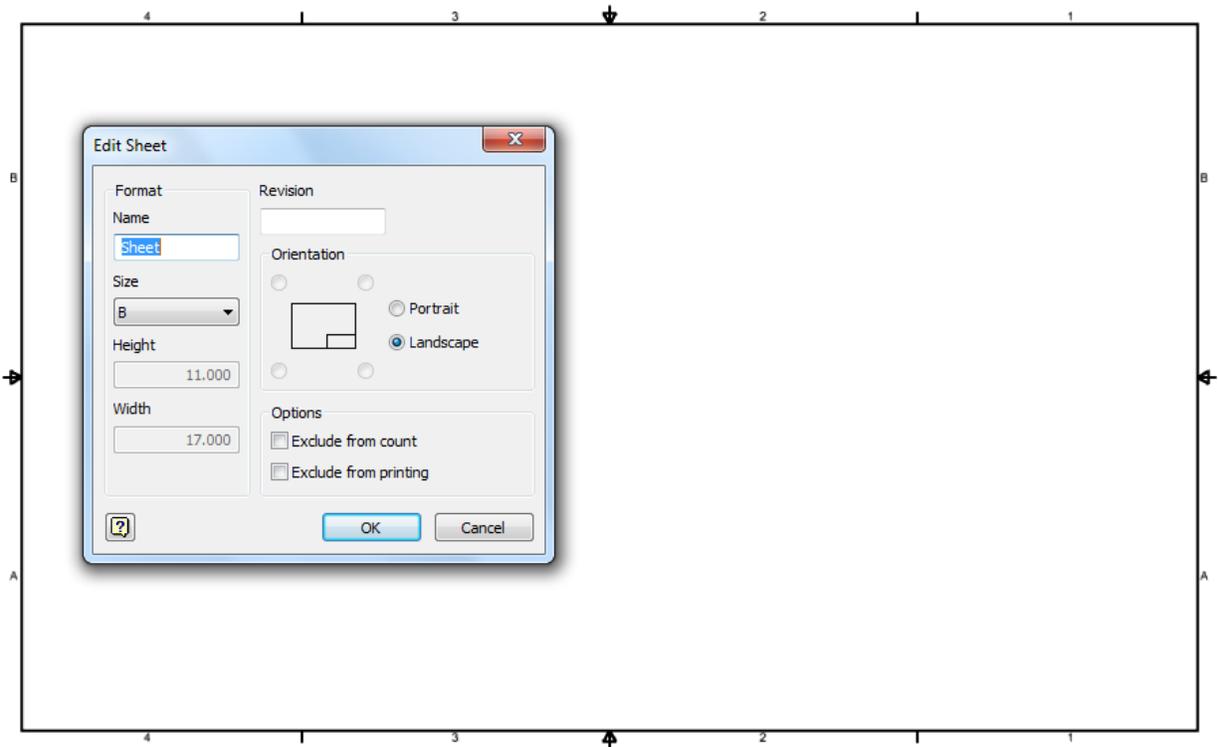
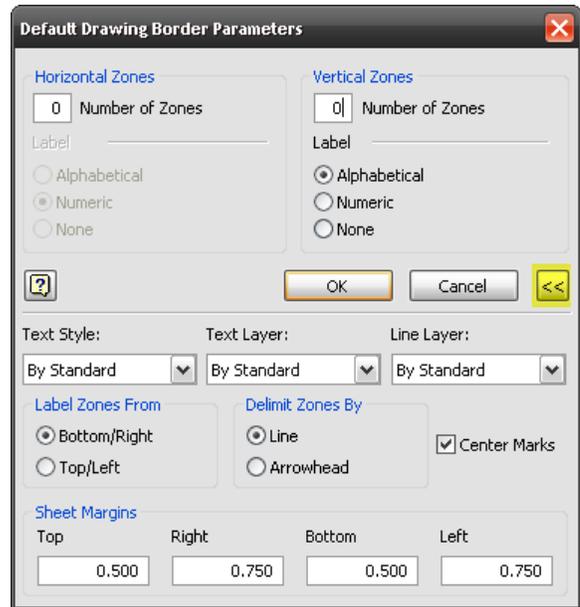


Setting the number of zones to 0 will remove the zone marking from the inserted border.

Click the >> button to expand the dialog box and display the additional options.

In the expanded dialog box, use the options to specify the text style and layer to use for the border text and adjust the location from which the zones are labeled. This is an example of why you should create custom Text Styles before implementing any Drawing Resource creation so that the correct style can be selected.

Additionally, toggle off **Center Marks** if they are not required and adjust the **Sheet Margins** (the offset of the border geometry from the outside edge of the paper).

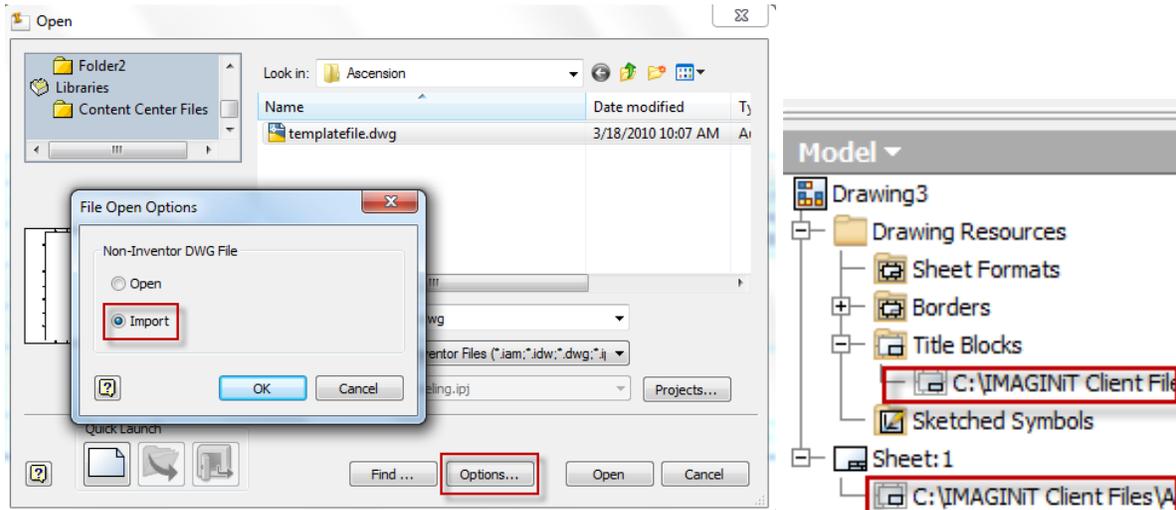


Create a Title Block

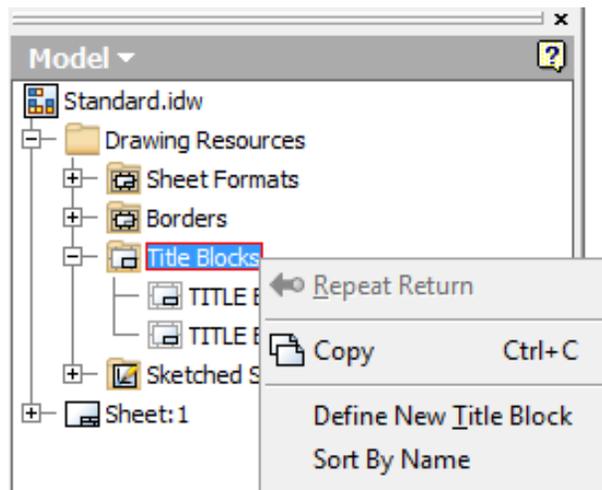
Main Title Block

It would be wise to determine size and spacing before starting this procedure. Print a copy of the Border without the Title Block and sketch a rough idea of what you want to have in your Title Block by hand. You might want to use a legacy AutoCAD Title Block as a starting point.

If you want to use an existing AutoCAD Title Block in Inventor, you can import one, but it will place all of the attribute definitions as prompted entries and will not be indexable in Data Management software. If the AutoCAD Title Block was created correctly, using Fields rather than Attributes, they would import as Custom iProperties and would be searchable but not necessarily mapped correctly. In addition, not everything will be scaled correctly when imported from AutoCAD into Inventor. It will be close, but not close enough for me. For this reason, I usually start from scratch when creating Title Blocks.



To create a new Title Block Definition, right-click on the Title Block area of the Drawing Resources and select **Define New Title Block**. You will be placed into a Sketching environment, similar to but not equal to the ones found in Part and Assembly modes (see the Drawing Resource Construction Tools at the end of this handout).

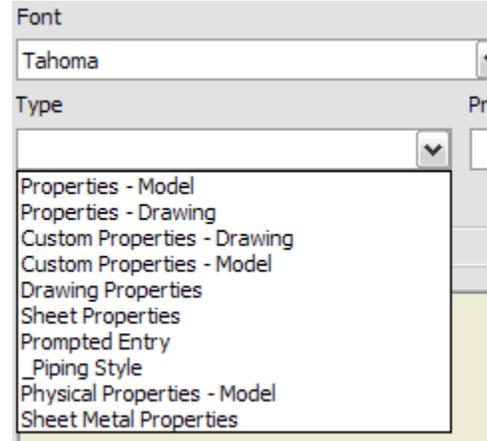


Start by sketching out a rough shape and size for your Title Block. Use dimensions where necessary and even dimension expressions are available here. Try to avoid too many dimensions or constraints as we all know that too much of either can slow Inventor down and in the drawing environment has been known to cause errors if equations are overused. I tend to use construction geometry a lot here to avoid this.

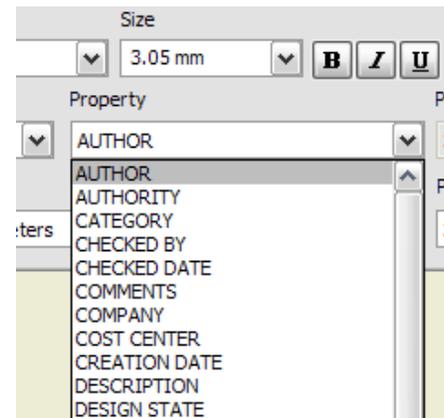
Text can be used in a drawing resource to create standard notations and report on company design tracking properties. When using text in a drawing resource, use the Type selection box to select the required category and the Property selection box to select the required property.

Note: Use the Add Text Parameter button () to add values to the text.

Option	Description
Properties - Model	iProperty values from a given model reference.
Properties - Drawing	iProperty values from the drawing file.
Custom Properties - Drawing	Custom iProperty value from the drawing file.
Custom Properties - Model	Custom iProperty value from a given model reference. (You might have to insert a model to get this in a template and then delete the View.)
Drawing Properties	Property: <i>Number of Sheets</i>
Sheet Properties	Properties: <i>Sheet Number, Sheet Revision, and Sheet Size</i>
Prompted Entries	Creates Custom in place user entries (similar to ACAD attributes).
_Piping Style	Properties associated with the Routed Systems package.
Physical Properties - Model	Properties: <i>Area, Density, Mass, and Volume</i> . (Hint: make sure the Application Setting is <i>Update Mass Properties on Save</i> .)
Sheet Metal Properties	Properties: <i>Flat Pattern Extents Area, Flat Pattern Extents Length, and Flat Pattern Extents Width</i> .

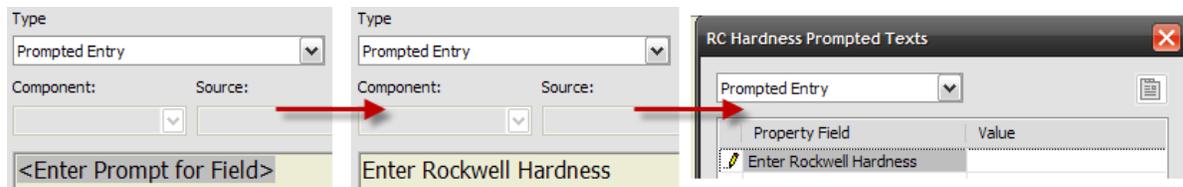


Types of Properties (insertion of a source file may be required to obtain specific Type Properties)



Property Selections

The Prompted Entries category allows you to create custom user-defined text and prompts for users. Select **Prompted Entry** and enter the required question or information from the designer in the highlighted area.



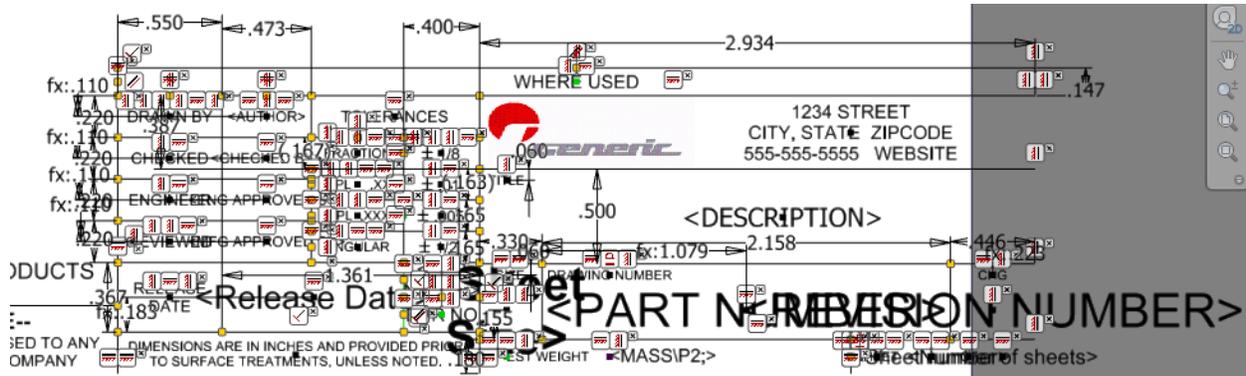
Note: Only one Prompted Entry is allowed per Text Box.

When using text, make sure you use the correct justifications, line spacing, single or multi-line options, colors, text overrides, and any predefined text styles that you previously created.

Use Horizontal and Vertical constraints with the text objects and their justifications to align them in addition to dimensional relationships (these can be done but not edited by a Parameters box).

Note: You might have to finish the sketch and validate the work to make sure you are leaving enough space for the designated values (placement and quantity of text).

Eventually this:



Will become this:

STANDARD PRODUCTS		WHERE USED		1234 STREET CITY, STATE ZIPCODE 555-555-5555 WEBSITE	
DRAWN BY	mflayler	TOLERANCES			
CHECKED	XXX	FRACTION	± 1/8		
ENGINEER	XXX	2 PL .XX	± .01		
REVIEWED	XXX	3 PL .XXX	± .005		
RELEASE DATE		ANGULAR	± 1/2'	SIZE	DRAWING NUMBER
-- COMPANY PROPRIETARY NOTE-- INFORMATION ON THIS DRAWING MAY NOT BE DISCLOSED TO ANY PARTY WITHOUT WRITTEN AUTHORIZATION FROM COMPANY		EDR NO	B	N/C	
DIMENSIONS ARE IN INCHES AND PROVIDED PRIOR TO SURFACE TREATMENTS, UNLESS NOTED.		EST WEIGHT	SHEET 1 OF 1		

Multiple Title Block

Commonly, it is necessary to create multiple sheets so that more extensive data can be recorded. It is not usually necessary to repeat all of the information contained in the Main Title Block in subsequent sheets. This creates more room for detail work and reduces the amount of repetitive information across the drawing package. While this is a company's prerogative, there are long standing roots in industry and standards to simplify the multiple sheets in a drawing.

SIZE	DRAWING NUMBER	CHG
B		N/C
EST WEIGHT	SHEET 2 OF 2	

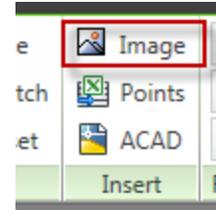
Repeat the procedures for creating a main Title Block, but apply less information in order to follow this documentation method.

Company Logo: Bitmap or Redraw?

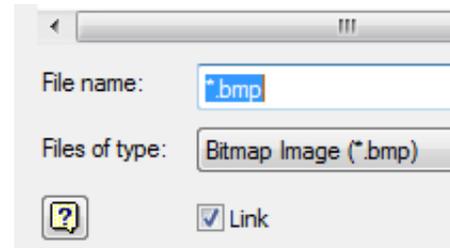
When choosing a placement method for a Company Logo in a Title Block, there are a few things to watch out for. These are small things that can cause great annoyance if not foreseen. The image below shows a Bitmap and a Sketched Company Logo. I'll leave it to you to decide which one is which.



When a Bitmap is placed in any file using the **Insert Image** command, you can either **Link** or **Embed** the image. Linking will always force Inventor to look for a location for the image outside the file, based on your project workspace. In companies in which project files are not used correctly, it is prone to losing the reference.



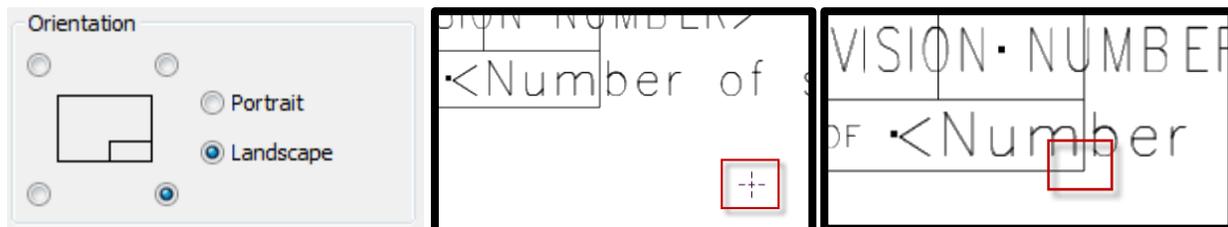
To embed the file instead of linking it, clear the checkbox when first placing the image. Another disadvantage of an image, whether it is linked or embedded, is that it normally creates extra bitmaps when files are exported to AutoCAD.



In the end, to avoid these issues, redrawing the image from scratch or inserting an older AutoCAD rendition is best.

Title Block Insertion

The Application Options or sheet overrides are used to control the insertion of the Title Block. The overall geometry controls the placement into the border. The extents of the geometry will determine the block placement, and the outermost entity thereof will be used as the insertion point. Therefore, if you have a lingering point in your definition outside your design area, it will become the point at which the definition will be inserted.



Title Block Insertion Setting

No!

Yes!

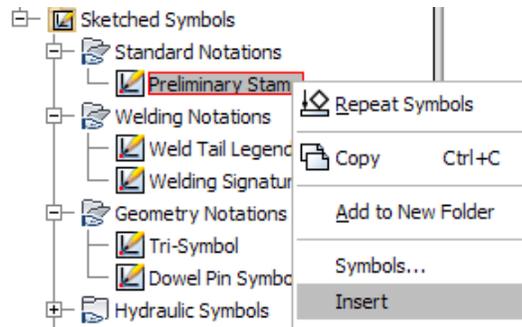
Create Sketched Symbols or Blocks

See last year's class at AU2009, which focused solely on Sketched Symbols and AutoCAD Blocks.

Class ID: MA314-1: It's Not Sign Language.

Its Not Sign Language goes further in depth on this topic than is permitted by the time available for this session. It also details how to use the Inventor Sketcher inside the Drawing Manager of Inventor. Since there are some key differences between Drawing and Part sketching.

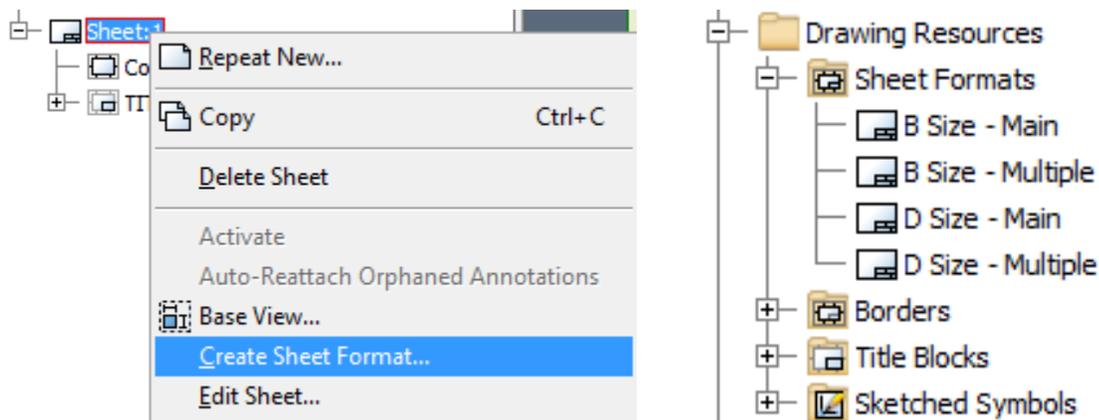
Make sure you capture your company's industry or private symbols in this form to use throughout the Drawing Manager. This will avoid having to do heavy reproduction and copy and paste operations between files. If there are a large number of symbols, you can organize them into folders within the Drawing Resources folder as well.



Create a Sheet Format

Once a Sheet has been created with proper sizing for the layout, and proper Borders, Title Block, and default Sketched Symbols have been placed, you can create a saved Sheet Format from the current set up. If required, you can also add a default set of views to the sheet and have Inventor prompt for a part or assembly file when the sheet is activated. Although this is a designed method, it seldom works perfectly and the majority of Inventor users would rather place the views manually themselves to convey their intended design.

To create a basic Sheet Format without views right-click on the Sheet header in the Drawing Manager and select the **Create a Sheet Format** option. Name the Format per the decisions made during its creation, such as **B-Sized Main** or **B-Sized Multiple** sheet.



Double-check your intent

Always double-check your intent for the template before it is saved. Ideally include the entire Inventor team in the decision making process so that everyone understands what went into the template so that if changes need to be made after initial inception that the user base will understand how to ask, and what not to ask for.

Styles Library Administration

Depending on the project's *Use Style Library* setting, Autodesk Inventor templates take on a slightly different role, whether you want Template control or Style Library control for Styles.

If you set Use Style Library = No in the project

Templates interact with styles as they do in pre-Autodesk Inventor R9 templates. Styles are stored in the templates and are copied to new documents when created. If a style in a template is changed, that change does not propagate to existing documents based on that template. Required document styles must be added to the template files.



If you set Use Style Library = Yes or Read Only

Templates are not the primary source of styles for new documents. They are the source of the default styles that the document uses. When a new document is created, any styles in the template are copied from the Style Library as part of the document creation process.

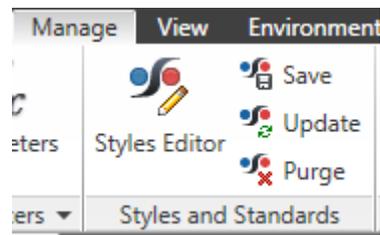


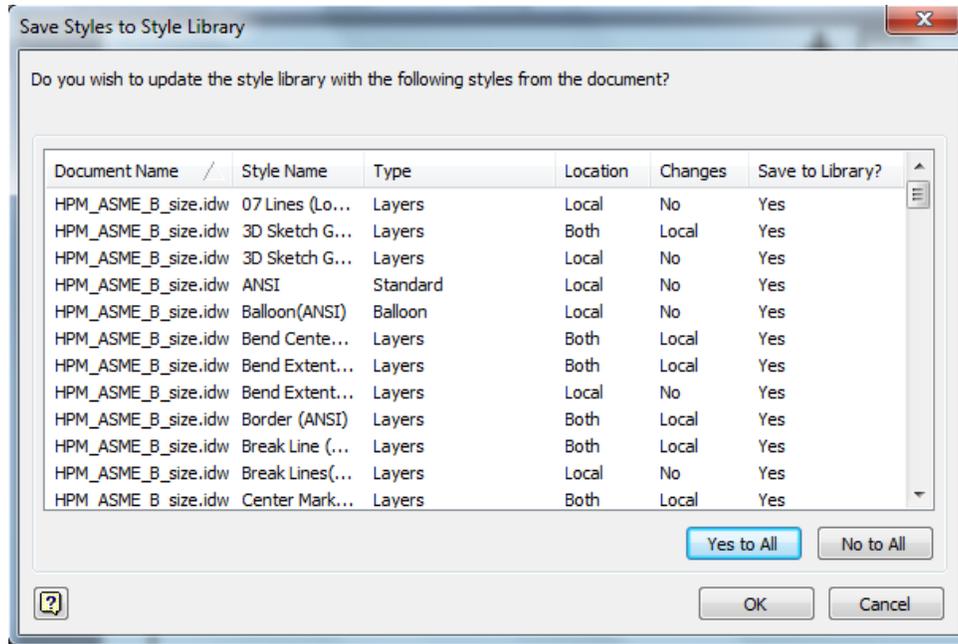
When using a Style Library, styles that are in the template are refreshed to match the style library when a new document is created. If a style in a template does not exist in the library, it is added to the new document. If there is a mismatch between a corresponding style in the template and the library, you will get a message stating that the Style in the library will be used.

If you chose to not use the Styles Library, no further action is necessary. However, this usually prevents Inventor from adding other items to the Styles Library. In my opinion, this is not the best method because if new styles need to be added, they will also need to be added to the legacy files in individually. Using the Styles Library allows you to use the Style and have it update or be added to the files when they are next opened.

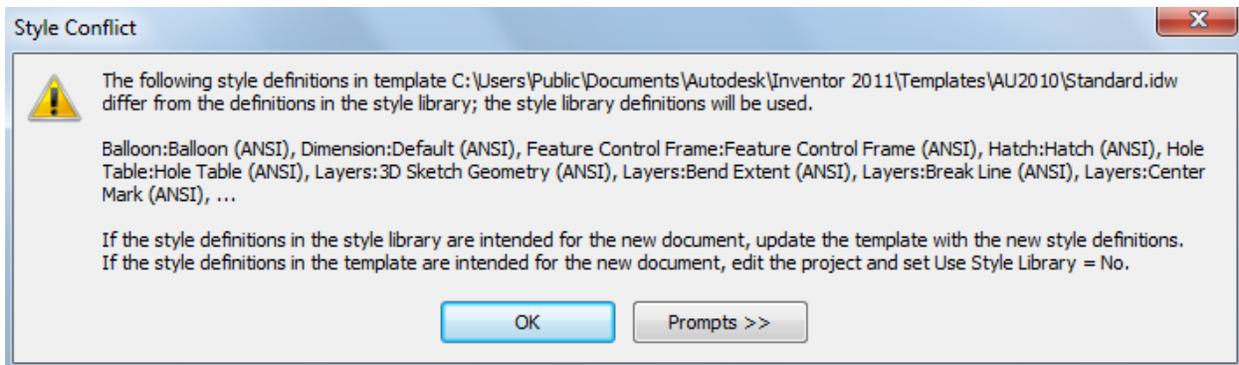
To save the newly created Styles and Active Standards from the template to the library, go to the Manage Tab and the Styles and Standards Panel.

For the Save command to be active, you must have the setting in your Project file set to **Yes** for Styles and Standards and not to **Read Only**. Any changes you have made or any new styles that were added to your file can then be saved to the Library.





If this is not done, every time the template is started with the settings set to **Yes** or **Read Only**, an error message opens prompting you that some styles do not match. It also prompts you that it will use the default styles in the Library rather than the modified ones. This is another great reason why the template should contain all new styles for your company, rather than a modification of the default ones.

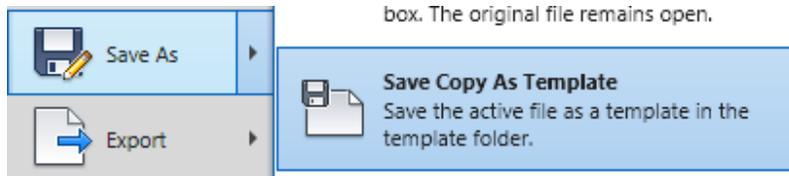


Set your template to the default opening settings

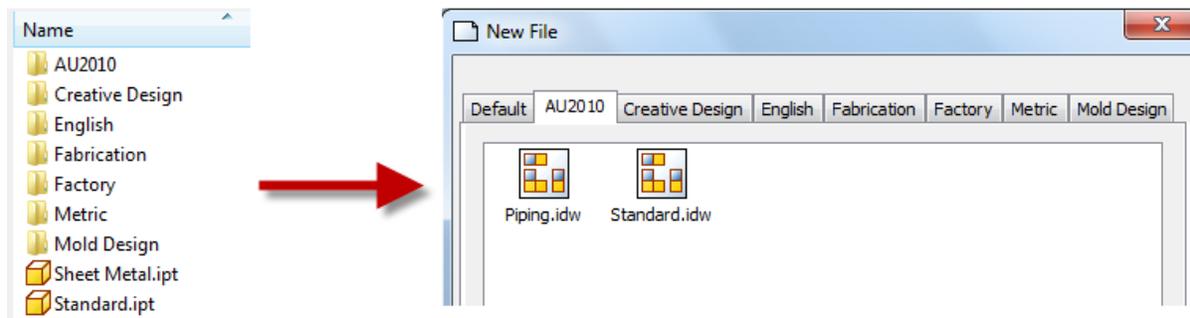
Make sure your template is designed for the most likely Sheet Format and Active Standard, so that when a new document is created from it, these will be used by default. When it is saved and then loaded as a template, it will take the last settings from file to start with. This is a simple step, but unless mentioned it is easily forgotten.

Save Template in template directory

Once completed, save the template in your template directory either manually or with the new *Save Copy As Template* option in the Application Menu.



Remember to create Folders specific to the template's usage and not to overwrite the Default templates as they are used for some higher level functions in Inventor, including Content Center. To create a New Folder, use normal Windows procedures. The folder structure will display as additional tabs in the New File dialog box.



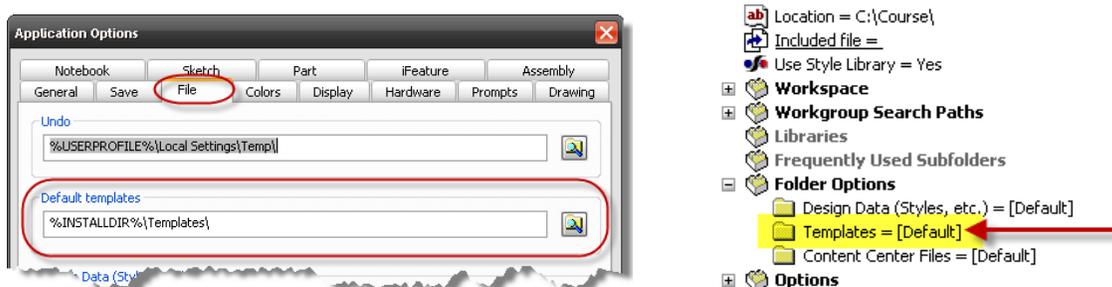
The default location of the Inventor templates is:

Microsoft Windows XP: Program Files\Autodesk\Inventor [version]\Templates

Microsoft Vista/7: Users\Public\Documents\Autodesk\Inventor [version]\Templates

If there is only a single user of Inventor, the default location may suffice. However, with multiple users the templates should be stored in a common location accessible by all Inventor users.

The default template location is set in the Application Options in the File tab. Project files can be configured to override the default template location specifying a different location for Inventor templates. This allows you to create project specific templates.



iLogic

Certain tasks that Autodesk has not built into the drawing environment can be programmed based on your company's policies. Essentially telling Inventor how to work instead of intelligently tricking Inventor to work the way your company documents. Make sure you understand the difference between controlling your rules externally or internally to a file. In the examples below the rules are all saved locally to the template before it is saved so that every file created with the template has this rule to drive the logic.

One use for iLogic is to make sure iProperties are filled in whether they are Custom or Standard as shown below...

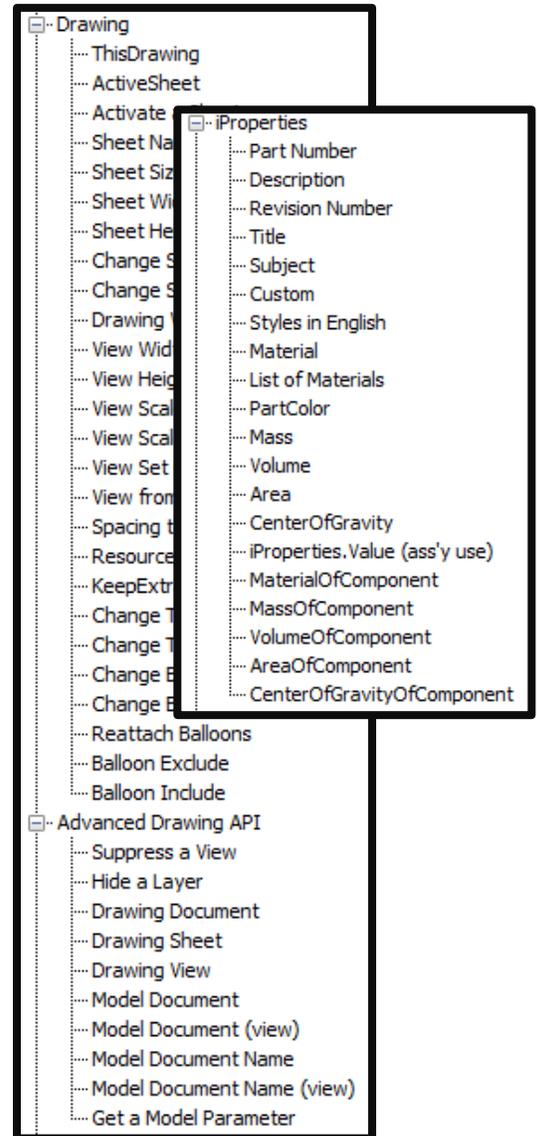
Title iProperty Example:

```
Title = iProperties.Value("Summary", "Title")
If Title = "" Then
Title = InputBox("Please Enter the iProperty Title", "iProperties Check - Title Filled Out?", Description)
iProperties.Value("Summary", "Title") = Title
End If
```

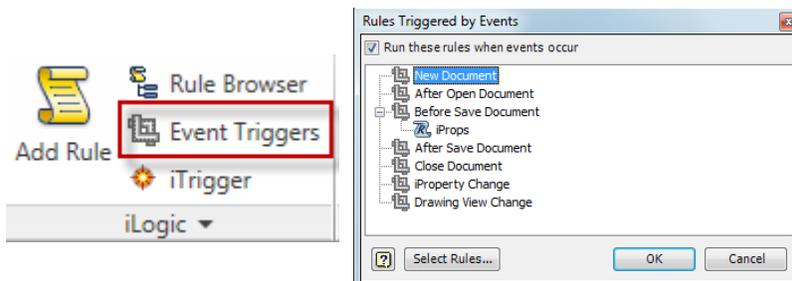
Part Number iProperty Example:

```
PartNumber = iProperties.Value("Project", "Part Number")
If PartNumber = "" Or PartNumber =
ThisDoc.FileName(False) 'without extension
PartNumber = InputBox("Please Enter the iProperty Part Number", "iProperties Check - Part Number", PartNumber)
iProperties.Value("Project", "Part Number")= PartNumber
End If
```

An important note about iLogic is how rules are triggered; the above rules are controlled by the Event Trigger to run Before Save Document.



List of Drawing and iProperty Snippets



Update Drawing iProperties based on Model iProperties (Standard or Custom)

This code snippet has the functionality to update the drawing iProperties with the Model iProperties info. Usually this isn't a big concern since Title Blocks and other properties can be placed into the drawing. The problem with this method is when you are indexing iProperties in Vault and need to search to find files with certain values, the Assembly or Part will show up but not the Drawing. This snippet will take care of that so searches on iProperties will return both results. Also note that this only works well with a one model file / one drawing file documentation approach.

If (ThisDrawing.ModelDocument Is Nothing) Then Return

```
modelName = IO.Path.GetFileName(ThisDrawing.ModelDocument.FullFileName)
```

```
iProperties.Value("Custom", "Property1Name") = iProperties.Value(modelName, "Custom", "Property1Name")
```

```
iProperties.Value("Custom", "Property2Name") = iProperties.Value(modelName, "Custom", "Property2Name")
```

Place 1st View Scale Based into Custom iProperty for Title Block Scale Value

Use this to place the scale string value shown in your 1st Base View dialog box to a custom property (in this case it must already be created), that is used in your Title Block. This way you don't have to baby sit the value for the scale as it will update before saves with the Rule Triggers.

```
iProperties.Value("Custom", "1st View Scale") = ActiveSheet.View("VIEW1").ScaleString
```

The screenshot displays two dialog boxes in Autodesk Inventor. The left dialog, titled 'P1-3 - CD240 Aux View Lab.idw Properties', shows the 'Custom' tab with a table of iProperties. The right dialog, 'Rules Triggered by Events', shows a list of rules with '1stViewScale' selected. A red arrow points from the '1stViewScale' rule in the right dialog to the '1st View Scale' property in the left dialog. Another red arrow points from the '1stViewScale' rule to the '1st View Scale' property in the left dialog. The background shows a drawing template with a title block containing fields for 'DRAWN BY', 'CHECKED', 'ENGINEER', 'REVIEWED', 'RELEASE DATE', 'EDR NO', 'DRAWING NUMBER', 'SCALE', 'SHEET', and 'CHG'.

Name	Value	Type
1st View Scale	1 / 2	Text
EDR Number		Text
Release Date		Date
RM Desc		Text
RM Length	ABC	Text
RM Part #		Text
Sht Size	B	Text

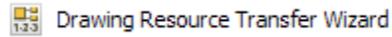
Event	Rule
New Document	1stViewScale
After Open Document	
Before Save Document	
After Save Document	
Close Document	
iProperty Change	
Drawing View Change	

WHERE USED			
DRAWN BY	Number	TOLERANCES	1234 STREET
CHECKED	XXX	FRACTION ± 1/8	CITY, STATE ZIPCODE
ENGINEER	XXX	2 PL ± .01	555-555-5555 WEBSITE
REVIEWED	XXX	3 PL ± .005	
RELEASE DATE		ANGULAR ± 1/2	
EDR NO	B	DRAWING NUMBER	CHG
		102650	N/C
DIMENSIONS ARE IN INCHES AND PROVIDED PRIOR TO SURFACE TREATMENTS, UNLESS NOTED.		SCALE 1 / 2	SHEET 1 OF 1

Drawing Resource Transfer Wizard

If a Drawing Resource needs to be populated through multiple drawings it would be rather tedious and inefficient to manually copy and paste the updated symbol. This is where the Drawing Resource Transfer Wizard (DRTW) is used. This wizard can batch process the addition or replacement of Borders, Title Blocks, and in our case Sketched Symbols to multiple existing files.

Note: DRTW does not work on AutoCAD blocks or Sheet Formats.



The DRTW has to be launched externally to Inventor while the Inventor application is closed. When initialized, the program runs Inventor in the background to process the intended operations. The utility can be found in the Start Menu.



As a rule of thumb, start with a good source file that contains everything you need to transfer to the destination files. This is usually a drawing template that has been updated with the new or updated symbols. Here is a breakdown of the steps in the dialog boxes:

1. **Select the Resources to transfer.**
Choose your source file and check the resources to transfer to the target files.
2. **Select the Target Files.**
These can be IDW or DWG and you can use <Shift> or <Ctrl> to select multiple files.
3. **Same name replacement/ignoring symbols.**
If updating symbols, some data loss is possible if the prompted text fields do not match.
4. **Start and view the process.**
A progress bar will display and a log file will be processed.
5. **Review the log file for error checking.**
Check for any missed or unable to update target drawings.

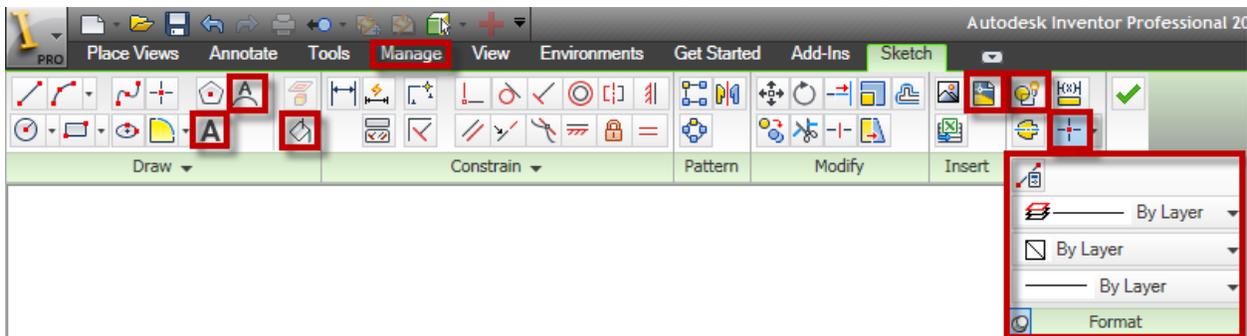
Class Summary: The creation of Drawing Templates and the utilization of Styles and Standards increases the conformity of the documentation process. The various tools and methods allow for a flexible drawing environment that will meet your company's needs now and as standards change throughout the years. Mapping correct data from either the model or drawing and using Autodesk Vault for indexing these values can greatly speed up searching for data in a large file store. By using these templates inside Inventor's Drawing Manager, companies ultimately save time and reduce errors in product life cycles and reduce time spent constantly modifying the same styles as well as having to recreate or manage multiple sheets and formats.

Appendix: Drawing Resource Construction Tools

How many times have you found yourself drawing the same entities for a symbol? Can you do it with your eyes closed? This class aims to teach you how to reduce repetitiveness in your design documentation using user created custom symbols in Inventor and from legacy AutoCAD drawings. Using these tools is excellent for productivity, standardization, and conformity.

Sketching Tools for Drawing Resources

The sketching environment that is activated is slightly different to that found in the modeling environment. For example, the Manage tab does not contain **Parameters** for sketched dimensions even though dimensional relationships can still be made. The highlighted commands below show some of the important commands that are used when creating sketched symbols.

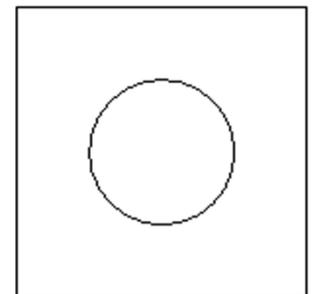
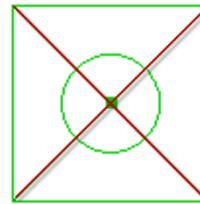
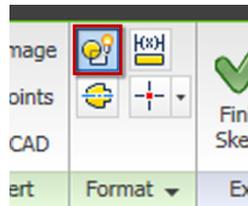


Sketch Only

The **Sketch Only** command in the Format panel allows creation of geometry to be toggled for visibility in the symbol once the definition has been created.

This geometry toggle is similar to the Construction toggle () in the normal modeling sketch environment.

NOTE: Dimensions are always **Sketch Only**.



The 2 red lines in the Sketch on the left are set to Sketch Only so that when the symbol above is created it does not show them.

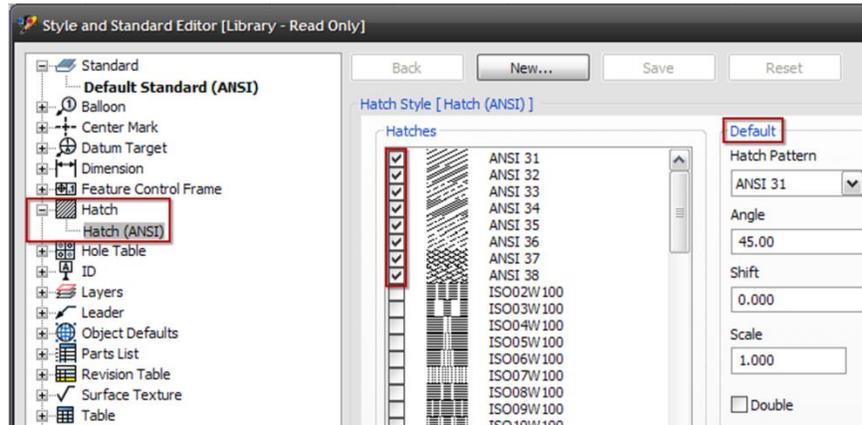
Constraints and Dimensions



The use of Geometric Constrains and Dimensions does not differ from normal Inventor sketch usage, with the exception of dimensional relations that are not tracked in the Parameters Manager. To improve performance in large Title Block sketches, consider using dimensions and constraints along with correctly justified text to create a professional border. Simply eyeballing it does not really work.

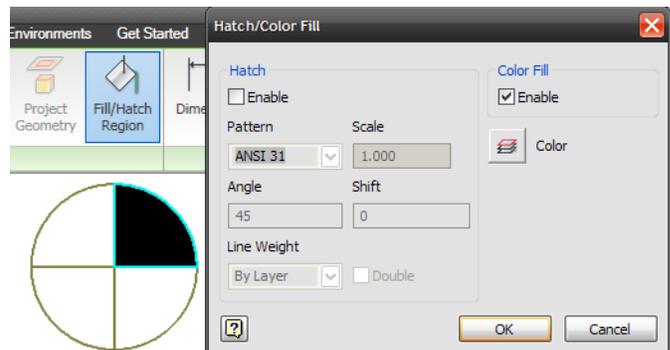
 Fill/Hatch Region

The **Fill/Hatch Region** command hatches an enclosed boundary with a pattern or a solid fill color. Sometimes, when the command cannot distinguish which boundary to hatch or fill, a **Split** may be required to break up some boundary lines. When the command is invoked it will only show the hatches available in the current Standard. In this case, only ANSI 31 - 38 appears in my list and ANSI 31 is the default. In Inventor 2011, you can add custom .PAT files from AutoCAD as well.



Once the **Fill/Hatch Region** command is started, select a single boundary. If the boundary is valid it will highlight as sketched.

Selecting the boundary opens the dialog box to enable **Hatch** or **Color Fill** and their options. This only works for one boundary at a time so repeat as needed.



Related Classes

Styles and Standards Management:

Class ID: CM422-1 – AU2010

Speaker: Stan Wile, MSD Sr. Application Engineer

Class Description:

This class will discuss and demonstrate how to properly configure and use the Design Data folder. We will dive into the folder and review its inner workings. This will take you through Tube and Pipe styles, Hole and Thread data, Sheet Metal styles, Studio Lights and Scenes, as well as each tab of the Styles and Standards dialog box for parts and drawings. Proper migration of the Design Data folder for future releases will be discussed as well.

Learning Objectives:

- Understand the benefit shared styles and standards
- Fully understand how to setup, configure, and use the styles and standards
- Learn about some little known settings hidden in the Design Data folder
- Understand how to setup Inventor to better work for you
- Modify your templates to meet your own company standards

Sketched Symbol Creation and Drawing Sketch Tools:

Class ID: MA314-1 – AU2009

Speaker: Mark Flayler, MSD Application Engineer

Class Description:

Drawing Manager in Autodesk Inventor has the functionality to create Custom Sketched Symbols to document your designs or standardize your company's notations and standards without the monotony of recreation. This class takes a look at the tools available in the Drawing Manager that are designed to facilitate this need. Learn to create and use custom notations and geometry to communicate your design. Use blocks from your AutoCAD® drawings for similar usage with little redesign for use in Inventor. Learn how to organize and update your symbol libraries using Inventor's Drawing Resource Transfer Wizard.

Learning Objectives:

- Learn to create custom sketched symbols in the Drawing Manager
- Use existing blocks from AutoCAD files in Inventor drawings
- Learn how to use prompted entries, iProperties, and connection points effectively
- Identify Design Intent usage in your drawings
- Learn how to update and replace symbols across multiple drawing files