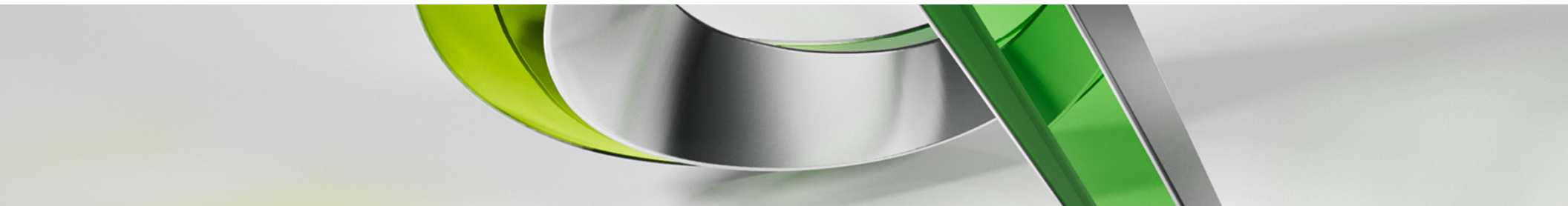




Autodesk® Inventor® Drawing Manager Tips and Tricks

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Class summary

This class covers several workflows that answer the most common questions from the Inventor users about the Drawing Manager.

Key learning objectives

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

- Insert desired properties in drawing title block, parts list, and revision tables
- Update automatically the model properties in the drawing
- Associate materials to solid hatch in section view by default
- Create Sheet Formats with views

Add the scale value from the first view in the title block

Add the scale value from the first view in the title block

- Adding the scale value from the first view in the title block
- Two methods
- Simple iLogic rules

Designed by gassa	Checked by	Approved by	Date	Date 7/3/2013	Scale 1 : 4
		Arbor_Press	Edition	Sheet 1 / 1	

Add the scale value from the first view in the title block

■ Method 1

1. New drawing

2. Add Rule

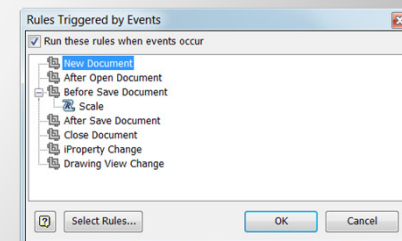
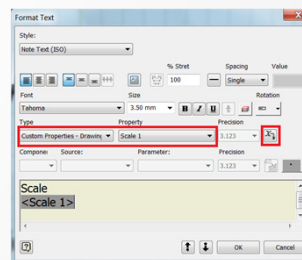
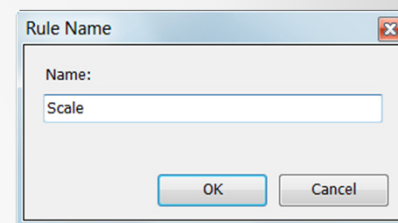
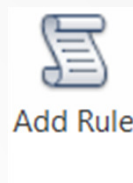
- Script from file "Scale Title Block 1.txt"
- Custom drawing property "Scale x" on sheet x (i.e.: Scale 1 on Sheet:1)

3. For each sheet

- Add "Scale x" to title block
- Save as title block with new name
- Replace title block with new one

4. Add rule to Event Trigger (i.e.: Before Save Document)

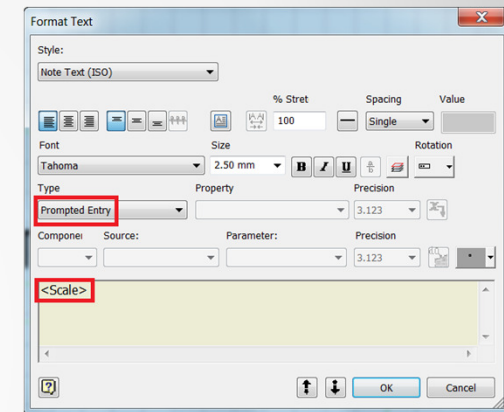
5. Save drawing as Template



Add the scale value from the first view in the title block

■ Method 2

1. New drawing
2. Add Prompted Entry to title block
 - Text should contain <Scale>
3. Add Rule
 - Script from file “Scale Title Block 2.txt”
4. Add rule to Event Trigger (i.e.: Before Save Document)
5. Save drawing as Template



Add the scale value from the first view in the title block

- Limitation of Method 1
 - One different title block for each sheet
 - Caveat when adding extra sheets and / or changing their order
- Limitation of Method 2
 - Need to close Prompted Entry dialog
 - Adding title block
 - Adding new sheet
 - Starting new drawing (one dialog per sheet)

Plot the date stamp in the title block

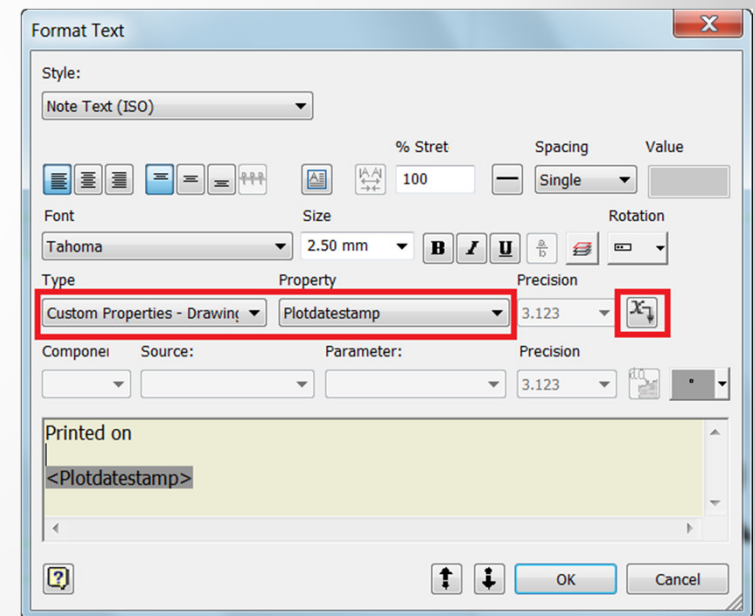
Plot the date stamp in the title block

- Include the plot date in the title block
- “plot date” is not “create date”
- Simple iLogic rule

Date	Printed on 7/5/2013	Date	7/3/2013	Scale	1 : 2.5
Arbor_Press			Edition	Sheet 2 / 5	

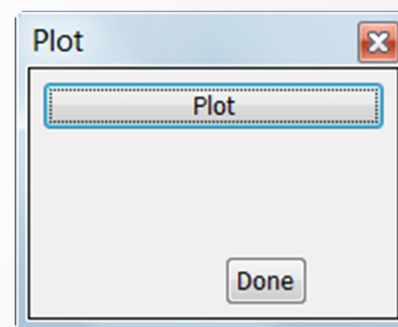
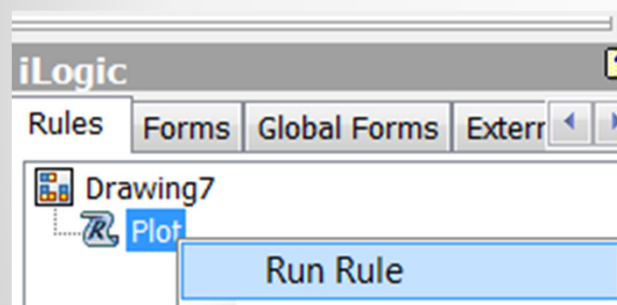
Plot the date stamp in the title block

1. New drawing
2. Add Rule
 - Script from file "PlotDateStamp.txt"
 - Custom drawing property "Plotdatestamp"
3. Add "Plotdatestamp" to title block
4. Save drawing as Template



Plot the date stamp in the title block

- Run the rule for printing the drawing
 - Update the plot date
 - Initiate the print dialog



Add Density Property to Parts List & BOM

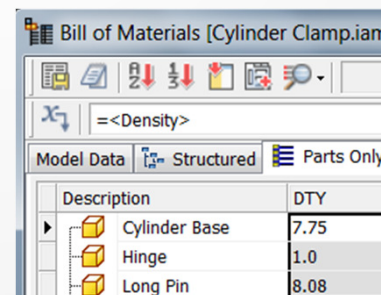
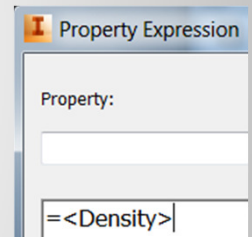
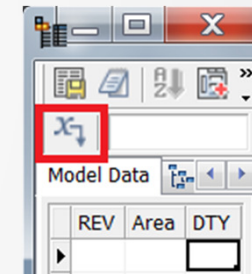
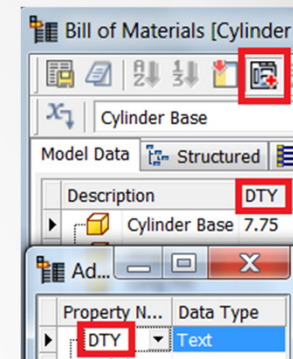
Add Density Property to Parts List & BOM

- Expose component Density in BOM and Parts List

PARTS LIST				
ITEM	QTY	PART NUMBER	DESCRIPTION	Density (g/cm ³)
1	1	TU-007	Cylinder Base	7.75
2	1	TU-008	Hinge	1.0
3	1	TU-009	Long Pin	8.08

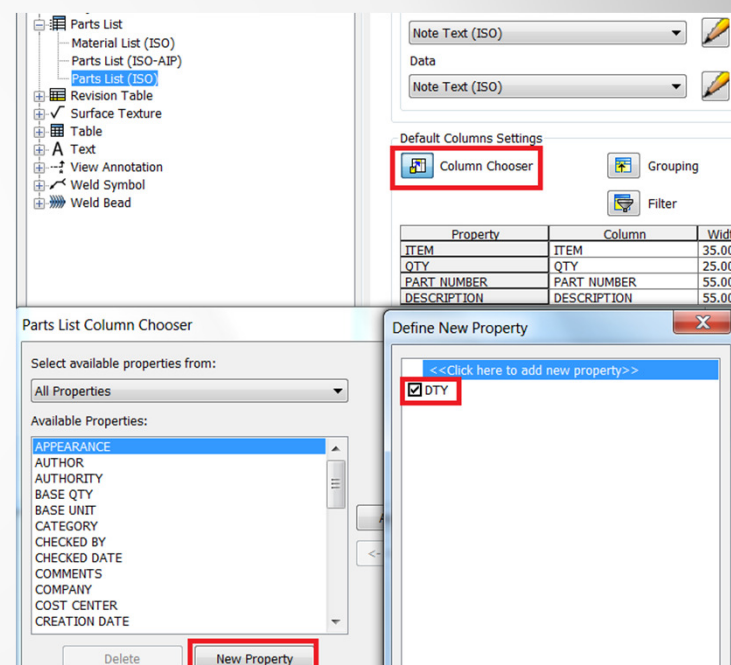
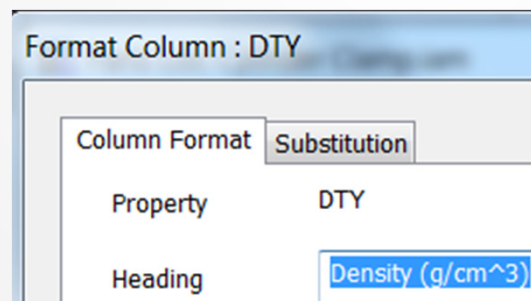
Add Density Property to Parts List & BOM

1. Open the assembly
2. Add Custom iProperty Column to BOM
 - Name not equal to “Density” or “DENSITY”
 - Type is text
3. Select the first cell
4. Create following expression
 - “=<Density>”
 - Expressed in g/cm³
5. Drag it down to the last cell (Part Only tab)



Add Density Property to Parts List & BOM

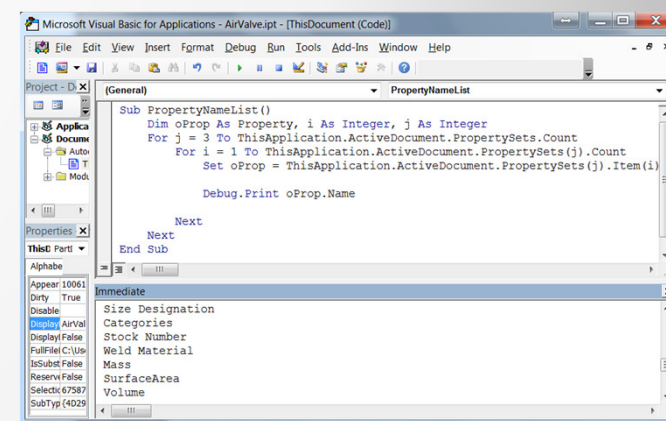
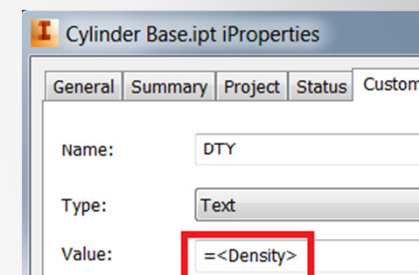
6. Add the custom column to the Parts List style (drawing)
7. Rename the column heading



Add Density Property to Parts List & BOM

Additional information

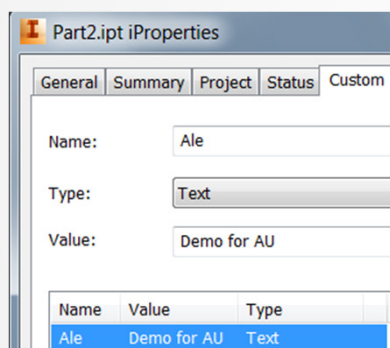
- Settings for templates
 - Define Custom Property in part template
 - Add Custom iProperty Column (step 2) in assembly template
 - Parts List settings (step 5 and 6) in Style Library
 - Or add style in template
- Other component properties
 - Same procedure
 - VBA "Attribute.txt"
 - List available properties in Immediate Window



Add model iProperties to the drawing Revision Table

Add model iProperties to the drawing Revision Table

- Include model iProperties in Revision Table



Part2.ipt iProperties

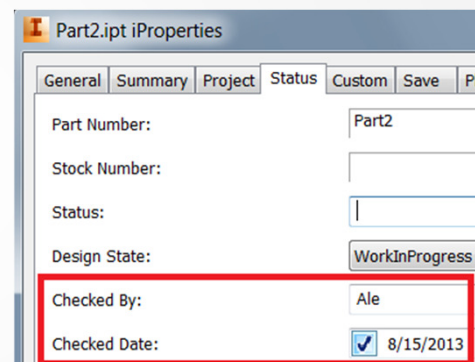
General Summary Project Status Custom

Name: Ale

Type: Text

Value: Demo for AU

Name	Value	Type
Ale	Demo for AU	Text



Part2.ipt iProperties

General Summary Project Status Custom Save PL

Part Number: Part2

Stock Number:

Status:

Design State: WorkInProgress

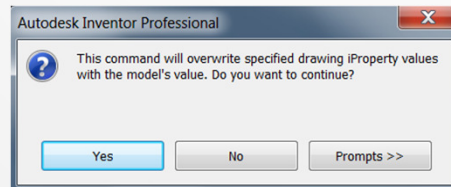
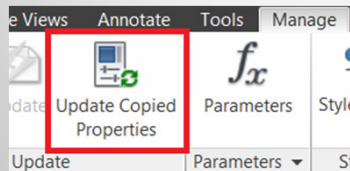
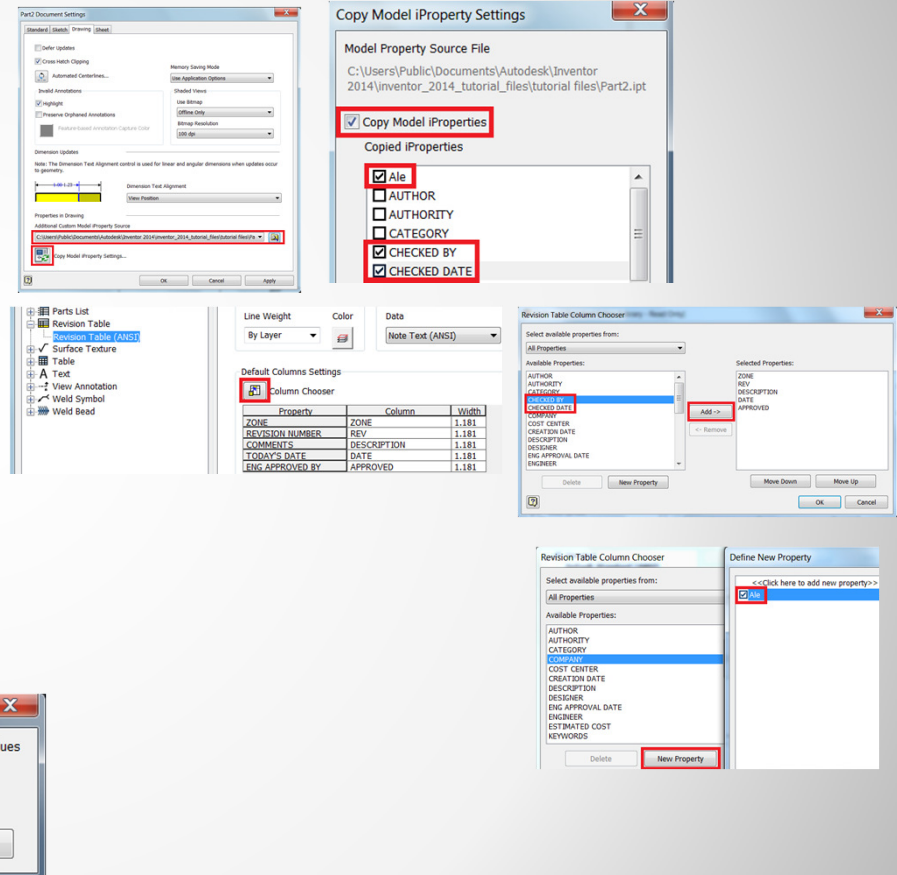
Checked By: Ale

Checked Date: ☒ 8/15/2013

REVISION HISTORY							
ZONE	REV	DESCRIPTION	DATE	APPROVED	CHECKED BY	CHECKED DATE	Ale
	1		8/16/2013		Ale	8/15/2013	Demo for AU

Add model iProperties to the drawing Revision Table

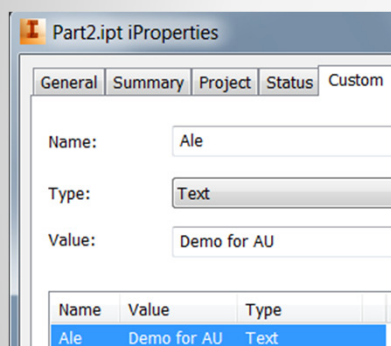
1. Open the drawing
 - With model views
2. Copy Model iProperties
3. Add columns to Revision Table style
 - Or to the existing table
4. Update Copied Properties
 - When the model properties change
 - Update the last added row



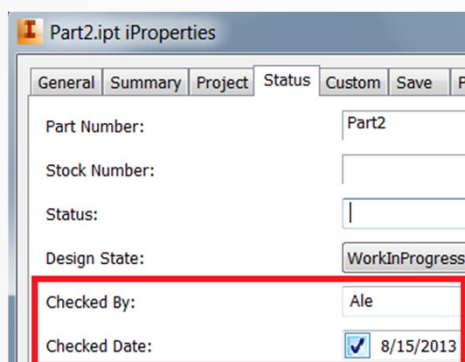
Update automatically drawing properties from model iProperties

Update automatically drawing properties from model iProperties

- Keep model and drawing properties in sync
- “Update Copied Properties” not needed

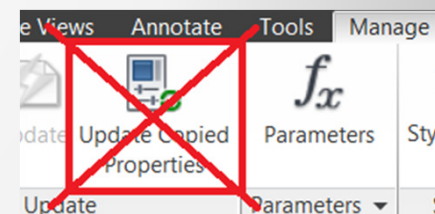


Name	Value	Type
Ale	Demo for AU	Text



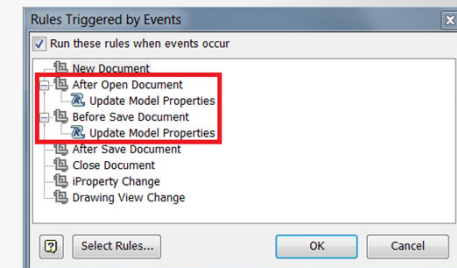
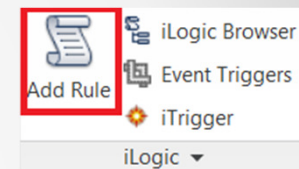
Checked By:	Ale
Checked Date:	<input checked="" type="checkbox"/> 8/15/2013

REVISION HISTORY							
ZONE	REV	DESCRIPTION	DATE	APPROVED	CHECKED BY	CHECKED DATE	Ale
	1		8/16/2013		Ale	8/15/2013	Demo for AU



Update automatically drawing properties from model iProperties

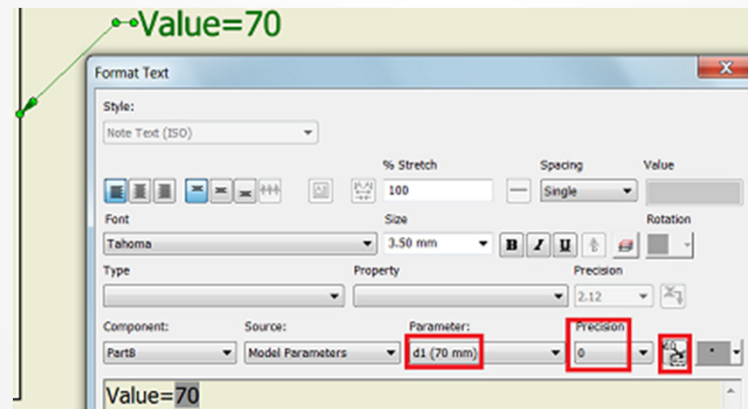
1. New drawing
2. Add Rule
 - Script from file “Update Model Properties.txt”
3. Add rule to Event Trigger
 - After Open Document and/or Before Save Document
4. Save drawing as Template



Remove trailing zeros of referenced parameters in leader text

Remove trailing zeros of referenced parameters in leader text

- Model parameter value with no trailing zeros
- Add Parameter with desired precision not recommended
 - Display value not accurate if changed in the model



Remove trailing zeros of referenced parameters in leader text

■ Recommended Workflow

1. Select the export checkbox for the parameter

2. Change the format removing trailing zeros and unit

3. Insert the value as Custom Property – Model

Parameter Name	Unit/Type	Equation	Nominal Value	Tol.	Model Value	Key	Export Parameter
Model Parameters							
d0	mm	100 mm	100.000000		100.000000		
d1	mm	70 mm					<input checked="" type="checkbox"/>
d2	mm	10 mm					
d3	deg	0.0 deg					

Custom Property Format

Property Type:
Text

Preview
70

Units:
mm

Format:
Decimal

Precision:
3.123

☐ Units String

☒ Leading Zeros

☐ Trailing Zeros

Value=70

Format Text

Style:
Note Text (ISO)

Font:
Tahoma

Type:
Custom Properties - Model

Property:
d1

Precision:
2.12

Component:
Part8

Source:
Model Parameters

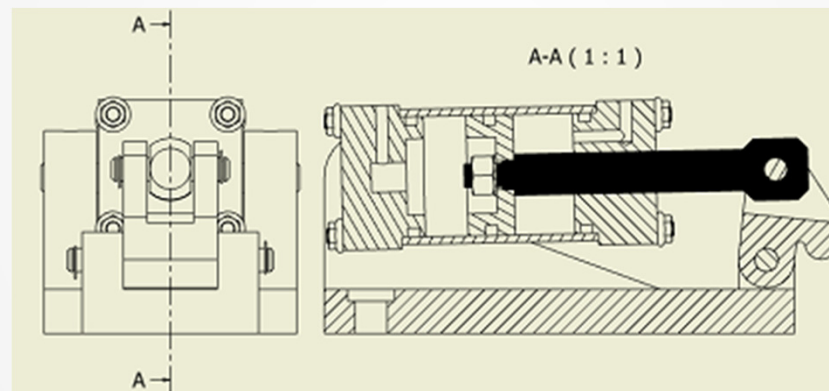
Parameter:
d0 (100.00 mm)

Value:
Value=<d1>

Associate by default materials to solid hatch in section views

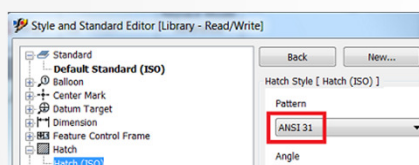
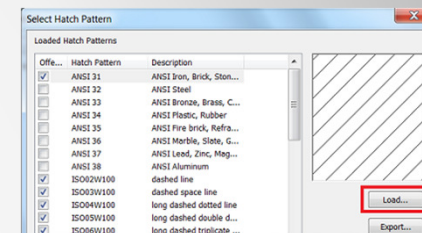
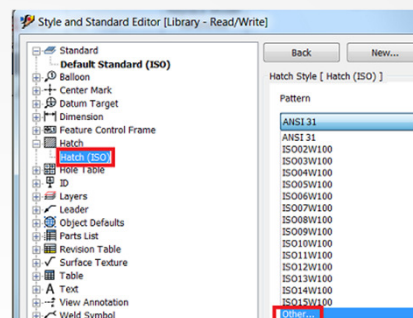
Associate by default materials to solid hatch in section views

- Solid hatch for components with specific materials

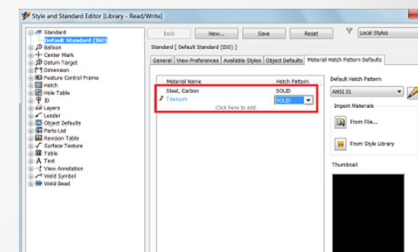


Associate by default materials to solid hatch in section views

1. New drawing
2. Add the solid Hatch Pattern to the active Hatch style
 - Load "SOLID.PAT"
3. Set the default Pattern back to the usual one



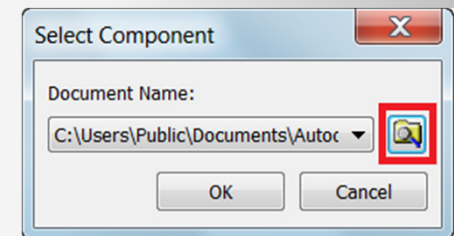
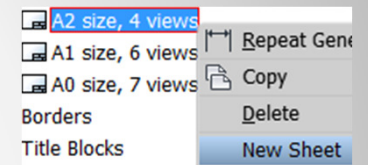
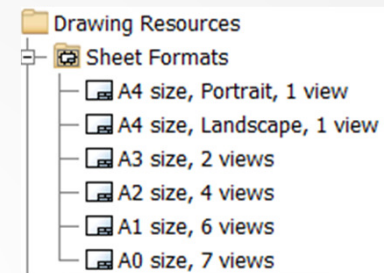
4. Associate the materials to the solid hatch
5. Save the edits in the Style Editor
6. Save settings in Style Library
 - Or add style in template



Create Sheet Formats with views

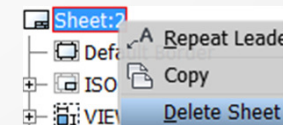
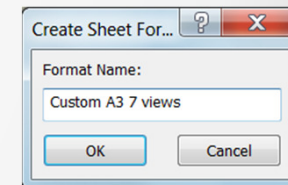
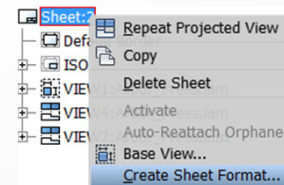
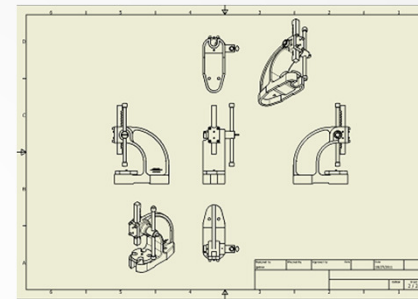
Create Sheet Formats with views

- Predefined Sheet Formats not suited
- Create new Formats



Create Sheet Formats with views

1. New drawing
2. New sheet
 - Desired size and orientation
 - Desired border and title block
3. Place model views
 - Base and projected
 - Desired position scale and style
 - Within the border
4. Create Sheet Formats
 - Assign name
5. Delete the sheet
6. Save drawing as Template

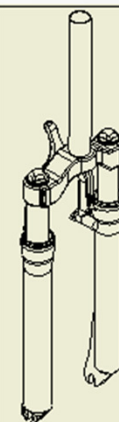


Parts Only and Structured Parts Lists in one drawing

Parts Only and Structured Parts Lists in one drawing

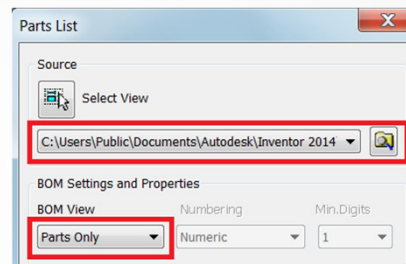
- Both BOM Views in the same drawing

STRUCTURED				PARTS ONLY		
ITEM	QTY	PART NUMBER	DESCRIPTION	QTY	PART NUMBER	DESCRIPTION
1	2	Dial-Elastomer		2	Dial-cap	
1.1	1	Dial-cap		2	Dial-Cover	
1.2	1	Dial-Cover		2	Dial	
1.3	1	Dial		12	Elastomer	
1.4	6	Elastomer		2	Dial-Base	
1.5	1	Dial-Base		10	Spacer	
1.6	5	Spacer		1	Fork-Slider	
2	1	Fork-Slider		1	Fork-Piston	
3	1	Fork-Piston		1	Fork-Crown	
4	1	Fork-Crown		1	Head Tube	
5	1	Head Tube		1	Fork-Tube	
6	4	NF E 25-125 - M3 x 10	Slotted cheese head - Product grade A - Symbol C S	1	Fork-Damper	
				1	Fork-Brace2	
				1	Fork-Slider_MIR	
7	2	UNI 5933 - M4 x 12	Countersunk Screw	1	Fork-Piston_MIR	
8	1	Fork-Tube		1	Fork-Tube MIR	
9	1	Fork-Damper		1	Fork-Damper MIR	
10	1	Fork-Brace2		2	UNI 5933 - M4 x 12	Countersunk Screw
11	1	Fork-Slider_MIR		4	NF E 25-125 - M3 x 10	Slotted cheese head - Product grade A - Symbol C S
12	1	Fork-Piston_MIR				
13	1	Fork-Tube_MIR				
14	1	Fork-Damper MIR				

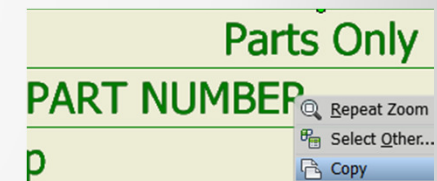
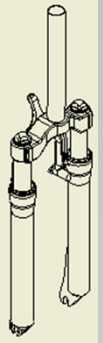


Create Sheet Formats with views

1. Parts List with one BOM View in main drawing
 - Balloons will follow this View
2. New drawing
3. Parts List with other BOM View
 - No need for creating view
4. Copy & Paste it in main drawing
 - Remove the ITEM column



STRUCTURED			
ITEM	QTY	PART NUMBER	DESCRIPTION
1	2	Dial-Elastomer	
1.1	1	Dial-cap	
1.2	1	Dial-Cover	
1.3	1	Dial	
1.4	6	Elastomer	
1.5	1	Dial-base	
1.6	5	Spacer	
2	1	Fork-Slider	
3	1	Fork-Piston	
4	1	Fork-Crown	
5	1	Head Tube	
6	4	NF E 25-125 - M3 x 10	Slotted cheese head - Product grade A - Symbol C S
7	2	UNI 5933 - M4 x 12	Countersunk Screw
8	1	Fork-Tube	
9	1	Fork-Damper	
10	1	Fork-Brace2	
11	1	Fork-Slider_MIR	
12	1	Fork-Piston_MIR	
13	1	Fork-Tube_MIR	
14	1	Fork-Damper MIR	



Summary

We had a good look at several workflows that answer the most common questions from Inventor users about the Drawing Manager.

For more tips and trick about these topics, refer to the [Being Inventive](#) Inventor blog, under the [Drawing](#) Category.

Thank you for attending!

