

Technical Drawing notes

To produce notes on drawing is often a tedious job that includes placing similar texts on drawings. This job can be done a lot more efficient by using a library of some sort. There are of course several options:

1. Creating Blocks under the drawing resources so engineers can place them from there.
2. Creating a central text document where the most common notes can be found.

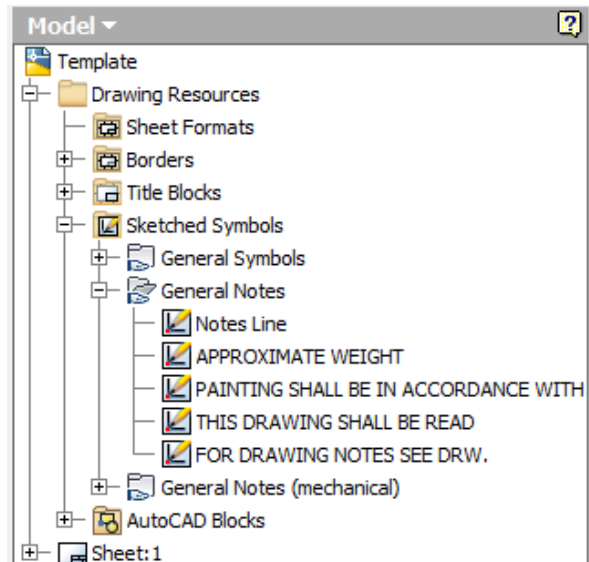


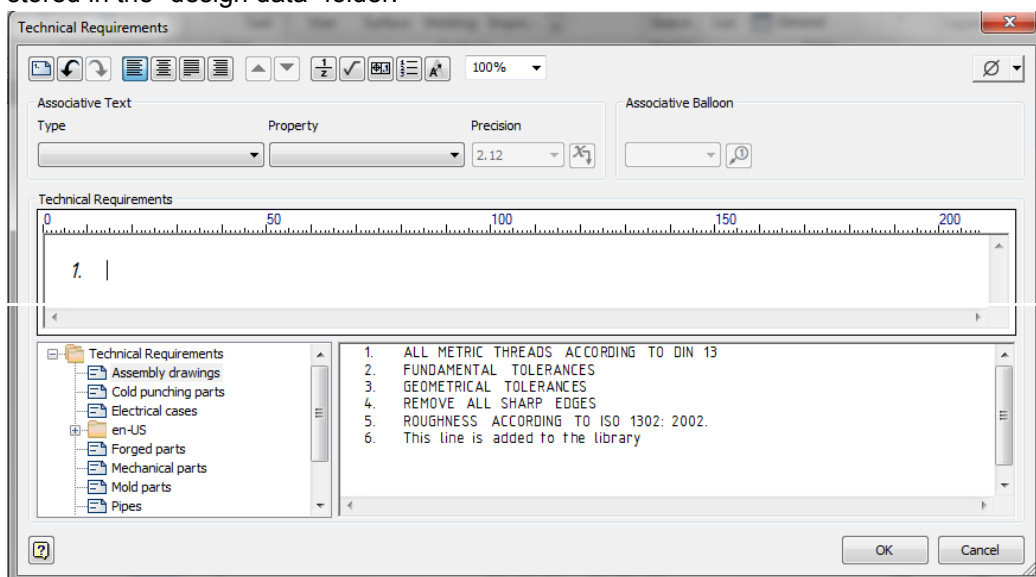
Figure 0.1 creating a library in the template can be hard to manage and increases the size of each drawing

However there are some symbols that cannot be placed by either one of these methods. The symbols I am talking about are of course the symbols that define the roughness and the feature control



Figure 0.2 Roughness and Feature Control symbols

There is a 'hidden' tool available in Inventor under the ESKD plugin (GOST annotations) called "Technical requirements". This tool allows users to create these specific notes with the appropriate symbols. And maybe even more important: it allows users to create a library. This library is then stored in the "design data" folder.

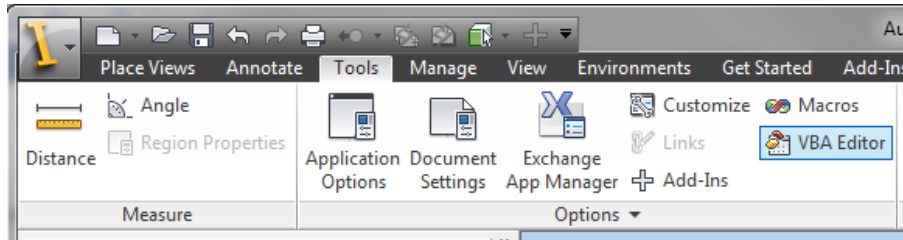


However the tool is only available when this plugin is activated. This means that those of us that do not use GOST standards would need to turn the plug-in off and on.

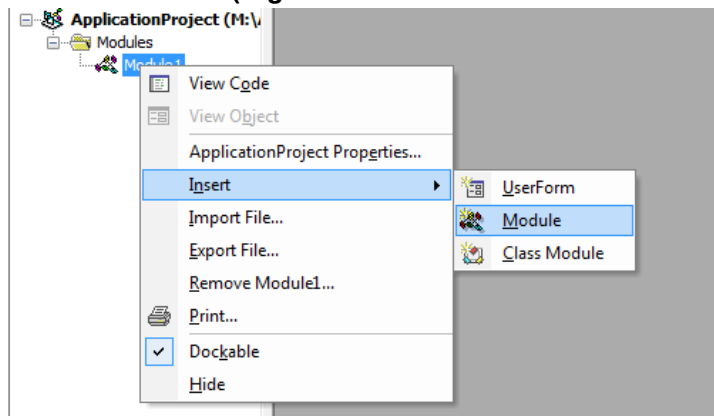
This, of course, eliminates the efficiency that can be gained from this tool. This is where the macro that can be found in the link below comes in to play. This allows you to create a button to activate the 'Technical Requirements' tool and then close it without having the ESKD plugin active all the time:

To apply the code to your Inventor installation:

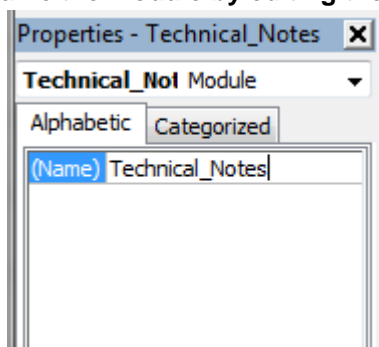
Go to the "VBA Editor" (*Tools Tab, Options Panel*)



Create a new module. (*Right Mouse Button -> Insert -> Module*)



Rename the module by editing the properties:



Insert the VBA code in the module:

```
(General) Deactivate

' Get the DWF AddIn using its ID
Dim AddIn As ApplicationAddIn
Set AddIn = addins.ItemById("{005B21FC-8537-4926-9F57-3A3216C294C3}")

' Activate AddIn
If AddIn.Activated = True Then
    Exit Sub
Else
    AddIn.Activate
End If
End Sub

Public Sub Deactivate()
    Dim app As Application
    Set app = ThisApplication

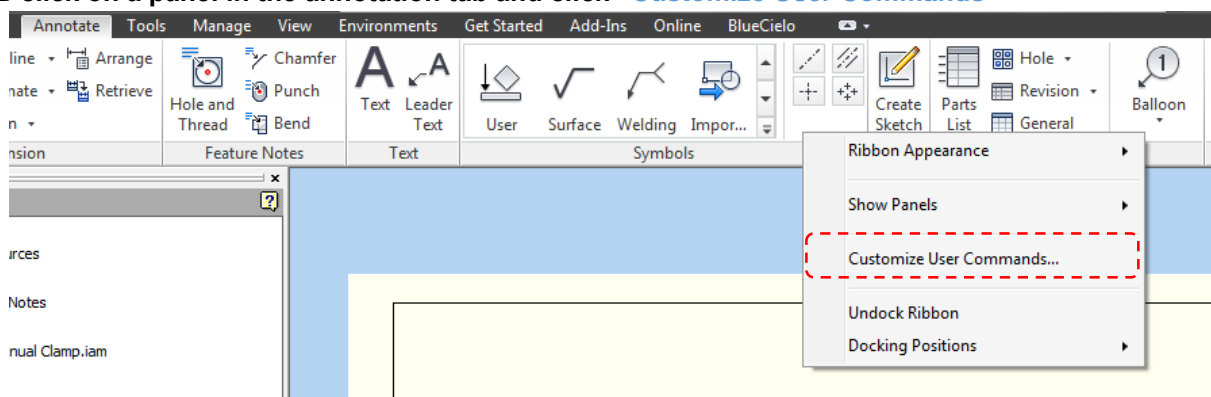
    Dim addins As ApplicationAddIns
```

You can find the code at: http://hoppend.nl/Technical_notes_VBA.txt

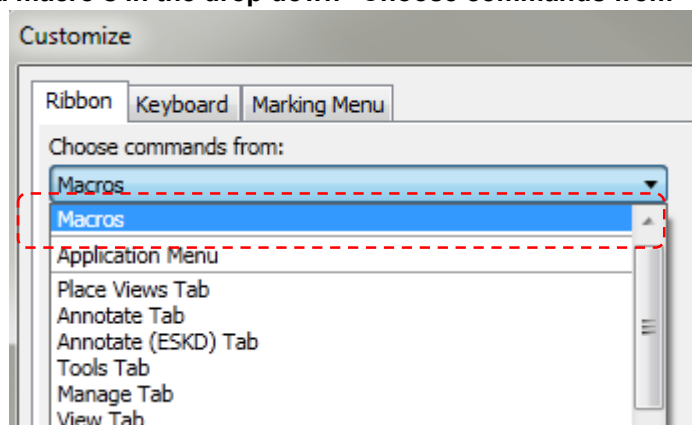
Save your project and close the VBA editor

Open a drawing

RMB click on a panel in the annotation tab and click “Customize User Commands”



Find macro's in the drop down “Choose commands from”



Add the macro to the ribbon of your liking:

