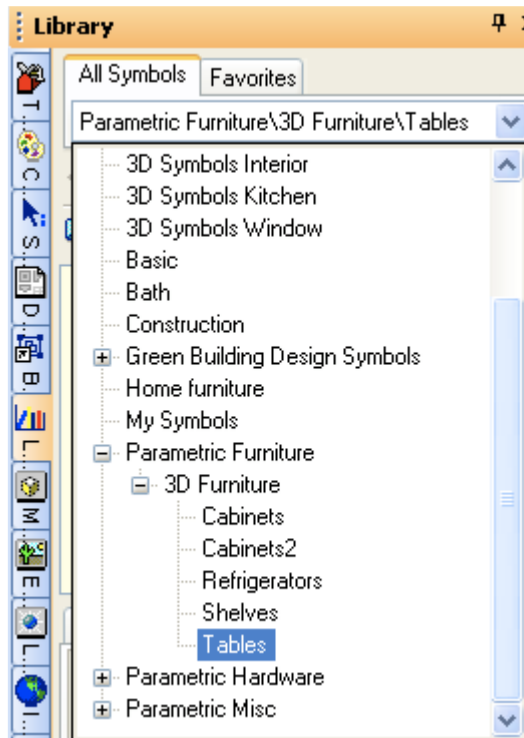
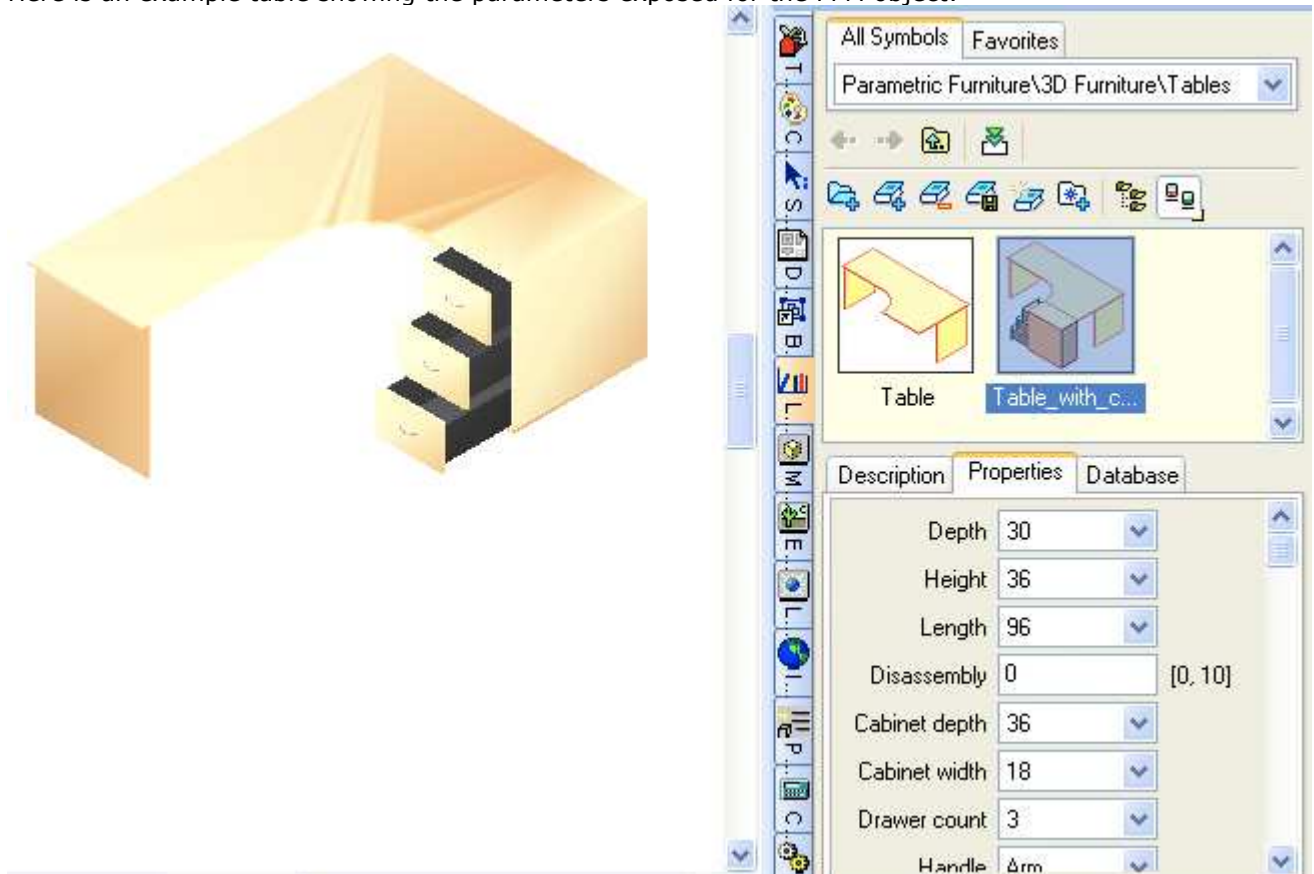


Parametric Part Manager TurboCAD Pro 15 saw the introduction of the **Parametric Part Manager**. The PPM can be used to create parts or objects that only exposes a select number of parameters for the user to change. PPM objects can be defined using the [PPM Scripting](#) language or with a wizard using the [PPM Wizard](#). TurboCAD comes with a number of predefined parametric objects that can be accessed from the [Library Palette](#):



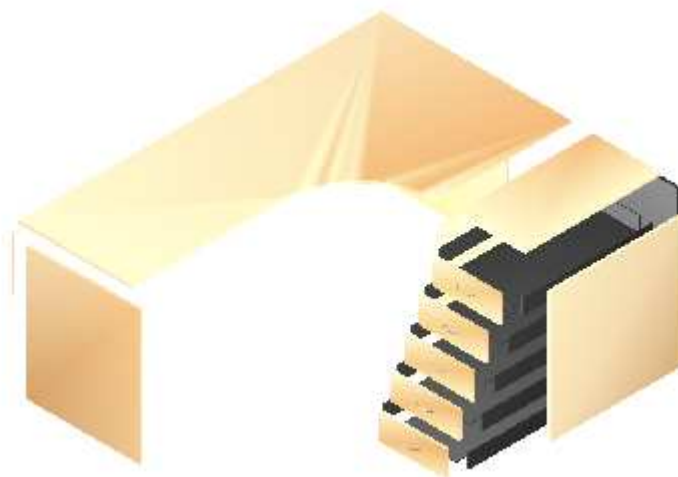
Here is an example table showing the parameters exposed for the PPM object:



In the part tree in the [Selection Info Palette](#), I changed the **Disassembly** parameter to 6 and the **Drawer Count** parameter to 5:

Properties		Constraints
Property	Value	
General		
Pen		
3D		
Parametric 5...		
Symbol Par...		
Depth	30 in	
Height	36 in	
Length	96 in	
Disass...	6	
Cabinet..	36 in	
Cabinet..	18 in	
Drawer...	5	
Handle	Arm	

and I get the following result:

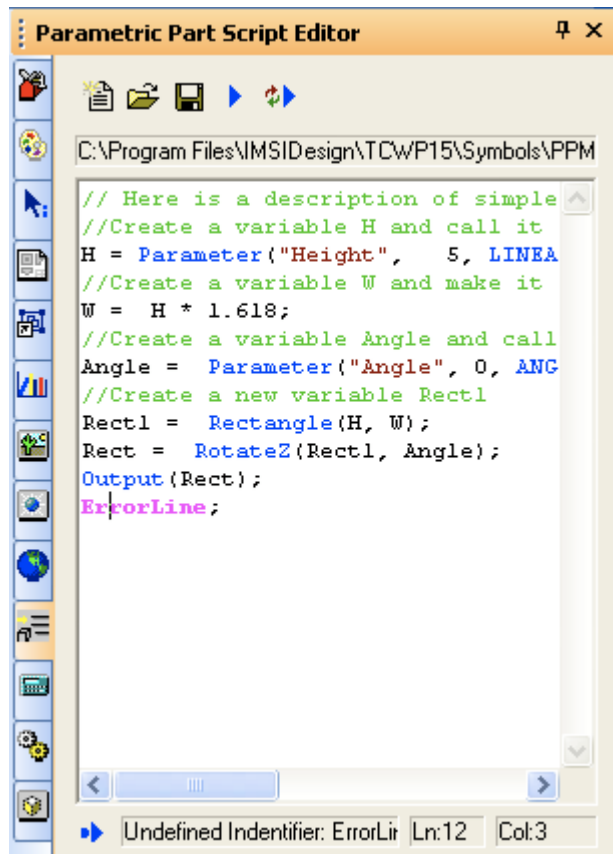


The PPM Scripting language is a very simple geometric description language with keywords to define geometry and parameters associated with the geometry.

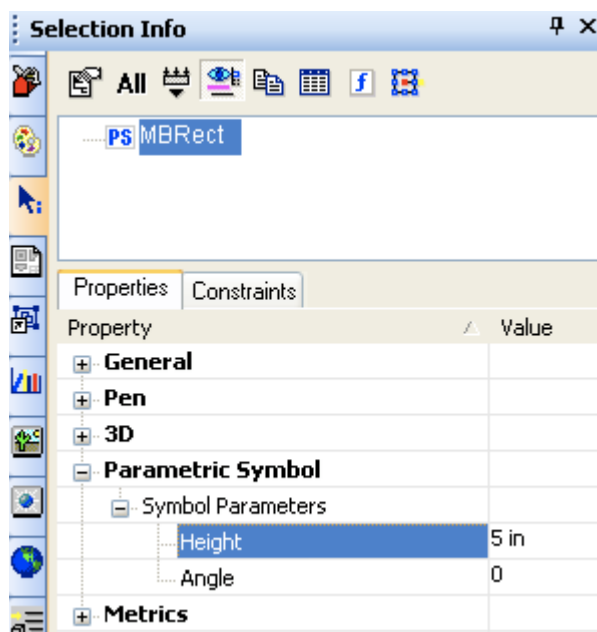
To use the PPM Script editor, select **Parametric Part Script Editor** from the **View** menu. Here is a simple example, for more detail go to the [PPM Scripting Reference](#) page:

```
// Here is a description of simple rectangle.
//Create a variable H and call it Height. The minimum value is 10, the maximum 100
H = Parameter("Height", 5, LINEAR, Interval(10, 100));
//Create a variable W and make it the golden ratio of H
W = H * 1.618;
//Create a variable Angle and call it Angle. The minimum value is 0, the maximum 360
Angle = Parameter("Angle", 0, ANGULAR, Interval(0, 360));
//Create a new variable Rect1
Rect1 = Rectangle(H, W);
Rect = RotateZ(Rect1, Angle);
Output(Rect);
```

Type the above into the editor window and click on the disk icon to save. You will notice that the text is color coded. Black text indicates variables, brown indicates text and blue indicates reserved words. Magenta text indicates an error. The status line of the palette will indicate the reason for the error. In the example of the screen shot it is **Undefined Identifier:ErrorLine**



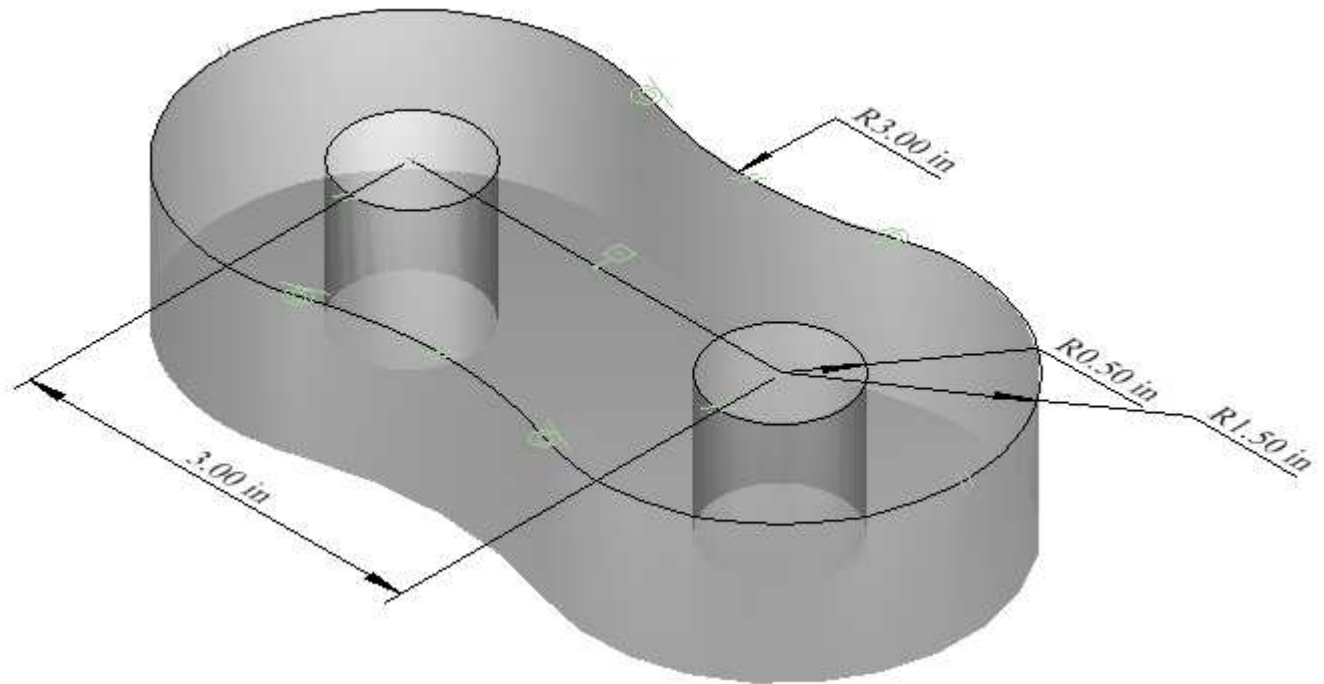
Once it is saved you can click on the blue arrow to run the script. It will draw a rectangle on the screen. If you select the rectangle and go to the Selection Info Palette² you will see that you now have the two parameters: **Width** and **Angle**. If you change **Width** the rectangle's width will change and so will the height, according to the golden ratio.



The PPM Wizard

The PPM Wizard allows you to create parts or symbols where you can specify which parameters can be changed by the end user. If you think about the geometry of a simple table, then you have an infinite number of possible values for the different dimensions. In reality when you have to manufacture the table, you are left with a couple of models that share the same basic geometry, the same height, and only the length or the width can be changed.

For this example, i will use a simple chain link shown below:

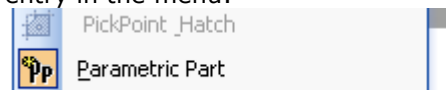


The specification that I have is that only the inside hole diameter variable.

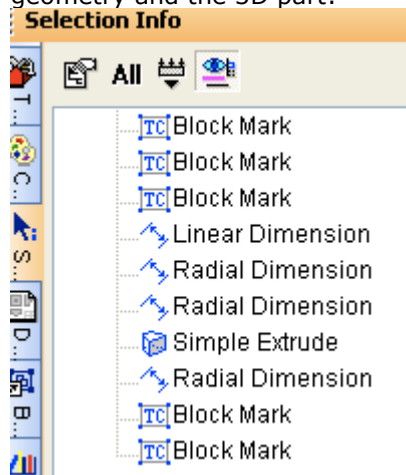
The second value that can change is the thickness of the chain link.

To set this up, I've used a constrained based design (see the green marks) and used driving dimensions.

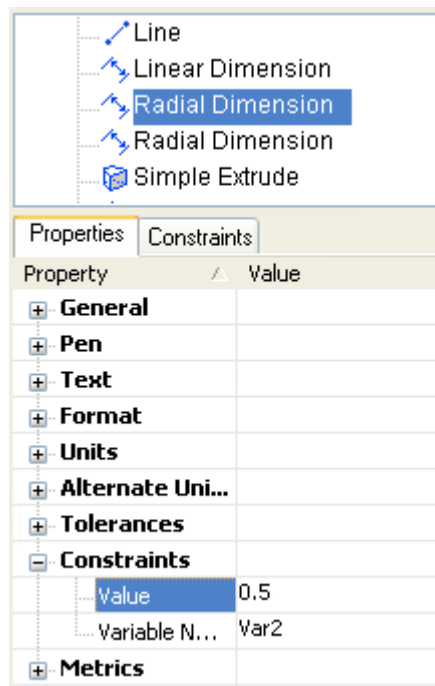
The first step is to start the wizard. Choose **Parametric Part** from the format menu, it is the last entry in the menu:



I now have two main options. The first is to select the 3D part, but then I will only be able to change the thickness because that is the only information in the part tree. The second option is to select everything because that also includes the 2D geometry that was used for the extrude operation. I draw a selection rectangle around the whole part to select everything, then select the **Parametric Part** and in the [Selection Info](#) palette I can see the part tree contains the 2D geometry and the 3D part:



The next step is to select all the parameters that I want to expose in my part. I click on the **Radial Dimension** entries in the part tree until I see the appropriate radial dimension highlighted in the drawing. Then I open the **Constraints** branch because I want to add the **Value** parameter of the dimension as a parameter.

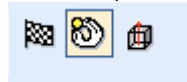


A right click on the **Value** will open a dialog box prompting me for a name for the variable. I call it **HoleR** and click **OK**.

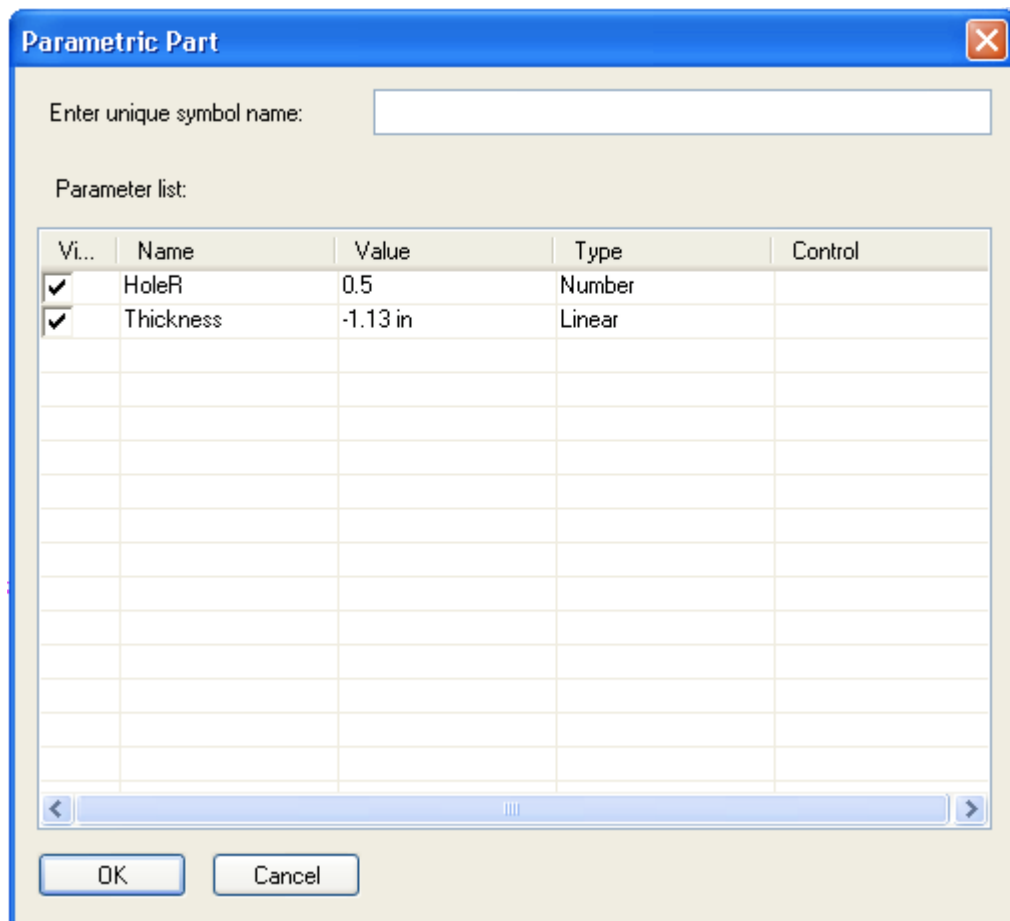


Next, I repeat the same steps for the extrude operation. Select **Simple Extrude** in the part tree and open the **Simple Extrude** branch in the details window, right click and call this parameter **Thickness**.

Done with the definitions and in the status bar I have three options, the first option is to finish the selection, the second is to reposition the reference point and the last is to set the assembly axis.

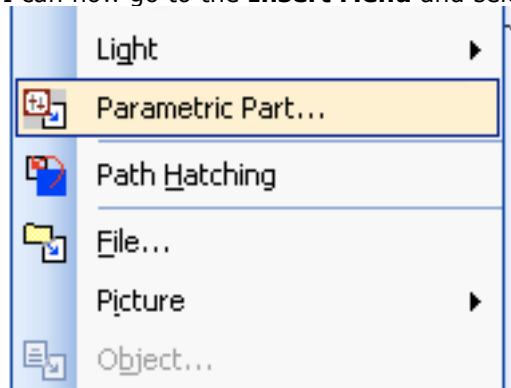


Click on the **Finish** button and it brings up a new dialog box.



Call the part **Chain Link** and click **OK**

I can now go to the **Insert Menu** and select **Insert Parametric Part**



And from the list I select **Chain Link**. If I now click on the inserted part, go to the [Selection Info Palette](#) and open the **Parametric Symbol** branch, I see that there are only two parameters available to change, **HoleR** and **Thickness** and I can use them to manipulate the part.

Selection Info

T

C

S

D

B

L

M

All

f

PS Chain Link

Properties

Constraints

Property	Value
General	
Pen	
3D	
Parametric Symbol	
Symbol Paramete...	
HoleR	0.5
Thickness	-1.13 in