Inventor 2015 – Pattern Along An Object

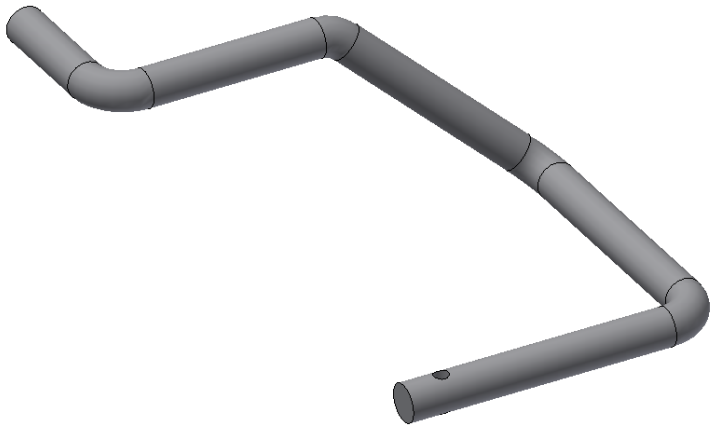
How to use the rectangular pattern command to pattern a feature along an object.

When modelling in Inventor if the same feature appears multiple times then the pattern commands can really help save time.

There are 2 commands. Rectangular Pattern and Circular Pattern.

But do not be fooled. A Rectangular Pattern can be any shape pattern you want. The example below is an often asked support query on how to pattern a feature along or around an object.

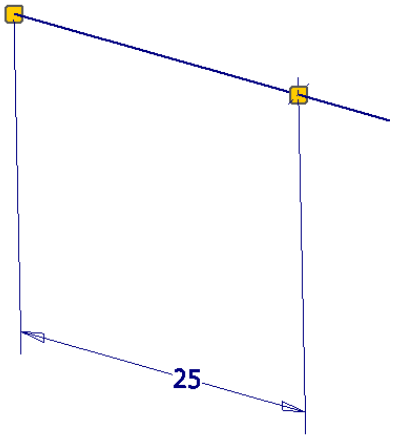
Lets start with along. Lets say you have a swept object like a tube.



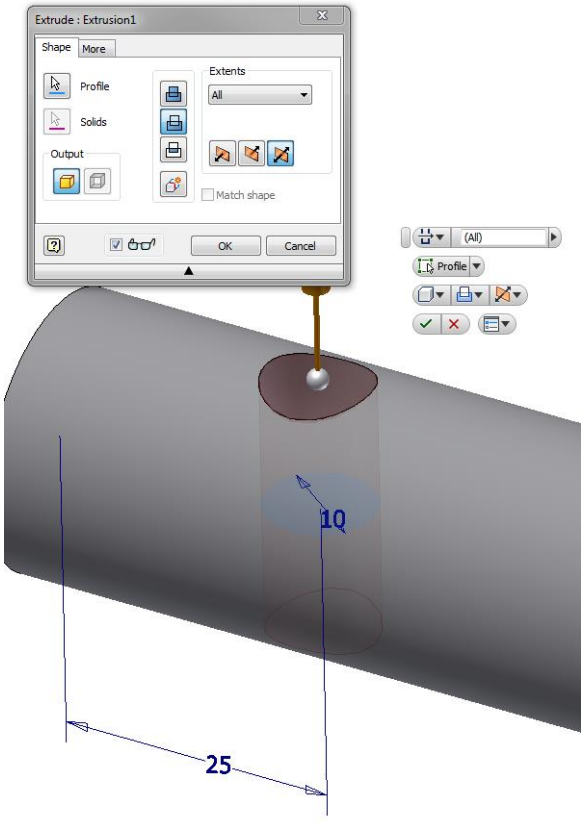
You can see in the above picture I have a single hole. If I want that hole to be spaced equally along this swept feature then I can use the Rectangular Pattern command.

Now there are a number of things to take into account. You will want your sweep to have been created with a single feature and sketch (2D or 3D).

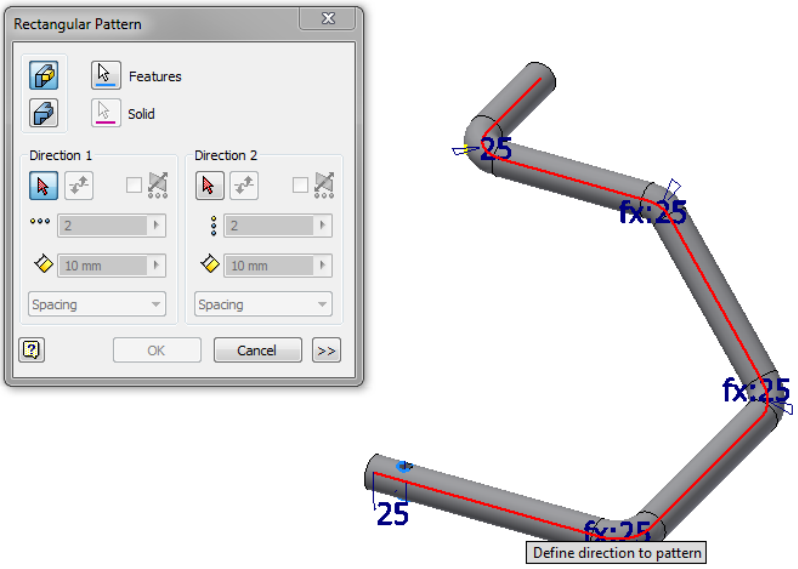
You will also want your first position indicated on the sketch as a point, in this case the location of my first hole (which was made referencing this point).



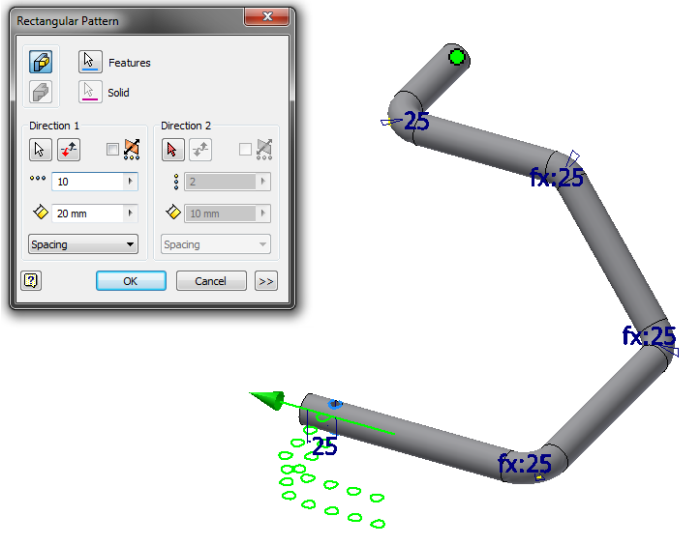
Now I have also been mentioning the term hole. In some cases the hole doesn't behave well, so an extrusion gives you loads more options, so infact my hole is a circle cut extrusion (note that I extruded in both directions and my cut was through all).



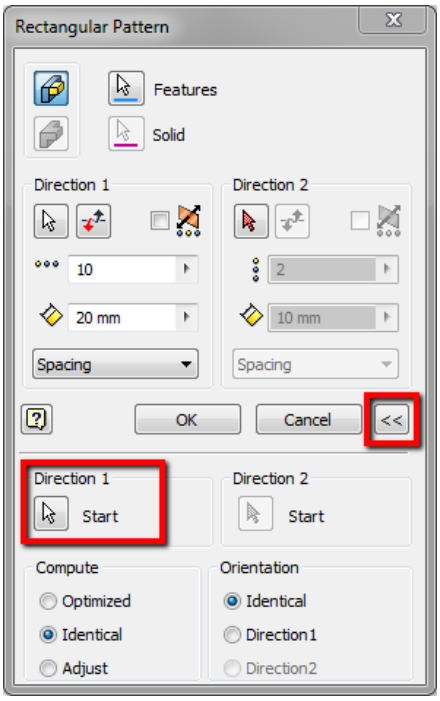
Now if we start our Rectangular Pattern, first pick the feature to copy. Then pick the sweep path used (it is safest to pick this from the graphics window and make sure the whole length of the path highlights).



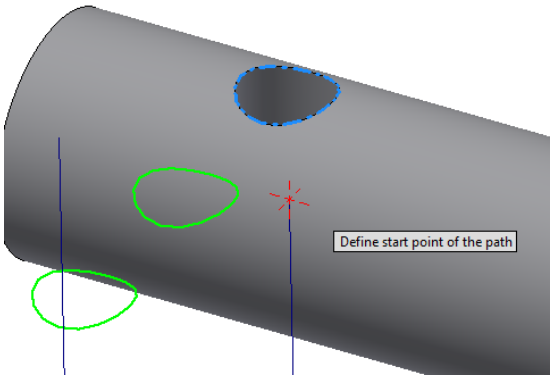
Then just up the number of instances and the spacing to see the effect. Do not be worried if it looks wrong like below.



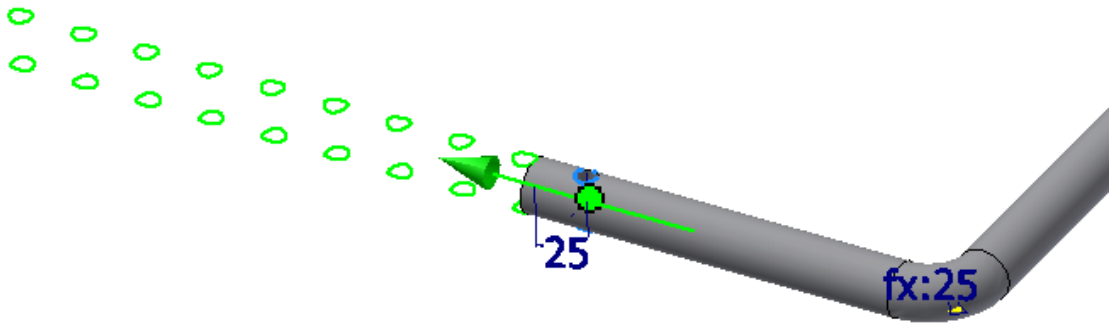
Currently our pattern needs a few more bits of information to correct it. If you expand the dialog, and select Direction 1 Start point.



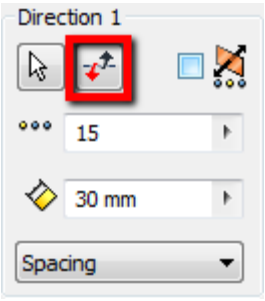
Then pick the point defining the location of your first feature.



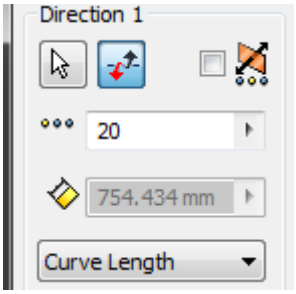
It then may either follow the path nicely or go off in the wrong direction.



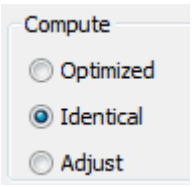
Just use the flip button next to Direction 1 to sort this.



Your pattern should now be coming together nicely. Sort out your spacing and number of instances correctly. Note that you have the option to pick curve length and it will work out the spacing for you.



Now the final bit to decide is the other options on the expanded dialog.



Compute defines how the new features are calculated.

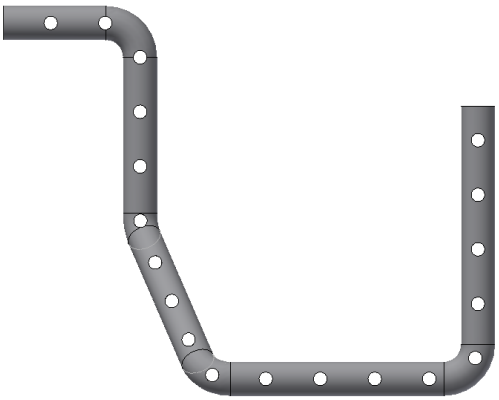
Optimized creates identical copies of selected features by patterning feature faces. Optimized is the fastest compute method. Limitations are the inability to create overlapping occurrences, or occurrences that intersect different faces than the faces of the original features. When possible, speeds up the pattern compute. So in this case I cannot use it, optimized really only works for simple features on flat faces.

Identical Creates identical copies of selected features by replicating the results of original features. When the optimized method is not possible, Use for identical features. In this case if I use this I get the following result. Again as this keeps everything the same it doesn't work items that require the feature to change for each instance.

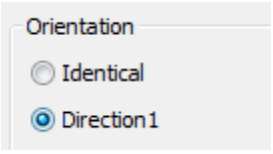
Adjust Creates potentially differing copies of selected features by patterning features and calculating extents or terminations of each pattern occurrence individually. Computation time is lengthy for patterns with large numbers of occurrences. Preserves design intent by allowing pattern occurrences to adjust based upon feature extent or termination conditions, such as a feature that terminates on a model face.

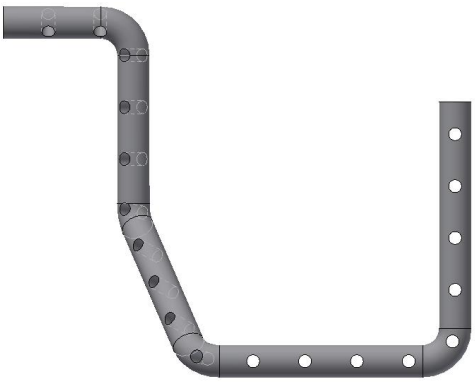
This gives me the result I want.

But note that all the holes are still vertical.



If I want them to be all pointing in a different direction (often you want them to stay normal to the face they are on). To achieve that change the orientation to match Direction 1.

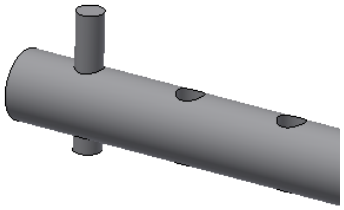




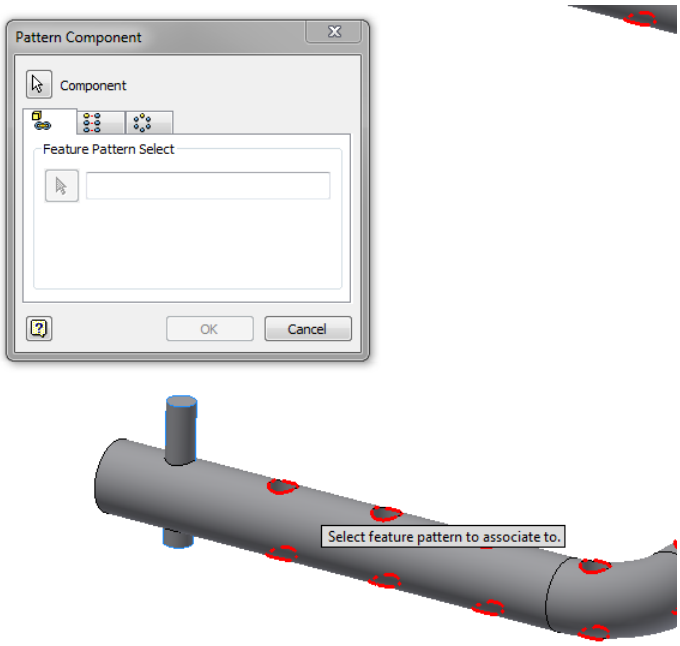
Now it is probably quite likely that you will be placing this part into an assembly if it isn't already in one. It is also likely that the same part fits into each hole.

Well there's a tool to achieve that quickly and easily so that it always updates and matches the pattern above.

In your assembly make sure you fully constrain 1 instance to the start instance of your part pattern feature.



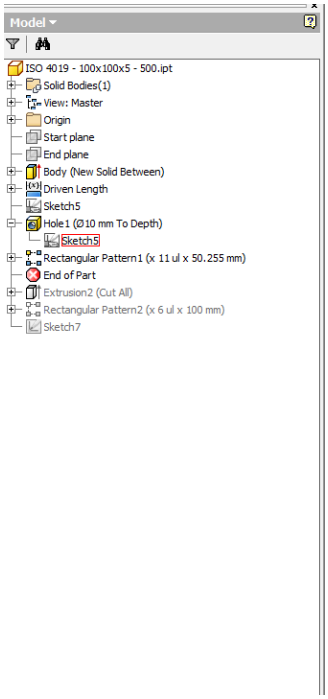
Then use the assembly pattern command. Pick your first instance and use the feature command to select the patterned holes. Voila.



Please see GIF on Cadline Community

The same above process works on most shaped parts and with most features as long as you have a sketch path to pattern along. Here is an example of a ribbed conduit that is also changing cross section from circular to elliptical and some steel box section with holes.

In all cases make sure you have a path to pattern along and also you define the direction, start point and compute and orientation options.


Model
ISO 4019 - 100x100x5 - 500.ipt
Solid Bodies(1)
View: Master
Origin
Start plane
End plane
Body (New Solid Between)
Driven Length
Sketch5
Hole1 (Ø10 mm To Depth)
Sketch5
Rectangular Pattern1 (x 11 ul x 50.255 mm)
End of Part
Extrusion2 (Cut All)
Rectangular Pattern2 (x 6 ul x 100 mm)
Sketch7