

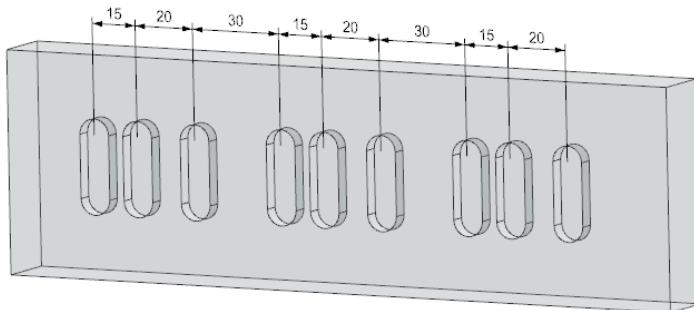
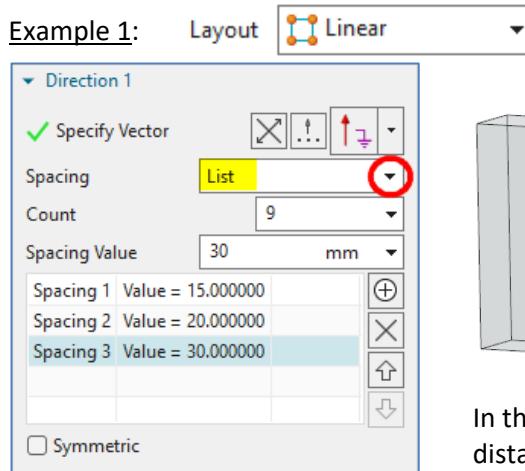
NX Pattern

The geometric pattern commands (pattern feature, pattern surface, pattern geometry) in NX offer a wide range of options. In this NX Smart Tip, I will take a closer look at some of the less common options.

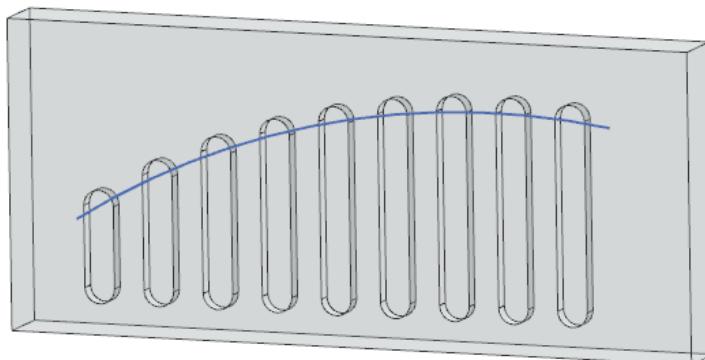
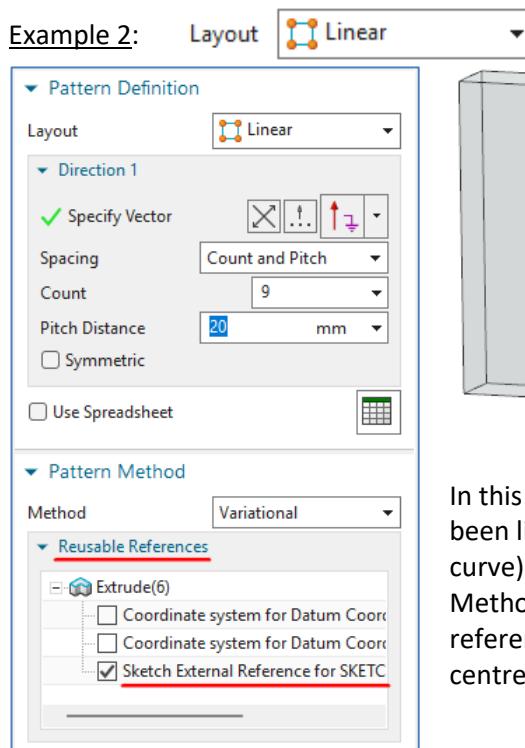


In our examples, we use the command 'Pattern Feature'.

When selecting the layout options 'Linear', 'Circular' or 'Along', the option 'List' is available in the "Direction" section of the 'Spacing' parameter list. With this option, the spacing is defined using a list of values that is repeated until the desired number of elements is reached.



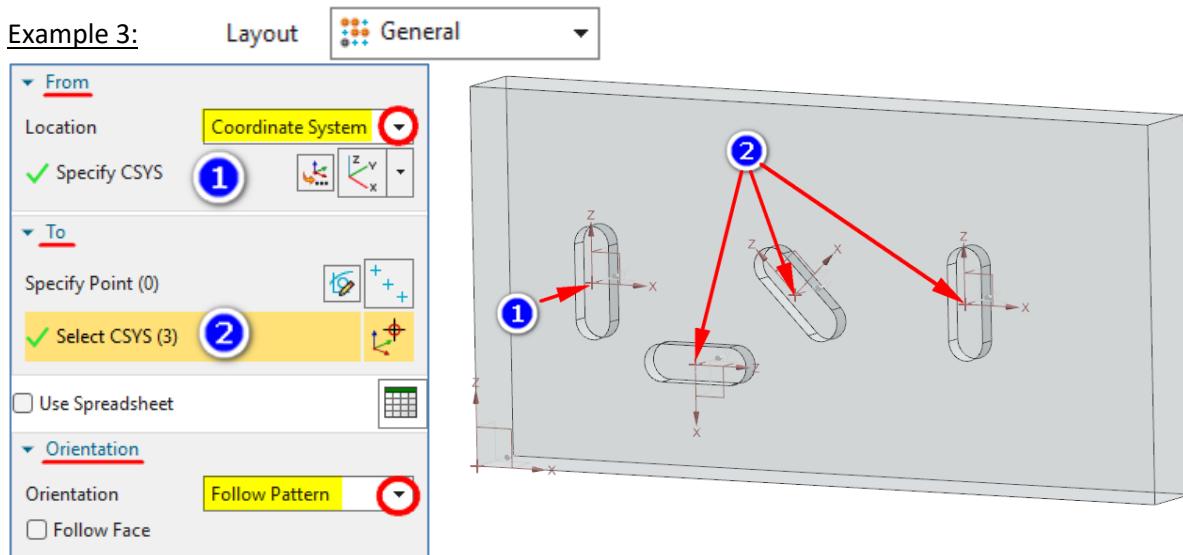
In this example, a total of 9 instances are created and the 3 distances in the list (15, 20 and 30) are repeated 3 times.



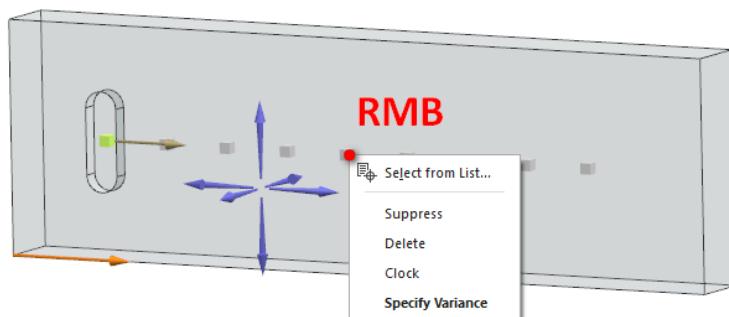
In this example, the upper arc centre of the slotted hole has been linked to the spatial curve within the sketch (point on curve). The reusable references are listed in the 'Pattern Method' section. In this case, the bottom entry, the sketch reference curve, is activated. This activation causes the centre point of the arc to follow the curve during patterning.

In example number 3, the slotted hole is patterned as a 'general' pattern using previously defined and aligned coordinate systems (KSYS). The origin axis system (1) is specified in the 'From' section.

Several coordinate systems (2) can be specified as targets in the “To” section. The spatial alignment of the slotted hole is based on the respective axis system, as the ‘Follow pattern’ orientation has been set in this example.



Are you familiar with the options available in the right-click menu in edit mode? When the blocks are displayed, you can use the right mouse button to modify this specific instance in the pattern, i.e. suppress, delete, move or even set parameters separately.



I would be delighted if this NX Smart tip was useful to you. Do you have any questions?
Just get in touch with me.

Your NX trainer, consultant and service provider

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