

Control Formfeature variable

You want to design a component with variable dimensions and hide individual feature elements depending on the situation. In this NX Smart Tip, I'll demonstrate how to do this using a perforated plate. The external dimensions are variable. If the length is less than 100 mm, no holes are created. If the length is 100 mm or greater, a hole is created in the centre of the plate every 50 mm. The width of the plate depends on the length and can be 50 mm or 100 mm.

Step 1:

Create expressions

Command:

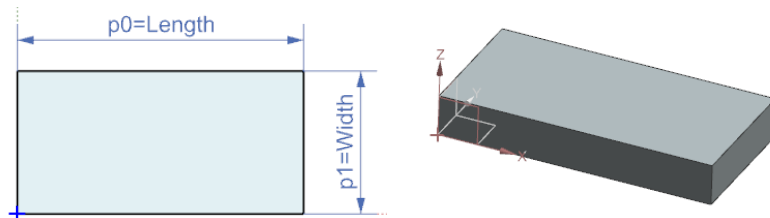
Tools + Expressions...

Please pay particular attention to the "Hole" variable.

	Name	Formula	Value	Units	Dimensionality	Type
1	Default Group					
2				mm	Length	Number
3	Height	15	15	mm	Length	Number
4	Hole	if(Length<100)(0)else(1)	1		Unitless	Number
5	Length	100	100	mm	Length	Number
6	Number	floor(Length/50)	2	mm	Length	Number

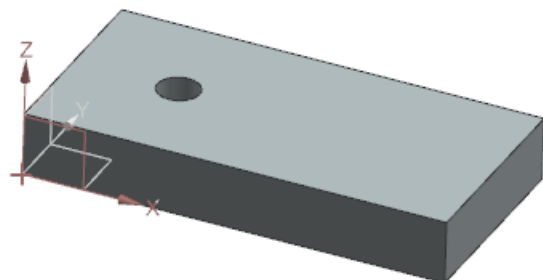
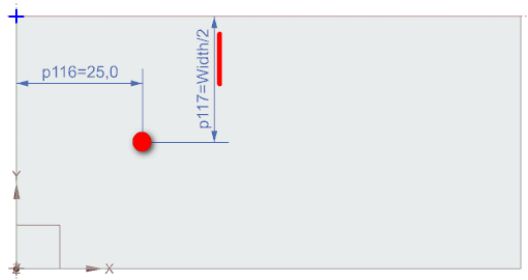
Step 2:

Create a rectangle in a sketch using control dimensions and variable Length and Width.

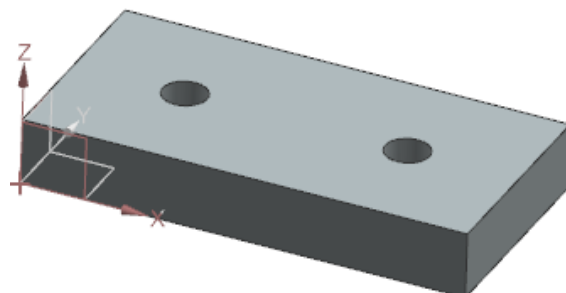
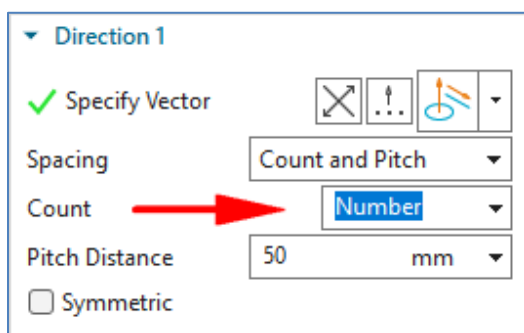


Extrude this sketch and create the block shown in the image.

Step 3: Drill the first hole (D=10 mm)



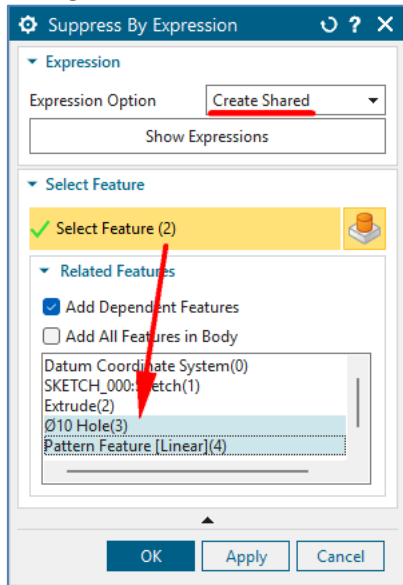
Step 4: Duplicate the hole using 'Pattern feature'



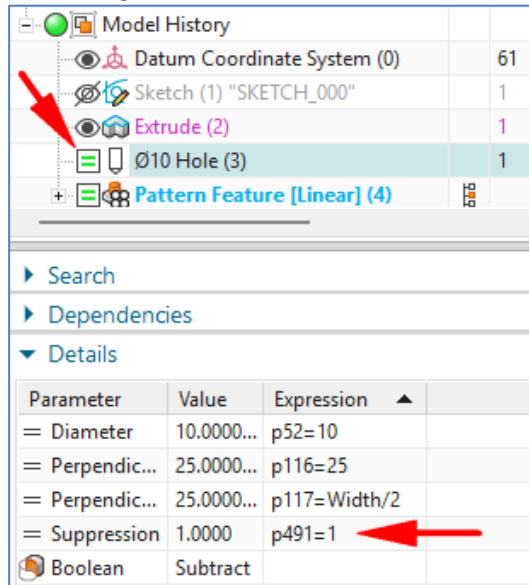
Step 5: Hide the holes if the length is less than 100 mm

Menü + Edit + Feature + Suppress by Expression

Dialog window:



Part-Navigator:

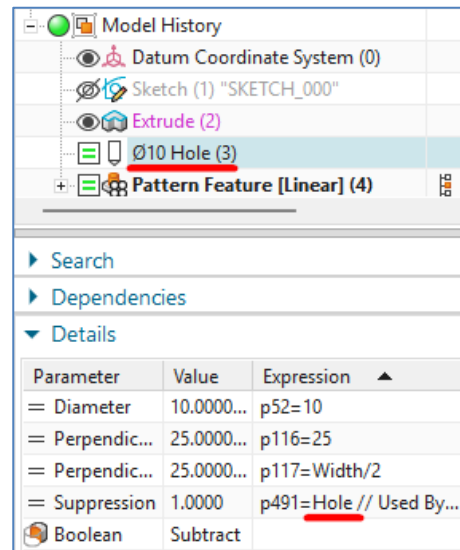


Step 6: Control the suppression via the user variable "Hole".

Select the hole in the Parts Navigator. Under 'Details', there should now be a variable called 'Suppression' with a value of 1 (TRUE) or 0 (FALSE). Double-click to select this value and replace it with the variable 'Hole'. The suppression setting is now controlled via the user variable.

Test:

If the length specified is less than 100 mm, the holes are now automatically omitted.



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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