\*\*How to speed up your modelling process\*\*

This how to is for modelling in Solidworks and will provide you with a few handy shortcuts.

First of all let’s talk about sketches. To speed up your sketches you need to know about the quick access wheel. When you hold your right mouse button this wheel will appear (see pic below). As standard this wheel will contain smart dimensions (on top), line (to the left), circle (to the right) and square (on the bottom). Your can change these in **Tools -> customize -> mouse gestures**. This will greatly lower you mouse movement and therefor increase your speed and you ability to model for langer periods of time.

Afbeelding met schermopname, tekst, software, Multimediasoftware

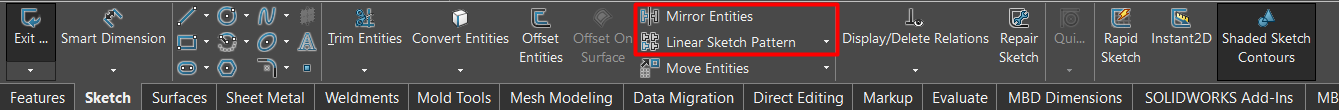
Automatisch gegenereerde beschrijving

Another quick access is if you have drawn a line and you want your the next line curved, you can do this. Pick line (if not active already) go to where you want your curve to start and start a line. Drag it out a bit but than go back on to the starting point of that line. Of done correctly you’ll have a curved line when you drag it back out. So click ones on a existing line to start drawing a line go back to the starting of that line and then draw the curve you want.

Afbeelding met tekst, diagram, schermopname, lijn

Automatisch gegenereerde beschrijving

Next let’s talk about symmetry. If a sketch is symmetrical you might want to use the **mirror entities** option. This will make so you’ll have less dimension in your sketches, so if you want to change something in you sketch you don’t have to change multiple dimensions. This also applies to reoccurring patterns. Like hole. If you have lots of hole in a line you could use **linear sketch pattern** in your sketches.



But symmetry is not always the case. Somethings you have a few dimensions that are the same, but somewhere else in a sketch. In this case use relation to “connect” these line to have the same length. To do this select both lines and look an your left of your screen. Here you’ll see a few options to make relation between the lines. Horizontal/vertical will make them parallel to each other and on one of the asses (x or y asses). Collinear will make them in line with each other. And equal will make them equal length. Multiple relation can be applied to the same pair of lines.

Parts

Now try to make as minimal features. Sometimes you can make the hole already in the sketch of the extrusion. So you don’t need to make a cut. This will make your model tree more organised.

Also a good shortcut is if you have multiple file open in Solidwork you can easily switch between them by using **ctrl + tab**. If you tap it you switch to your last opened one and if you hold it you will see al the open tabs and can select the one you want.

If you find it hard to get on the right angle or plane to make a feature you can make reference planes.

Afbeelding met Multimediasoftware, software, tekst, Grafische software

Automatisch gegenereerde beschrijving

When you are building a part you can sometimes need a already existing surface. In this case you can use the **convert entities** feature. When in a sketch select the surface you want sketch line of. Then go over to your toolbar and select covert entities. Now these line are connected to that surface so if it changes your sketch will change with it.

