Gap Filtering.

This section will explain the proper use of gap filtering. Firstly, what are gaps. Gaps are transitions between two parts of a toolpath. Say we are running a surface finish contour around 100 parts nested in a 12”X12” plate. Without being enabled, Mastercam sees all of the features as needing to be completed at the same time by height. That means that it will run one small cut on one feature, then do another small cut on the second feature and so on. This leads to a lot of wasted transitions and is represented by a bunch of yellow rapids around the part. If you enable optimize toolpath, it will find the shortest distance from cut to the next which usually is represented by less vertical retracts and more, shorter transitions in the program. It also takes longer for Mastercam to process but this option drastically changes the toolpath when enabled, so I would suggest to just leave it on by default unless you are working with a very detailed toolpath.

Now the different types of transitions.

There are only three that I have ever found a use for.

The first is the default, Broken. This option makes the machine stop, retract, rapid over, plunge, and then start machining every time the gap tolerance is exceeded. Most times, this is what we want.

The second is arc and line entry. This is exactly the same as broken except it adds an arc and line to the entry/exit of the toolpath, so the tool does not just plunge down to the finished height. This is useful for finishing parts where the surface must be free of dwell marks.

The last option is follow surface. This one creates a transition that follows the model. This is great for roughing, but can leave witness lines when used on a finishing pass. The transitions here are represented by a purple line, which is a feed at rapid code that will not create a dogleg rapid that could gouge the part.

Direct is basically the same as follow surface, except it moves without regard for the solid model. Why this is an option, I have no clue. It causes witness lines and gouges in most toolpaths so don’t use it unless you have a good reason, and you run the verify to look for gouges.