

# Program created

[Technology](#), [Development](#)



Next to be considered is the software needed. For a small company, where the equipment may be a stand alone PC most CAD/CAM's will run on an operating system of Windows 9x/ME or NT. The Windows 9x operating systems do, however, have significant security flaws. Therefore, whilst they would be fine for a stand alone PC if the PC were to be connected to a Company network or intranet then they would not be secure. The best option for a secure network would be Windows NT4 or higher. The speeds and feeds of the tools, Creating an Operations Sheet, Devising a tool list

Due to the fact that these three aspects are generally achieved hand in hand with each other they have been brought all under the same heading. Let us consider an operations sheet. The operations sheet, for a CNC programme, denotes a list of operation numbers, the tool used for those operations, their speeds and feeds and a brief description of what each operation does. The software can now be used to create the program. This is performed by first either creating the drawing within the Cad/Cam package or by importing the drawing from a suitable Cad package such as AutoCAD(r) or Catia(r). See Fig. 1.

The next step is to define tool paths that need to be cut. These vary from package to package. Some Cad/Cam packages deal specifically with tool paths whilst others create groups and curves (such as CamTech Peps(r)) and then generate their tool paths from these. See Fig. 2. Next, it is necessary to define the tools that are going to be used and then apply them to the tool paths or curves and state what process is required to operate. E. g. Pocket mill; finish profile; drill etc. When this stage is reached it is possible to view the cutting process on the PC to observe how the tools are going to run.

See Figs. 3-8. This provides a clear (assuming a quality Cad/Cam package is used) indication of any problems that may exist. It is related specifically to the component that is required to be created within this project. Any alterations may be achieved by entering the Cad/Cam's code window. An example of the code generated in Camtech Peps(r) is shown below with a description of what the code means. The program can now be " post processed". This involves, as described on page 1, placing the code generated in the code window through a language translator.

This translated code is now in the language form the CNC machine can now understand. It has just been " post processed". The translator is known as a post processor and each different type on CNC control system has its own post processor. Whilst this would, at first glance, appear cumbersome it does allow for great flexibility within any company that has a wide range of CNC machines with different controls. The code generated in the code window by the Cad/Cam is specific to the Cad/Cam package but generic to the job.

This generic code can be accessed and then post processed by many different post processors allowing the job to be passed to many production machines within a company. There are two examples of post-processed code listed here. One is in the ISO format that is often called FANUC(r) (as FANUC(r) closely follows the ISO standard). The other is in the language Heidenhain(r). This language is often used of CNC mills and machine centres due to its ease of reading and writing. It can almost always be found on CNC mills manufactured by Bridgeport(r) who have a partnership with Heidenhain(r).