An introduction to CAM Software.

Lex Lennings, May 2001.

1. Historical perspective of CAM.

CNC milling machines have of course been developed based on conventional milling machines, where the tool is moved through the material by operating a hand wheel for each of the available axes (X, Y, Z, ...). The basis of adding NC (Numerical Control) is very simple: replace the hand wheel by a positioning motor and add some electronics (the NC controller) to control the position. Nowadays many conversion kits are available to perform such operation.

The first NC controllers were very simple: no user-interface, just an option to read the tool-positions to go to from a punched tape. Very soon new controllers were introduced, including a simple computer with a special-purpose keyboard and a display. The operator now could enter the sequence of movements to make (the NC program) on the controller, which was called CNC (Computerized Numerical Control). In some cases a tape puncher / reader still was available for external storage.

Next step was of course to use a PC for external storage, communicating with the CNC controller via a serial cable. This allowed the operator to conveniently sit at his desk to enter a new NC program, while at the same time the CNC machine was executing a previous job. Do note that for most desktop machines the situation is a bit different, as here a PC is used for the CNC controller (much cheaper than a special purpose control computer).

In the situation just described the operator enters the complete NC program on the computer, using a plain editor like Notepad or a special purpose NC editor. Each movement has to be separately entered, like "G0X10Y20Z30" for "Machine in a straight line from the current position to position (10,20,30)". This is of course very labor intensive, and this is where CAM software comes in to 'automatically' generate NC program files.

2. The position of CAM Software in the process.
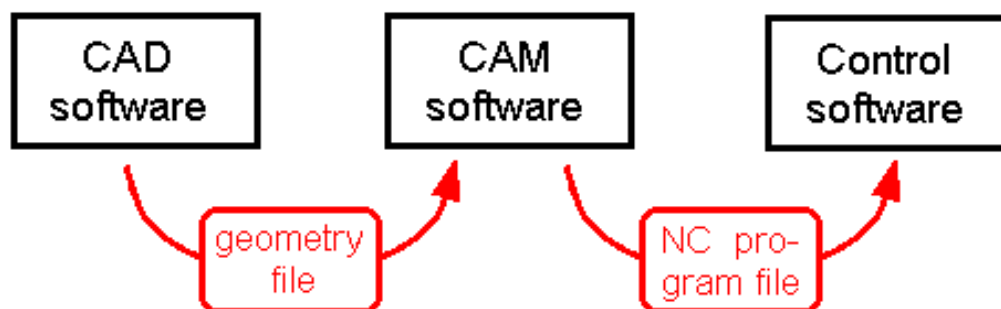
In order to manufacture a part, nowadays typically three different software programs are used (see the illustration above):
- First the **CAD software** to make the design of the part
- Next de **CAM software** to calculate the toolpaths based on the design, compensating for the cutter's geometry, adding feedrate and spindle commands, etc
- Third the **control software** to read the toolpaths and let the machine actually move along these paths.
This subdivision of tasks by three different programs is the same for both 2D and 3D applications.

The Control software comes with the milling machine, while in contrast the CAD and the CAM software have to be bought separately. In case an NC controller with built in special purpose computer is used, the Control software is integrated. In case of a PC the Control software has

to be installed, still it "belongs" to the machine as the machine cannot function without the Control software. Do note that many lowcost CNC machines are delivered with MS Dos based Control software, as the realtime control needed for machining is difficult to program under Windows.

Communication between the three programs is done using files. From CAD to CAM the design is transferred using a file format for geometry data exchange. For instance file types like IGES, STL and STEP for 3D, DXF for both 2D and 3D, and Postscript and HPGL for 2D applications. These are standard formats that in most (!) cases can be used without any special configuring needed. Much more can be said about these file types. Perhaps an idea for a next article of the month I won't add any explanations here.

Communication from CAD to Control software is done using NC program files, for which many formats do exist. In most cases the format will be a (minor) variation on the ISO / DIN G-code format. G-code is supposed to be a standard, however in practice each manufacturer chooses a bit different implementation. In other cases a proprietary format is used. So for this communication the CAM software has to fine tune its output in order to meet the requirements of the NC controller used. This fine tuning is done by the Postprocessor.

The **Postprocessor** is the part of the CAM software that translates the toolpath data into the correct file format when saving (in fact an export filter). This functionality is the same as used in the (Windows) device driver that comes with any printer, to translate the wordprocessor's output to the format required by that printer. In many current CAM systems the postprocessor can be configured by the user, making it easy to connect to any new machine. This in contrast to the older CAM packages where the user has to pay (much) money to the supplier to order a new postprocessor.

I some specific situations one of the three programs just described (CAD-CAM-Control) may be omitted. For instance some machines can be used without control software (like the small Roland models, where a plain "print" command is sufficient to start the machine). Or in some setups a plot file (or even a 2D DXF file) from the CAD system can be immediately sent to the control software, skipping the CAM step. Still it makes sense to distinguish the three basic steps, for a clear picture of the process involved.

3. Types of CAM software.

Many different CAM software packages are available, showing large differences in price. It is a very difficult job to get a clear view about each system's capabilities and it's price/performance ratio. The CAM software comparison table on this website is a great help, still only doing some real-life testing gives good evaluation information. To give some guidelines for choosing, two basic categorizations can be made.

A clear difference is present between CAM software for 2D and for 3D applications. With 2D is meant that the CAM system imports a 2D drawing file and calculates a toolpath with all movements taking place on a constant Z-level. Obviously several toolpaths on different Z-levels can be combined to create a 3D result, which is called 2.5 D machining (note that more definitions of the notion 2.5D do exist). In that case to user has to enter the correct Z-level to be used for each toolpath. A 3D CAM system in contrast imports a full 3D CAD model and calculates toolpaths to create a 3D result. Note that in this case also toolpaths on constant Z-level may be used (waterline machining), however these are automatically generated. Many CAM packages do offer both 2D and 3D, however still have their clearly recognizable foundation in one of both fields.

A second distinction is between simple and high-end CAM software. The high-end stuff is meant for professional toolmakers, who know about all possible milling parameters, want to be able to control any parameter for an optimum result, and are willing to pay for that. These extra high-end parameters do include options like
- support for a fourth axis, or for full 5 axis machining
- optimization for High Speed machining (constant tool load)
- special sequences for approaching and leaving the geometry (lead-ins)
- automatic stepover calculation
- a wide choice of machining strategies, like parallel, spiral, radial, pencil tracing, flat surface

recognition, offset machining, plunge milling and automatic smoothing of almost vertical surfaces.
- automatic detection and removal of rest material
- management of undercuts
- rendered machining simulations.
The more simple programs offer less options, and are thus both cheaper and easier to use.

4. A few more Remarks.

The word "NC Program" can be used in two senses, which can be quite confusing when communicating about CAM. In this article it is used for the 'File that contains all toolpath information', exported by the CAM system and imported by the Control software. As the word "Program" in most cases means Software, sometimes the word "NC program" is used for a 'Program to calculate NC data', so for the CAM software. Be aware of possible confusions.

The capabilities of your machine must of course match the requirements of the CAM software. One requirement should be checked, as some older CNC machines do not offer this: 3D line interpolation. This means the possibility to travel from point A to point B in a straight line in full 3D. This is not easy, as all three axes will have to keep up a different speed. Some machines are only capable of straight lines involving 2 axes (2D line interpolation).

A second important machine capability is on-line machining: the capability to handle large NC program files directly from the computer's hard disk. This involves some handshaking between the PC and the controller, as in most cases the data transfer will be faster than the actual machining. No big deal: any simple printer can do so. However be aware that many CNC machines cannot: they have been designed for NC programs that are completely entered by hand, so consider a 100 Kb NC program file as very large. For CAM this is not large: it is easy to create 10 Mb NC programs, or even larger. Those older machines require the complete NC program file to be transferred before the machining can start, limiting the file size to the (say) 256 Kb of available internal memory. The option of on-line machining is also called drip-feed, or DNC (for Direct Numerical Control).

I do hope to have offered some valuable insights in the world of CAM software. Please feel free to comment on this article (info@deskproto.com), and also to play around with the fully functional trail version of our own 3D CAM package DeskProto, that you can download free of charge.