Pro/DESKTOP® Tutorial
Introductory Level - CAM
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>INTRODUCTION</td>
<td>4</td>
</tr>
<tr>
<td>CAM IN INDUSTRY</td>
<td>4</td>
</tr>
<tr>
<td>Removing Material</td>
<td>7</td>
</tr>
<tr>
<td>Machining</td>
<td>7</td>
</tr>
<tr>
<td>3 Axis Machining</td>
<td>8</td>
</tr>
<tr>
<td>2½D vs 3D</td>
<td>8</td>
</tr>
<tr>
<td>Applications of 2½D</td>
<td>9</td>
</tr>
<tr>
<td>3D Machining</td>
<td>10</td>
</tr>
<tr>
<td>QUIZ</td>
<td>12</td>
</tr>
<tr>
<td>MODIFYING DESIGNS FOR PRODUCTION</td>
<td>13</td>
</tr>
<tr>
<td>Designing for Machining</td>
<td>14</td>
</tr>
<tr>
<td>Quality vs Speed</td>
<td>15</td>
</tr>
<tr>
<td>CAM IN THE CLASSROOM</td>
<td>17</td>
</tr>
<tr>
<td>Tools/Settings</td>
<td>17</td>
</tr>
<tr>
<td>Work Piece</td>
<td>17</td>
</tr>
<tr>
<td>MULTIPLE OBJECT MACHINING</td>
<td>19</td>
</tr>
<tr>
<td>Long Machining Times</td>
<td>20</td>
</tr>
<tr>
<td>FROM CAD TO CAM</td>
<td>21</td>
</tr>
<tr>
<td>EXPORTING AN STL FILE</td>
<td>22</td>
</tr>
<tr>
<td>POST PROCESSING</td>
<td>22</td>
</tr>
<tr>
<td>Post Processing Principles</td>
<td>22</td>
</tr>
<tr>
<td>Importing a 3D Model</td>
<td>23</td>
</tr>
<tr>
<td>Size of Model (scaling)</td>
<td>24</td>
</tr>
<tr>
<td>CALCULATE TOOL PATH</td>
<td>30</td>
</tr>
<tr>
<td>On Screen Simulation</td>
<td>30</td>
</tr>
<tr>
<td>Machining</td>
<td>31</td>
</tr>
<tr>
<td>QUIZ</td>
<td>32</td>
</tr>
<tr>
<td>POST PROCESSORS IN SCHOOL</td>
<td>33</td>
</tr>
<tr>
<td>Modela Player</td>
<td>33</td>
</tr>
<tr>
<td>Mill Wizard</td>
<td>33</td>
</tr>
</tbody>
</table>
INTRODUCTION

Computer aided manufacture (CAM) in schools includes a range of techniques including printing, embroidery, vinyl cutting, machining of resistant materials, etc. All of these are based on graphical images, most in two dimensions (2D).

This booklet concentrates on techniques for taking three dimensional (3D) designs from Pro/DESKTOP and machining them out of resistant materials. In order to place this work in context, reference will be made where appropriate to 2D techniques.

Designing is an important aspect of design and technology but equally vital is the capacity to manufacture. Computer Aided Manufacture (CAM) is the logical extension of CAD but requires expensive machines. The software that controls CAM machines usually comes as part of the package but link software is required to convert Pro/DESKTOP files into machine instructions.

CAM IN INDUSTRY

Stereo Lithography

Stereo Lithography is a term coined to describe a laser process for solidifying resin to form complex 3D components. It creates durable, highly accurate models but can take days to create large, detailed objects. The file format for rapid prototype models is known as STL and breaks the surface of an object into thousands of triangles.

Stereo Lithography machines are still the realms of large companies, often costing well over £100,000 each.

The STL file format however is now being used to transfer files to machining equipment.
LOM

Laminated object manufacture (LOM) is a well-established industrial process that slices a component into wafer thin slices. These are cut from sheet material, usually paper or card, and bonded together to form a copy of the computer model.

Models formed by this process are robust with properties similar to softwood or medium density fiberboard.

A school version of this process is now available. Boxford a UK based machine tool manufacturer has produced RapidPro software. This operates like a software wizard guiding the user through a series of screens. The slices are cut into sheets of self-adhesive cardboard or paper. A clever technique of guide pegs is used to align the slices when, with a bit of patience, they are glued together 're-assemble' the model.

**Fuse Deposition Modelling (FDM)**

This process is used extensively and produces dense, robust models that can be handled without fear of damage. Some plastics used in this process can be sterilized and used to prototype food products.

FDM machines build models from very fine strands of thermoplastic. The technique could be likened to using a hot glue gun to build up layers of plastic onto a surface, gradually creating a component. Because models are built in air the software adds 'scaffolding' to support overhangs.
Models created using this technique are robust enough to be used for testing purposes. Automobile companies routinely produce FDM models of air ducts and manifolds to check flow characteristics.

**3D Printers**

These machines use a technique where fine powder is laid down in layers with adhesive bonding the particles for each ‘slice’. Materials include cornstarch and plaster.

Early examples of these machines cost up to $250,000 but the price is now below $50,000. Predictions suggest a machine will be available below $5,000 within 5 years, a point where schools may find them affordable. Pottery manufacturers make extensive use of this process to create prototypes and molds directly in plaster. The ‘Hot House’, a design center near Stoke on Trent, England offers a range of CAD and CAM services to the ceramic industries including 3D printing.
Removing Material

Spark erosion

This technique is used to shape materials that are difficult to cut by mechanical methods. Titanium is one such material.

A thin wire acts as the anode and the titanium blank is the cathode. A high voltage supply is connected between the electrodes and an electrolyte liquid sprayed over the work piece.

The ‘cutting’ wire is gradually moved to create the required shape. For internal holes, like the one above in a suspension upright, the wire can be threaded through a pilot hole prior to starting the cut.

Machining

Machining components manually is a complex process even for experienced operators. Defining the instructions for a computer-controlled machine can also be very difficult.

Programming tool paths and machine instructions is a highly specialized job, needing well-qualified engineers with high-level ICT skills.
Computerized milling machines are often described as Computer Numerically Controlled (CNC). Machines need a set of instructions to determine the movement and speed of cutters and positioning motors.

Picture courtesy of: The Manufacturing Engineering Centre – University of Wales.

3 Axis Machining
To explain CNC machining this diagram shows the worktable and cutting head.

This shows the three axes of movement. The movement of each axis can be controlled independently.

2½D vs 3D
Early CNC machining in schools was confined to 2D cutting due to the complexity of converting 3D co-ordinates into machine instructions. These instructions consist of a standardized language of “G-Codes”. Simplified, a straight cut would be programmed as a start point, end point and depth of cut.
Wherever this instruction appears in the program the cutting tool moves downwards cutting into the material to the required depth and the x and/or y movement is made creating a slot in the material.

Most modern 3 axis machines come with software that will convert a 2D line drawing into G-code instructions removing the need for a knowledge of programming. The technique of converting a drawing into machine instructions is called “post-processing”.

Working from 2D drawings, a number of different depths can be determined, hence 2½ dimensional. The different depths are often identified by color of line.

**Applications of 2½D**

A common school project employing these 2½D techniques is a rolling ball maze game.

The base is machined from a 10mm thick block of acrylic with a 6mm diameter cutter at three depths of cut.
• 9mm deep for maze track (Red)
• 3mm deep for lid recess (Green)
• 10mm deep for the outside shape of the block (Blue)

Note: A lid is cut separately from 3mm clear acrylic.

Another example of 2½D might be creating a landscape model. How would you set about this? If you are familiar with the Loft feature in Pro/DESKTOP you could easily model a shape similar to this.

Machining in 2½D would look something like this, resembling the contour lines shown on topographical maps.

A small 2½D model of this would not take very long to machine, perhaps a few minutes.

What would the tool paths look like for each layer? Remember, at each layer you need to clear away all the material out to the edge of the rectangle.

3D Machining

Machining in full 3D would take far longer as most software uses Raster cutting techniques. Software analyses the model in vertical slices. This can be in the X or Y-axis. The illustration below shows raster cuts in the X-axis with the step-over between cuts increased to show the principle.
This shows more typical step-over spacing.

The above step-over would produce this quality of surface finish.
For a very high quality finish the step-over should be no more than 25% of the cutter diameter and cutting carried out in the X and Y-axes.

However, raster machining has a significant cost in terms of the time needed for cutting. The above model is likely to take well over an hour to machine.

**Industrial machining**
Many schools are now buying 3 axis machines. Some machine tool suppliers have an optional fourth axis on their products.
Engineering companies increasingly need four, five and even six axis equipment for highly specialized work.

**Quiz**

1. What other forms of CAM are there open to schools other than using resistant materials?
2. Name and describe how one form of rapid prototyping operates.
3. What letters are used to identify the three axes of a CNC milling machine?
4. Which direction of movement does each letter identify?
5. Explain the difference between 2½D and 3D machining.
6. Sketch a component that lends itself to 2½D machining.
7. Sketch a component that lends itself to 3D machining.
8. Describe raster machining in the X-axis.
9. What factors determines the surface finish of raster machining?
10. Give one advantage and one disadvantage of four/five axis machining.
MODIFYING DESIGNS FOR PRODUCTION

As a design concept your model may be fine. If however you were designing for production particularly in quantity you could not finalize the shape without considering the limitations of the production process.

**Machining**

Limitations of 3 axis machining include the diameter, shape and length of cutter.

Machining every component would normally be impractical, unless perhaps each customer wanted a unique design. How much would each one cost? At this stage you are unlikely to have the information you would need to work this out.

For example, would potential purchasers of humble fridge magnets pay a very high price? Can you think of small machineable products people will pay very high prices for?

**Injection molding**

A metal mold shaped like the object has molten plastic forced in under pressure.

Limitations include corners where the plastic may not flow before setting.

Illustration: [TEP injection molding kit](#)

The entry and exit holes for the plastic may need to be re-positioned to make the plastic flow more smoothly and sharp corners may need to be rounded to prevent voids.

Pro/DESKTOP can be used to design the mold ‘in context’ so that clamping bolts and entry/exit holes are visible throughout the design process.

Molds for commercial machines are very expensive. This must be taken into account when setting the price for individual parts.
**Vacuum forming**

A thin sheet of plastic is heated until soft and ‘sucked’ onto a former by vacuum.

Limitations include sharp external corners cutting the plastic and webs forming around corners.

*Formech* and *C R Clark* make vacuum forming machines for use in schools.

To avoid cuts, sharp edges on the mold should be rounded.

Reducing the height of the shape would help prevent webs forming and some plastics such as styrene are less prone to forming webs.

**Designing for Machining**

When using Pro/DESKTOP to design products for machining it is important the limitations of the machining process are taken into account. There would be little point designing fine detail if the finishing tool is a 3mm- diameter ball nosed cutter.
**Quality vs Speed**

Another limitation of Computer Aided Manufacture (CAM) is the time machines take to cut out a 3D shape. As we have discussed, a high quality finish requires very small step-over and subsequently long machining times.

The telephone shape in the example below is approximately 60mm long and 30mm wide. Maximum depth was approximately 6mm. The total machining time was 3 hours 45 minutes!

The finish achieved was excellent and making the block size in Pro/DESKTOP almost exactly the same as the mould could have reduced the cutting time by as much as 50%.

Very fine detail can only be achieved with small diameter cutters but a 1mm cutter with a 25% tool diameter stepover will have to make over 400 profile cuts to finish an x axis raster cut. A 3mm tool with 25% stepover will need 132 passes across the workpiece. The detail possible with the 3mm cutter will however be considerably less.

<table>
<thead>
<tr>
<th>Ball nose cutter</th>
<th>Stepover (% of tool Ø)</th>
<th>Length of model</th>
<th>Number of passes</th>
<th>Time to machine</th>
</tr>
</thead>
<tbody>
<tr>
<td>3mm Ø</td>
<td>25%</td>
<td>100mm</td>
<td>136</td>
<td>1h 15m</td>
</tr>
<tr>
<td>1mm Ø</td>
<td>25%</td>
<td>100mm</td>
<td>404</td>
<td>3h 45m</td>
</tr>
</tbody>
</table>

The x, y, and z movements of the 3-axis machine will also restrict the overall size of model. Typically a 3-axis machine for schools would have a working area (in mm) in the region of:

\[
\begin{array}{ccc}
X & Y & Z \\
300 & 200 & 100 \\
\end{array}
\]

In practice the need to clamp material will reduce the X and/or Y working area.
The length of the tool often limits the useful Z movement. A 100mm long 10mm ball nose cutter would probably be acceptable whereas a 3mm ball nose tool 100mm long would break.

10mm diameter tool acceptable for roughing.

3mm diameter tool vulnerable to breakage due to excessive length

It is no accident that the cutting flutes of a 3mm ball nose cutter are only 10-15mm long.

This problem is the limiting factor for surface finish when machining deep features.

In this situation, if surface finish were important, industry would probably employ a four/five axis machine to allow a standard length 3mm ball nose cutter to finish the steep sides of the component.
Managing CAM effectively presents a number of problems particularly when class projects are involved.

**Tools/Settings**

Default settings for speeds and feed-rates are usually very conservative. New users of CNC would be advised to use the defaults while they are learning and let experience suggest when changing the default settings will increase productivity without damage to work piece or tooling.

**Path Spacing (Stepover)**

As a general rule the smaller the path spacing the longer machining will take but the finer the finish. Draft cuts will usually have a 50% path spacing (step-over). Fine finish is usually 20% of cutter diameter for the step-over.

**Speeds and feeds**

The materials being cut and the power of the machine govern speeds and feeds. Settings for most machines are very conservative and only experience will determine optimum settings for a particular machine. Extreme care must be exercised when increasing speeds and feeds, particularly with light duty machines.

**Work Piece**

**Size**

It goes without saying that models should be no bigger than the maximum size of the machine. Larger models can be scaled but where size is important Pro/DESKTOP can be used to chop the model into slices.

Machine each slice separately and glue them together to recreate the model to the required size. This technique is particularly useful where the tooling cannot accommodate the Z depth in one go.
**Reduce border to a minimum**

When raster machining molds, leave the smallest margin possible around the mold cavity. Unnecessary area will increase the machining time.

Large flat top surface will increase machining time

Minimum top surface significantly reduces machining time

**Fixing**

Double-sided tape is the most common method used in schools.

- Workpiece
- Double sided tape
- Sacrificial sheet
- Double sided tape
- Machine table
Where multiple identical objects are being machined one after another, consider fixing fences to the base as a simple jig. Each block can then be aligned accurately against them before fixing down. This will cut down on set-up time.

Some objects lend themselves to bolting down. A good example of this would be moulds for the TEP injection-molding kit. Use plastic bolts like the ones used on automobile number plates. They are less likely to damage cutters if a collision with the cutting tool occurs.

**MULTIPLE OBJECT MACHINING**

Set-up time is non-productive. Here are a number of strategies to ameliorate the time taken to set-up before and dismantle after machining.

**Assemble several designs**

Wherever possible fill the machining area with models to cut down on setting up time. Several small designs from students can be assembled in Pro/DESKTOP to be machined in one go.
Alternatively, duplicates of a single model are another way to maximize the use of a CNC machine.

A game of two halves

Where two halves need to be machined as with the mobile phone example earlier, create a Pro/DESKTOP file with two copies of the same object, one being inverted. A front and a back can then be machined in one go.

Long Machining Times

Some machining cycles can take a very long time. To minimize sound disruption in lessons consider the position of a CNC machine very carefully. Noise that seems no problem at first can be very annoying after many minutes or even hours for complex designs!

Schools are increasingly setting their CNC machine to run overnight or even across a weekend.

Companies use their expensive CNC machinery 24 hours a day. There is no return on capital investment while a machine is unused. Many modern factories are also fully automated. A further advantage of CNC machines (and robots) is they don’t get tired or irritable!
Fact or Fiction?

There is a story about fully automated hi-tech factory where the company employs just one security guard and a dog.

The man is there to feed the dog.

The dog is there to bite the man if he tries to touch any of the machines!

FROM CAD TO CAM

Computer aided manufacture used to be a lengthy complex process requiring in-depth knowledge of a machine control language that was based on G-codes.

Specialist software now makes this conversion very easy. In outline there are three stages involved:

1. Export the Pro/DESKTOP model as an STL file
2. Post processing the STL file into machine instructions
3. Use the machine instructions to machine your model.

The Boat hull from one of the other Pro/DESKTOP tutorials will be used to show the complete process from start finish.
**EXPORTING AN STL FILE**

The model must now be exported from Pro/DESKTOP as a Stereo Lithography file (.stl).

- With the model **Dinghy Hull.des** open in Pro/DESKTOP
- Open the **File...** pull-down menu
- Click on **Export**
- Select **Stereo Lithography file...**
- Note which folder the file is being saved in and give the file an appropriate name.
- Click on **OK**

The next step is to open the STL file in your Post processor software.

**POST PROCESSING**

**Post Processing Principles**

Post processing involves converting a 2D CAD drawing or 3D model into a set of machine instructions (G-codes) that will control a milling machine or lathe.

The following instructions have been written to describe a generic procedure for creating a CNC file. It does not refer to any particular suppliers software.

Post processing involves making decisions about how a 3D model will be machined. They fall into the following broad headings although some software may deal with them in a slightly different order.

1. Import 3D model (STL).
2. Direction (Z axis).
3. Size of model (scaling).
4. Depth of cut.
5. Material to be machined.
6. Tooling.
7. Roughing/finish cuts.
8. Calculate Tool path.
10. Output CNC machine instructions.
The machine instructions are then used control the 3-axis machine when cutting your design in a resistant material.

Note: Depending on the post processing software and machine tool you are using, the steps above may occur in a different order and the terminology may vary slightly.

**Importing a 3D Model**

The most common file format for transferring 3D models from CAD programs to post processors is Stereo Lithography format (STL). This was created for early rapid prototyping machines that used lasers to cure liquid resin creating complex solid models directly from CAD data. STL has become the de-facto industry standard for transferring 3D CAM files.

Most post-processing software accepts STL files and this is the format you should use from Pro/DESKTOP in STL:

- With the post processor software open.
- Open or Import the STL file (previously exported from Pro/DESKTOP).

The example used here is **Dinghy Hull.stl**. You could use this to try out these instructions with your own post-processor.

A 3D view of your model will appear as a wire frame or an STL surface view.

![Wireframe](image1.png) ![STL surfaces](image2.png)

An STL file stores the 3D shape with every surface made up of triangles. Rectangular flat surfaces will need only two triangles whereas tightly curved surfaces will need hundreds of triangles. That is why the front of the hull in example above is so dark.
**Direction (Z axis)**

This option allows models with detail on front and back to be cut twice from different directions. The two parts can then be glued together to form the final model.

Most software provides four or more directions for machining.

- Select the direction for machining

**Size of Model (scaling)**

Large models can be too big to be machined in one go. The model could be broken into smaller, full-size bits, machined and re-assembled to produce a component of the original designed size.

However, there are occasions when the machining time to create a full-size component is not necessary and a scale model will serve the purpose. A smaller, scale model can be machined in a fraction of the time for a full-scale component.

At the simplest level, scaling can be determined as a percentage of the original model. 50% will create a half-scale component whereas 200% a model twice full size.

Some software provides the option of setting the scale from a linear distance, usually the X, Y or Z-axis.
This is particularly useful where a model is to be scaled to suit an existing component.

Here a picture frame has been rescaled to suit a photograph.

• Select an appropriate scale for your model.

**Depth of cut**

It is important to be able to set the depth of cut. For example where only one surface of a model will be produced on CAM.

In a previous example of a two part machined mobile telephone, the depth of cut would probably be chosen to coincide with the ‘shut line’ or ‘join’ in the plastic casing for the phone.

• Set the appropriate depth of cut for your design.
Material to be machined

Most software will have default values for the optimum linear cutting for a wide variety of materials.

The optimum linear cutting speed relates to the highest speed a shaper tool can be moved.

Factors affecting the linear cutting speed include the need to produce a good finish, machine tool power, time between sharpening/bit replacement and minimizing the risk of tool breakage or damage.

Choosing the correct material is important, as it will influence the rotational speed of the cutting tool and the feed rates for the three axes. These will be worked out in subsequent steps.

• Select the material you will be machining

Tooling

The range of cutters for machining is extensive. CNC machines cutting metals commonly use High Speed Steel (HSS) slot drills or end mills. End mills are preferred for CNC work because they will cut in the Z-axis when entering the material. This removes the need for tool changing to create pilot holes.

As well as conventional end mills ball nosed cutters are very useful for roughing and finishing. The natural fillet ball nosed cutters leave in internal corners can be used to advantage.
Routers

Routers operating with timbers use solid carbide or carbide tipped tools at high speed for longer life. Specialist work can also make use of specialist tool bits such as Ogee, Ovolo or Veining profiles. Check with your machine tool supplier for the full range.

Tool changing

CNC machines vary in the way tools are held and changed.

The simplest system holds the cutting tool in a collet chuck. After every tool change the Z “tool offset” for the new tool must be set either through “touching on” to the material or using jigs to set tool overhang.

Manual quick-change chucks are extremely useful. Before machining, a number of tools are fitted into special collets and the offsets for each tool set in the software.

During machining when a tool change is necessary, machining will stop, the tool will move clear of the work piece and the computer will prompt for the appropriate tool.

Note: Software with older machines may not support tool changes. A workaround for this would involve exporting a Pro/DESKTOP model and post-processing it at each stage requiring a different tool. Providing tool offsets are correctly set and the material is not moved between machining each separate CNC program, a successful outcome should result.

Roughing/finish cuts

Machining is usually carried out in two distinct stages, roughing and finishing.
**Roughing**

This usually involves a large diameter tool to remove the majority of waste material quickly. Surface finish is not an issue so high speeds and feeds can be employed. The step-over can be up to 50% of the cutter diameter, reducing the number of ‘passes’ to a minimum.

This will help reduce the machining time. In industry this is particularly important where capital investment in machinery and human costs are significant elements in the final component cost.

**Step down (Z axis)**

Removing deep amounts of material may require more than one cutting pass at successively deeper cuts.
Specify the roughing tool parameters. These may include:

1. Diameter (mm)
2. End shape (Square/ball nose)
3. Rotational speed (RPM)
   (The software may calculate this for you from the material specified previously, cutter diameter and X, Y feed rate).
4. X and Y-axis feed rate (mm/sec)
   (See comment for the previous point).
5. X, Y stepover (% tool diameter)
6. Z axis feed rate (mm/sec)
7. Z axis step down (mm)

**Finishing**

A small diameter cutter is used for finishing to reach into corners wherever fine detail is required.

The step-over for small diameter cutters when finishing should be 20% or less.

Ball nosed cutters are often used because they do not leave such sharp edges on faces that slope in the Z-axis.

![Slot drill finishing](image1)

![Ball nosed finishing](image2)

Many of the parameters set for finishing will be similar to roughing. Exceptions include step-over already discussed and Z-axis step down.
Z-axis step down does not apply to finishing as the last layer is cut in one pass. Instead you may be asked to specify an uncut margin.

Roughing will leave material represented by the gray area all over the surface of the model to be removed in finishing.

- Specify smoothing tool parameters. These may include:
  1. Diameter (mm)
  2. End shape (Square/ball nose)
  3. Rotational speed (RPM)
     (The software may calculate this for you from the material specified previously, cutter diameter and X, Y feed rate).
  4. X and Y-axis feed rate (mm/sec)
     (See comment for the previous point).
  5. X, Y step over (% tool diameter)
  6. Z axis feed rate (mm/sec)
  7. Uncut margin (fractional mm, e.g. 0.2mm)

**CALCULATE TOOL PATH**

This is the final stage, creating the co-ordinate information and paths for the tip of the tool to follow. You may not be aware of this stage other than the computer stopping to do some heavyweight calculations. An egg timer on screen may be the only outward sign this stage is being carried out.

**On Screen Simulation**

The post-processing software now knows how the object will be machined. From this it can create an accurate ‘picture’ of what the finished surface will look like.
Not only that, most software will show a real-time or speeded up version of the cutting taking place. Some will even work out how long machining will take.

### Machining

You will need to ensure the post processing software knows the make and model of the machine being used and any additional facilities such as auto tool change.

You may also need to specify other parameters so check with your machine software.

The example on the right is the first few lines taken from a G-code program to machine the dinghy hull:

```
T1M6
G025.000
G0X0.000Y0.000S2600M3
G0X0.001Y0.001Z5.000
G1Z-26.446F200.0
G1X0.101F300.0
X105.999
Y0.100
Y1.491
X105.899
X63.600
X63.400Z-26.418
X61.100Z-25.902
X60.700Z-25.863
X58.300Z-25.848
X56.300Z-25.848
X55.200Z...........
```

The complete CNC file is 5738 lines long. Imagine trying to program every line of that manually!
1. Describe one limitation of 3 axis machining
2. Give one example of a subsequent manufacturing process a CNC component may be used for.
3. How might the machining time be reduced for a given component?
4. What, if any, would be the down side to this change?
5. What modifications might be necessary to make a model suitable for vacuum forming?
6. What problems do you foresee if a very long small diameter tool is used?
7. How would you cope with machining a model that is too tall for the tool length/Z capacity of your CNC mill?
8. Describe two methods of fixing the work piece to the CNC table.
9. What is the name of the process that converts a drawing or 3D model into a set of machine instructions?
10. What is the export file format used to transfer Pro/DESKTOP designs into the next stage of manufacturing?
POST Processors in School

There are two easy to use post processors available to schools, Roland's Modela Player and DelCAM's Mill Wizard. Both pieces of software operate in a very similar way. A wizard format asks the user to make a series of choices leading to a set of machine instructions to cut out the Pro/DESKTOP design.

Modela Player
This is software provided free with Roland 3 axis machines including the MDX and CAMM2 models. It accepts files exported from Pro/DESKTOP and controls Roland machines directly through the printer or serial port from the computer.

Mill Wizard
Mill Wizard (known as MiniCAM in the UK) is software from a company called DelCAM. It works in a similar way to Modela but does not control machines directly. Instead it saves a set of machine instructions in a range of formats suitable for a wide range of machines likely to be used in schools.

The file produced by Mill Wizard is loaded into the CAM machine control software in order to execute the manufacture process.