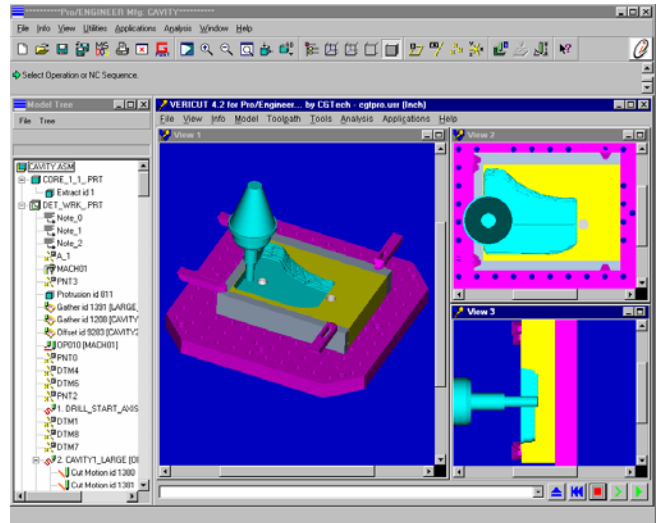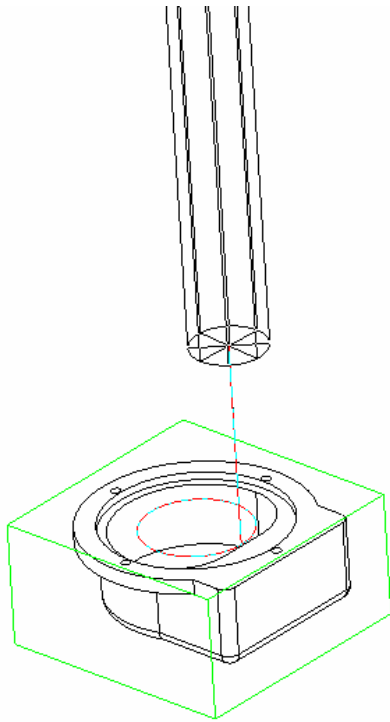# Introduction to CAM
(Computer Aided Manufacturing).


```
$$*        Pro/CLfile  Version 2001 - 2001320
$$-> MFGNO / POMPHUIS
$$-> FEATNO / 41
MACHIN / UNCX01, 1                                          Cutter location file
UNITS / MM
LOADTL / 1,  OSETNO,  1
SPINDL / RPM, 2345.000000,CLW
RAPID
GOTO / 14.5000000000, -0.0000000000, 50.0000000000
```

```
                                                                    %*
                                                                  O231*
                                    N0010G00G28G55G40G91X0.Y0.Z0.M19*
                                                              N0015G99*
                                                          N0020 T1 M06*
                                                   N0025 G00 X0. Y0. Z30.*
        ISO-programme file                                N0030 S2345 M03*
                                                   N0035 G00 G43 Z50. H01*
                                                          N0040 X14.5 Y0.*
                                                              N0045 Z10.*
                                                      N0050 G01 Z-5. F234.*
```

1. Goals for this introduction CAM:

1.1 Study the structure for the generation of a tool path by means of a CAM-software module.

1.2 To understand the technological jargon of a CAM-module.

1.3 Understand the importance of work preparation

1.4 Understand and use a postprocessor.

2. CAM: Computer Aided manufacturing.

An important aspect of a computer aided design and production environment is the integration of design and manufacturing (machining).

This is where CAM-software comes it.

This software provides ways of generating programmes for machine tools starting from (3D)CAD-data. Information data flow in a CAM-software can be found in figure 1.

Such a CAM-software can be a standalone application, where geometrical information is inputted in a standard protocol (DXF, STEP, IGES…).

It can also be an integrated part of the design (CAD) software.

Either form is equal for the output to the CNC-machine (Computerized Numerical Controlled machine tool), only organisation of data is more or less integrated.

For this introduction we use the integrated Pro Manufacturing-CAM-software in Pro Engineer Wildfire.

Every commercial CAM-software uses the same structure in 3 steps:

Input of geometrical and technological data.

-   Geometrical describe what has to be produced
-   Technological data describe how to produce the part

Calculation of the tool path and creation of the tool paths.

These tool paths then can be edited and simulated to result in the most efficient and economic machining.

Translation of the tool paths to a suitable ISO-protocol for the machine tool. This step is called "post processing". This is a translation done by independent software outside of Pro Engineer. It is a necessary step, since each machine tool uses its own interpretation of the standard ISO-DIN 66025 code.

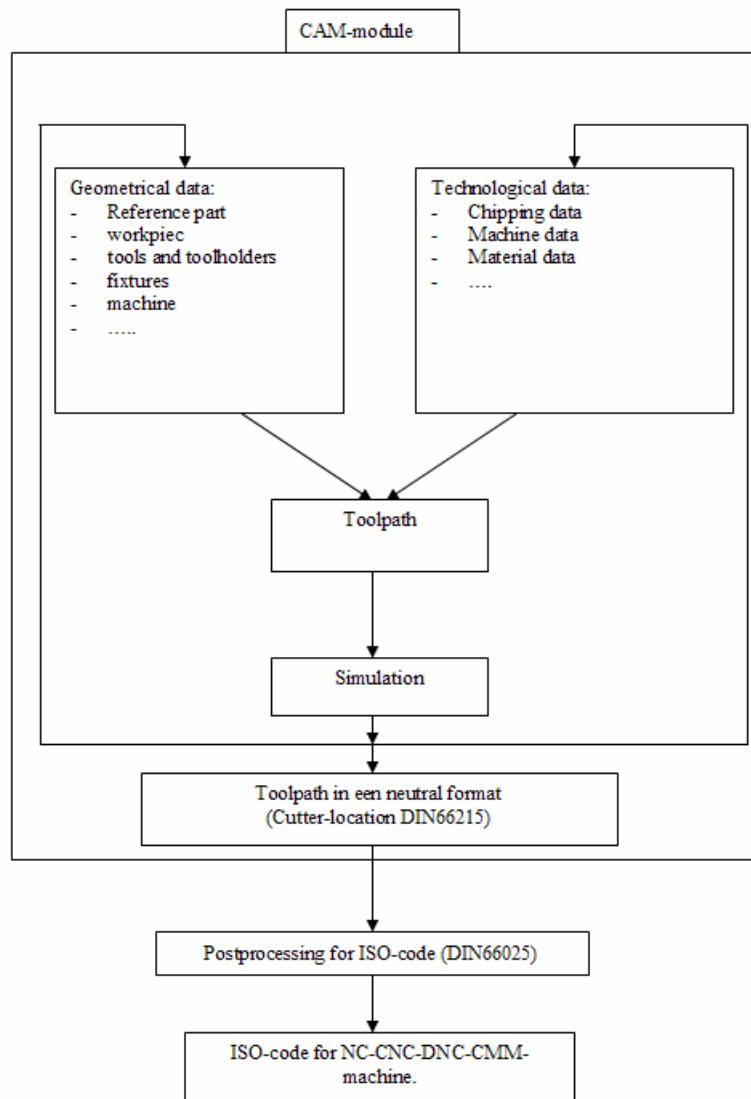Figure 1 A Cincinnati Milacron Arrow 750 milling machine

Figure 2 Flow of data in a CAM-module

The CAM-module of Pro-Engineer is called Pro-Manufacturing or Pro/NC.
This module is completely integrated in the design software, and is as such associatively linked to the design. If the design changes (design model) tool paths will change too.
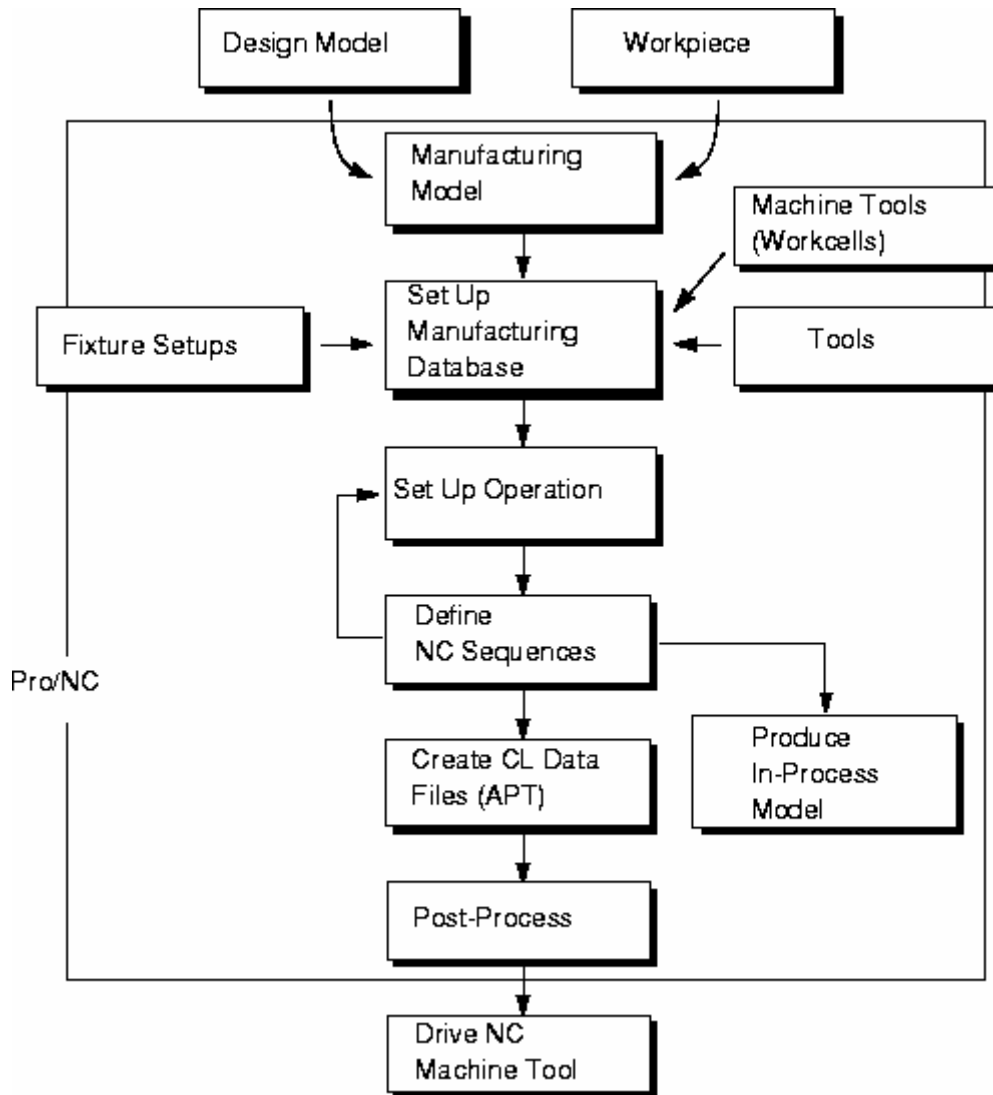
Figure 3 Structuring of the Pro/NC module

3. Data flow in ProNC

3.1.   3.1 Set up of the manufacturing model:

The manufacturing model consists of the Design Model (reference part): this is the product after machining.
If a simulation of the material removal is important, a work piece (rough part) has to created or assembled on top of the design model.
ProNC will remove the excess material of the work piece up to the reference part.

3.2.   3.2 Setup of the manufacturing data base and operation.

The manufacturing database contains the meta description of the machine operations environment.
De machineparameters zijn de parameters die eigen zijn aan de machine. Naam, controllertype, maximale snelheiden en verplaatsingen….

Three sets of data can be inputted:
- Tooling data: which tools (data and description) are available on the machine tool, what do them look like, what position in the tool stock are of tool are they in….
- Machine tools/Work cells: type of machine tool, machine parameters (limits, speeds…)
- Fixture Setups: fixture tools to fix the work piece on the machine tool

An operation is the definition of the set of tool paths that will be executed on a specific machine tool in one fixture position. This is, the work piece will remain fixed in the same position on the machine during the operation.
Operation setup includes definition of the user coordinate system in which coordinates of the tool paths will be outputted. This UCS on the CAD-model reflects the actual coordinate system on the machine tool.

3.3.   3.3 Definition of NC-SEQUENCES:

A SEQUENCE is a tool path with its geometrical and technological data. These data include the type of tool, surfaces to machined and speed at which the tool is to operate and the tool path.
E.g. a drilling tool path for the drilling of a pattern of holes.

3.4.   3.4 Creation of tool path/Cutter Location files:

A TOOL PATH is the specified direction to which the tool moves around the reference model. The path can be displayed on the screen by a path of red lines with arrows indicating the direction of tool.
Cutter Location files are being created. The CL-file is a calculation of the position of the tool, where geometry, machining parameters and other technological data are taken into account. These CL-files can be simulated and edited to optimise the machining.

3.5.   3.5 Postprocessing:

Post processing converts CL-files to the ISO-protocol used in a specific machine tool (MCD-file= Machining Coordinate Data file).
Every machine tool (controller) has its own postprocessor.
Post processing from within ProE is done with GPOST, a general postprocessor which can be customised to fit every machine tool controller that uses ISO-DIN66025 protocols.
Mill tools, lathe tools, WEDM tool…: almost any machine tool can be customised.

3.6.   3.6 Machining:

After uploading the MCD-file (.tap) to the machine tool controller, the machining can be done, and thus the part can be produced.

Example of a CL-file (Cutter-Location) and a MCD-file after postprocessing.

CL-file of the tool path on the title page:

(File extension .NCL.1)

```
$$*        Pro/CLfile  Version 2001 - 2001320
$$-> MFGNO / POMPHUIS
PARTNO / POMPHUIS-STOCK
$$-> FEATNO / 41
MACHIN / UNCX01, 1
$$-> CUTCOM_GEOMETRY_TYPE / OUTPUT_ON_CENTER
UNITS / MM
LOADTL / 1,  OSETNO,  1
$$-> CUTTER / 20.000000
$$-> CSYS / 1.0000000000, 0.0000000000, 0.0000000000, 0.0000000000,  $
       0.0000000000, 1.0000000000, 0.0000000000, 0.0000000000,  $
       0.0000000000, 0.0000000000, 1.0000000000, 0.0000000000
SPINDL / RPM, 2345.000000,  CLW
RAPID
GOTO / 14.5000000000, -0.0000000000, 50.0000000000
RAPID
GOTO / 14.5000000000, -0.0000000000, 10.0000000000
FEDRAT / 234.000000,  MMPM
GOTO / 14.5000000000, -0.0000000000, -5.0000000000
CIRCLE / 0.0000000000, 0.0000000000, -5.0000000000,  $
0.0000000000, 0.0000000000, 1.0000000000,  14.5000000000
GOTO / 14.5000000000, -0.0000000000, -5.0000000000
GOTO / 14.5000000000, -0.0000000000, 50.0000000000
SPINDL / OFF
$$-> END /
FINI
```

MCD-file generated after post processing (File extension .TAP)

```
%*
O231*
N0010G00G28G55G40G91X0.Y0.Z0.M19*
N0015G99*
N0020 T1 M06*
N0025 G00 X0. Y0. Z30.*
N0030 S2345 M03*
N0035 G00 G43 Z50. H01*
N0040 X14.5 Y0.*
N0045 Z10.*
N0050 G01 Z-5. F234.*
N0055 G03 X14.5 Y0. I-14.5 J0.*
N0060 G01 Z50.*
N0065 M09*
N0070G00G28G55G40G91X0.Y0.Z0.M19*
N0075M30*
%*
```

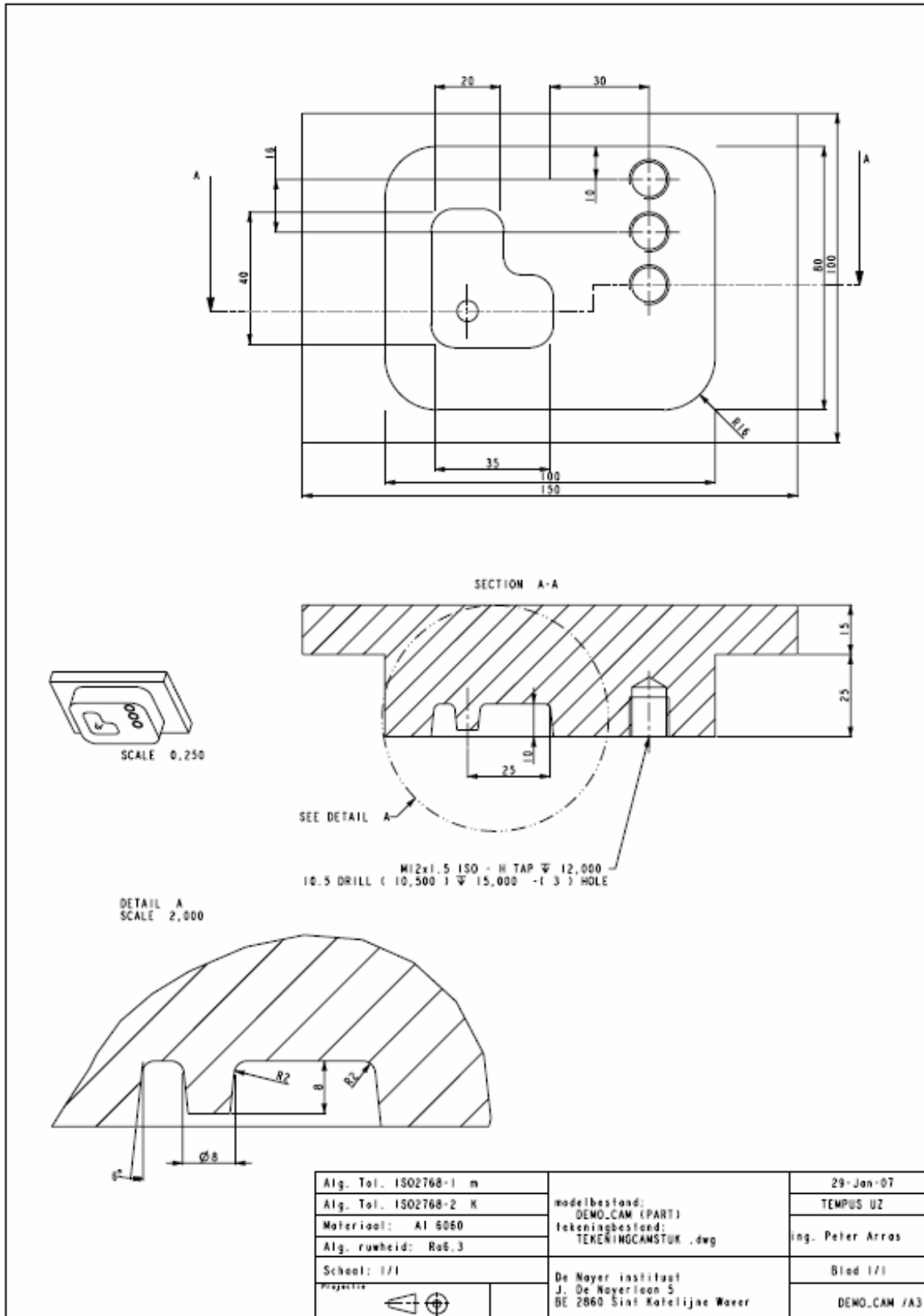## 4. Example of the creation of the (virtual) machining of a part.

Figure 4 Drawing of the demo part

In this example the following object will be set up on a 3-axis milling machine and machined with several typical milling sequences.
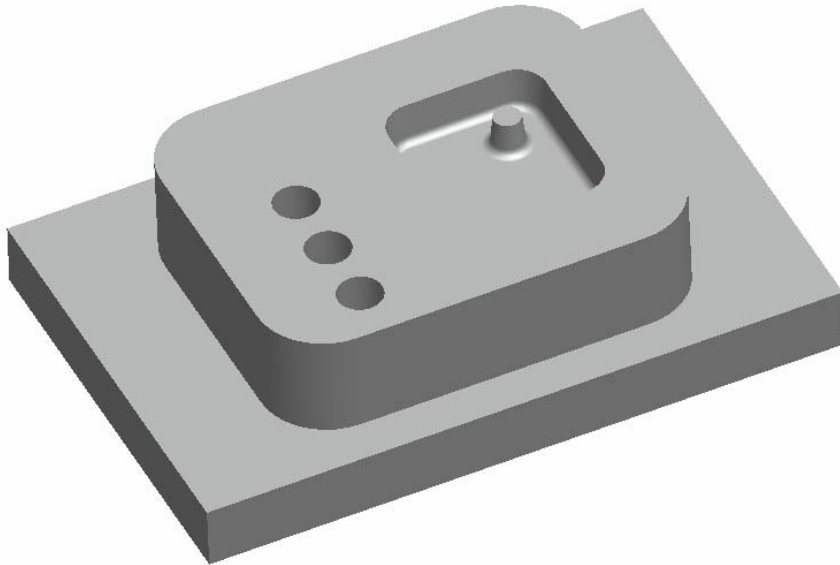

Figure 5 Shaded image of demo part

Objectives:
To create a Workpiece around an existing Reference Model (design model)
To create a co-ordinate system for the operation (machine tool paths)
To put up tools
To generate a face cut
To drill a series of holes.
To generate a profile cut
To create a pocketing cut

4.1.   *Starting the manufacturing file:*

Create a new file for the machining data.
Choose **NEW** from the **FILE** menu and select **MANUFACTURE**, type **NC ASSEMBLY** and type in a name such as **DEMO_UZ**.

Select a mmNS template.
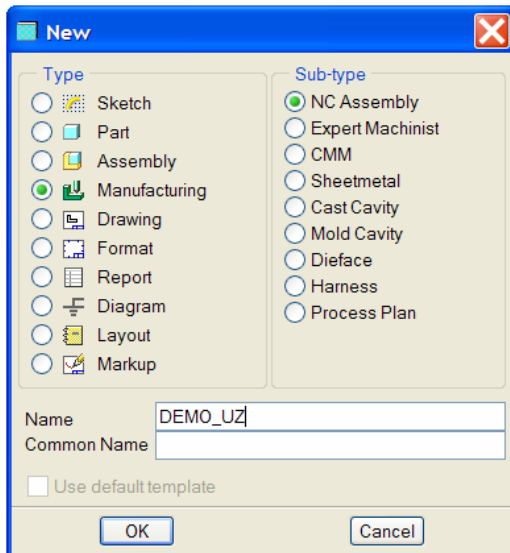
Figure 6 Start new object

The file created is ready to store all the manufacturing information.

4.2. *Setup the manufacturing model (reference and stock part):*

The first data to be inserted into the file is the model to be machined. This is specified by the Command MFG MODEL>ASSEMBLE> REF.MODEL, DEMO_CAM.prt>DONE/RETURN .
Assemble the reference part at the default location (with assembly constraints: <RMK>Default Constraint)
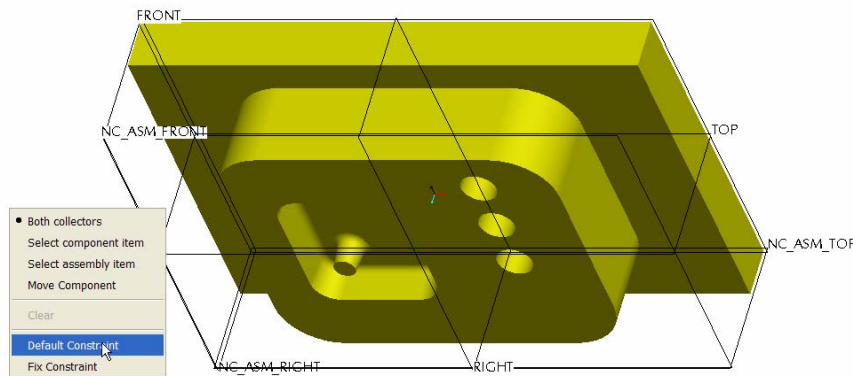
The second piece of data to be inserted into the file is the Work piece. The work piece is a part created to represent the stock material which is removed to by a series of cuts to form the design model (DEMO_CAM.prt). A work piece can be assembled or created. In this example there has no work piece been created to insert in the current manufacturing file, so the work piece is to be created in within this file.
Select MFG Model> CREATE>WORKPIECE. Enter the name of the work piece, type in stock_demo_cam.
By means a extrusion we will now make the stock material for the reference part.
Select SOLID>PROTRUSION (this is the equivalent of INSERT>EXTRUDE in part modelling mode), EXTRUDE, SOLID, DONE and create the stock from the bottom surface

of the part up. Use the bottom surface as the sketch plane, accept default orientation, ignore
the prompt for references and from the sketcher menu select USE EDGE. Select the outer
edges of the base extrusion.
By using the use edge command the work piece will be created exactly the same size as the
outer profile of the design model. Extrude (Blind) to a depth of 42. The external
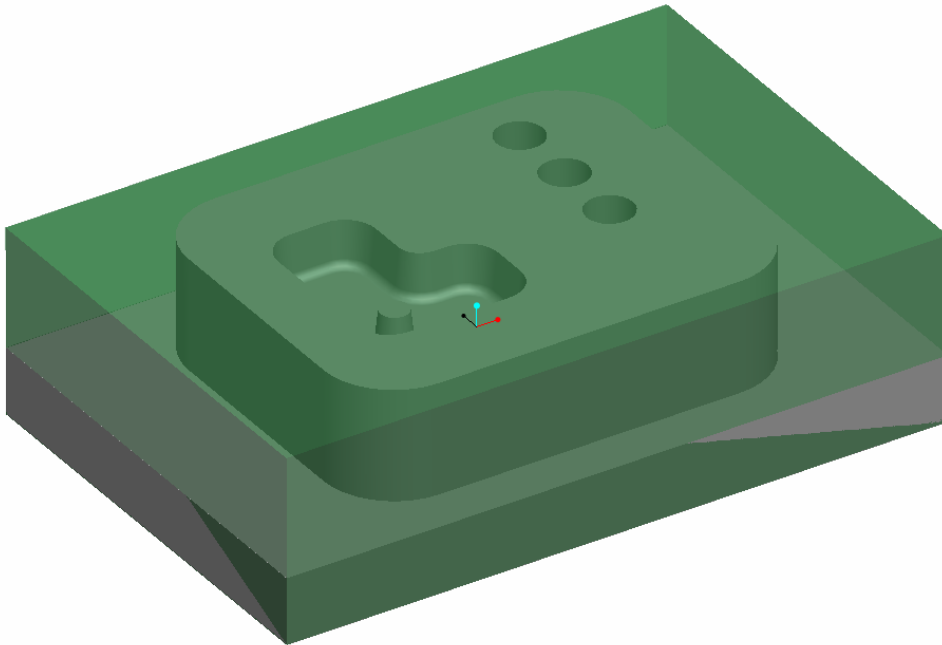profile/surfaces of the work piece will be displayed in green.

Figure 7 Reference part and work piece assembled.

Now that the work piece is made, we define an UCS which will be used on the machine tool
to produce the reference part.
INSERT>MODEL DATUM>COORDINATE SYSTEM.
Use the top, front and side plane of the work piece, and orient the ucs as in the figure. It is
important to define the direction of the Z axis correctly since the Z axis determines the
vertical motion of the machine tool!
Rename the UCS to machining_UCS1
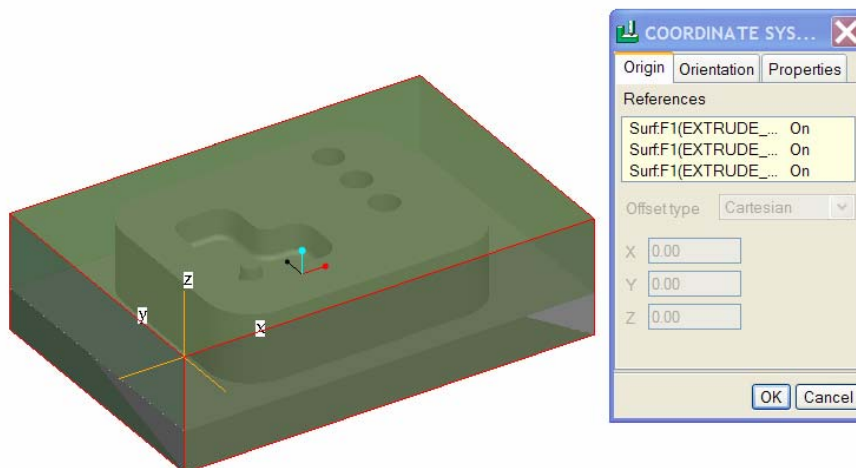
Figure 8 Orientation of the new created UCS

4.3.   *Setup the manufacturing database and operation:*

When selecting MFG SETUP we automatically enter the Operation Setup.
An operation is the definition of the set of tool paths that will be executed on a specific machine tool in one fixture position. This is, the work piece will remain fixed in the same position on the machine during the operation. Operation setup includes definition of the user coordinate system in which coordinates of the tool paths will be outputted. This UCS on the CAD-model reflects the actual coordinate system on the machine tool.
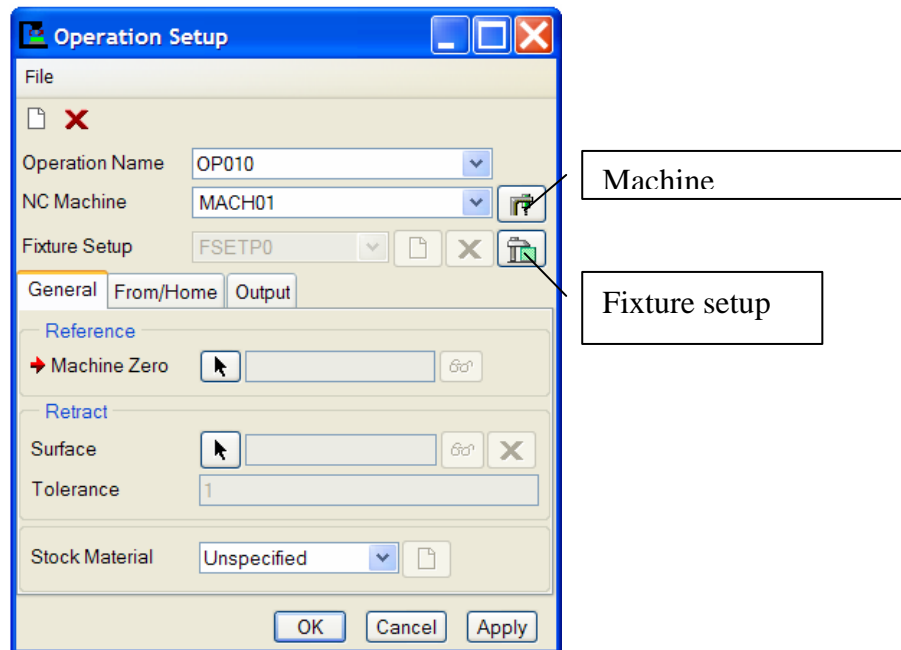
Figure 9 Operation setup.

NC machine/Workcells:

In the nc-machine/workcell we setup the machine tool: MFG SETUP>WORKCELL
In the setup we specify type of machine (mill, lathe, WEDM….) and machine parameters such as limits, maximum speeds….
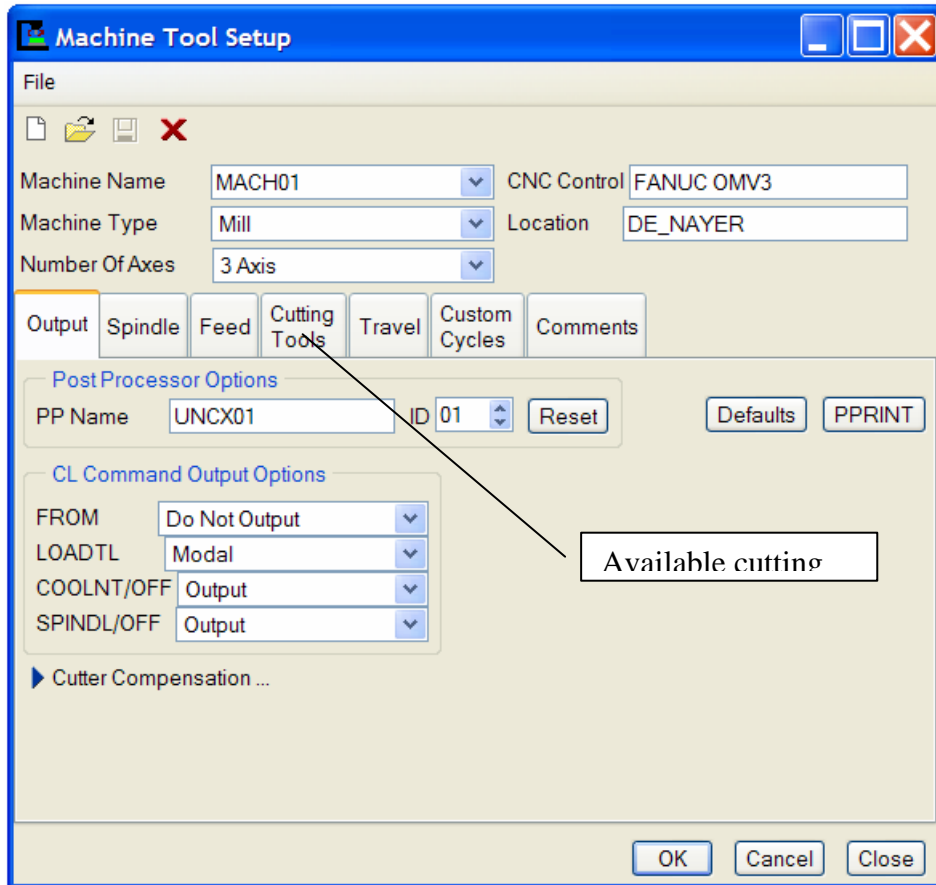
Figure 10 Machine tool or work cell setup

Cutting Tools/tooling:
In cutting tools we set up the available cutting tools: MFG SETUP>TOOLING
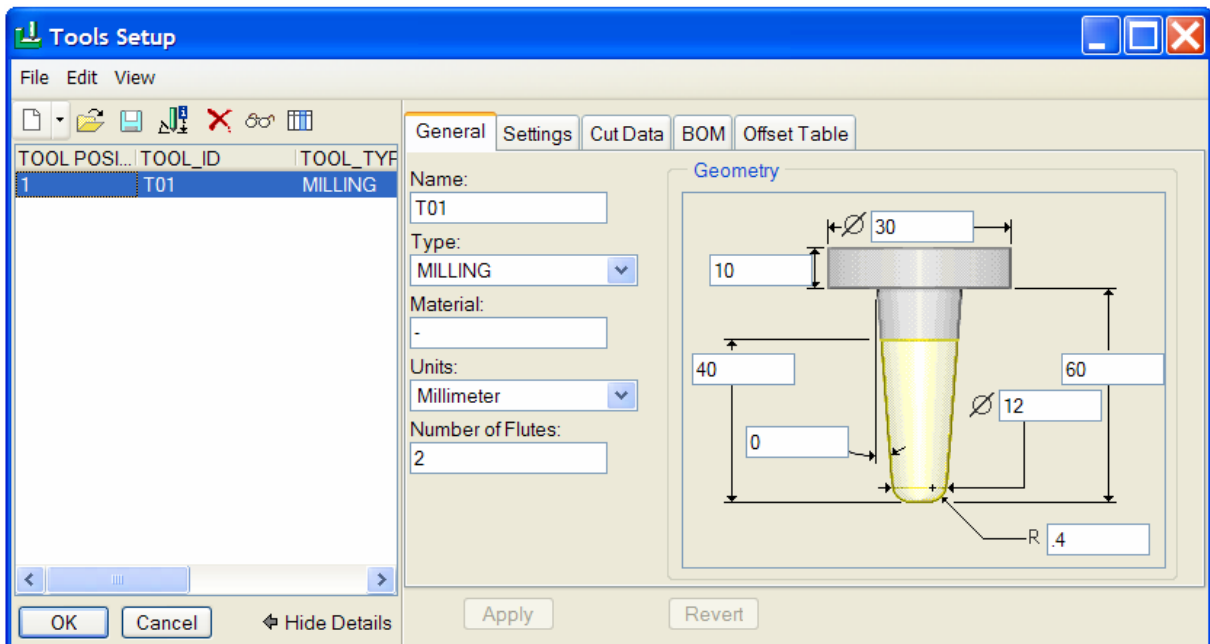The setup is done for a certain machine/work cell.

Figure 11 Tool setup dialogue box

Operation data:

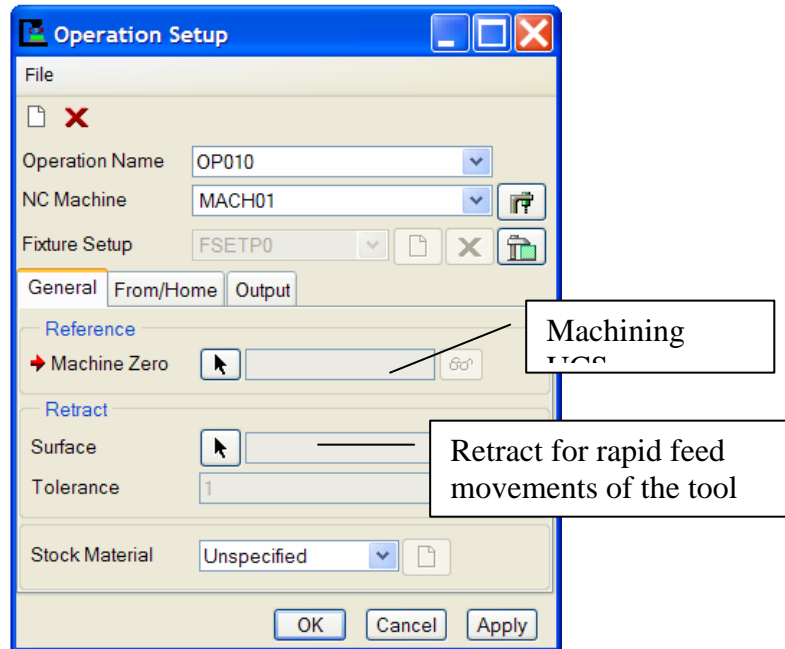We now put up the operation data UCS and retract level.

Figure 12 Operation setup dialogue box

For the machine zero we select the machining_UCS1 coordinate system.
For retract surface, select ALONG Z AXIS, with an offset of 60 mm.

4.4.   NC SEQUENCES.

Now that the Operation (machine setup) has been defined the "NC Sequences" can be generated.

FACE CUT.

The first cut is a FACE CUT to machine the top surface of the model.
MACHINING>NC SEQUENCE>(NEW SEQUENCE)> FACE>DONE.
From the dialogue SEQ SETUP ensure to tick Tool, Parameters, and Surface, then select DONE.
In the tool dialogue box select or setup a milling tool with diameter 12 mm.
In the Param Tree enter the values below, and under "Advanced", change the parameter "TRIM_TO_WORKPIECE" to "yes". As such, the tool path will span the reference part, and will mill the top surface up to the work piece boundaries.
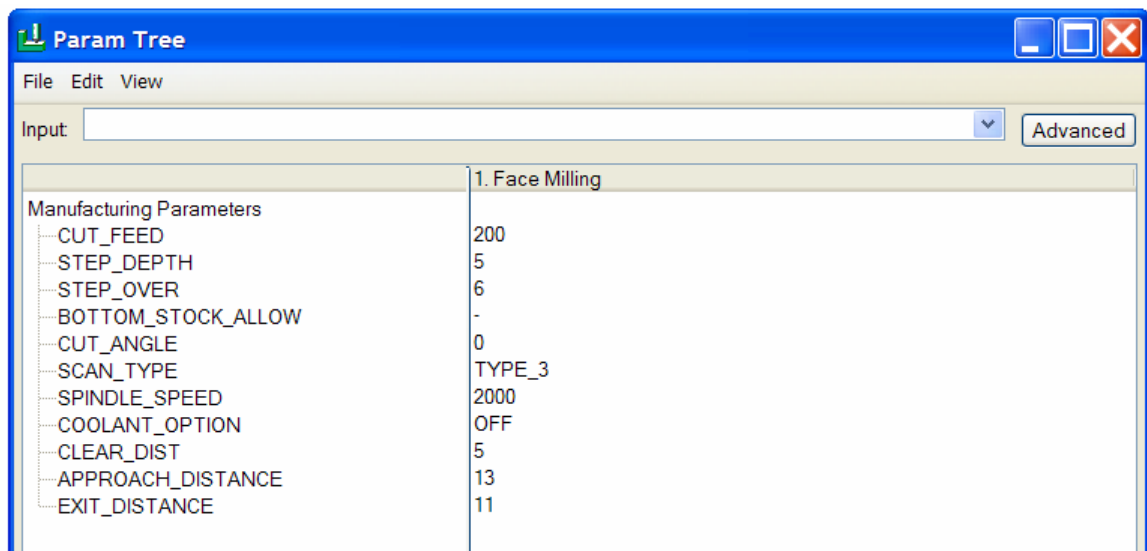
Figure 13 Parameter setup for face milling sequence

In the "SURF PICK" box, select MODEL>DONE and select the top surface of the reference part to be machined. (Then select DONE)
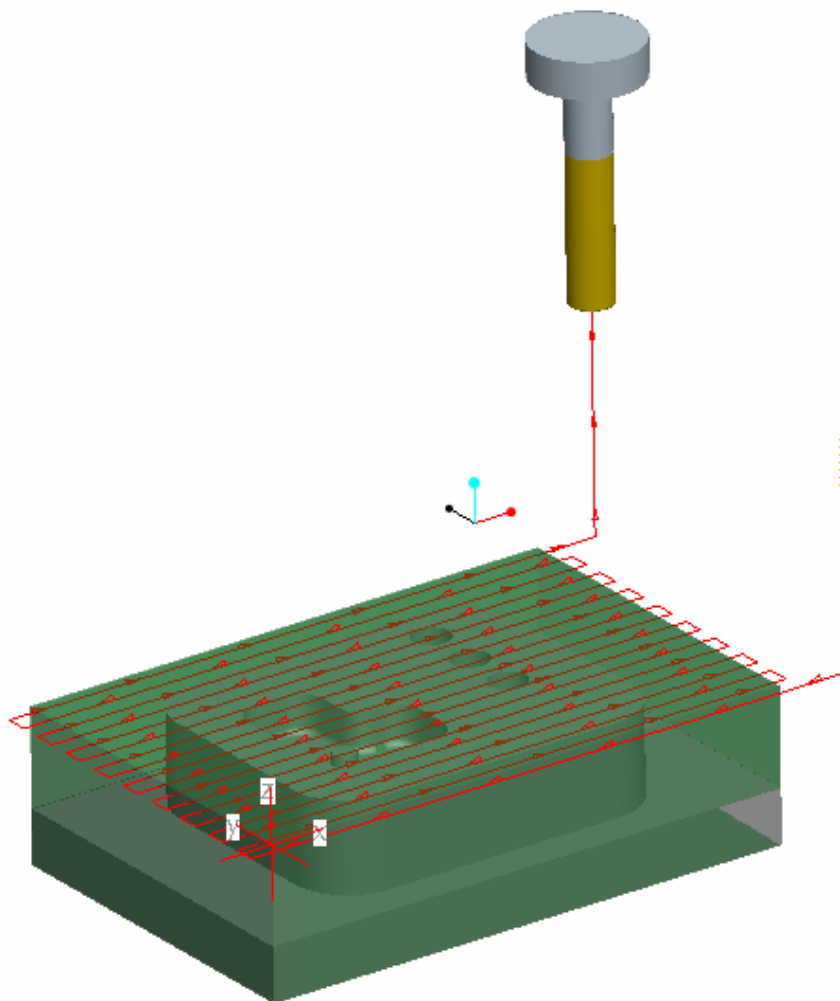

Figure 14 Face milling sequence (play path)

To view the tool path/NC sequence created PLAY PATH>SCREEN PLAY> DONE.

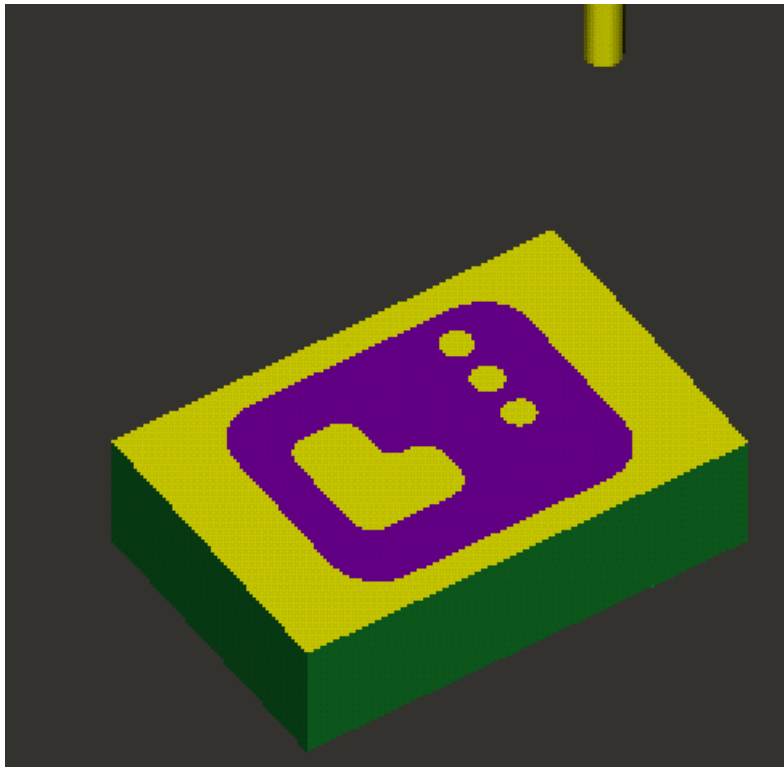To view the simulation in colour select NC CHECK>DISPLAY>RUN to view the tool path in colour.

Figure 15 Face milling sequence (NC check)

The NC check will display the machining NC Sequences in colour, listed below is what the following colours indicate in he NC Check.

Yellow = Colour of the Workpiece where the tool is cutting during the NC Sequence.
Green = External colour of the Workpiece.
BLUE = Colour of the Ref.Model where it becomes visible as the tool cuts closer to the Ref.Model. If this colour appears there is no need to worry.

Cyan (light blue) = Designates the area where the tool may be grazing or gouging the Ref.Model. If this colour appears the tool path has to be altered to avoid the tool clashing with the Ref.Model.

The Cutter Location file can be viewed by selecting "CL Data".
"DONE SEQUENCE" to finish the face milling.

HOLEMAKING.

The next NC Sequence to be created is for the drilling of the holes.
MACHINING>NC SEQUENCE>NEW
SEQUENCE>HOLEMAKING>DONE>DRILL>STANDARD>DONE.
From the dialogue SEQ SETUP ensure to tick Tool, Parameters, and Holes, then select
DONE.
In the tool dialogue box select or setup a drilling tool with diameter 10.5 mm.
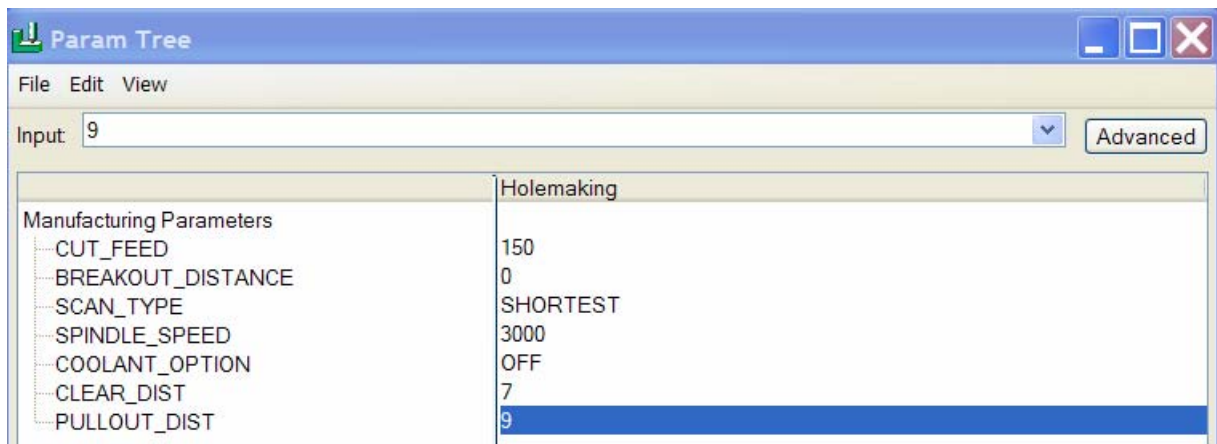In the Param Tree enter the values below.

Figure 16 Parameter setup for Holemaking sequence

Now we need to select the holes to machine.
Selection of the holes can be done in many ways: manual (select circular surfaces or axis
lines), by rules…
We will leave it to ProE to automatically find the hole set to be machined by specifying the
desired drill diameter.
Select on the tab page "Diameters" the drill diameter 10.5.
Under options, select "Start hole for scan" and select the left most hole for the start of the tool
path.
In the depth tab, select the wanted depth for the holes. If selected auto, ProE will drill the
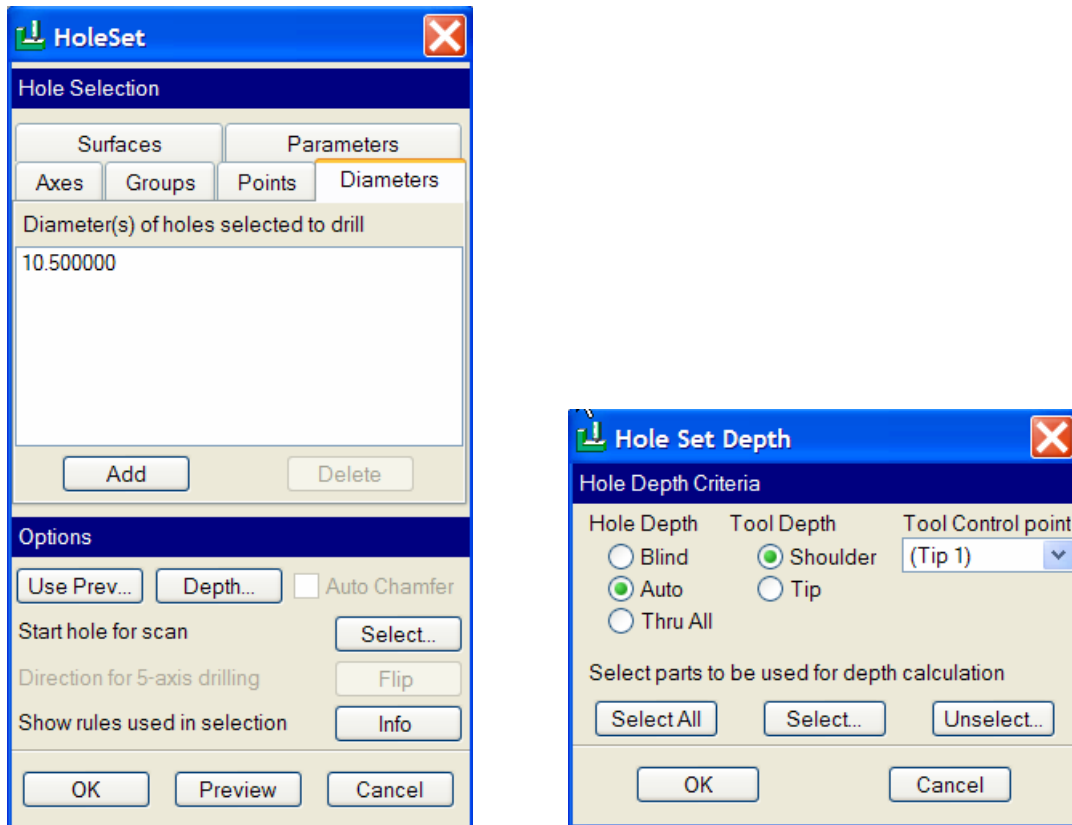holes to the depth specified in the part.

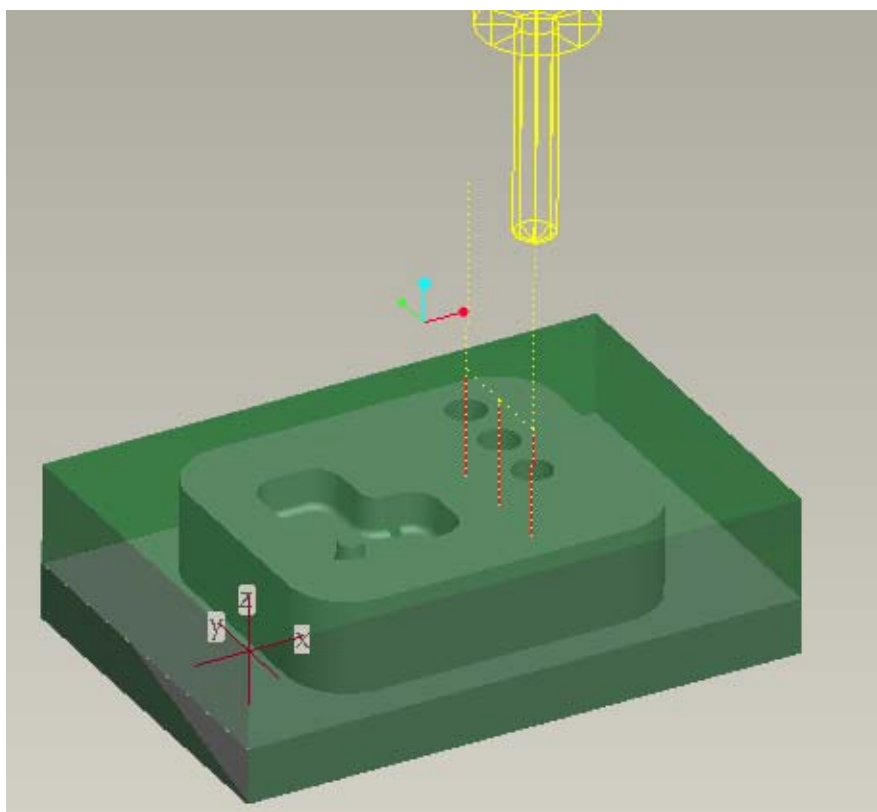Figure 17 Selection of holes and depth for holes to drill

Figure 18 Hole making sequence.

To view the tool path select PLAY PATH>SCREEN PLAY>DONE.
Done Seq finishes the NC Holemaking Sequence.

PROFILE MILLING.

The next NC Sequence to be created is for the profiling of the outside of the part.
MACHINING>NC SEQUENCE>NEW SEQUENCE>PROFILE>DONE.

From the dialogue SEQ SETUP ensure to tick Tool, Parameters, and Surfaces, then select
DONE.
In the tool dialogue box select or setup a drilling tool with diameter 10.5 mm.
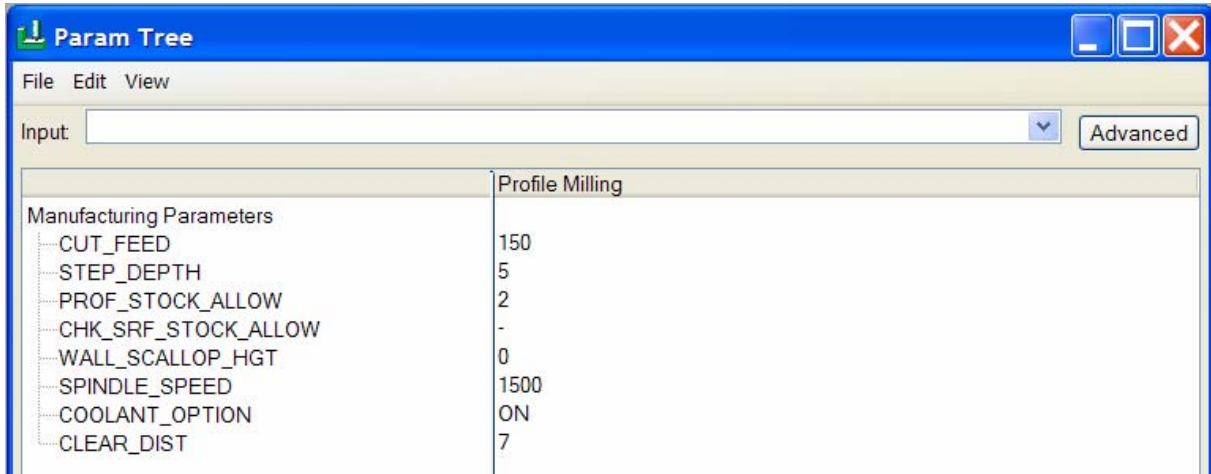In the Param Tree (simplified) enter the values below.

Figure 19 Parameter setup for profile (simplified)

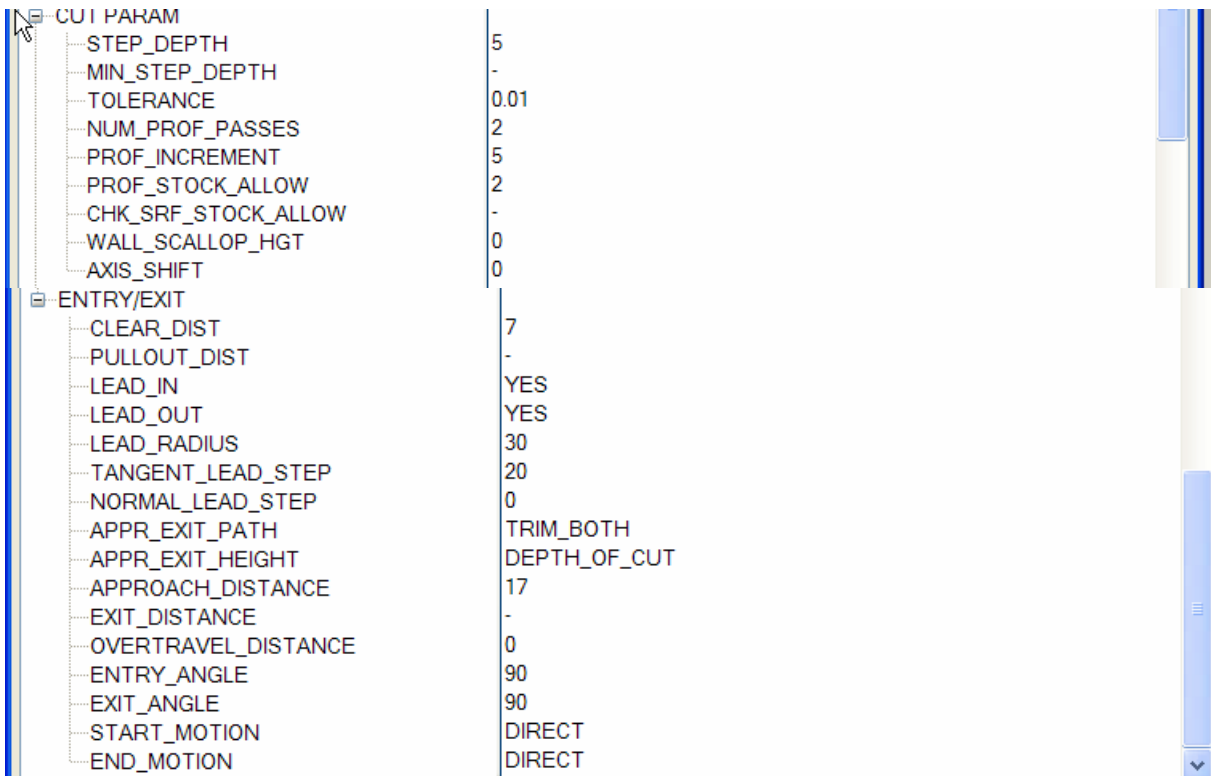In the Param Tree (advanced) enter the values below.

Figure 20 Parameter setup for profile (advanced)
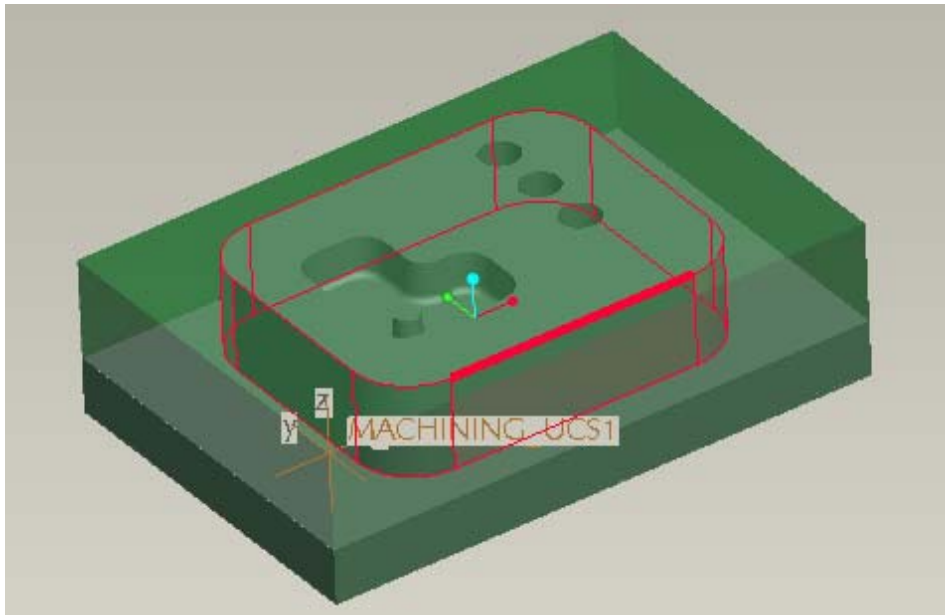
Next select the surfaces to be machined:
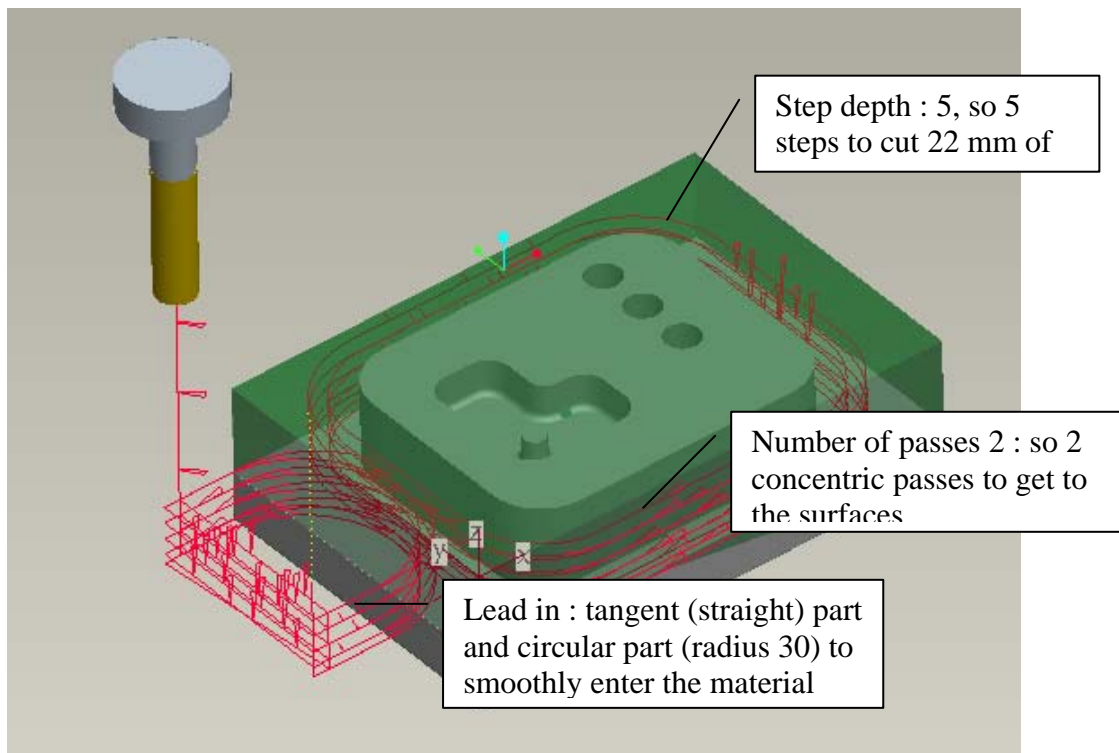

Figure 21 Surfaces to profile.

Step depth : 5, so 5 steps to cut 22 mm of

Number of passes 2 : so 2 concentric passes to get to the surfaces

Lead in : tangent (straight) part and circular part (radius 30) to smoothly enter the material

Figure 22 Profile milling sequence: effect of parameter setup

4.5.   *EXERCISE:*

Make a POCKET milling sequence for the pocketing of the L-shape pocket.
Use a tool with diameter 8, and corner radius 0.2.