

FACULTY OF ENGINEERING
UNIVERSITY OF STELLENBOSCH
USER INSTRUCTIONS
FOR
LEADWELL VMC40 NUMERICALLY CONTROLLED
MILLING MACHINE

18 Junie 2014
Prof AH Basson

TABLE OF CONTENTS

1. Safety Precautions	2
2. Basic User Instructions.....	3
2.1. Jog.....	3
2.2. Handle.....	3
2.3. MDI (Manual Data Input)	3
2.4. Tool Changing	4
2.5. How to Load and Remove Tools from the Tool Changer	4
3. Procedure for Turning the Machine On	5
4. Procedure for Shutting the Machine Down	5
5. Procedure for Switching on the Controller	6
6. Procedure for Running a NC Program.....	7
6.1. Safety Precautions	7
6.2. How to Set Up the Machine	7
6.3. Setting Up Program Feed	8
6.3.1. Running a Program in NC-Machine Memory	8
6.3.2. Running a Program from the PC.....	8
6.4. To Run a Program	9
7. Program Preparation	10
8. Example Program.....	11
Appendix A: List of Utility Programs	12
Appendix B: Summary of Programme Codes for Leadwell VMC40 with Meldas Controller.....	13
Appendix C: Arrangements for Supervision and Management of the NC Milling Machine.....	16
C.1 Booking Procedure and Usage Fees	16
C.2. Maintenance and Purchases.....	17
C.3 Cleaning.....	17
C.4. Authorization to Operate the Machine	18
C.5. Safety Aspects	18

1. SAFETY PRECAUTIONS

- The user instructions in this document must be strictly adhered to.
- Only persons authorized by Prof Basson or Mr Blom may operate the NC-machine.
- The person that operates the machine is responsible for ensuring that nobody in the vicinity of the machine is exposed to any risks. The person has to ensure that all precautions regarding safety and health are enforced.
- Nobody may use the machine alone after hours. There must be at least one other person nearby who has to periodically check if everything is still fine.
- The room may never be left open except if a person that is authorized to use the machine is present.
- When the machine vice, 4th axis or any heavy object is mounted or removed from the machine bed, all persons nearby must wear safety shoes.
- All potential safety risks must be reported to Prof. Basson.
- All programs must be checked using NCEdit before feeding it to the machine.

When the machine's door is open and the **EMERGENCY STOP** button is not activated, the following is applicable:

- Everyone near the machine must wear safety glasses.
- Keep loose clothes (such as tie's, sleeves, etc.) away from the spindle.
- Keep a distance of 300 mm between any part of your body and the spindle.
- Keep a distance of 300 mm between any part of your body and the machine bed.

Safety duties of Staff and Students:

Art. 14 of the Act on Occupational Health and Safety, Act 85 of 1993, stipulates that a student or staff member:

- May not put his/her safety or that of any other person or staff member at risk;
- Must co-operate to fulfil the safety requirements;
- Must carry out instructions given by a responsible person;
- Must report all unsafe or unhealthy conditions to a responsible person (Prof. Basson or Mr. Blom);
- Must report any incident that may harm his/her safety/health as soon as possible to the persons named above.
- Will be guilty of a criminal offence according to the Act if the rules above are not adhered to (negligence may be regarded as purposeful disregard of the Law.)

2. BASIC USER INSTRUCTIONS

2.1. JOG

- Switch **MODE SELECTION** to **JOG**.
- Switch **AXIS SELECT** to the axis that has to be moved.
- Set the speed of movement by adjusting **JOG & FEED RATE OVERRIDE**.
- Push the **MANUAL FEED** + or - buttons.
(Directions are shown on figure on the front of machine bed. NB the directions are not the same as those used by programs.)

2.2. HANDLE

- Switch the **MODE SELECTION** to **HANDLE**.
- Switch the **AXIS SELECT** to the axis that has to be moved.
- Set the speed of movement by adjusting **INCREMENTAL & RAPID**.
- Turn the hand wheel with the winder.
(Directions are shown on the figure on the front of the machine bed. NB the directions are not the same as those that are used by programs.)
- Do **not** operate any other switches while turning the hand wheel.

2.3. MDI (MANUAL DATA INPUT)

- Switch the **MODE SELECTION** to **MDI**.
- Push the **EDIT/MDI** button on top of controller panel.
- Select **MDI** on the screen options.
- Edit the program where needed:
 - Push the **SHIFT** and **INS** to place characters between other characters.
 - To create a new line you have to add the **EOB** character where the line has to be, and then push **INPUT/CALC**
 - Push **SHIFT, C-B/CAN** to clear the screen.
 - Remember to specify the absolute commands (M90) or relative commands (M91) if appropriate.
 - If a tool-change command (M06) is given, you have to give at least M30 thereafter.
 - Remember the decimal point for coordinates (for example X10 means X0.010).
- When the program editing is finished, push **INPUT/CALC**. The following must appear at the bottom right of the screen: **MDI SETTING COMPLETE**.
- Then you can follow all the instructions that have to do with running the MDI program.

2.4. TOOL CHANGING

Precautions:

- Make sure that the coolant pipe and vacuum cleaner inlet are not going to obstruct the tool change.
- Make sure that the position on the tool changer that is opposite the spindle, is empty.
- If necessary, write the Z-coordinate of the tool point at a reference position (e.g. on the tool setter) so that the same value can be set for the new tool.
- NB: when there is already a tool in the spindle, make sure that the spindle can rotate freely, because the machine rotates the spindle before changing tools.

Tool changes can be programmed by **MDI**. However, there are programs in memory to load every tool. The program numbers are 100 + tool number. (For example: 101 for tool 1). How to use these programs:

- Switch **MODE SELECTION** to **MEM**.
- Push **MONITOR** at top of controller.
- Choose **SEARCH** on screen options.
- Enter program number and push **INPUT/CALC**. Make sure that the appropriate program number shows on top of the screen (example: **O 101**).
- Choose **COORD**
- Make sure that the correct program is shown, for example

```
M06 T18;  
M30;  
%
```

- Push **CYCLE START**.

Warning:

If the controller switches off, by accident, while busy changing tools, it will lose some of the parameter settings. These settings have to be reset before further work can be done. See Prof. Basson if this happens.

Warning:

If the compressed air pressure is too low while the machine is busy with tool changing, it will come to a standstill. If the pressure then rises above 5 bar again within a short period, the machine can continue with the tool change. If not, you have to switch off the Controller. As previously explained, some of the parameter settings will have to be reset before further work can be done.

2.5. HOW TO LOAD AND REMOVE TOOLS FROM THE TOOL CHANGER

- The tool holder is placed/removed through an opening in the tool changer's case. Tools may **never** be mounted in the position opposite the spindle.
- The following programs that have been programmed into the machine's memory can be used to rotate the tool changer to place/remove the tool holder in the chosen tool position: to get tool

position (i) opposite the mounting opening you have to run program 200+i. This program will load the tool that would leave tool position (i) opposite the opening in the tool changer's case.

NOTE: this program loads another tool in the spindle. Make sure that the tool that is in the spindle in the beginning of the procedure can turn freely, and make notes of the reference Z-position if necessary.

- Always keep tool position 1 empty. In this way all programs can be ended with Tool 1 and the spindle can be left empty.

3. PROCEDURE FOR TURNING THE MACHINE ON

- Open compressed-air valve at wall.
- Check air pressure (typically higher than 5 bar) and lubricating oil level on the NC-machine.
- Switch on the electrical main switch.
- Follow the "Procedure for Switching on the Controller" before running any program.

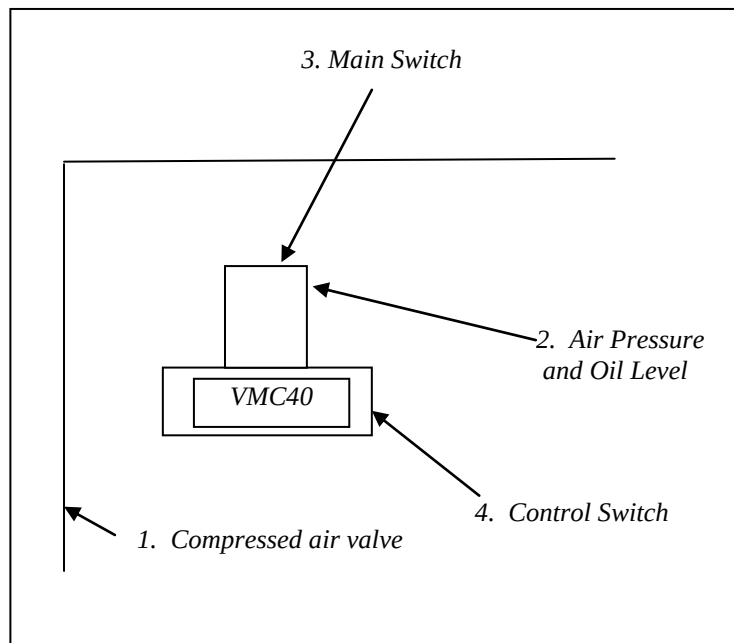


Figure 3.1 Top View of VMC40 in corner of the room

4. PROCEDURE FOR SHUTTING THE MACHINE DOWN

- If a tool is still in the spindle, let the machine replace it in the tool changer (select tool 1; NB: keep tool position nr. 1 empty).
- If a work piece is left on the bed, make sure that the spindle is higher than any point of the work piece.
- Switch the Controller off.
- Switch off the main switch.
- Vacuum up all dirt or sawdust, or wash all metal chips off the bed.

5. PROCEDURE FOR SWITCHING ON THE CONTROLLER

This procedure must be followed each time that the controller is switched on.

NB Do not switch the **AXIS SELECT** to B before the fourth axis is unclamped.

NB If an axis is moved beyond its limit and the red light starts flashing, keep the **EMG RELEASE** switch up while the axis moved away from its limit.

NB The figure on the front of the machine bed shows the direction of the bed's movement for + or- movements of the different axis.

- Switch the controller on (front right on the machine).
- Make sure that the tool changer position opposite the spindle is empty.
- Make sure that there is nothing loose on the machine bed.
- Make sure that the fourth-axis cables are not damaged while moving the bed to **HOME**.
- Move the Z-axis at least 100mm away from its **HOME** position by means of **JOG** or **HANDLE**.
- **HOME** the Z-axis:
 - Switch **MODE SELECTION** to **HOME**.
 - Switch **AXIS SELECT** to **Z**.
 - Push any of the two buttons at **HOME/MANUAL FEED**.
 - Wait till the Z-axis comes to a standstill.
- Make sure that the Z-axis is higher than anything on the machine bed.
- Move the X- and Y- axes at least 100 mm away from their **HOME** positions by means of **JOG** or **HANDLE**.
- Let the X- and Y-axes find their **HOME** positions (procedure as for Z-axis).
- Unclamp the B-axis brake by using M41 as follows:
 - Switch **MODE SELECTION** to **MEM**.
 - Push the **MONITOR** key on the controller.
 - Choose **SEARCH** on the screen options.
 - Type 2 and push **INPUT/CALC** (the message **SEARCH COMPLETE** will show).
 - Choose **COORD** on the screen options.
 - Make sure that the part of the program that is shown on the screen is the following:
M41;
M30;
 - Push **CYCLE START**.
- Let the B-axis find **HOME** position (procedure is the same as for the Z-axis).
- Make sure that all the axes' green **HOME** position lights are lighted on the control panel.

6. PROCEDURE FOR RUNNING A NC PROGRAM

6.1. SAFETY PRECAUTIONS

- Make sure that the whole area that is being machined is within reach of the machine, and that there is no part that reaches over the machine bed.
- Make sure that the work piece is very firmly clamped and that the clamps are not in a position that they could be in the way/or be cut.
- Remove all loose objects from the machine bed (remember the vice handle and turn-table spanner).
- Make sure that the fourth-axis cable will not be damaged when the bed moves.

6.2. HOW TO SET UP THE MACHINE

- Load the tool into the spindle if necessary (see the procedure described elsewhere in this document).
- Move the tool until it touches the top of the tool length setter with **JOG** or **HANDLE**, adjust the tool length setter's scale to 0 and note the Z-axis **MACHINE COORDINATES** so that you can reset the tool length later on (push **MONITOR** on the controller and choose **COORD** on the screen options to show the coordinates).
- Move the tool to the given start position through **JOG** or **HANDLE** (if it is the first time that the program is being run, set Z higher than normally so that the tool does not touch the work piece).
- **NB:** Record the **MACHINE COORDINATES** of the start position so that the machine set-up can be repeated exactly (push **MONITOR** on the controller and choose **COORD** on the screen options to show the coordinates).
- If cutting in wood, plastic or wax:
 - Position the vacuum cleaner inlet and switch it on (at the switch on the vacuum cleaner).
 - Make sure that the **COOLANT** switch is **OFF**.
- If cutting in metal:
 - Remove the "Tool Setter" from the machine table.
 - Position the coolant nozzles and switch it on, or put the **COOLANT** switch on **AUTO** if the program is going to control it automatically.

HINT:

The following programs can be used to set the coordinates of the various axes to zero:

Program 10: zero X, Y en Z

Program 11: zero X

Program 12: zero Y

Program 13: zero Z

6.3. SETTING UP PROGRAM FEED

6.3.1. Running a Program in NC-Machine Memory

- Set **MODE SELECTION** on **MEM**.
- Push **MONITOR** on the top of the controller.
- Choose **SEARCH** on the screen options, type in the program number, and push **INPUT/CALC**. The correct program number should appear on the top of the screen. (e.g. O 1).
- Choose **COORD** on the screen options to show the program and the coordinates.

6.3.2. Running a Program from the PC

- Switch **MODE SELECTION** to **TAPE**.
- Push **MONITOR** on the controller and choose **COORD** on the screen options.
- Switch **SINGLE BLOCK** on at the control panel.
- Place a copy of the program in the computer's **C:\LEADWELL** subdirectory.
- If Norton Commander is not running, type "**NC**" and **ENTER**.
- Choose the "Menu" option in Norton Commander by pushing F2.
- Choose the option for up and downloading of **NC** programs.
- When **TERM95** is running, press **PAGE UP**, select **RAW**, and click with mouse (only once) on the file that contains the tool path. Do not push **ENTER** yet.
- Push the **CYCLE START** button.
- Push **ENTER** on the computer to begin the program transfer.
- Make sure that the start of the correct program is shown on the controller screen, noting in particular that there is nothing preceding the program start. If not, push **CANCEL** on the **NC-machine's** control panel, and **ESC** on the computer's keyboard. Close **TERM95** and return to the step in the procedure before **TERM95** is started.
- Follow the instructions in the following section that describe how to run the program.
- After the program has finished cutting, push **ESC** on the PC.

6.4. TO RUN A PROGRAM

NB: Keep the machine's door closed except, when everybody near the machine is wearing safety glasses.

- Set the following functions as indicated for either the first (trial) runs or the other:

FUNCTION	FIRST RUN	OTHER RUNS
SINGLE BLOCK	On	On
COOLANT	Off	On / Auto
JOG & FEED RATE OVERRIDE	20%	100%
INCREMENTAL & RAPID	25%	100%
SPINDLE OVERRIDE	100%	100%

- First read all the instructions that follow, because you should focus your attention on the machine when proceeding.
- Push the **CYCLE START** button. If the program is run for the first time, keep your finger on the **FEED HOLD** all the time and watch the machine's movements through the whole program.
- Push **FEED HOLD** if any potentially dangerous movements are performed. To stop the program implementation temporarily, push **FEED HOLD**. Push **CYCLE START** to re-start program implementation again. The controller screen shows the distance that the spindle has to move to finish each program step (**DIST TO GO** on the **COORD** screen).
- When **SINGLE BLOCK** is on:
 - **CYCLE START** must be pressed for each program step.
 - The cursor on the screen stops at the line that has just been executed.
 - The **CYCLE START** button's light keeps burning while the machine is busy with the instruction.
 - If the program is run for the first time, keep **SINGLE BLOCK** on for at least the first 10 program steps.
- After the program is finished and the yellow warning light flashes, push **RESET** on the controller.
- If you want to interrupt the program and start over, push **FEED HOLD** and wait till the machine comes to a standstill. Push **RESET**. If the program transfer is coming from the PC, push **ESC** on the PC. Move the Z-axis till the spindle is clear of the work piece. Do it with **JOG** or **HANDLE**. Close TERM95 in the PC. Repeat the procedures to run the program.

7. PROGRAM PREPARATION

An example of a program is given in the following section.

Note that the program number must contain the following: the date that the program was finalised and a two-figure serial number, or a student number and a single-figure serial number. The program name must contain the name of the owner. This information will be used to clear the hard drive of the computer periodically. Do not store programs in the machine's memory before getting permission from Prof. Basson.

Axis system: Right handed with Z vertically upwards, X to the right and Y in the direction of the machine.

Warnings:

- Although comments between brackets, such as the program name in the example, are acceptable in principle, comments in programs often produces strange errors when the cutting program is executed.
- It is the safest to give only one instruction of each letter (for example G) in each block.

Programs can be tested in NCEdit, which is available on the PC at the NC Machine. This program is fairly strict and will show errors that could be accepted by the **NC-machine**, but there are very few errors that will bother the **NC-machine** that will not be shown by **NCEdit**.

The following instructions must be given to the machine operator so that component can be manufactured.

- The program file name.
- The program's start position. Physical position relative to the work piece and corresponding coordinates.
- The tool types and diameter, and in what tool positions they are to be loaded.
- D and H registers for the tools when compensation is going to be used.
- Orientation of the work piece on the machine bed.
- Acceptable clamping positions.

8. EXAMPLE PROGRAM

Example Program for Leadwell VMC 40 with Meltas Controller	
PROGRAM	EXPLANATION
% O96092501 (NAME)	Start program feed Program number (date + 2 figures or student number + 1 figure); Program name
G90 G80 G40 G49 G17 G54 G92 X0 Y0 Z100. B0 G00 Z110.	Absolute commands, Cancel sub programs, Cancel tool diameter-compensation Cancel tool length compensation Set XY-contouring Choose axis system 1 and set coordinates for current position Lift tool above work piece/reference point before M06 or G49
/M06 T18 (16MM END MILL)	Load tool 18 (/⇒ skip if BLOCK SKIP is on)
G43 Z130. H17 F1000	<i>Optional:</i> Set length compensation from register 17, move to Z130. And set feed rate to 1000 mm/min
S1200 M03 M08	Spindle speed at 1200 rpm Spindle on, clockwise Coolant on
N10 G00 X34.679 Y28.988 Z3. G01 Z-1. F100 X35.867 Y29.151 F500 G02 X62.91 Y8.639 I3.26 J-23.7 X60.301 Y-19.9 I-62.91 J-8.63	Line numbers are optional but show on the controller and help trace with the program implementation Rapid movement.. NB: not linear interpolation. Remember decimal points! Linear interpolation, Feed rate 100mm/min Linear interpolation, Feed rate 500mm/min Circular interpolation. Note that I and J have signs
G00 Z110.M09 M05 /M06 T1 /G49 G40 G00 X0 Y0 Z105. G01 Z100. M30 %	Lift tool above work piece. Coolant off Stop spindle Load tool 1, which is always kept empty Cancel length compensation except if tool 1 is not loaded (/⇒ skip if BLOCK SKIP is on) Cancel tool diameter compensation Rapid movement to above start position Move to start position Stop program and rewind End of program feed

APPENDIX A: LIST OF UTILITY PROGRAMS

The following programs are loaded in the NC-machine memory. To run the programs in the NC-machine memory, follow the instructions given above.

PROGRAM	FUNCTION
2	Disconnect B-axis
6	Load tool 1 and move to X0, Y0, Z100
10	Zero X,Y, and Z
11	Zero X
12	Zero Y
13	Zero Z
100+i	Load tool i
200+i	Load a tool so that tool holder i is opposite the opening in the tool changer.

**APPENDIX B: SUMMARY OF PROGRAMME CODES FOR
LEADWELL VMC40 WITH MELDAS CONTROLLER**

CODE	PURPOSE	CODES	REMARKS
G00	Positioning	X,Y,Z,B	Can move with angles of 0°, 45° or 90° Moves at "Rapid Traverse" speed.
G01	Linear interpolation	X,Y,Z,B	Moves in a straight line between old position and new one. Moves with speed indicated by F.
G02	Clockwise circular interpolation	X,Y,Z, I,J,K	I,J,K is the position of centre relative to starting point.
G03	Anti-clockwise circular interpolation		G17, G18, G19 determines the plane containing circle. Moves at a speed indicated by F.
G04	Dwell	X	X gives a time in seconds (if in decimal point) or milliseconds (if no decimal point is given)
G17 G18 G19	Circles and Compensation in XY level Circles and Compensation in ZX level Circles and Compensation in YZ level		use only one of these commands per line.
G40 G41 G42 G43	Cancel tool diameter compensation Tool compensation left Tool compensation right Tool length compensation	X,Y,D X,Y,D Z,H	Compensation Left means tool is to the left of work piece, if you look in direction of tool motion. Coordinates are the end of a straight line motion with compensation
G54	Choose work-coordinate system 1		
G80	Cancel build-in cycles		
G90 G91	Absolute commands Incremental commands		
G92	Set the position registers	X,Y,Z,B	The current tool position is taken to be these coordinates (the origin of the coordinate system is moved)

CODE	PURPOSE	REMARKS
M00 M01 M02 M30	Stop program Optional stop Program end Program end and rewind tape	Controlled by switch on control panel
M03 M04 M05	Spindle on - clockwise Spindle on - ant-clockwise Spindle off	Must be preceded by an S code to set the speed of rotation.
M06	Tool change	Tool number must be given in T-code Note that the X- and Y- positions are unchanged, but a tool change leaves Z in the maximum position
M08 M09	Coolant on Coolant off	
M40 M41	4 th axis clamp on 4 th axis clamp off	
X Y X B	Coordinates that the point the tool must move to. If preceded by G90, coordinates are measured relative to the origin set by G92. If preceded by G91, coordinates are measured relative to current position.	Followed by + or - and up to seven digits. X,Y,Z: If no decimal point given, the measurement is in μm . If a decimal point is given, measurement is in mm. B: Measured in degrees, right-hand screw rule. If a coordinate is left out, the previous value is used. Plus-sign may be left out.
I J K	Position of centre of circle, measured <u>relative to the current position</u>	Followed by + or – up to seven digits. Leave out only if 0.

CODE	PURPOSE	REMARKS
O	Program number	8 digits maximum Recommended: use the date + 2 digits
N	Line number	Up to 4 digits Has no effect on the program implementation, but is displayed on controller
D H	Diameter compensation register Length compensation register	Operator must have loaded the radius / length of the tool in the machine memory before starting the program. For lengths: relative to the tool used for setting the start point.
F	Feed rate	The number is the rate in mm/minute. Operator may change feed rate by a factor of 0 to 120%.
S	Spindle speed in rpm	Operator may change rate by a factor of 0 to 120%.
T	Tool number	
/	Optional line	Must be at the beginning of the line Line is skipped if " OPTIONAL BLOCK SKIP " on the control panel is switched on.
()	Comments	BEWARE: Sometimes gives problems in the middle of a program

APPENDIX C: ARRANGEMENTS FOR SUPERVISION AND MANAGEMENT OF THE NC MILLING MACHINE

The responsible artisan (Mr Graham Hamerse) manages day-to-day matters of the machines. All enquiries, complaints or suggestions must be addressed directly to him.

C.1 BOOKING PROCEDURE AND USAGE FEES

- Bookings for the use of the machine have to be arranged with the responsible artisan. Bookings are indicated on the white board in the NC Room.
- No user may rely on using the machine for more than three consecutive days per week. Only if no other jobs are waiting, a user may occupy the machine for more than three consecutive days.
- Conflicts about access and bookings should be referred to Prof Basson.
- If the machine is used for pre- and postgraduate training, pre- and postgraduate student projects/theses or bona fide research that is funded solely from university funds, the maintenance fees indicated below must be paid into cost point R812. For any private work or work that is done for external payment (including contract research), the second tariff must be paid into cost point R812.

FEES FOR USE OF NC-MACHINE	
TYPE OF USAGE	TARIFF PER HOUR THAT MACHINE IS OCCUPIED
Bona fide academic usage by departments of the Engineering Faculty (only maintenance and cutting tips for available tool holders)	From 1 Jan 2002: R20/hr From 1 Jan 2003: R22/hr From 1 Jan 2010: R30/hr From 1 Jan 2014: R30/hr if cutters are provided by job, else R45/hr if NC Machine's cutting tips are used
Bona fide academic usage by other departments of US (operating cost and cutting tips for available tool holders)	From 1 Jan 2002: R30/hr From 1 Jan 2003: R33/hr From 1 Jan 2010: R40/hjr From 1 Jan 2010: R55/hjr

- The machine operator must record the time (start and stop time) that a job used in the log book. Jobs that occupy the machine for more than one day, must be recorded separately for each day.
- The time recorded in the log book must normally include the set-up time and the machining time (i.e. when the machine is running and/or when the operator is working with the machine). Full tariff is applied to this time. Non-productive time when a job occupies the machine (e.g. while making program corrections), must be paid at 15% of the full tariff. Non-productive time is calculated as follows: (number of work days that the machine was occupied x 6 hr/day) minus (hours paid for as productive time).
- The time used for making program changes is not added in if no jobs are waiting for the machine.
- The responsible artisan must recover the costs from the users on a monthly basis.

- Whenever possible, jobs should be set up in such a way that more than one job can be accommodated at the same time. This is particularly important with jobs that require much program development and testing.

C.2. MAINTENANCE AND PURCHASES

- The responsible artisan must arrange for the purchase of lubricants and cutting fluids so that there are enough in store at all stages.
- The machine operator must ensure that the machine's compressed air supply, lubricant level and cutting fluid level are acceptable before and while jobs are being cut. If the machine will be running overnight, these aspects must be checked after 16h00 too.
- The responsible artisan must arrange for the renewing of the coolant at least once per term. If a lot of cutting in wood has been done, the coolant must be replaced more often.
- The responsible artisan must arrange for the purchase of cutting tools and must ensure that there are always cutters in a range of sizes available.
- Normally each job should pay for its own cutter, i.e. the job must pay for the replacement of the machine's cutters that were used. However, small jobs that use only a small part of a cutter's life, will not be charged for the cutter. Specialised cutters have to be bought by the job.

C.3 CLEANING

- The general rule is: leave the machine in a clean condition.
- NB: Compressed air may not be used to clean the machine. The coolant must be used to wash metal chips off and the vacuum cleaner for wood chips.
- The machine operator must ensure that all fixtures are removed from the machine bed after the job has been completed. This includes the large block and the 4th axis.
- The machine operator must ensure that the machine bed, spindle and cutting tools are washed clean, using the coolant, when the job is finished and at least once per day.
- The responsible artisan must ensure that the room is cleaned thoroughly once per week (e.g. every Friday between 15h30 and 16h30).
- The responsible artisan must ensure that the milling machine are cleaned once per day (e.g. every afternoon after 16h00).
- The machine operator must personally arrange for cleaning the machine when required at other times.

C.4. AUTHORIZATION TO OPERATE THE MACHINE

- Only persons who have undergone the safety training may operate the machine.
- No job may be started on the machine unless an authorised operator has checked the set-up, clamp configuration, spindle speed, feed rates and machine movements (at least one trial run must be completed).
- The following persons are currently authorised to operate the milling machine: Ferdi Zietsman, Anton van den Berg, Graham Hamerse and Anton Basson.
- Only Ferdi Zietsman or Anton Basson may authorise persons to operate the machine.
- Final year or post graduate students that must have repeated runs done, may be authorised to operate the machine just for their jobs, subject to all the other conditions stipulated in this section.

C.5. SAFETY ASPECTS

- The responsible artisan is the safety coordinator for the NC Machine room. He serves in this capacity on the Design & Mechatronics Division's safety committee.