STANDARD TEST PART FOR TENSILE TEST

HAMK
HÄMEEN AMMATTIKORKEAKOULU
HÄME UNIVERSITY OF APPLIED SCIENCES

Bachelor's thesis

Mechanical Engineering and Production technology

Riihimäki
Spring, 2017

Olorunmuyiwa Igunnuoda

HÄMEEN AMMATTIKORKEAKOULU
HÄME UNIVERSITY OF APPLIED SCIENCES

ABSTRACT

Mechanical Engineering and Production Technology
Riihimäki

Author	Olorunmuyiwa Igunnuoda	**Year** 2017
Subject	Standard test parts for tensile test	
Supervisor(s)	Väisänen Tapio	

ABSTRACT

The goal of this thesis was to make standard test parts for tensile test as the topic suggests. The task included among other things creating NC-programs which would be used in the manufacture of these parts on a CNC machine, programming the machine, machining the test parts, as well as the documentation related to this project. This thesis was commissioned by HAMK University of Applied Sciences and was chosen as a thesis topic by the author who is a sheer CNC, CAD and CAM enthusiast and hobbyist, and who is also planning to deepen his knowledge in that field.

The NC codes were generated both by a CAM software called surfCAM and manually for both the milled and turned parts respectively. The CAM software was used as opposed to manual programming for the milled part because it is faster, one can simulate the tool paths before loading the program to the machine and higher precision and speed. For the turned part, manual programming was used because of the simplicity of the part.

The results of this thesis project are milled and turned tensile test specimen parts which will be used for tests, along with the NC-program which could be used to produce more test parts if need be.

Keywords CNC machine, CAM, machining, NC

Pages 26 pages

List of abbreviations

CNC - Computer numerical controlled

CAM - Computer aided manufacturing

CAD - Computer aided design

NC – Numerical control

CONTENTS

1 INTRODUCTION

The need to carry out a tensile test was what brought about this thesis work. The tensile test is one of the most widely used tests in the field of engineering to determine the overall strength of a material. The test also reveals the yield point, ductility, toughness, strain-hardening capabilities, proof stress, elastic limit and so on. The test involves applying some amount of tensile load or force on both ends of the test piece and then gradually increasing the load or force till the material fractures.

The test specimens were made according to ISO specifications, although there are also other well-known standards. The specimens had rectangular and cylindrical cross sections, the former cross section could also have been made by other manufacturing methods such as abrasive water jet cutting, laser cutting and perhaps plasma cutting which would have left more kerf on the workpiece, likewise, laser cutting and waterjet cutting although less when compared to plasma cutting. So the aim was to machine both parts on a CNC machine as requested by the commissioning organisation of this thesis work.

1.1 Objectives

The objectives of this theses work were to create a NC programme that was to be used to manufacture standard tensile test pieces. The job entailed programming the CNC machine, determining the best possible and the most economical way of fixturing the parts on the machine, manufacture of the test parts, and a documentation of how the job was done so that if there is a need to make these parts in the future, the documents will be helpful there.

1.2 Dimensions of parts

As a requirement by the commissioning organisation, both parts were to have an overall length of 250mm. For the turned part, the canned cycle for the NC- code was to be that of a single line format, probably due to the machine available for use at that time.

2 THEORETICAL OVERVIEW

To review the theoretical studies about this work, the concept of metal cutting will be briefly delved into, most especially machining. In addition to that, definitions of terms as regards to the test pieces and other related items are given in this chapter.

2.1 Machining

Machining is one of the numerous manufacturing processes which consists of the removal of a material and modification of the workpiece surface after it has been produced by various methods (Kalpakijan & Schmid 2009, 553). Machining happens to be a very important process in modern day manufacturing particularly in cases where dimensional accuracy, smooth and shiny surface, sharp features, special surface finishes are desired. In instances where small batch sizes are needed, machining is the best manufacturing process to call upon. However, despite the merits inherent in this process, the drawbacks have always been that it wastes materials though in some instances very small, takes longer time than other manufacturing processes and it generally requires more energy, capital, and labour when compared to other processes.

“In some cases of machining, motion is given to the work piece and tools remains stationary, while in some other cases, the work piece is stationary and the machine tool provide motion to the cutting tool. in yet other cases, motion is given both to tool as well as the workpiece”. (Gupta, Gupta, & Mittal 2009, 91).

Machining is generally carried out with the aid of machines called machine tools examples of machine tools are the drill press, the lathe machine, milling machine, slotting machine and so on. Each machine tool is designed and suited for a particular application. Turning, boring, facing, threading, parting off, knurling, drilling, grooving, cutting with a form tool, taper turning operations and the likes are operations related to the lathe machine. Hole making or simply drilling are operations in which the drilling machine is known for. Gear cutting, face milling, profile milling, pocket milling and other numerous milling operations are carried out on a milling machine. The slotting machine on its part is simply used to make slots or keyways. More lights will be shed on the milling and lathe machine operations later in this chapter particularly as it relates directly to this thesis work.

Machine tools are referred to as either conventional (or manual) or CNC types. The conventional ones require the operator to use hand operated wheels or levers or engaged gear transmission to perform machining operation. They can usually perform only straight-line movements in one plane or direction at a time. While the CNC machine tools, on the other hand, are directed by computerised controls. They can produce intricate and complex shapes with extreme accuracy and efficiency. When properly configured, they can also perform many operations with many different types of cutting tool while running without the need of an operator. *Figures 1* and *2* illustrate a CNC machine centre and a manual lathe machine respectively.

Figure 1. CNC machine centre (Haas automation n.d.).

Figure 2. Manual lathe machine (XYZ machine tool n.d.).

2.1.1 **Turning**

Turning is the most basic operation of a lathe machine whereby cylindrical parts are made by feeding a single point tool to a rotating workpiece. “The tool travels along the outside of the workpiece to shave off material and produce cylindrical parts” (Hoffman, Hopewell, Janes & Sharp 2012, 355).

Figure 3 shows the numerous operations that can be carried out on a lathe machine.

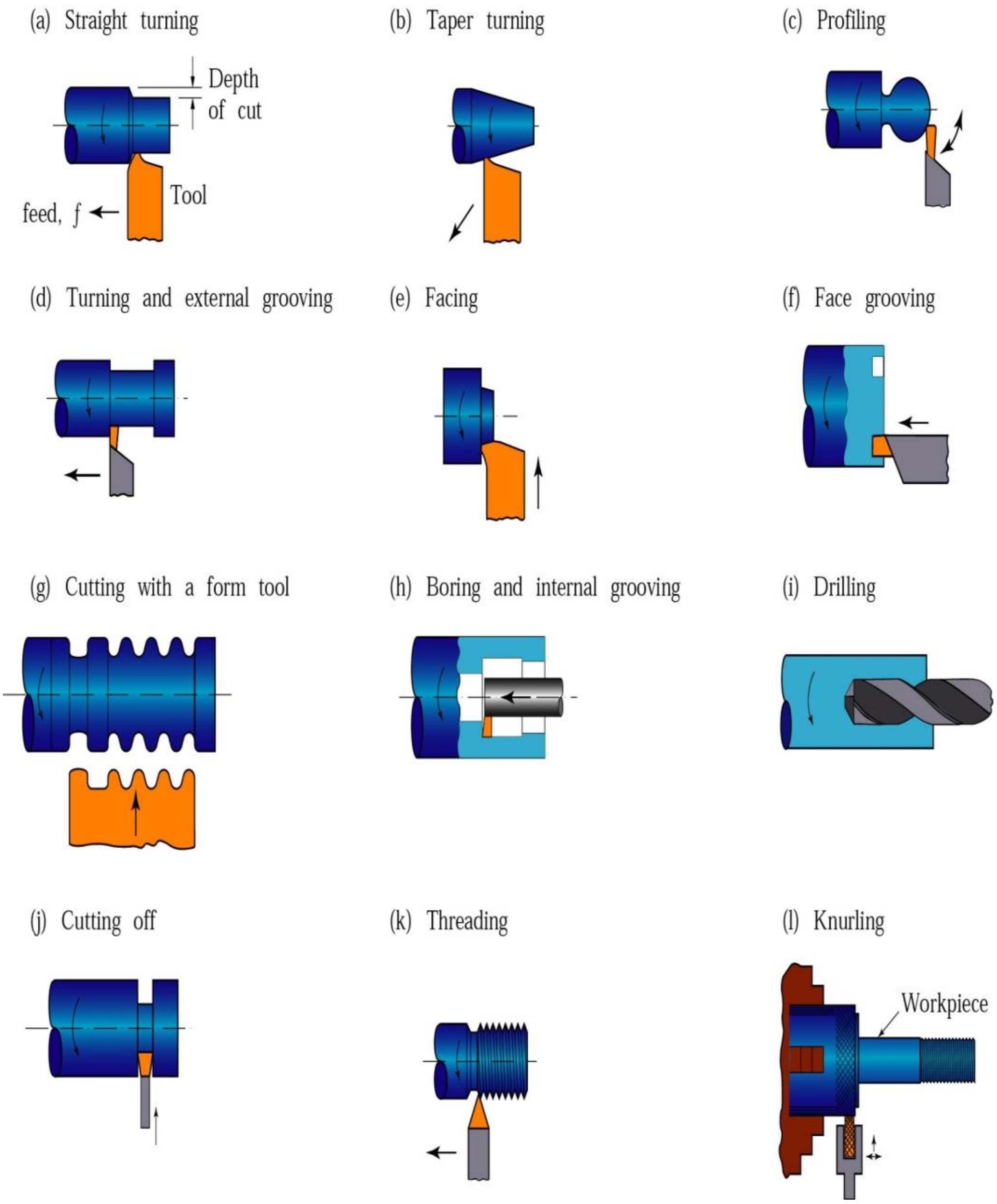

Figure 3. Lathe operations (Kalpakijan & Schmid 2009, 616).

2.1.2 Milling

Milling is a machining process used to machine flat and angled surfaces by feeding a workpiece into a rotating cutting tool to remove materials. (Hoffman et al. 2012, 451). The types of tools used therein are referred to as multipoint cutting tools. Milling machines are one of the essential machines in any modern-day machine shop.

Generally, there are two main types of milling process: The up milling or conventional milling process and down milling or climb milling process.

In conventional milling, the direction of rotation of the milling cutter and the direction of workpiece feed are opposite to each other; whereas, in climb milling, they move in the same direction at the point of contact of the cutter and the workpiece. In conventional milling, the cutting teeth tries to uproot and lift the workpiece from the machine table, while in climb milling reverse is the case (Gupta et al. 2009, 111). In up milling, the chip thickness at the start is nil and is maximum when the cutting teeth leave the surface of the workpiece, while in climb milling, the reverse is the case (Gupta et al. 2009, 111).

In practice, the up-milling process is commonly used. *Figure 4* describes the difference between conventional milling and climb milling process.

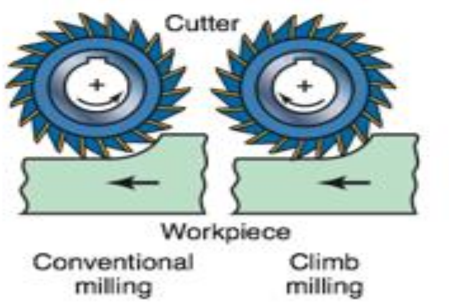

Figure 4. Milling process difference (Kalpakijan & Schmid 2009, 661).

2.2 **Cutting parameters**

The most important parameters in machining are: depth of cut, feed and cutting speed. These parameters play important roles as they determine the quality of the machined part, tool life, efficiency and so on. These three parameters are briefly explained in the following:

Depth of cut:
The depth of cut is the distance the tool is fed into the workpiece. It is the thickness of the layer of metal removed in a single cut. It is measured in millimetres.

Feed:

The feed is the time rate at which the tool is moved into the workpiece (for lathe machine) or at which the part is moved into the tool (milling machines). Feed is measured in millimetres per revolutions (mm/rev).

Cutting speed:

Cutting speed is the linear speed at which cutting takes place. The optimal cutting speed is dependent on the tool material, work piece material, and whether a cutting fluid is used or not.

Figure 5 illustrates some turning and milling formulas.

Cutting data formulae

Turning

$$\text{Cutting speed}, v_c = \frac{\pi \cdot D \cdot n}{1000} \quad (m/min)$$

$$\text{Spindle speed}, n = \frac{1000 \cdot v_c}{\pi \cdot D} \quad (rev/min)$$

$$\text{Material removal rate}, Q = v_c \cdot a_p \cdot f \quad (cm^3/min)$$

$$\text{Surface roughness}, R_a \approx \frac{f^2 \cdot 50}{r_\varepsilon} \quad (\mu m)$$

Legend

v_c = Cutting speed (m/min)
n = Spindle speed (rev/min)
f = Feed per rev (mm/rev)
a_p = Axial depth of cut (mm)
D = Workpiece diameter (mm)
Q = Material removal rate (cm^3/min)
R_a = Surface roughness (µm)
r_ε = Nose radius (mm)

Milling

$$v_c = \frac{\pi \cdot D \cdot n}{1000} (m/\text{min})$$

$$n = \frac{1000 \cdot vc}{\pi \cdot D} (\text{rev/min})$$

$$vf = fz \cdot z \cdot n = f \cdot n (\text{mm/min})$$

$$h_m = fz \cdot \sqrt{\frac{ae}{D}} (\text{mm}) \; \frac{ae}{D} < 0{,}3$$

$$Q = \frac{ap \cdot ae \cdot vf}{1000} (\text{cm}^3/\text{min})$$

Legend

v_c = Cutting speed (m/min)
n = Spindle speed (rev/min)
v_f = Feed speed (mm/min)
a_p = Axial depth of cut (mm)
a_e = Radial depth of cut (mm)
f = Feed per rev (mm/rev)
z = Number of teeth
f_z = Feed per tooth (mm/tooth)
D = Cutter diameter (mm)
h_m = Average chip thickness (mm)
Q = Material removal rate (cm^3/min)

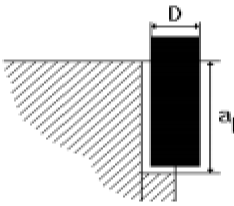

Figure 5. Formulas (Uddeholm 2007).

With these formulas and some tool manufacturers' recommendations, the optimal cutting parameters can be determined.

2.3 Work holding

Work holding devices simply hold or secure the workpiece to facilitate ease of machining. These include clamps, vices, chucks, fixtures and so on. How a part is fixed to a machine tool is very important as it affects the number of set-ups and hence, time and cost. From the economic point of view, one of the best and the easiest way to minimize costs as far as a machine shop is concerned is to minimise the number of set-ups.

The tool path, the allowable cutting forces, the tool shape and size, the overall accuracy of the machining process, and some other important factors determine how to secure or hold a part for machining (Sme n.d. 1).

2.3.1 Lathe work holding

“On a lathe machine, the workpiece rotates instead of the cutting tool, so the work holding device must transmit torque to rotate the workpiece and withstand the centrifugal force developed by the rotation” (Sme n.d. 3).

Chucks are the primary work holding device on the lathe machine. Other work holders on a lathe are collets and mandrels.

2.3.2 Milling work holding

On a milling machine, the tool rotates contrary to that of a lathe machine. As the tool rotates, pressure is generated, which is quite high when compared to the single point tool used on a lathe machine. Therefore, the workpiece should be rigidly fixed to avoid vibration.

The number and variety of work holding and clamping devices on a milling machine is quite extensive. Among them are vices which are the most basic devices, the vee block, angle plates, step clamps, toggle clamps, F-clamps, G-clamps, and so on. Below are a brief description of the above mentioned milling work holding devices.

Vice
Vice are the most basic and common work holding device on a milling machine. Loading and unloading of workpieces on a milling machine is arguably faster on a vice than most other work holding devices.

Vee-block
Vee-block are work holding device used chiefly for holding cylindrical workpiece on a milling machine. They are used when milling a keyway or when a flat is to be machined on a shaft.

Angle plate

The angle plate is a very useful tool in the workshop as it serves as a work holding device, serves as a stopper and as well as an alignment tool. It is a simple device with two flat surfaces at right angles to each other.

Figure 6 shows an angle plate with a vee block.

Figure 6. Angle plate and vee block (Qhunt media 2016)

Step clamp

A step clamp assembly usually contains Tee nut, stud, clamp, riser block and clamping nut. In all milling machines, there are tee-slots along the entire length of the milling table into which these assembly are fastened to. A typical step clamp set is shown in *Figure 7*.

Figure 7. Step clamp set (Hoffman, Hopewell, Janes and sharp 2002, 474).

Toggle clamps

The toggle clamps are mostly designed to clamp a workpiece very quickly and easily by hand without the need of any other tool. Most toggle clamps manufacturers specify on those clamps the permissible clamping force. *Figure 8* shows a typical toggle clamp set up.

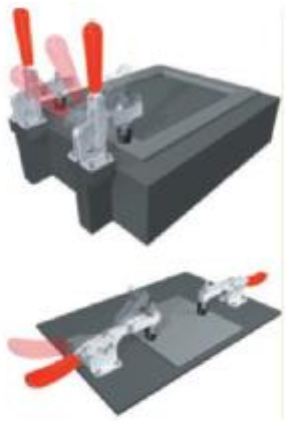

Figure 8. Toggle clamp set up (Hoffman, Hopewell, Janes and sharp 2002, 475).

F-clamps

As the name implies, they are F-shaped clamps used to fasten parts to the milling table. It has two jaws which are used for clamping. one is fixed, and the other is adjustable with the aid of a screw bar. These types of clamps are available in various sizes, and the clamping force is dependent on the size of the clamp. The bigger the clamp; the bigger the clamping force and vice versa. *Figure 9* below shows a picture of a 3-D Modelled F clamp.

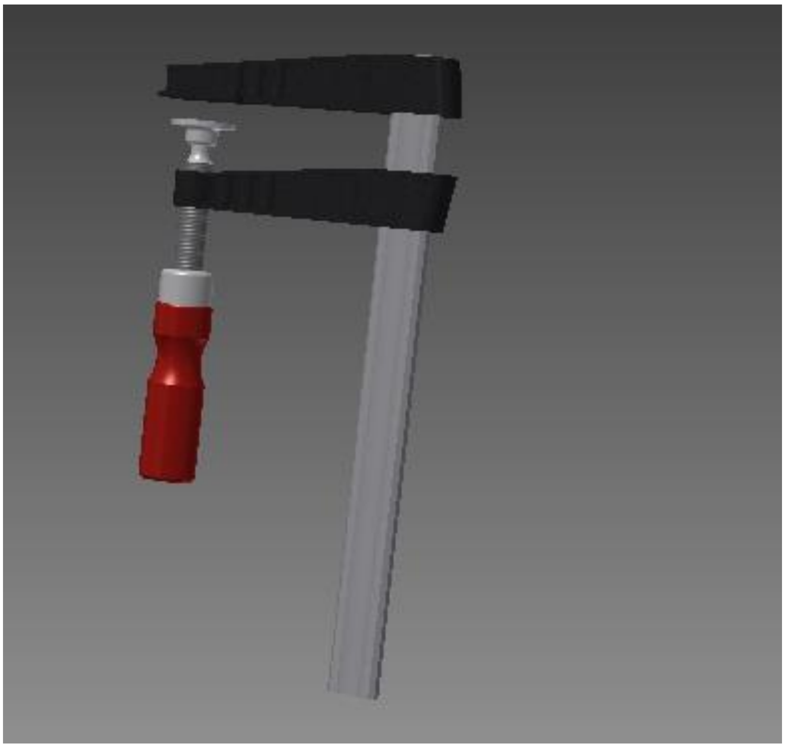

Figure 9. F clamp

G-clamps

The G-clamps which are sometimes called C-clamps are G-like or C-like shaped clamps used to fasten parts to the milling table. They have similar application to the F-clamps mentioned above. A picture of a 3-D modelled G-clamp is shown in *Figure 10.*

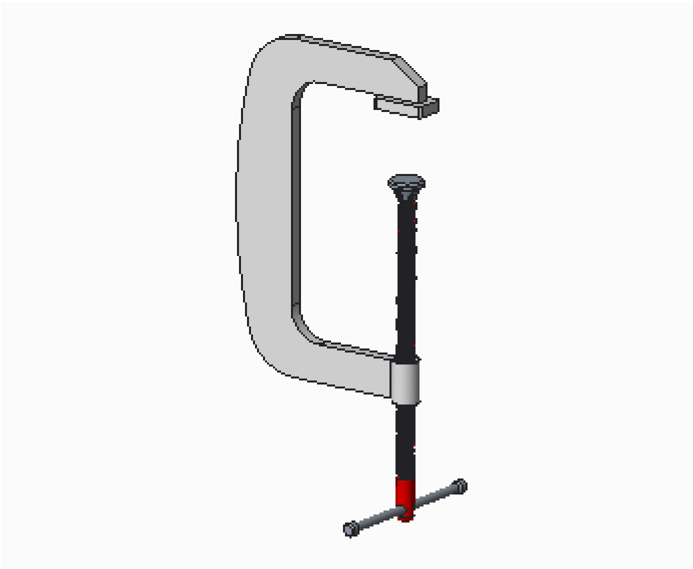

Figure 10. G clamp

2.4 Test part geometry

This chapter explains the shape and dimensions of a standard tensile test piece as defined by the ISO standards.

Figure 11 shows a description of the terms for a flat test piece.

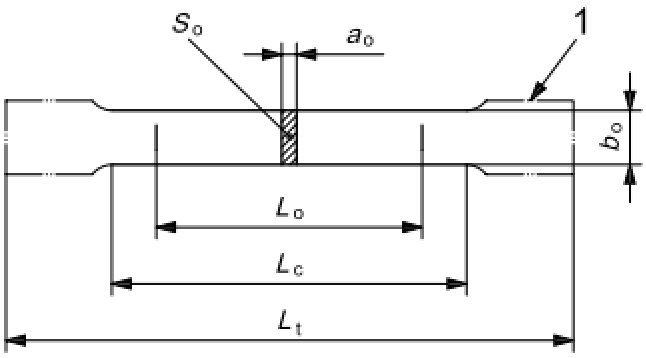

Figure 11. Definition of terms (ISO 6892-1/2009, 28.)

Overall length L_t:
The overall length is simply the overall length of the test piece as the name implies.

Cross-sectional area S_o:
This is the cross-sectional area of the parallel length. And it is equal to the product of b_o and a_o for a rectangular test piece and $\pi d^2/4$ for the cylindrical test piece, where d is the diameter of the parallel length. a_o is the thickness and b_o is the width of the parallel length for the flat test piece.

Gauge length L_o:
The gauge length as defined according to ISO standard as the length of the parallel section of the test piece in which elongation is measured at any moment during the test. ISO suggest that the gauge length as a rule should be according to the equation below.

$L_o = k\sqrt{S_o}$………………………………………………… (equation 1)

where the value of k is 5.56

Parallel length L_c:
Parallel length is defined according to ISO standard as the length of the parallel reduced section of the test piece. ISO recommends that the parallel length should be at least

$L_o + (d_o/2)$ for cylindrical test piece……………. (equation 2)

And

$L_o + 1.5\sqrt{S_o}$ for rectangular test piece…………. (equation 3)

Transition radius:
The transition radius is the radius between the gripped ends and the parallel length. According to ISO specification, the minimum transition radius for a flat test piece should be 12mm, while the minimum transition radius for a cylindrical test piece is given as $0.75b_0$.

3 TURNED PART

The turned part of this project was programmed manually owing chiefly to the fact that the part is quite simple and efforts to program it by a CAM software did not generate the desired tool paths. Before the advent of the CAD and CAM software, manual programming was the norm, and it is still of much importance as CAM generated codes can easily be edited and adjusted on machines if need be, but one of the drawbacks there is that it is time-consuming and might be an uphill task to program longer programs.

The blank size for this part was 250mm in length with a diameter of 20mm. The part was pre-spot drilled at one end to enable it to be fixed to the live centre on the tail stock. The spot drill dimension is not critical; it should be something reasonable enough for the tailstock to hold or support. The part with its dimensions is given in *Figure 12.*

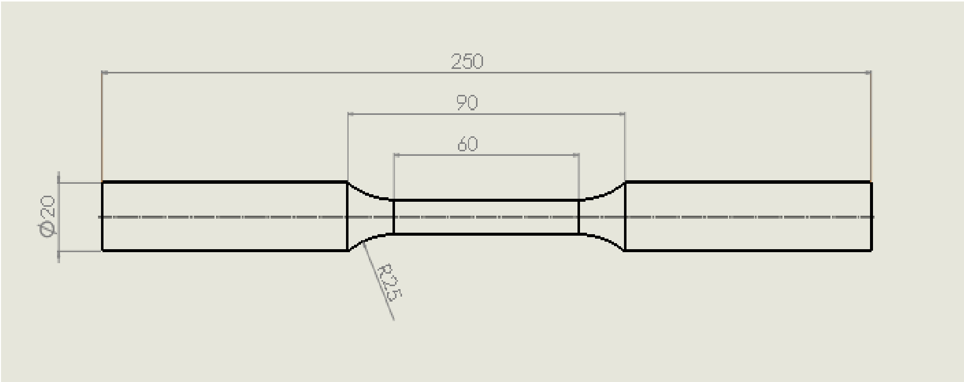

Figure 12. Cylindrical test piece with dimensions

How the part is clamped and set-up is shown schematically in *Figure 13*.

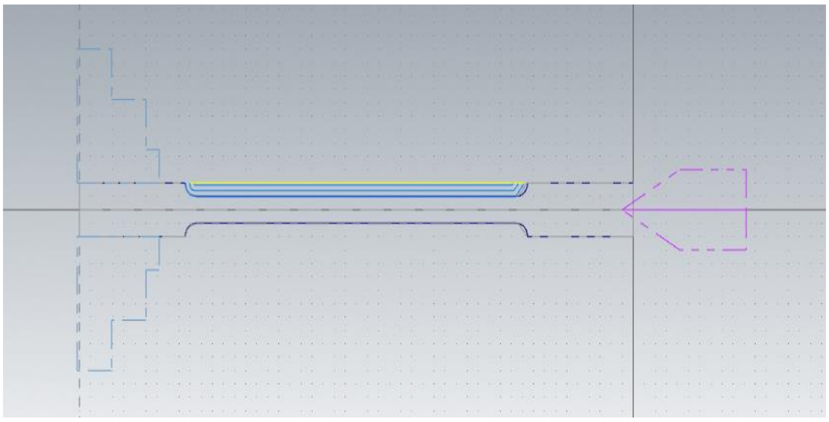

Figure 13. Lathe set up with tool path

3.1 Selection and parameters of cutting tool

Tool selection can sometimes prove to be a daunting task especially for those who are just cutting their teeth in the trade, but today, most tool manufacturers provide hints and clues on how to select the best tool, inserts and tool holders for a particular job.

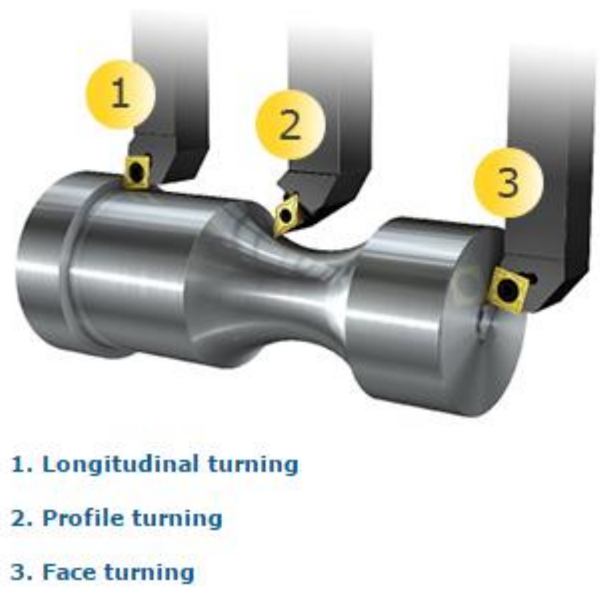

Figure 14. Turning tool selection (Sandvik, n.d.)

Figure 14 illustrates a clue from Sandvik coromant. The cutting tool insert most suitable for this job is a 35° degree diamond insert which is shown in *Figure 15*.

Figure 15. 35 degree turning insert (Sandvik, n.d.)

It is also worth mentioning that different inserts have different tool holders so therefore the appropriate insert holder has to be used.

And now to the cutting parameters. Cutting parameters play a very vital role in the machining process as they affect the surface finish, cutting force, tool life and so on. In tools manufacturers' catalogues, recommended cutting speeds are given. In cases where one cannot lay his or her hands on a manufacturer's catalogue, an educated guess might do, though this is not always advisable.

From the formula for lathe operation in *Figure 5,* the cutting speed was calculated as follows.

$$\frac{3.142 \times 20 \times 3000}{1000} = 188.5\text{m/min}$$

3.2 Results

This chapter will consist basically of the NC code that was used to produce the cylindrical part and the picture of the machined part.

The G-code for the cylindrical part is as follows:

```
%

O1234 (lathe program)

G54

G00 X100. Z-20.

T0606

G50 S3000

G96 S150 M04

G00 Z-80.

X20.5

G71 P10 Q20 U0.2 W0.2 D800 F0.15

N10 G01 X20. Z-80.

G02 X10. Z-95. R25.
```

```
G01 Z-155.

N20 G02 X20. Z-170. R25.

G00 X22.

G42 Z-80.

G70 P10 Q20 F0.15

G00 G40 X50.

M30
```

%

The machined part is shown in the *Figure 16*

Figure 16. Machined cylindrical part

4 MILLED PART

The G-code for the rectangular milled part was made with a CAM software called surfCAM. The blank size was 250mm by 45mm of which 2.5mm was milled away from it on both sides so that the final width became 40mm. The dimensions of the final part are given in *Figure 17*.

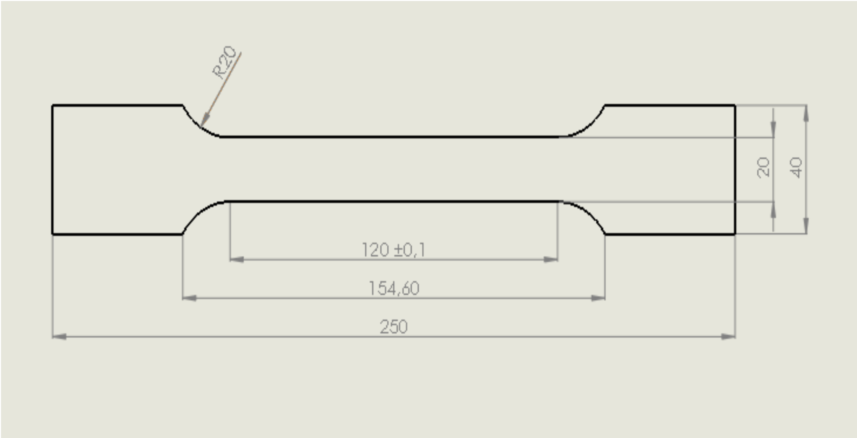

Figure 17. Flat test piece with dimensions

4.1 CAM programming and tool path

As the case with most CAM software, the part to be programmed is first modelled in 3-D modelling software and then saved in a compatible format such as IGES. Same was the case here as well; the part was modelled in CREO parametric and then imported to surfCAM.

4.1.1 Work zero

The work zero used in programming is a 2.5mm offset on the negative Y-axis from the edge of the blank as shown in Figures 18 and 19.

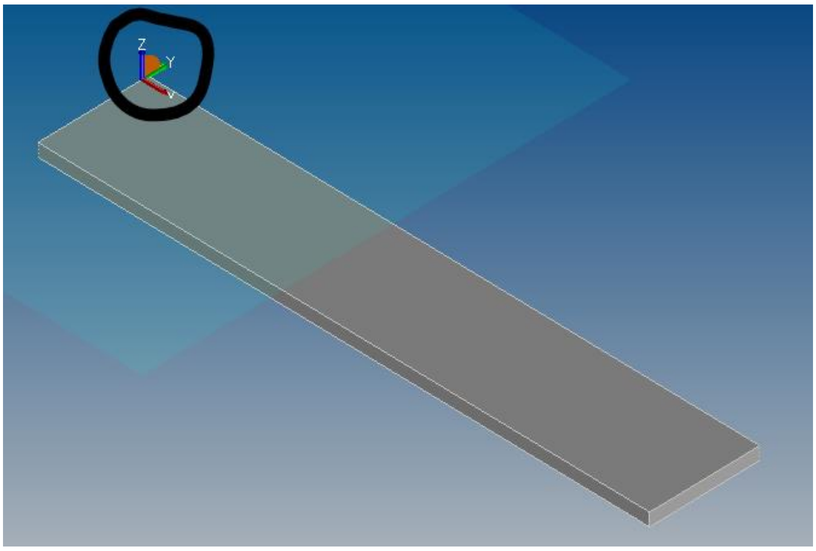

Figure 18. Work zero

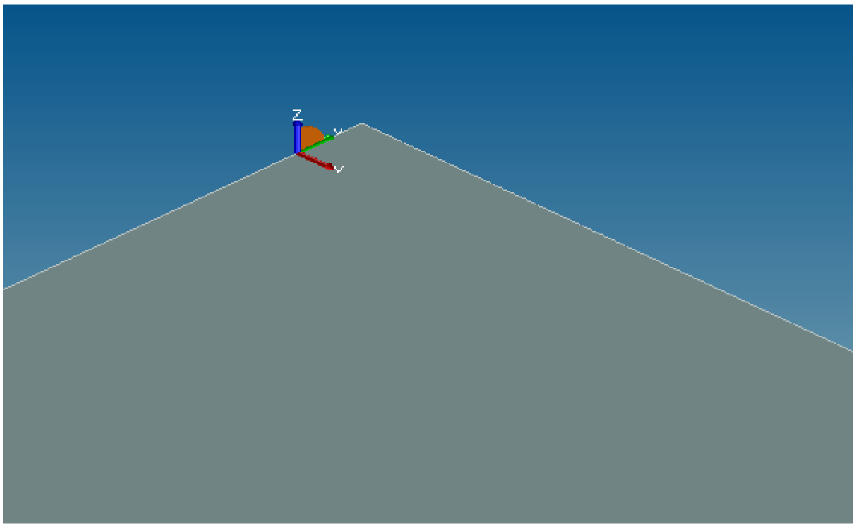

Figure 19. Zoomed work zero

4.1.2 Tool path

The tool path is shown in *Figure 20*, with the green line representing the cut profile or feed move, while the red line represents rapid move.

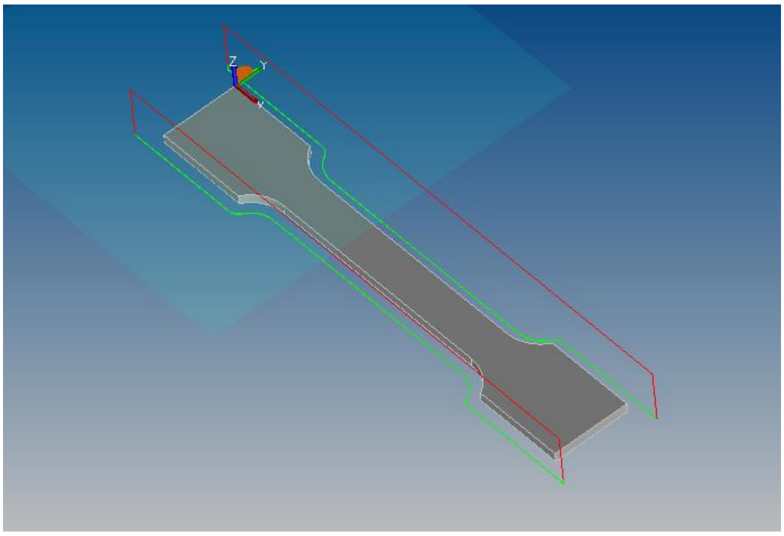

Figure 20. Tool path

From the tool path in Figure 20, it is noticeable that the feed moves start from some distance away from the material and also end at some distance after the cut, this is due to the lead-in and lead-out programmed in it which is shown in *Figure 21*.

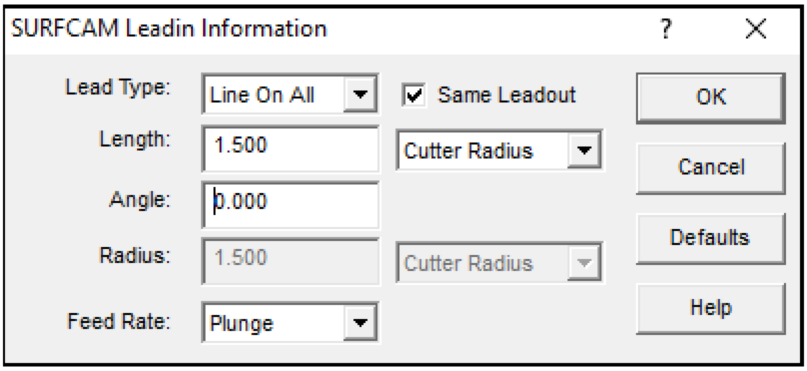

Figure 21. Lead-in and lead-out

4.1.3 Simulation

One of the great advantages of the CAM software is the ability to simulate the tool path to show how the tool will travel in a real situation, through this, costly mistakes that might arise by manual programming such as tool breakage and damage to machine can be reduced to some extent. Another interesting advantage with it is that it eliminates wastage as a result of error in the handwritten code. *Figure 22* shows a part simulation in surfCAM.

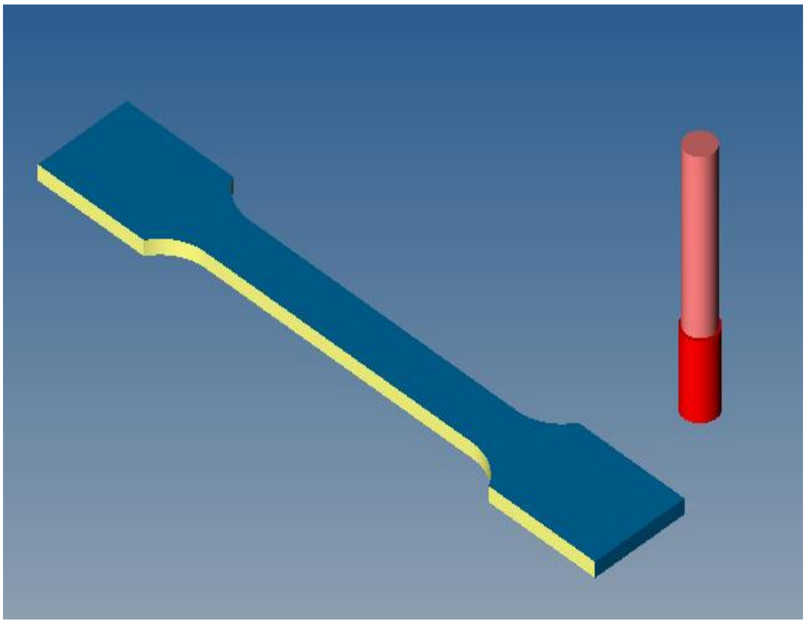

Figure 22. Simulation

4.2 **Fixturing**

The easiest way to clamp the part could have been just to use a milling vice, but unfortunately, the thickness of the parts in the project did not permit that, as the required clamping force would have bent the material even before machining. Considering this, a step clamp, a F- clamp, a G-clamp, or even a toggle clamp would be okay provided the width of the clamping area does not exceed 38mm, and provided that there is enough room for the tool to pass at the sides of the part without colliding with the clamp. For this thesis work, the step clamp was used. The final set up before machining is shown in *Figure 23*.

Figure 23. Final set-up before machining

An advantage of step clamp was that up to five or more parts could be machined at once in just one set up thereby reducing the machining time, set-up time and ultimately cost. What is needed to be done in this regard is just to change the negative Z-depth in the NC-program to the desired one depending on the number of parts and the thickness of the blank, and not forgetting to put the flute length of the tool into consideration. However, the drawback inherent in this case was that the work zero needed to be confirmed repeatedly.

As mentioned earlier, the width of the fixture should not exceed 38mm, the width of the clamp used here was 35mm so that there was roughly about 2.5mm between the milled surface and fixture from the sides.

The *Figure 24* shows the set up after machining

Figure 24. Set-up after machining

4.3 Cutting tool and parameters

For the milled part profile, any cutting tool with a radius less than 20mm would have been okay, but a 14mm radius end mill was used in this project as seen in *Figure 25*.

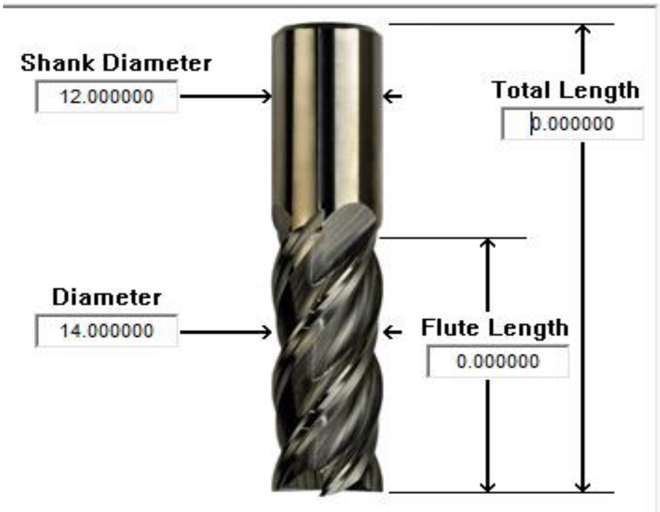

Figure 25. 14mm End mill

The total length and flute length were left as zero as they could be anything reasonable. It is worth mentioning again that the diameter 14 mm end mill should be strictly used as tool compensations were not used in the generated program.

The software recommended the cutting parameters.

The cutting parameter used were:

Spindle speed = 700 rev/min

Cutting speed from the formula in *Figure 5* was given as

$$\frac{3.142 \times 14 \times 700}{1000} = 30.79\text{m/min}$$

Feed rate 0.5 mm/rev

Figure 26 shows some more detailed cutting parameters as recommended by the software.

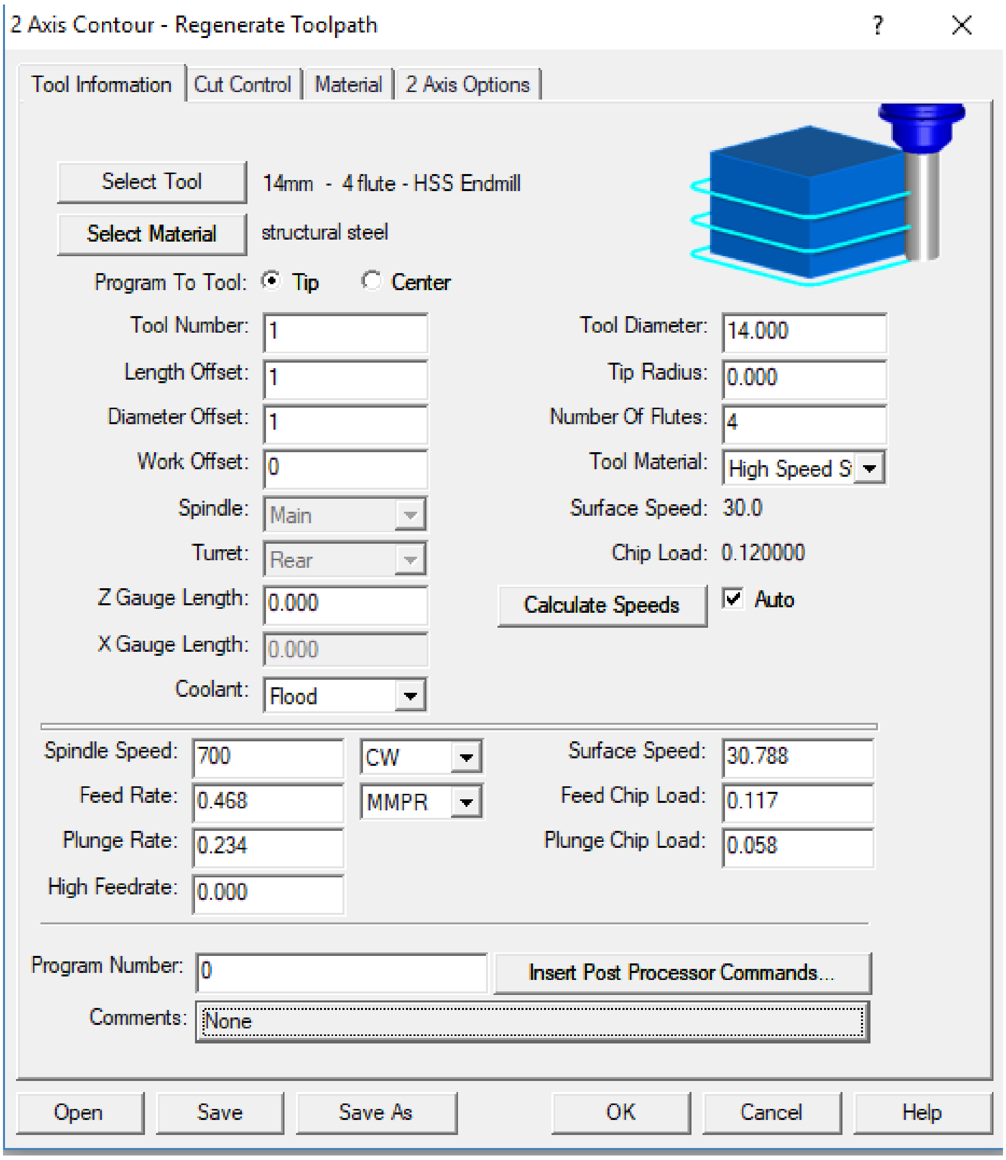

Figure 26. Tool information

4.4 **Results**

This chapter will consist basically the NC code generated by the software and a picture of the machined part.

The G-code is given as follows.

```
%
1001 (milling)
G17 G40 G80
G91 G28 Z0
G40 G28 X0
T01 M06
G90 G00 G54 G43 X250. Y-57.5 Z25. H01 S700 M03
M08
G00 Z2.5
G01 Z-6.5 F300
Y-47.
X202.3 F300
G02 X196.238 Y-43.5 I0 J7.
G03 X184.98 Y-37. I-11.258 J-6.5
G01 X65.02
G03 X53.762 Y-43.5 I0 J-13.
```

```
G02 X47.7 Y-47. I-6.062 J3.5

G01 X0

G00 Z50.

Y17.5

Z2.5

G01 Z-5. F300

Y7.

X47.7 F300

G02 X53.762 Y3.5 I0 J-7.

G03 X65.02 Y-3. I11.258 J6.5

G01 X184.98

G03 X196.238 Y3.5 I0 J13.

G02 X202.3 Y7. I6.062 J-3.5

G01 X250.

G00 Z25.

G91 G28 Z0

G90

M30

%
```

The machined parts are shown in *Figure 27.*

Figure 27. Final machined part

5 Conclusion

The G-code for the tensile test parts was made, a good way of fixturing the parts was devised, the CNC machine was programmed and the tensile test parts were made, which by all indications testify to the fact that the thesis project was completely accomplished. However, efforts should be made to come up with a better clamping method as regards the milled part, so that there would be no need for setting the zero position over and over again for each set-up or batch. In addition to the aforesaid, it is worth getting a tool manufacturer's catalogue and adhere to the recommended cutting parameters in order to achieve optimal cutting results.

This thesis work has been an eye opener and a learning curve for the author who happens to have minimal experience as far as numerical controlled machines are concerned, although he possesses some experience with manual machine tools. This draws attention to the fact that it is not just about programming, neither is it about just getting the right tool path nor just writing the G-code, there is actually more to it, ranging from the selection of the correct cutting parameters, work settings on the machine, fixturing, and so on. But all in all, this thesis project has increased my knowledge as far as CNC machining, CAM programming(surfCAM), work planning, and general workshop duties are concerned.

References

Dosarathi, G. (2015). *CNC etc*. Retrieved 14 April 2017 from http://cadem.com/cncetc/cnc-turning-cutting-parameters-selection/

Gupta, H., Gupta, R., & Mittal, A. (2009). *Manufacturing process second edition.* New Delhi: new age international(P) Limited publisher.

HAAS Automation. (n.d.). Retrieved 4 June 2017 from http://int.haascnc.com/vmc_mt.asp?intLanguageCode=1033&webID=50_TAPER_STD_VMC

Hoffman, P., Hopewell, E., Janes, B., & Sharp, K. (2002). *Precision manufacturing technology.* Delmar: Cengage learning.

ISO 6892-1 (2009). *Metallic testing - Tensile testing.* Geneva: International Standard Organisation.

Kalpakijan, S. & Schmid, S. (2009). *Manufacturing engineering technology sixth edition.* Washington: Pearson

Qhunt media. (2016). *what are the work holding devices used on milling machine*. n.d. Retrieved 17 July 2017 from http://www.qhunt.com/2016/01/what-are-the-work-holding-devices-used-on-milling-machine.html

Räisänen, S. (2015). *Semi-automated turning simulation production using catia V5*. Bachelor's thesis. Mechanical and production technology. Saimaa university of applied sciences.

Sandvik Coromant. (n.d.). Retrieved from 14 May 2017 http://www.sandvik.coromant.com/en-gb/knowledge/general_turning/choice-of-application/external-turning/pages/default.aspx

SME. (n.d.). Fundamental manufacturing process study guide. Retrieved 17 July 2017 from https://www.sme.org/WorkArea/DownloadAsset.aspx?id=73767

Uddeholm Corrax. (2007). Cutting data recommendations. Retrieved from 4 June 2017 http://www.uddeholm.com/files/Cutting_Data_Corrax_eng.pdf

Väisänen, T. (2017). Online learning material for Manufacturing module, Moodle. Häme University of Applied Sciences.Retrieved 10 Feb 2017 from https://moodle.hamk.fi

Väisänen, T. (2017). Online learning material for Workshop automation moodle, Moodle. Häme University of Applied Sciences. Retrieved 17 Feb 2017 from https://moodle.hamk.fi

XYZ machine tools. (n.d.). Retrieved 4 June 2017 from http://www.xyzmachinetools.com/machines/manual-machines/