National Technical University of Ukraine
«Igor Sikorsky Kyiv Polytechnic Institute»
EDUCATIONAL AND RESEARCH
INSTITUTE OF MECHANICAL ENGINEERING
Department of Manufacturing Engineering

«On the rights of manuscript»
УДК _ 658.512___

«**Admitted to the defense**»
Head of the department

____Oleksandr OKHRIMENKO

“___”_____________2024

Diploma project

for a bachelor's degree
according to the educational and professional program
«Manufacturing Engineering»
in the speciality 131 «Applied mechanics»

on the topic: Manufacturing Process Planning for a part " Double bearing housings "

Developed by:
the student of 4th year of study, group MT-03

Assifi Kasem Abdul Kareem Kasem________________________ __________
(full name) (signature)

Academic supervisor Phd, associate professor Danylova Liudmyla
(position, academic degree, academic title, surname and initials) (signature)

Reviewer Phd, associate professor Krasnovyd Dmytro ________
(position, academic degree, academic title, surname and initials) (signature)

I certify that in this diploma project there are no borrowings from the works of other authors without appropriate references.
Student _____________
(signature)

Kyiv – 2024

National Technical University of Ukraine
“Igor Sikorsky Kyiv Polytechnic Institute”
Educational and Research
Institute of Mechanical Engineering
Department of Manufacturing Engineering

Level of higher education – first (bachelor)
Specialty – 131 “Applied Mechanics”
Educational and Professional Program “Manufacturing Engineering”

APPROVED
Head of the department
_______ Oleksandr OKHRIMENKO
«___»____________20__ p.

ASSIGNMENT

for the student's diploma project

Assifi Kasem Abdul Kareem Kasem

1.Topic of the project: Manufacturing Process Planning for a part " Double bearing housings "

Project supervisor: Phd, associate professor Danylova Liudmyla

approved by the University Order «_»___05___ 2024 №

2.Deadline for submission of the project « 10» June 2024

3. Initial data for the project Part drawing and material, Production quantity per annum, working conditions of the part in the assembly unit.

4. Content of the text part (explanatory note): Design of the operational manufacturing process plan, calculation of the allowance, fixture design, economic calculation. Methods for engaging movements during milling.

5.List of the graphic material (indicating mandatory drawings, posters, presentations,etc.) 3-D drawing of the part and drawing of the workpiece.

Schematic representation of a technological operation. Assembly drawing of the machine tool. Overview of Machining and Milling Process.

6. Consultants for chapters of the project

Chapter	Surname, initials, and position of consultant	Signature, date	
		Issued the task	Accepted the task
Many manufacturing process plan	ass.prof. Danylova L.M.		
General questions of mechanical engineering	ass.prof. Danylova L.M.		

7. Issue date of the assignment ____«20 » April 2024______________

CALENDAR PLAN

No	The stage of the diploma project execution	The deadline for the stages of the diploma project	Notes
1	Analysis of design features of the part	20.04.24	completed
2	Determining the type of production	20.04.24	completed
3	Calculation of the allowance	30.04.24	completed
4	Design of the typical surfaces processing routes	10.04.24	
5	Design of the operational manufacturing process plan	15.05.24	
6	Setting cutting conditions	20.05.24	
7	Development of the fixture design	30.05.24	
8.	Calculation of the cost processing	30.05.24	
9	Study of the methods for engaging movements during milling.	10.06.24	

Signature:

Student Assifi Kasem Abdul Kareem Kasem

Supervisor Danylova Liudmyla

ABSTRACT

Course project on the topic: Manufacturing Process Planning for a part " Double bearing housings "

The project consists of text and graphics.

The graphic part is made on 4 sheets of A1 format and includes: 3-D models and drawings of parts and blanks, schemes of technological transitions of operation 005 of the technological process of manufacturing parts "linear bearing housing" and assembly drawings of devices for operations 005 and 010.

The explanatory note includes two sections: technological and design.

The explanatory note also includes:

- drawings - 35;
- tables - 11.

At the end of the course work is a list of references and appendices, including operational technological process and sketch maps.his paper presents an in-depth study of machining and milling processes, focusing on the essential principles and methodologies that underpin these critical manufacturing techniques. Starting with an overview, the paper explores different machining methods and the functions of milling machines, including the important movements between the tool and the workpiece and the role of lubrication in these processes.

Keywordsords: manufacturing, operational technological process, milling.

[redacted]

Table of Contents

CHAPTER 1: Study of the methods for engaging movements during milling. 1
1. Overview on Machining and Milling 1
1.1. Introduction.. 1
1.2. Machining 1
1.3. Machining Process 2
1.4. Machining Methods 3
1.5. Milling 4
1.6. Milling Machines 12
1.7. Relative Movements between the Tool and the Workpiece 13
1.8. Lubrication 14
2. Fundamentals of Metal Cutting 16
2.1. Principle of Metal Cutting 16
2.2. Cutting parameters 18
2.2.1. Cutting Speed Vc 20
2.2.2. Feed rate (Vf) and Feed per Revolution (f) 21
2.2.3. Depth of cut: 23
2.3. Formation and types of chips 23
2.4. Concept of Surface Finish 25
3. Milling operations on Siemens NX 29
3.1. Open Area 29
3.2. Closed Area: 34
CHAPTER 2: Manufacturing Process Plan 36
2.1. General analysis of the part 36
2.1.1. Geometry analysis 36

2.1.2. Part’s working condition in the assembly 37

2.1.3. Material analysis 37

2.1.4. Type of production determination 37

2.2. Selection of the base process and Blank design 39

2.2.1. Selection of the base process 39

2.2.2. Casting Tolerance and Required Machining Allowance calculations 40

2.3. Locating Scheme selection 43

2.3.1. Choosing General Manufacturing Datum (GMD) 43

2.3.2. Choosing MD for the first manufacturing operation 45

2.4. Design of the typical surfaces processing routes 47
2.5. Design of the operational manufacturing process plan 50
2.6 Machine and Tooling Selection 52

2.6.1. Machine Selection 52

2.6.2. Tooling Selection and Cutting Conditions 53

2.7. Cutting Conditions 56
2.8. Simulation and G-Code Generation 56

CHAPTER 3: Fixture Design 62

2.1. Purpose of the Fixture 62
2.2. Possible Fixture designs and solutions 62
2.3. Clamping Force calculation 64

2.3.1. Vise Fixture 64

CHAPTER 4: Economical Section 66

1. Cost Estimation 66
2. Time Estimation 67

REFERENCES 69

ANNEX 70

CHAPTER 1: Study of the methods for engaging movements during milling.

1. Overview on Machining and Milling

1.1. Introduction

Machining refers to any shaping operation involving material removal using a machine tool designed to give a part specific dimensions and surface conditions (shape deviation and roughness) within a given tolerance range. From an economic standpoint, the industrial machining sector is quite significant, producing about 2.5% of the gross national product in a developed country

Machining primarily concerns metallic materials, with most common metal objects having already undergone one or more shaping operations. Shaping by machining also applies, though generally to a lesser extent, to all other classes of materials (ceramics, polymers, wood and derived materials, composites, glass, semiconductors, etc.), according to specific methods depending on the characteristics of the process and the material.

During the machining of a part, material removal is achieved by the combination of two relative movements between the part and the tool: the cutting movement and the feed movement. There are two ways to generate the desired surface: either by form work or by envelope work. In form work, the shape of the tool determines the final surface obtained. In envelope work, it is the trace of the tool's edge (the generating point) that produces the final surface.

Today, numerically controlled machines, i.e., equipped with a computer system, allow for partial or complete automation of the procedure.

1.2. Machining

The methods for shaping materials through material removal have continuously evolved to meet industrial requirements, whether economic, ecological, or otherwise. Today, a manufacturing engineer must be able to answer numerous questions to produce mechanical parts quickly, with the required quality, and at the lowest cost.

For example, in the context of machining, some of the questions that arise include:

- What type of machine should be used, and will it suffice in terms of power and precision?
- What cutting conditions should be employed to minimize tool or material damage?
- What solutions should be adopted in the design of tools, and what materials should they be made from to improve their lifespan and/or the surface quality of the machined parts?
- What are the mechanical properties of the part after machining?
- Is it possible to machine without using lubricants?

There are many more questions that could be asked, making it difficult to compile an exhaustive list. Fundamental knowledge does not provide answers to all these questions. However, machining technology and techniques have evolved and optimized production.

Since the industrial revolution at the beginning of the last century, machine tools have significantly evolved to meet demands such as better efficiency, greater safety, increased rigidity, higher travel speeds, increased power, improved productivity, and reduced tool wear.

In turn, the enhancement of these performances has revealed a set of phenomena that were not critical in traditional manufacturing but are crucial under high-speed machining conditions, where the rotation speeds of tools, such as milling cutters, become very high.

1.3. Machining Process

Machining is a complex process involving several components:

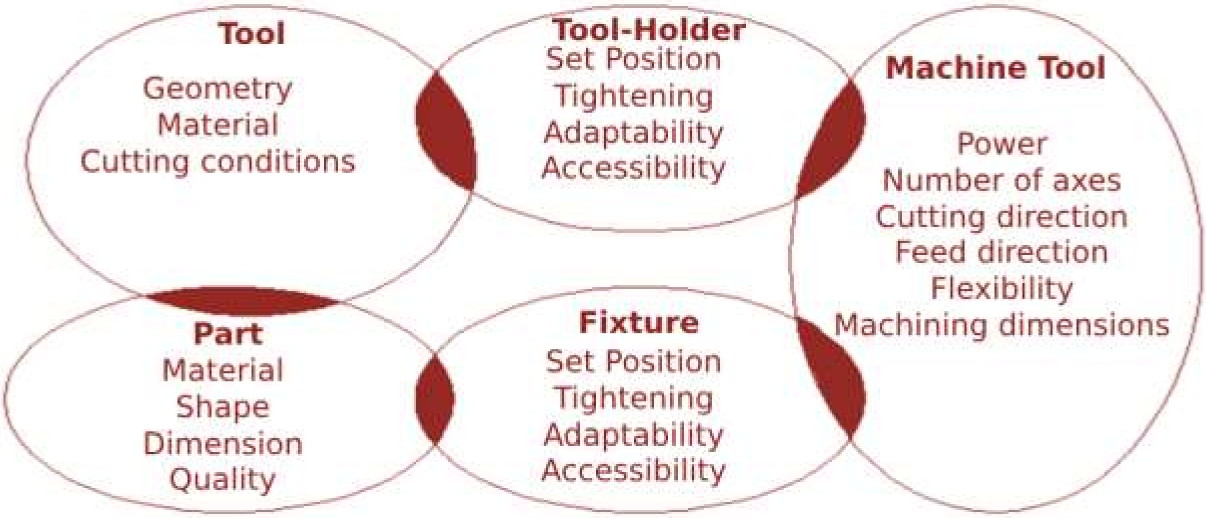


Fig.1. Definition of the POM system: Workpiece/Cutting Tool/Machine Tool

There are connections:

- Between the workpiece and the workpiece holder,
- Between the workpiece and the tool,
- Between the tool and the tool holder,
- Between the workpiece holder and the machine tool,
- Between the tool holder and the machine tool.

1.4. Machining Methods

Machining methods are extremely varied and can be distinguished according to three essential criteria:

- The date of their appearance: traditional methods and non-traditional methods.
- The corresponding physical phenomena: cutting processes and physicochemical processes.
- The type of machines and tools used.

Depending on the tools and machines used, different machining methods can be distinguished. The most common methods are turning, milling, drilling, grinding, and planing. Manufacturing begins with what is called a raw material and modifies it until it conforms to the detailed drawing, meeting the technical requirements of the design office.

1.5. Milling

Our work focuses on studying this process. Milling is a machining process involving material removal. It is characterized by the use of a machine tool called a milling machine and a special cutting tool with multiple edges called a milling cutter. The milling machine is particularly suited for machining flat surfaces and, if equipped with numerical control, can create all types of shapes, even complex ones.

In milling, the cutting is usually done with teeth placed on the periphery and/or on the end of a disc or cylinder.

Fig.2. Milling Process

1.5.1. Types of Milling

The feed movement is generally directed opposite to the rotation, but it can sometimes be directed in the same direction as the rotation. Here are some types of milling:

<u>Climb Milling:</u> In climb milling, the teeth of the cutter engage the workpiece tangentially to the surface to be machined. Before penetrating the material, the teeth slide on the workpiece, causing considerable friction. As the teeth advance, they penetrate the material and remove a comma-shaped chip.

Advantages:

- The engagement of the tooth is not dependent on the surface characteristics of the machined workpiece.
- Surface impurities or scales do not affect the tool's lifespan.
- The cutting process is smoother, especially if the cutter's teeth are sharpened.

Disadvantages:

- The tool tends to vibrate.
- The machined workpiece tends to be pushed upwards, which necessitates significant clamping.
- Faster tool wear compared to conventional milling.
- Chips fall in front of the cutter, making evacuation difficult.
- The upward force tends to lift the machined workpiece.
- More power is required due to increased friction.
- The surface finish is not as good because of chip evacuation.

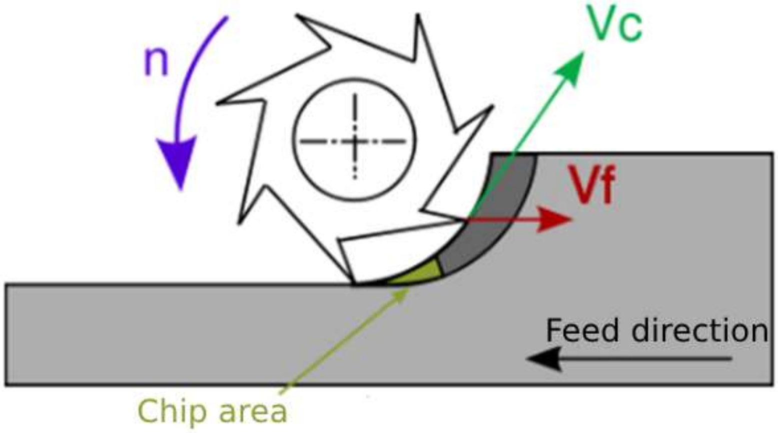


Fig.3. Climb Milling

Down Milling: In down milling, also known as conventional milling, the teeth engage the surface to be machined with a substantial thickness of material to be removed and undergo a shock. This system is adopted on milling machines that have a backlash elimination device between the control screws and their nuts.

Advantages:

- The downward cutting forces help to keep the machined workpiece in place, particularly for thin parts.
- Easier chip evacuation – chips are removed behind the cutter.
- Less wear – tool life is reduced by 50%.
- Improved surface finish – chips are less likely to be carried by the teeth.
- Less power required – a cutter with a high rake angle can be used.
- Down milling exerts a force on the workpiece.

Disadvantages:

- When the teeth engage the workpiece, the forces have a significant impact, requiring a rigid setup, and all play must be eliminated.
- Down milling is not suitable for machining workpieces with poor surface conditions, such as forged and cast parts. The scales on the material cause excessive wear and damage the teeth, thus reducing tool life.

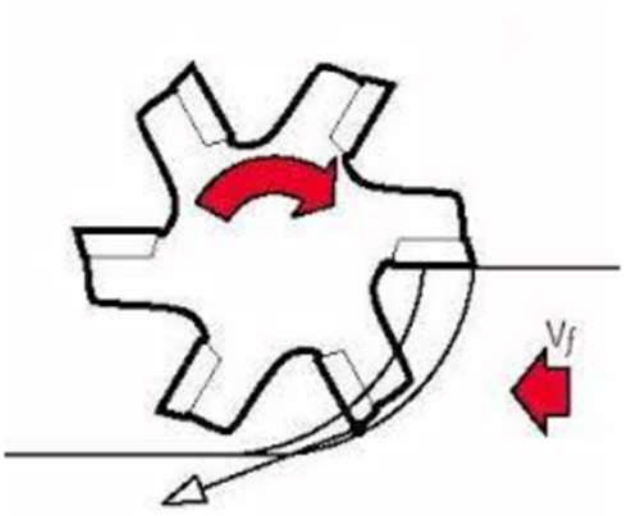


Fig.4. Down Milling

<u>Slotting</u>: Machining grooves or slots is a typical milling operation. The groove can be made using a three-edge cutter on a horizontal milling machine or with an end mill on a vertical milling machine. Using the three-edge cutter results in a better-machined groove, especially on the sides, providing better geometry compared to what is achieved with end mills. The latter can be subject to slight eccentric rotation and minor lateral flexing, which makes the machined surface and groove geometry less regular.

However, in certain specific cases, such as the one shown in figure B, it is not possible to use a disc cutter, and the use of an end mill becomes necessary.

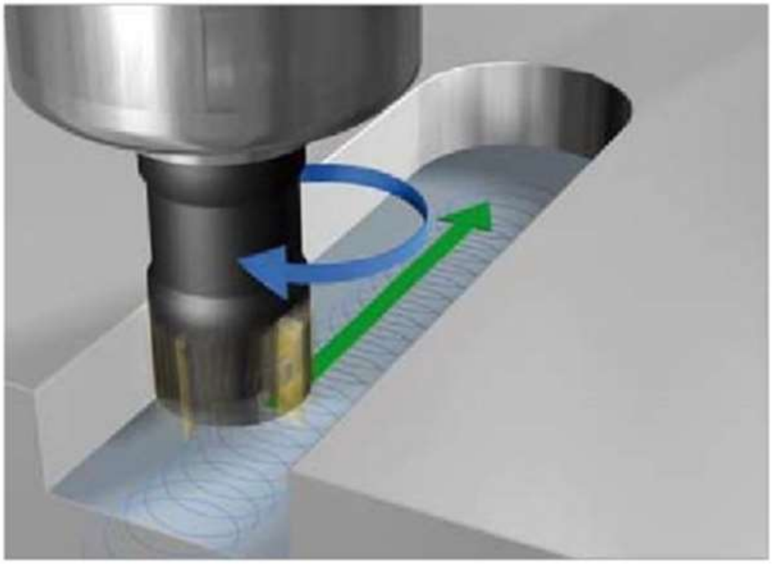


Fig.5. Slotting

1.5.2. Milling Cuter

The tool used for milling is the milling cutter. A milling cutter is a multi-edged tool with several cutting edges arranged radially around a circumference. With a milling cutter, it is possible to machine flat or curved surfaces, grooves, teeth, etc.

1.5.3. Characteristics of a Milling Cutter

It is important not to confuse the cutting direction with the direction of the helix.

a- Cutting Direction:

Milling cutters are said to be "right-hand cut" if, when viewed from above relative to the spindle of the machine, they rotate in a clockwise direction (also called the anti-trigonometric direction). This is the case for 95% of cutters. They are said to be "left-hand cut" if they rotate counterclockwise (trigonometric direction), which is the case for about 5% of two-edged cutters.

b- Type of Teeth:

The inclination of the cutting edges can vary from one milling cutter to another. The milling cutter can have:

- Straight teeth.
- Helical teeth (with left or right helix).
- Alternating teeth (double helix).

c- Shape:

Depending on the profile of the generatrices relative to the axis of the tool, milling cutters can be classified as cylindrical, conical, or shaped cutters.

1.5.4. Main Geometric Elements of the Milling Cutter:

The geometric shape of the cutting edges of a milling cutter is governed by three fundamental angles formed by the faces A and P, which determine the cutting or sharpening angles α, β, γ, as with all tools that remove chips.

For helical-tooth milling cutters, the angle δ is also considered. This angle determines the inclination of the cutting edge relative to the axis of the milling cutter and is called the helix angle or the attack angle.

a- Helix Angle or Attack Angle δ:

This is the angle between the longitudinal axis of the milling cutter and the inclination of the teeth. The angle will be small (about 5°) for machining short-chipping metals like cast iron and will increase for light metals (aluminum) (25°) or special steels like stainless steel.

b- Cutting Edge Angle β:

This is the angle that forms the cutting part of the milling cutter tooth. The sharper it is, the more fragile it becomes.

c- Rake Angle γ:

This angle affects how the chip forms on the cutting face. The larger the angle, the less cutting force is required. The rake angle is greater for materials like aluminum compared to cast iron.

d- Relief Angle α:

This angle prevents the back of the tooth from rubbing (heel dragging) on the already machined part of the workpiece.

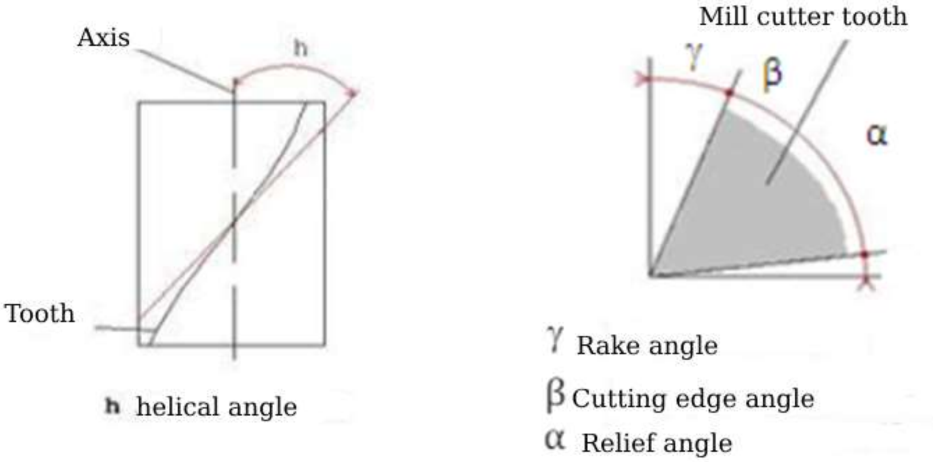


Fig.6. Sharpening angles

1.5.5. Examples of Main Milling Cutters

Milling cutters can have shanks or holes. There are mainly four families of milling cutters:

- End mills and profile cutters.
- Disk cutters.
- Slotting cutters.
- Profile cutters.

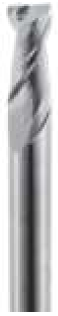
Here are some models of milling cutters:

a) The saw cutter:

b) The face milling cutter:

c) The two-edge milling cutter (Flat end mill):

d) The two-tooth slotting cutter:



e) The T-slot milling cutter:

This cutter, resembling a three-edge milling cutter, is used to machine the two recessed parts in a T-slot.

T-slots are widely used in machine tools (tables) and various tools for securing equipment.

For machining, you first need to create a straight groove (the body of the T-slot) with a three-edge milling cutter, then machine the two recessed parts of the T-slot.

f) The keyway milling cutter

g) The isosceles milling cutter

h) The chamfer milling cutter: available with two point angle values H: 60° and 90°.

For mounting, the most common setup is the chuck and taper mounting.

For example, a standard American socket is used, with the hollow taper being of Morse type, onto which the shank of the milling cutter and its tang are fitted.

Fig.7. Mounting Tool on Tool holder

1.6. Milling Machines

The machine tools used for milling are called milling machines. These milling machines are characterized by the position of the spindle or arbor and the movement capability of the worktable, as well as their efficiency (single-unit or series production). Essentially, three types of milling machines are distinguished:

A/ Horizontal Milling Machine: It uses cutters mounted on a horizontal arbor. It is used for surfacing and machining straight grooves and profiles.

B/ Universal Milling Machine: Derived from the horizontal milling machine, it allows for the use of cutters mounted on both a horizontal arbor and a vertical spindle. It can also assume various inclinations and is used for machining numerous milling forms, including helical shapes.

C/ Vertical Milling Machine: Equipped with a vertically and tiltable spindle arbor. It is used for surfacing or machining straight or circular grooves and contours.

D/ Toolroom Milling Machine: A highly flexible machine with multiple possibilities for horizontal and vertical head movements as well as the worktable. It is used for milling light but complex-shaped parts.

E/ Bed-Type Milling Machine (similar to a planer): A production milling machine. It can work simultaneously with one or more cutters mounted on one or more arbors. It is used for milling large-sized workpieces.

F/ Thread Milling Machine: Designed solely for cutting threads.

G/ Copy Milling Machine: Equipped with a device that allows the reproduction of even complex contours, following a template or pattern that guides the cutter along the shape to be executed. It is used for machining complex and irregularly shaped parts.

1.7. Relative Movements between the Tool and the Workpiece

The relative movements between the tool and the workpiece required for milling are the cutting movement, the feed movement, and the penetration movement.

-Cutting Movement (Mc): This is the main movement that removes material, provided by the rotation of the tool.

-Penetration Movement (Mp): This is the rectilinear movement that regulates the depth of penetration into the material. It is usually given to the workpiece, but in certain special machines, it is given to the tool.

-Feed Movement (Ma): This is the rectilinear movement given to the workpiece during machining. The tool thus constantly encounters material to be removed.

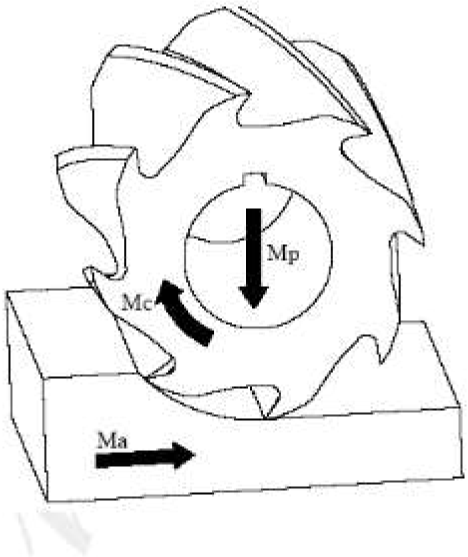


Fig.8. Relative Movements during Milling

1.8. Lubrication

Lubrication or greasing is a set of techniques used to reduce friction and wear between two elements in contact and moving relative to each other. It often helps dissipate some of the thermal energy generated by this friction and prevents corrosion. In these situations, fluid flows are parallel to the surfaces, simplifying their description and calculation (lubrication theory).

Lubrication refers to cases where the lubricant (mechanical) is liquid, while greasing refers to cases where it is compact. In mechanics, metal or ceramic parts are lubricated with a grease substance such as oil or grease. Lubricants come in liquid, paste, or solid forms and can be of mineral (mostly hydrocarbons), animal, vegetable, or synthetic origin.

Its role is to change the coefficient of friction between two elements to facilitate sliding or rolling between them and to prevent or minimize wear and heating. The physical laws governing this field (tribology) are very complex and are based on both materials' science and fluid mechanics. It's interesting to understand phenomena such as the formation of an oil wedge or oil film between two moving parts, the concept of lubricant layering, and the change in physical properties of oils depending on pressure and temperature.

Fig.9. Spot Lubrication

The first method involves applying the lubricant before or during the movement. This can be done manually, for example, by dropping oil with a dropper, applying grease with fingers (if it's not toxic), or applying the lubricant with a brush. This is the case, for example, when lubricating a bicycle chain, door hinges, etc. Lubrication can also be done by spraying using an aerosol (spray can).

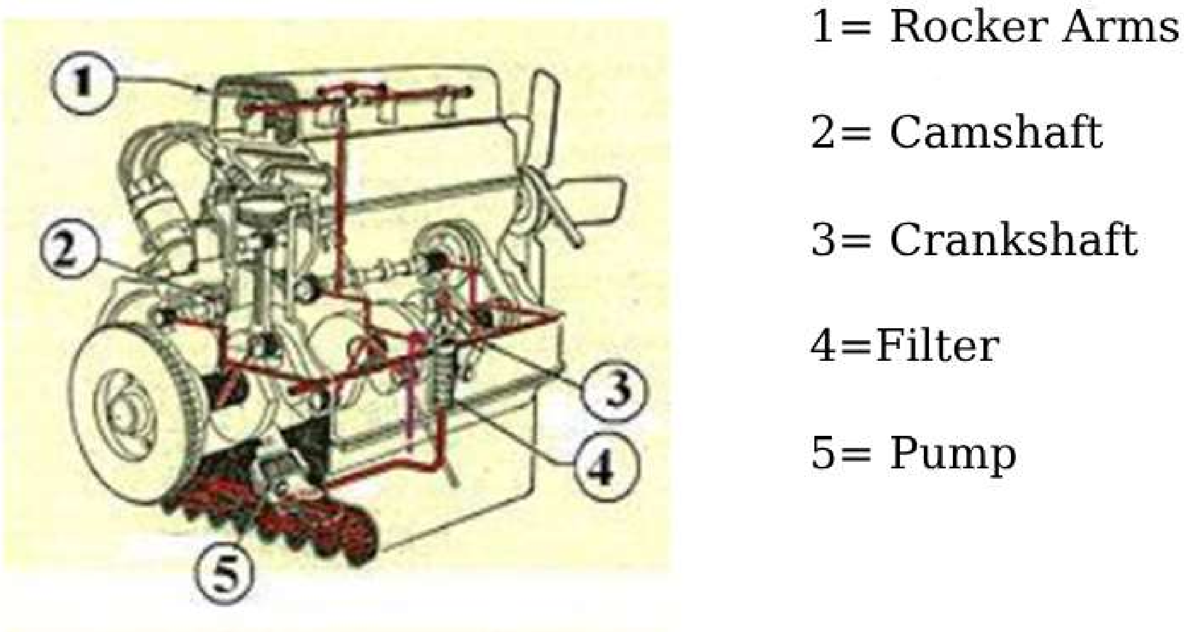


Fig.10. Continuous Lubrication

Continuous lubrication involves all moving mechanisms and is composed of a system of conduits that delivers the lubricant, via a pump, to various components (bearings, bushings, ball bearings) that need lubrication. The lubricant returns to the reservoir to be cooled and then passes through a filter that removes impurities.

- This is particularly the case with engine oil in an internal combustion engine. The lubricant degrades, especially due to high heat, and accumulates wear debris: therefore, it's necessary to regularly drain the reservoir and refill it with fresh lubricant. In some cases (which tend to become more common), the oil passes through a cooling radiator before returning to lubricate components that use the same fluid (engine, gearbox, turbo).

- In the case of two-stroke engines, fuel is mixed with a percentage of special oil (called 2-stroke oil) which provides lubrication to moving parts. There is no oil recovery afterward because it is burned along with the fuel.

- For stationary machines, we can also immerse the mechanics in a liquid lubricant, known as an oil bath or splash lubrication. As with internal combustion engines, the sump must be filled while respecting the minimum and maximum levels.

• Greasing with a ring: this system is used in bearings for lubricating the journals and consists of a ring, freely placed on the shaft, with a diameter large enough to dip into a reservoir at the bottom of the bearing. During the rotation of the ring, the oil adhering to it is carried to lubricate the journal.

• Machining of mechanical parts requires lubrication to ensure cooling of the tools and parts and to limit cutting forces to the minimum necessary for material removal (chips).

In the case of automatic lubrication, the device may be equipped with a recovery tank to reprocess excess or used lubricant.

Machine tools such as lathes, milling machines, etc., besides the lubricant for the operation of the mechanism, use cutting oils to facilitate cooling of the workpiece and tool, and the sliding of the chip on the tool, increasing the cutting speed.

• Continuous lubrication: The lubricant is usually poured in a continuous stream and then collected in a tank under the machine, filtered, and returned to the circuit. The fluid used is synthetic oil or a mixture of 5 to 10% soluble oil (lubricant) and water (coolant), called soap water due to its milky color.

• Spot lubrication: can also be done automatically, usually in the form of an aerosol; this is the case, for example, with machining using a CNC lathe. Or with a brush and specific fluid for the metal to be machined: linseed oil, petroleum (for aluminum).

2. Fundamentals of Metal Cutting

2.1. Principle of Metal Cutting

During material removal machining, we typically encounter the following configuration, as shown in Figure below:

- A cutting tool penetrates the material and removes a chip.
- The tool follows a trajectory relative to the workpiece, where the movements are ensured by the components of the machine tool.

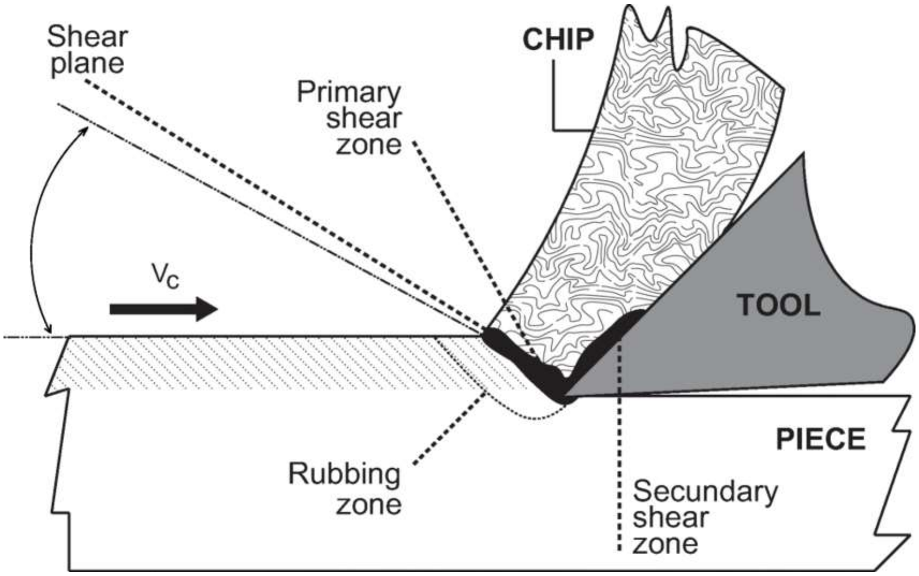


Fig.11. Chip formation

Upon closer examination of the cutting mechanism, we observe that it operates through three main perpendicular movements, as depicted in Figure II.2.

Cutting Movement "MC": This movement directly contributes to the removal of material in the form of chips during the working stroke.

Feed Movement "Ma": This movement laterally shifts a quantity a, known as the feed, so that the tool can detach additional chips during the next working stroke.

Penetration Movement "Mp": This movement determines the thickness of the metal layer to be removed in each operation, which is referred to as the pass.

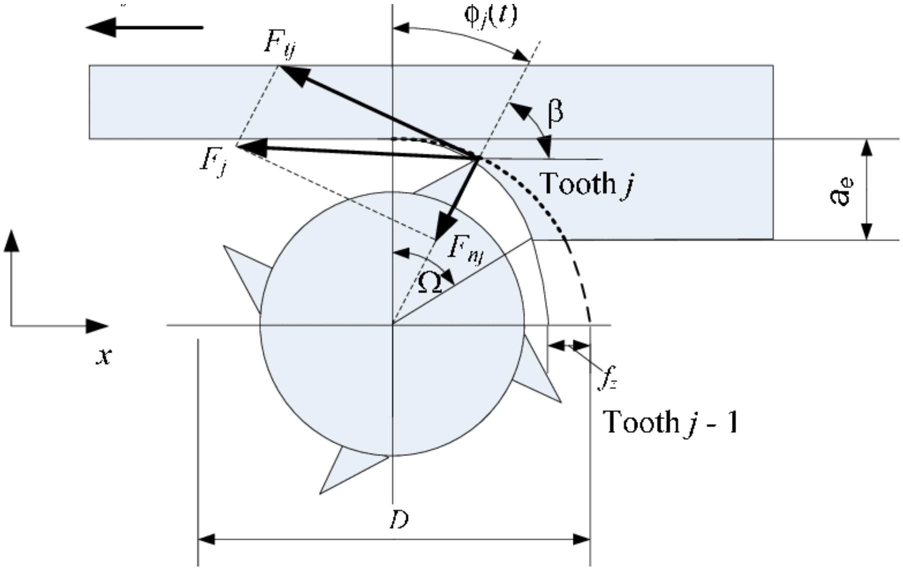


Fig.12. Cutting forces

In order to achieve satisfactory work (good surface finish, machining speed, moderate tool wear, etc.), the cutting parameters must be adjusted. There are several criteria that define these cutting parameters, as illustrated in the following figure:

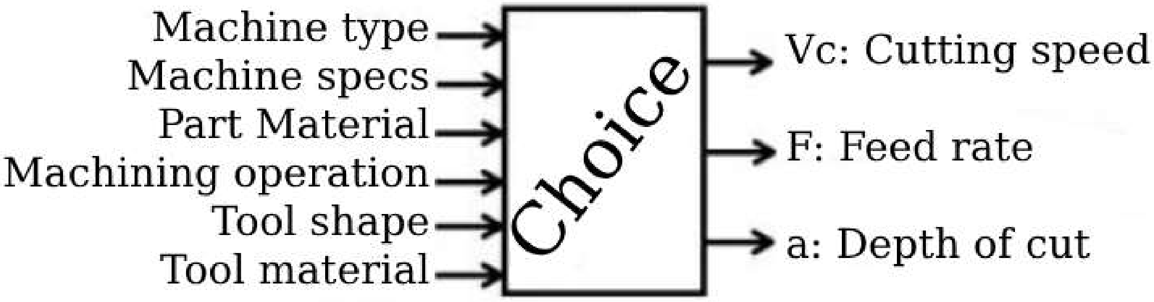


Fig.13. Cutting Parameters choice

Based on the type of operation to be performed, the machining method must be chosen, which in turn dictates the machine to be used. Therefore, the choice needs to be made between turning, milling, or drilling.

Material selection is crucial since cutting efforts vary significantly depending on whether you are machining a piece of polystyrene or steel. Consequently, material properties influence decisions related to machine power, among other factors.

Similarly, the machining operation selection follows the same principle as the choice of machine type. The tool's shape also plays a significant role. However, the tool material directly impacts tool wear and lifespan since it's the tool that cuts the workpiece and not vice versa.

All these criteria are interconnected, and since the ultimate goal is to obtain a well-machined part, specific parameters need to be determined:

- Cutting speed: Vc
- Feed rate: F (or Vf)
- Depth of cut: a

2.2. Cutting parameters

The cutting parameters, represented in Figure below, are, on one hand, quantities that characterize the movements of the tool and the workpiece (kinematic cutting parameters) and, on the other hand, the values of machining allowances and cutting dimensions (geometric cutting parameters).

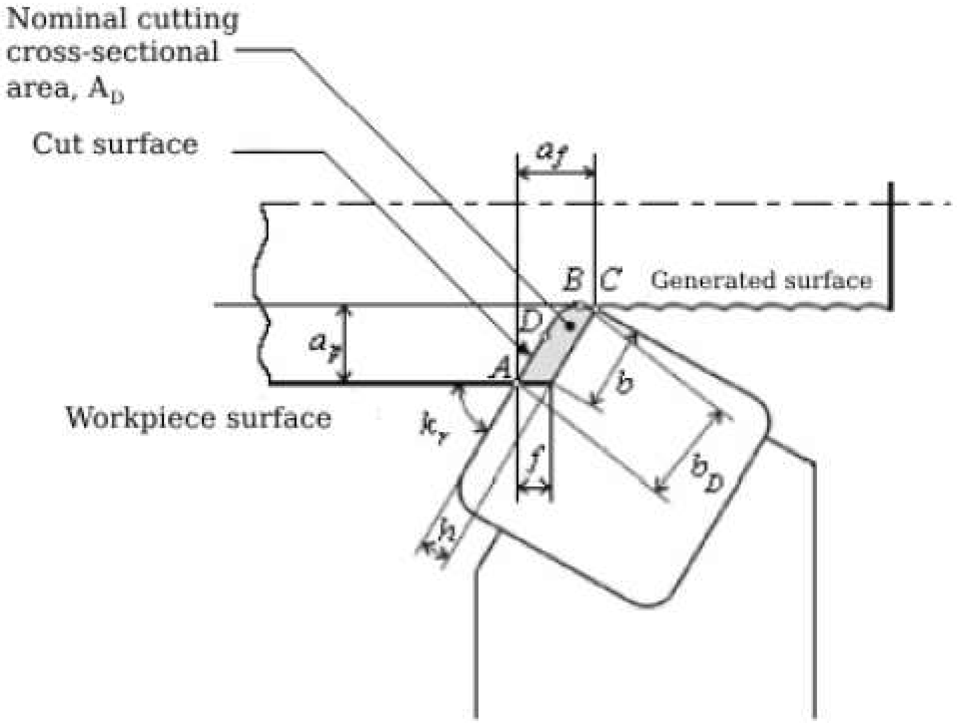


Fig.14. Cut dimensions in Turning

Where:

- a_p – depth of cut;
- a_f – cutting edge engagement;
- f – feed rate;
- h – cutting thickness;
- b – cutting width;
- b_D – nominal cutting width;
- k_r – cutting edge direction angle;
- D – main point of the cutting edge.

The general machining allowance refers to the layer of metal or material removed during all the machining operations on a given surface of the workpiece. It is equal to the difference in size between the rough piece and the finished machined piece. The inter-operational machining allowance is the layer of metal or material left after a given pass for the execution of the successive machining pass. The choice of allowances is a very important operation. The size of the allowance affects the amount of chips. Removing excess layers of metal requires time, electrical energy, etc. Excessively large allowances therefore reduce machining productivity and profitability. It is important to aim to reduce both the general and inter-operational machining allowances, especially in conditions of large and very large series production. In terms of dimensions and shapes, the rough pieces should be as close as possible to the finished piece.

The determination of cutting parameters is very important in industrial production because, for example, reduced regimes significantly increase the execution time of the workpiece and raise the cost price, while high regimes are also not advantageous because the tool wears out quickly, forcing frequent changes, resulting in the same outcome as before (increased cost price of the piece). Therefore, the adopted parameters must be optimal to ensure a minimum cost price of the piece with the highest productivity.

2.2.1. Cutting Speed Vc

It is the movement of a point on the cutting edge of the tool relative to the cutting surface in a unit of time.

2.2.1.1. Linear speed of a rotating point:

It is evaluated in terms of circumferential travel, that is, the length of the circumference of the piece traveled. Denoting D as the diameter of the circle described in millimeters [mm], N as the rotational speed in revolutions per minute [rpm], and Vc as the linear speed in meters per minute [m/min], we have:

a- Example of turning:

In turning, the cutting movement primarily drives the workpiece (rotating piece).

$$V_c = \frac{\pi \times D \times N}{1000} \; [m/min]$$

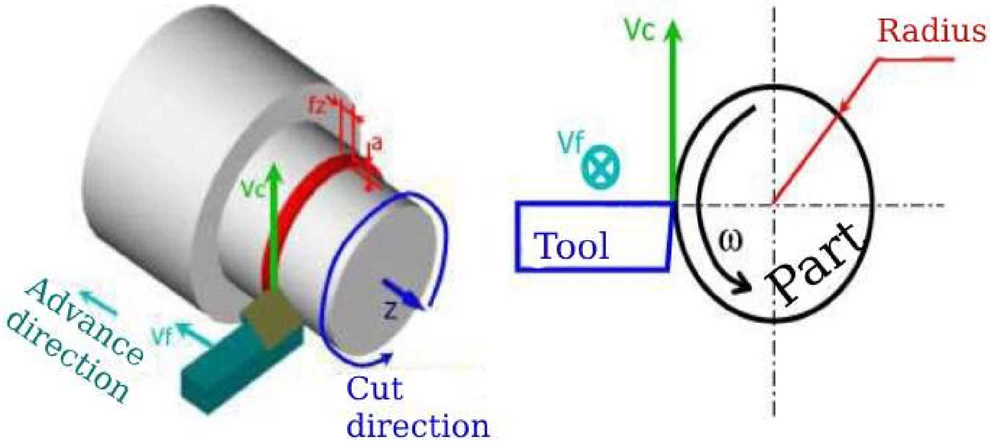


Fig.15. Cutting speed in Turning

We deduce the cutting speed Vc, which will determine the rotational speed of the workpiece that needs to be set on the machine.

$$N = \frac{1000 \times V_c}{\pi \times D} \; [rev/min]$$

The diameter "D" corresponds to the position of the tool tip, which leads to two scenarios:

- Machining parallel to the spindle axis. The generated surface is a cylinder, D = diameter of the cylinder.

- Machining perpendicular to the spindle axis. The generated surface is a plane, D = 2/3 maximum diameter of the plane.

b- Example of Milling:

In milling, the cutting motion drives the tool (rotating cutter), and the same formulation applies; however, the diameter "D" corresponds to the diameter of the cutter.

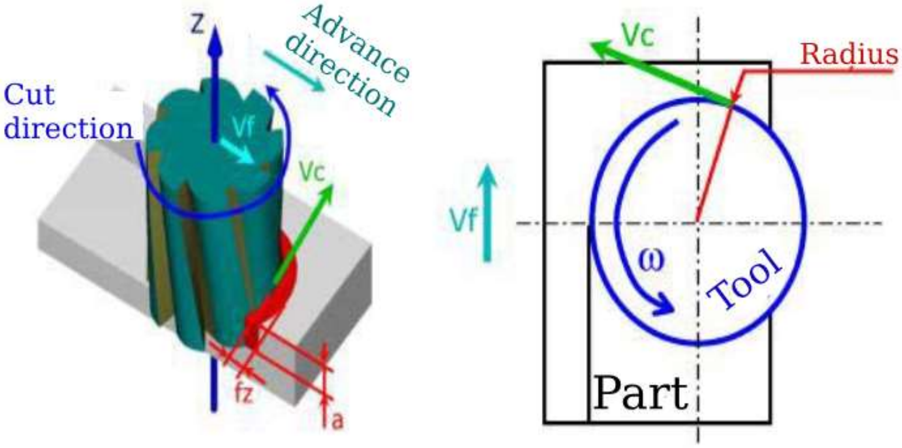


Fig.16. Cutting Speed in Milling

2.2.2. Feed rate (Vf) and Feed per Revolution (f)

Feed rate (Vf) is the instantaneous speed of the feed movement of a specific point on the cutting edge relative to the workpiece, as shown in Figure below. It is expressed either in millimeters per minute [mm/min] or in millimeters per revolution [mm/rev].

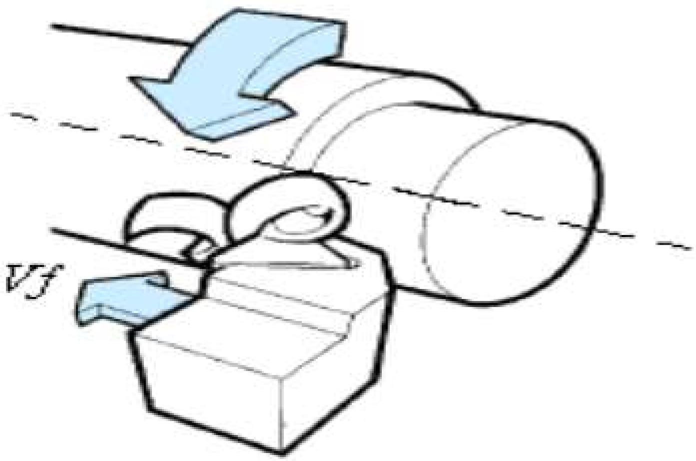


Fig.18. Feed rate

The feed, denoted as f (Figure below), corresponds to the difference in displacement of the tool between two iterations or two revolutions (one revolution of the workpiece in the case of turning).

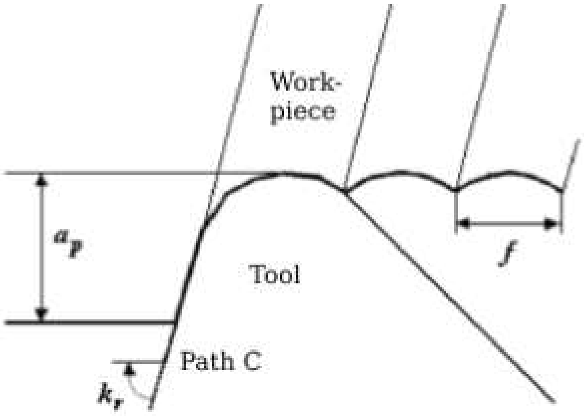


Fig.19. Rate f in Turning

2.2.2.1. Feed rate Vf in turning and milling

In turning, the feed rate Vf [mm/min] is given by the following formula:

$$Vf = fz \times N \text{ [mm/min]}$$

Where fz in [mm/(rev. tooth)] corresponds to the cutting capacity of the cutting edge for one rotation of the workpiece. In other words, it's the distance that the cutting edge will travel for each revolution of the workpiece.

In milling, the feed rate Vf [mm/min] is equal to:

$$Vf = Z \times fz \times N \text{ [mm/min]}$$

Where Z is the number of teeth of the cutter, and fz in [mm/(rev. tooth)] corresponds to the distance that the tooth will travel for each revolution of the cutter.

2.2.3. Depth of cut:

The depth of cut, noted as a in [mm], corresponds to the length of the cutting edge engaged in the material, in the case of orthogonal cutting, and to the difference between the radius of the workpiece before and after machining, in turning. The depth of cut is always measured perpendicular to the direction of feed, not along the tool edge.

2.3. Formation and types of chips

Analyzing the mechanisms of chip formation during the machining process is a fundamental step for any study in the field of cutting, for example, optimizing the selection of tools and predicting their lifespan. During this process, various phenomena can occur such as plastic deformation, contact and friction between the tool and the workpiece, thermal effects, wear, etc. If we examine the stress-strain curve of steel, we observe three zones:

- oa: Elastic deformation (reversible)
- ab: Metal flow
- bc: Plastic deformation (irreversible)
- starting from c: Rupture

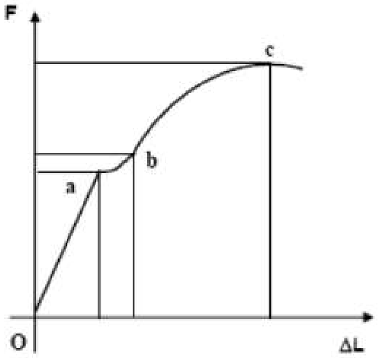


Fig.20. Tensile curve of steel

In the context of chip formation, we focus on plastic deformation, which involves the sliding of certain layers of material relative to others along shear planes. These slides cause:

- Changes in the shape, dimensions, and relative positions of the metal grains.
- Significant heating and modifications of the properties of the work material.

The following figure summarizes the principle of chip formation:

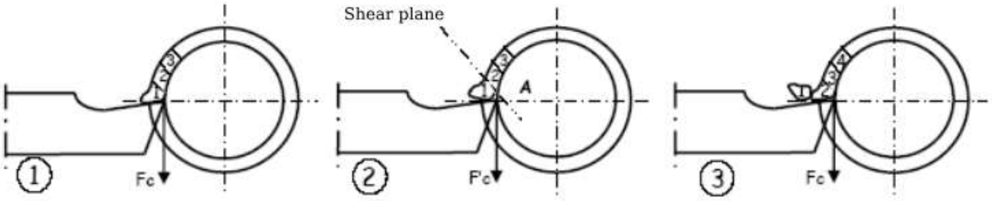


Fig.21. Mechanism of Chip formation

- At 1: the cutting edge has penetrated the material which, being unable to flow normally, braces against the cutting face and is heavily compressed. The cutting force increases to the maximum value Fc.

- At 2: a crack due to the shearing following compression appears at A, and the chip segment 1 slides on the cutting face, increasing the significance of the crack. The cutting force simultaneously decreases to the minimum value Fc'.

- At 3: the chip is compressed again and the cutting force increases. The cycle is continuous, and the forces due to the cutting action vary periodically very clearly for steels, with a higher frequency for cast iron (segmented chips), and less so for soft metals.

Chips are classified into three types:

a) Discontinuous chip:

Occurs during the machining of steel at a low cutting speed Vc = (5 to 10) m/min. The elements of the chip are very weakly connected.

b) Sheared chip:

Occurs during the machining of steel at a medium cutting speed Vc = (80 to 100) m/min; the surface of the chip facing the tool is smooth, while the opposite surface shows notches that clearly define the direction of the isolated elements of the chip bonded together.

c) Continuous chip:

Occurs during the machining of steel at a high cutting speed Vc > 100 m/min. During the machining of brittle metals (cast iron, bronze, aluminum alloy), the fragmented chip is observed. It consists of elements torn from the base material; of varied shapes that are not bonded to each other. The feed slope forms immediately, along the entire shear surface along which the separation between the chip and the base metal takes place. Such a chip leaves a machined surface rough, covered with significant pits and peaks.

2.4. Concept of Surface Finish

Surface finish refers to the irregularities on surfaces due to the manufacturing process of the part (machining, casting, etc.). These are most often measured with devices using a diamond-tipped stylus, called profilometers, which trace the profile of the surface.

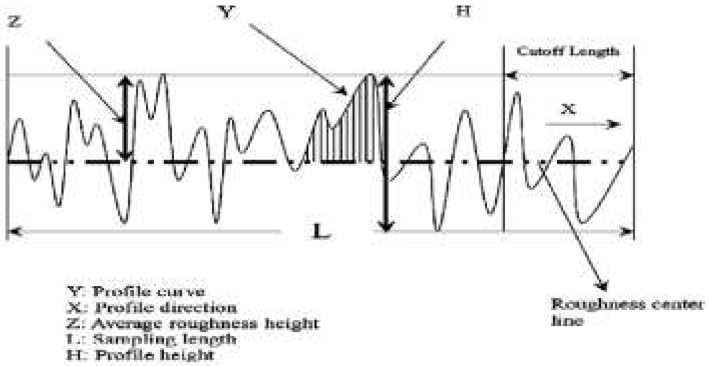


Fig.22. Profile of Surface finish

On the surface of a part obtained by machining, four main types of defects can be identified, classified as macroscopic or micro-geometric:

1. Deviations in shape and position.
2. Waviness (periodic defects).
3. Signature of the manufacturing process: striations, grooves (periodic or pseudoperiodic defects).
4. Accidental defects; tears, pittings, etc.

Roughness includes irregularities that are often detectable by touch, such as tool marks on a machined piece or the "grain" of a sandblasted or coated surface. For mechanical parts, these are generally irregularities less than 500 µm, with a width-to-depth ratio ranging from 100/1 to 5/1. Roughness affects the suitability for friction (especially during the running-in period), wear, adhesiveness of coatings, resistance to bending and alternating stresses, corrosion, etc.

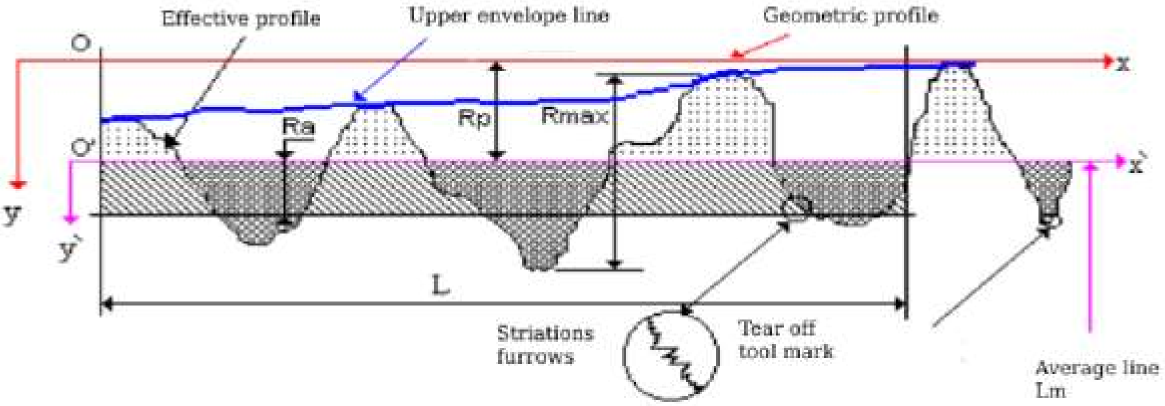


Fig.23. Roughness

During machining, factors that affect roughness include the feed rate of the tool, its sharpening, the waviness of its cutting face, the chip breaker, the nose radius, the quality and filtration of the lubricant, the dressing of the grinding wheel, high-frequency vibrations, etc. Depending on the machining processes and cutting conditions used, the machined surface can be more or less rough. The maximum value of the roughness defect is indicated by a number in microns, which increases as the depth of the striations increases.

The mode of generation of a surface can be identified by the shape of the striations according to the following figure:

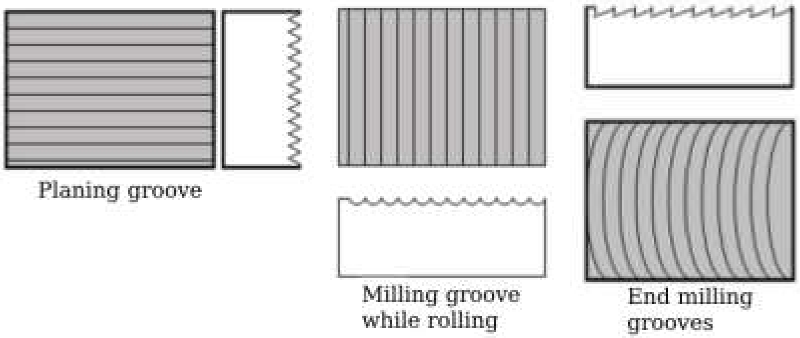
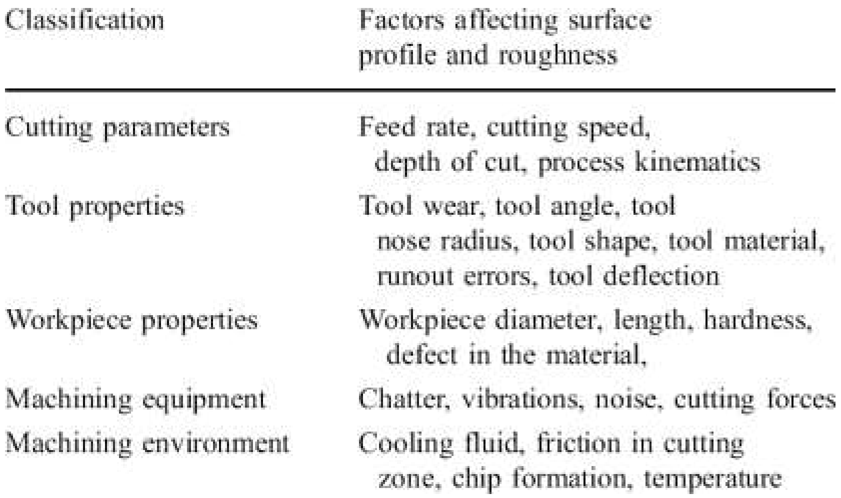


Fig.24. Shape of streaks obtained by different machining processes

The factors influencing the achievement of surface finish are:



Classification	Factors affecting surface profile and roughness
Cutting parameters	Feed rate, cutting speed, depth of cut, process kinematics
Tool properties	Tool wear, tool angle, tool nose radius, tool shape, tool material, runout errors, tool deflection
Workpiece properties	Workpiece diameter, length, hardness, defect in the material,
Machining equipment	Chatter, vibrations, noise, cutting forces
Machining environment	Cooling fluid, friction in cutting zone, chip formation, temperature

Fig.25. Factors affecting surface finish

Among the various factors affecting the surface finish of a mechanical part, the dominant factors are the geometry of the cutting tool, the feed rate, and the relative vibrations between the tool and the workpiece.

- Built-up edge:

Under the influence of disruptive elements such as temperature and friction, it may happen that the continuous chip adheres to the cutting tool, resulting in the formation of a layer called a "built-up edge" on the cutting edge of the tool. This can lead to disturbances in machining. The significance of this built-up edge increases until it is removed either towards the chip or towards the workpiece. In the latter case, it can result in an alteration of the surface finish of the workpiece. The built-up edge can be eliminated by increasing the cutting speed and reducing the feed rate, thereby reducing the machining efficiency. Sometimes, the use of appropriate lubricants can help avoid this reduction.

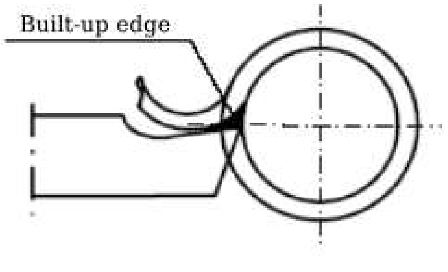


Fig.26. Built-up edge

- Tool Wear:

Tool wear is one of the most complex physico-chemical phenomena. It influences the geometry of the tool as well as the quality of the workpiece. It occurs due to the following reasons:

- Friction between tool and chip, tool and workpiece, and workpiece and chip (abrasive wear).
- Plastic deformation of the material in the active part of the tool.
- Significant heating due to cutting.
- Chipping of the tool (removal of small metal particles).

Wear can take several forms:

- Abrasive wear affects all tools and is the main cause of tool dullness.
- Wear due to plastic deformation of the tool material is more common in tools made of low-alloy steels (high-speed steel).

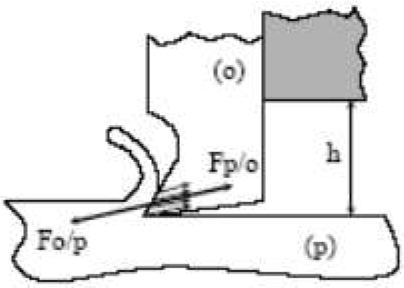


Fig.27. Curve

- Wear caused by high cutting temperatures is more pronounced in carbon steels and high-speed steels due to their relatively low heat resistance. Wear occurs more rapidly as the temperature of the active part increases, as shown in the figure:

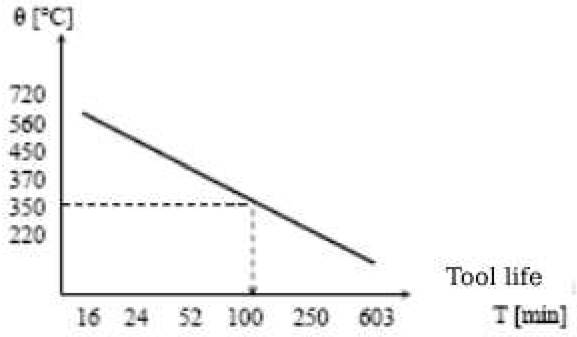


Fig.28. Tool life T[min]

A series of turning experiments aimed at determining the speed "vc" as a function of time T for dry machining and another with lubrication yielded the following results:

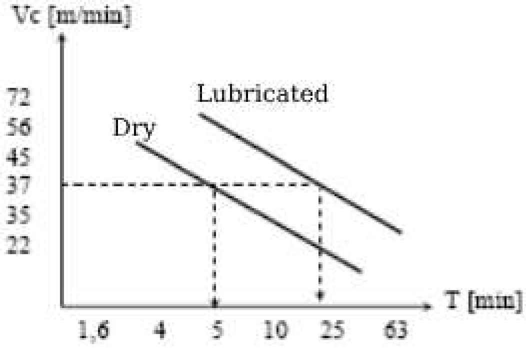


Fig.29. Influence of lubrication

What can be observed from the figure is that with well-adapted lubrication for the work to be done, the tool's lifespan can be significantly extended, resulting in reduced wear.

3. Milling operations on Siemens NX

Siemens NX CAM can be utilized to set up and manage an entire manufacturing series for both milling and turning machines. Its universal access to machining and manufacturing allows for the virtual simulation of the physical process setup. This feature enables users to identify potential physical issues, such as machine limits or collisions with fixtures and other parts. Additionally, the analytical tools help users assess the accuracy of the tool paths in relation to the original design, as well as identify areas that are under-machined or over-machined.

3.1. Open Area

- Linear, Linear Relative to Cut

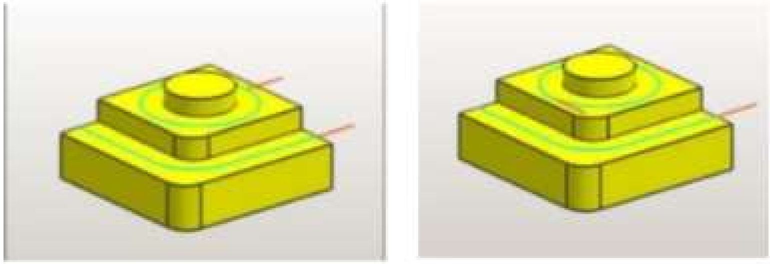
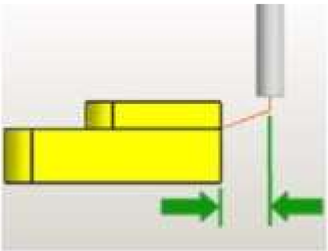
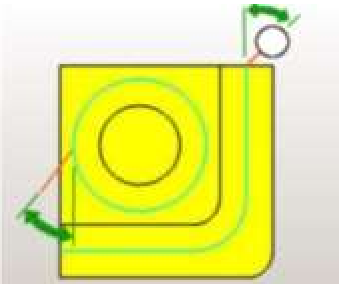
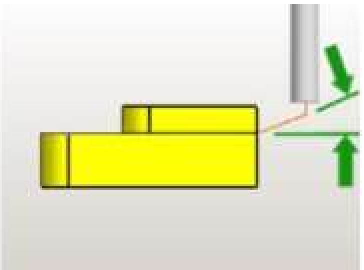
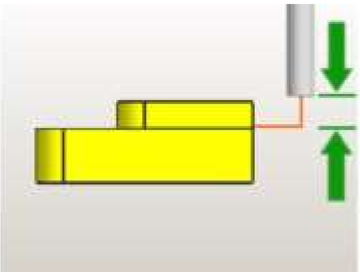
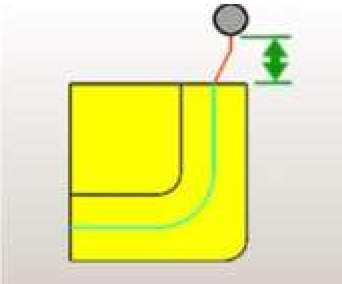


Fig.30. Linear and Linear Relative to cut

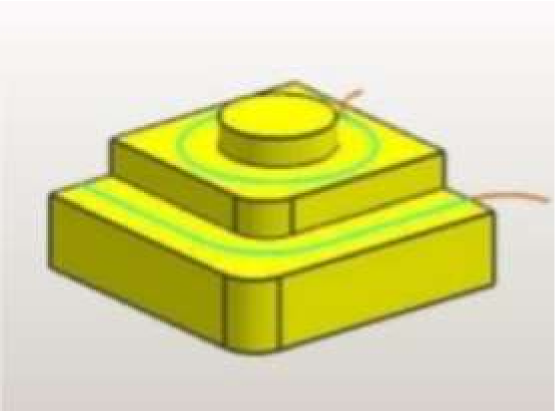
The tool engages with the cut in a straight line perpendicular to its axis. Conversely, with the "linear relative to cut" option, the tool engages with the cut along a line tangent to the generated toolpath. Both the "linear" and "linear relative to cut" options offer selections that allow for adjusting the chosen engagement strategy to meet the specifications of the machined feature.

Table.2. Linear options and Linear Relative to cut options

Option	Shape	Description
Length		This option limits the range of the linear line distance to the engage point.
Swing angle		This option determines the angle of the linear line that the tool follows to enter the cut relative to the chosen path.
Ramp angle		This option inserts the value of the ramp angle if chosen to approach the cut with a linear ramp entry.
Height		This option determines the distance length of the vertical line.

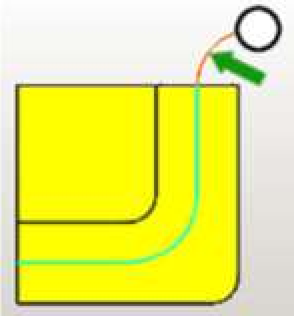
Minimum Clearance		This selection adjusts the distance specified for the tool to take from the workpiece.

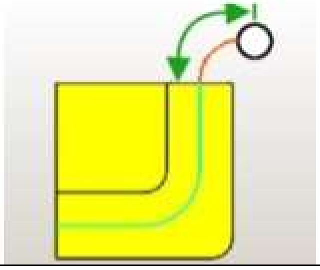
- Arc



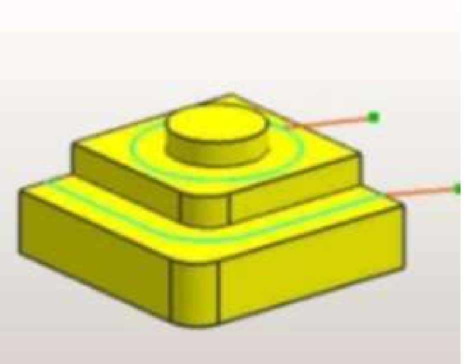
The tool engages in the cut with an arc either horizontally or vertically. Using this option requires more specifications to be determined:

Table.3. Arc Options

Options	Shape	Description
Radius		This option allows the determination of the radius of the arc generated.

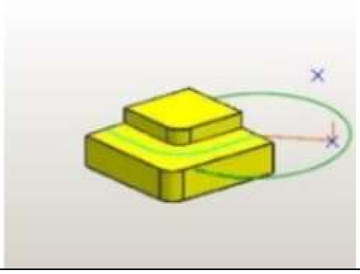
Arc angle		This option controls the value of the arc's angle.

- Point



This option allows the user to specify a point from which a linear line is generated for entering the cut. The most important option for this method

Table.4. Point options

Options	Shape
Effective Distance	

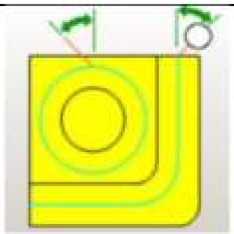
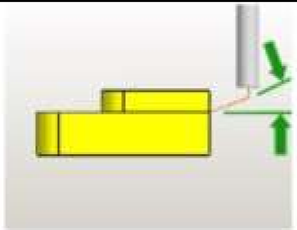
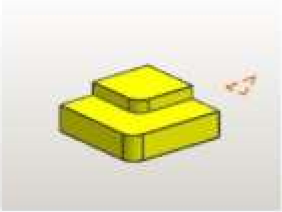
- Linear-Along Vector

This method uses a vector direction as an orientation for the linear movement generated.

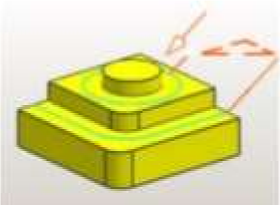
- Angle-Angle Plane

This method is generated by two angles. Specifying a plane is crucial for using this method:

Table.5. Angle-angle plane options

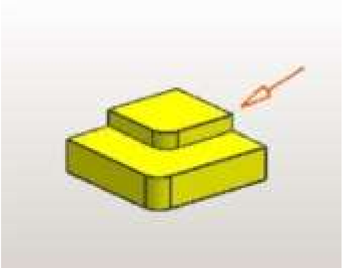
Option	Shape
Swing Angle	
Ramp Angle	
Specify Plane	

- Vector Plane



This method approaches the cut from the vector specified on the chosen plane.

3.2. Closed Area:



- Helical, Ramp on Shape

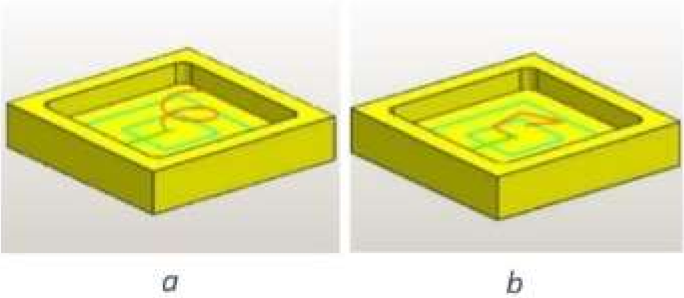
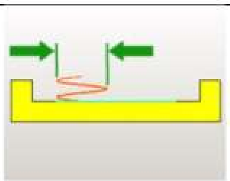
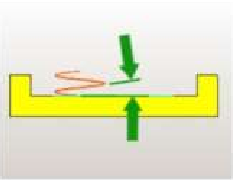
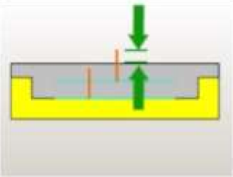
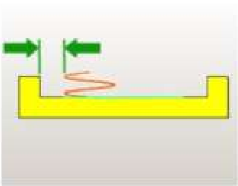
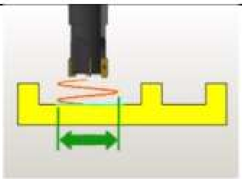


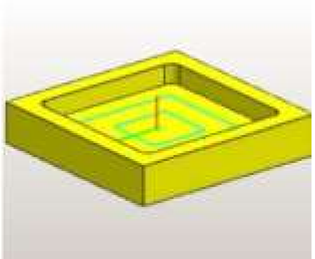
Fig.31. a-Helical, b-Ramp on shape

The tool engages in the cut with a ramping movement either linearly or helically. The options below specify the ramp form generated:

Table.6. Helical and Ramp on shape options

Option	Shape	Description
Diameter		This option specifies the diameter of the helical ramp.
Ramp Angle		This option specifies the Helical or Ramp on Shape ramp angle.
Height		This option determines the height of the ramp from the milled shape.
Minimum Clearacne		This option specifies the distance between the ramp movement and the wall of the cavity milled.
Minimum Ramp Length		This option sets the allowed value of the ramp length based on the cutting tool diameter.

- Plunge



In this method the tool enters the cut perpendicular to the surface cavity. The point where the plunge enters can be specified.

CHAPTER 2: Manufacturing Process Plan

2.1. General analysis of the part

2.1.1. Geometry analysis

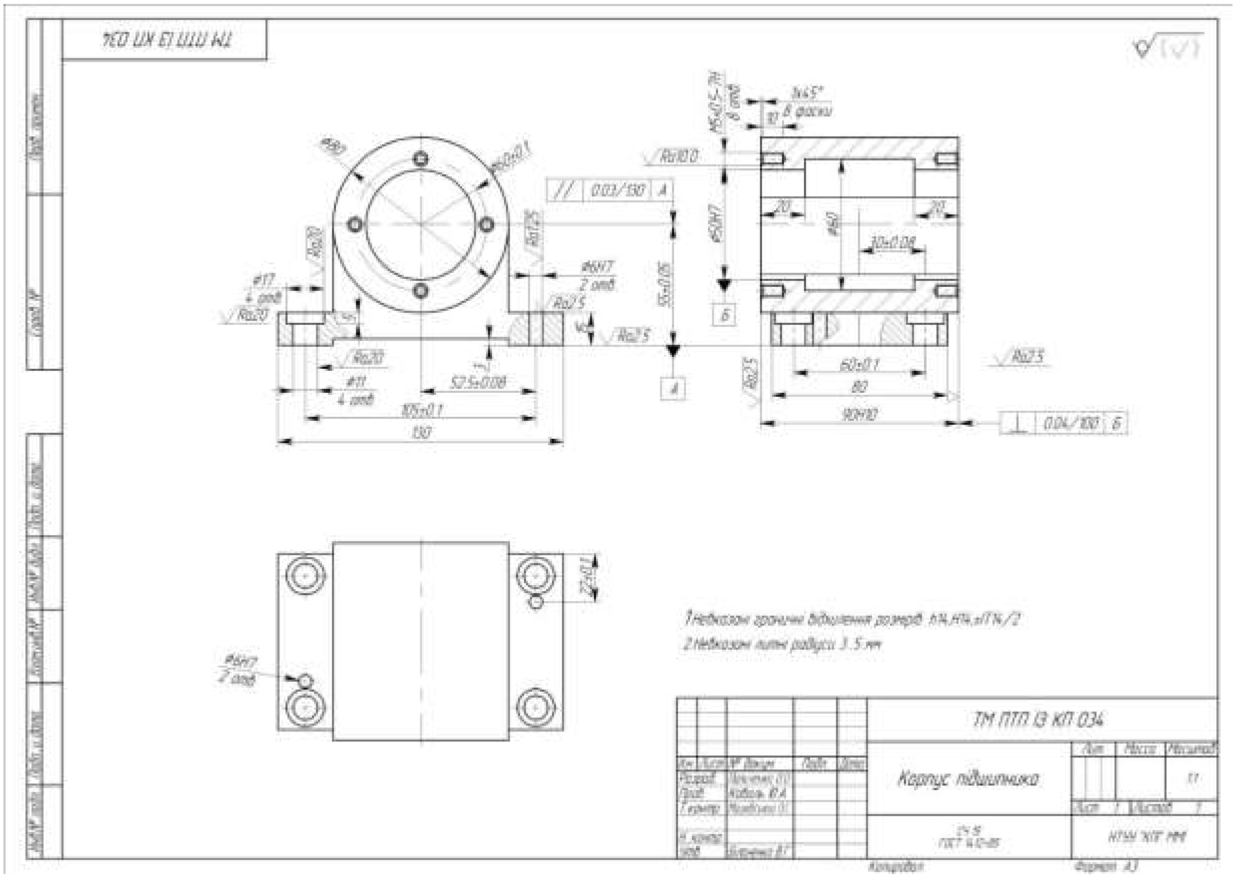


Fig.1. Initial variant drawing

Considering the configuration of the part ‘Bearing housing’ we defined that it belongs to the class “Body”.

In general, we conclude that the accuracy requirements and surface finish requirements are not very demanding, but there are a few surfaces that are subject to increased requirements. During manufacturing a special attention must be paid to the machining of holes ∅50H7, ensuring their perpendicularity to the ends, and alignment. As well as the surface “A” where Parallelism must be ensured.

2.1.2. Part's working condition in the assembly

The "Bearing housing" is designed to install and hold the gears, shafts and bearings in the appropriate position, as well as to retain the oil and prevent it from leaking. There are bearings inside the holes ∅50H7. Holes M5-7H are intended for fastening the bearing caps. The cover is connected using 4 threaded holes M5-7H.

2.1.3. Material analysis

Material of the part is "СЧ15" corresponds to a specific grade of cast iron according to the GOST 14.12-85 standard.
It has the following composition:

Table.1. Chemical Composition of the Material

Carbon	Silicon	Manganese	Phosphorus	Sulfur
3.5-3.7%	2.0-2.4%	0.5-0.8%	0.2%	0.15%

the tensile strength is specified as 150 MPa, which is equivalent to 15 kgf/mm²

$$\sigma_B = 150\ MPa$$

the density is specified as 7,0 g/cm3

$$\rho = 7\ {}^{g}/_{cm^3}$$

(HB) however varies based on the thickness of the casting. Here are the maximum hardness values for different wall thicknesses:

$$HB = 130 \dots 241$$

The analysis indicates that the part experiences periodic loading in a benign environment. The chosen material is suitable for these conditions and will ensure the part functions properly.

2.1.4. Type of production determination

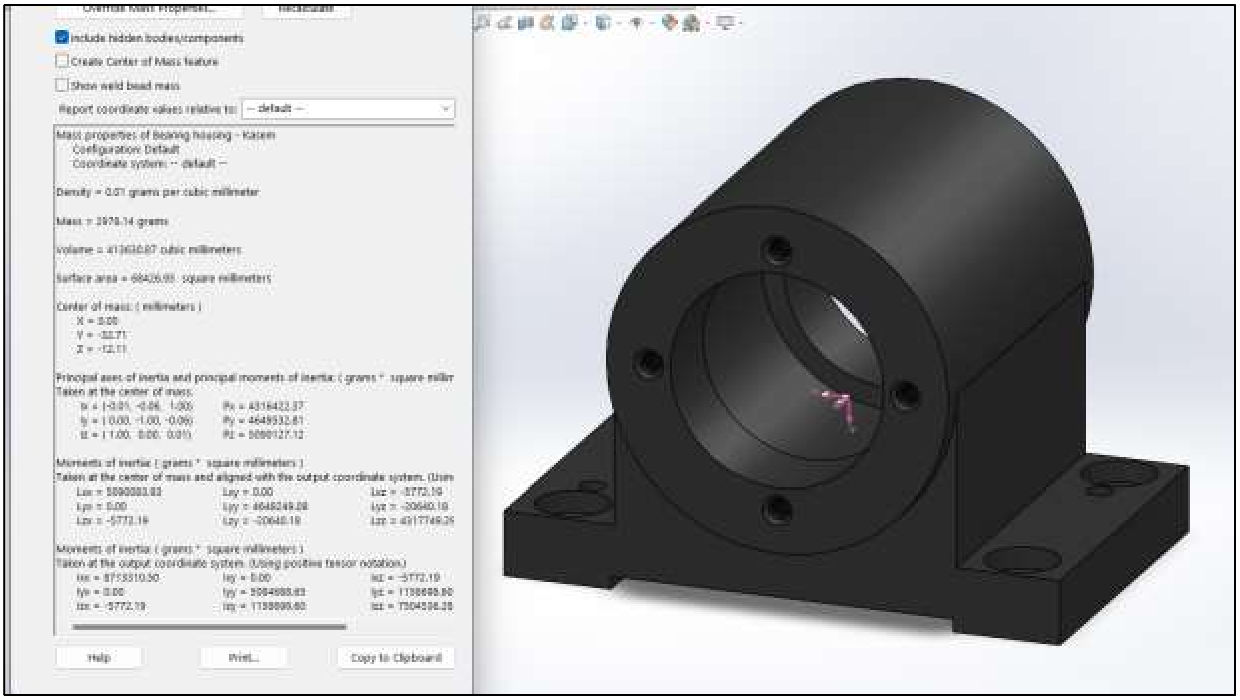
For educational purposes we will use analog methods of designation of production type based on weight of a part and production volume.

- Part weight m=2,978g (2.97 kg)
- Production volume Np = 200.

Let’s determine the type of production according to the following table

Table.2. Production type catalog

Weight of a part, kg	Type of production				
	Single	Small batch	Medium batch	High volume batch	Mass
<1	< 10	10 .. 2000	2000 .. 75000	75000 .. 200000	> 200000
>1 .. 2.5	< 10	10 .. 1000	1000 .. 50000	50000 .. 100000	>100000
> 2.5 .. 5.0	< 10	10 .. 500	500 .. 35000	35000 .. 75000	>75000
> 5.0 .. 10.0	< 10	10 .. 300	300 .. 25000	25000 .. 50000	>50000
> 10.0	< 10	10 .. 200	200 .. 10000	10000 .. 25000	>25000



☑ Include hidden bodies/components

☐ Create Center of Mass feature

☐ Show weld bead mass

Report coordinate values relative to: -- default --

```
Mass properties of Bearing housing - Kasem
    Configuration: Default
    Coordinate system: -- default --

Density = 0.01 grams per cubic millimeter

Mass = 2978.14 grams

Volume = 413630.87 cubic millimeters

Surface area = 68426.93  square millimeters

Center of mass: ( millimeters )
    X = 0.00
    Y = -32.71
    Z = -12.11

Principal axes of inertia and principal moments of inertia: ( grams *  square millir
Taken at the center of mass.
     Ix = (-0.01, -0.06,  1.00)      Px = 4316422.37
     Iy = ( 0.00, -1.00, -0.06)      Py = 4649532.61
     Iz = ( 1.00,  0.00,  0.01)      Pz = 5090127.12

Moments of inertia: ( grams *  square millimeters )
Taken at the center of mass and aligned with the output coordinate system. (Usin

Moments of inertia: ( grams *  square millimeters )
Taken at the output coordinate system. (Using positive tensor notation.)
```

Fig.2. Mass properties of the part

<u>Conclusion</u>: the production type – small batch, therefore, we will perform all further calculations and make technological decisions for the small-volume type of production.

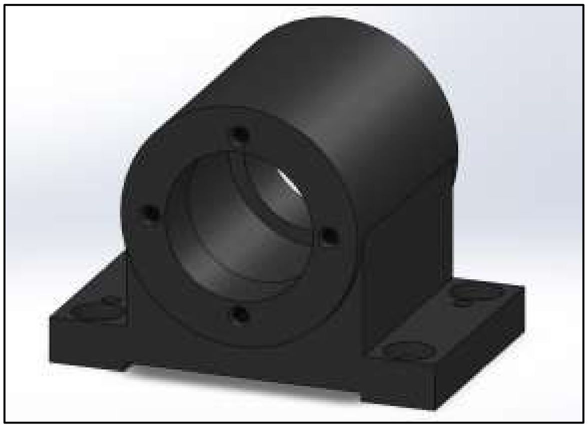


Fig.3. The 3D Model

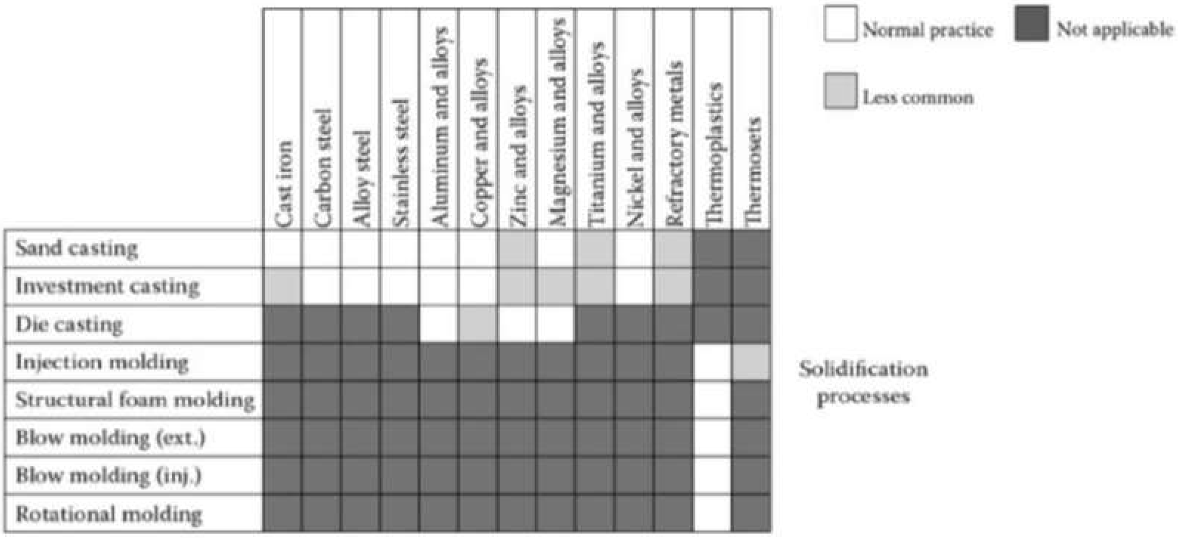
2.2. Selection of the base process and Blank design

2.2.1. Selection of the base process

Initial data for the process selection:

- drawing of a part + 3D Model;
- material of a part – Grey Cast Iron;
- Annual output – 200 pcs

We can now choose the Process with the help of The Solidification processes' chart in Fig.4



□ Normal practice ■ Not applicable

▨ Less common

	Cast iron	Carbon steel	Alloy steel	Stainless steel	Aluminum and alloys	Copper and alloys	Zinc and alloys	Magnesium and alloys	Titanium and alloys	Nickel and alloys	Refractory metals	Thermoplastics	Thermosets
Sand casting	□	□	□	□	□	□	▨	□	▨	□	▨	■	■
Investment casting	▨	□	□	□	□	□	▨	▨	▨	□	▨	■	■
Die casting	■	■	■	■	□	▨	□	□	■	■	■	■	■
Injection molding	■	■	■	■	■	■	■	■	■	■	■	□	▨
Structural foam molding	■	■	■	■	■	■	■	■	■	■	■	□	■
Blow molding (ext.)	■	■	■	■	■	■	■	■	■	■	■	□	■
Blow molding (inj.)	■	■	■	■	■	■	■	■	■	■	■	□	■
Rotational molding	■	■	■	■	■	■	■	■	■	■	■	□	■

Fig.4. Solidification processes

Considering the Geometry of the part and according to the chart above, we choose the Sand Casting as our base process.

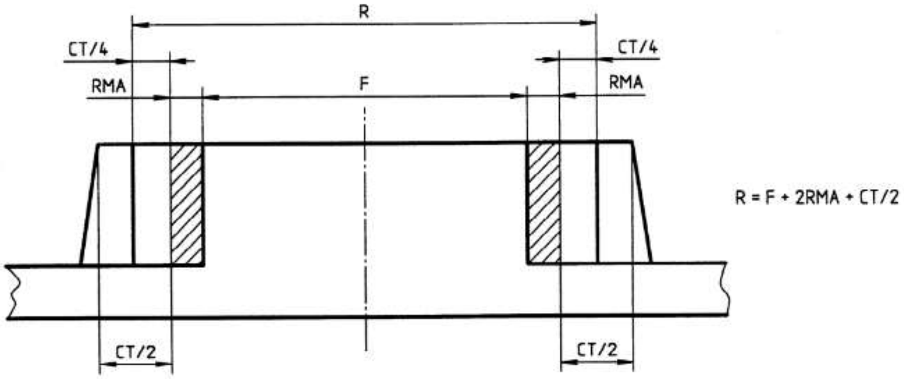
2.2.2. Casting Tolerance and Required Machining Allowance calculations

Given the part's material and relatively simple geometry, sand casting could be a suitable starting point for the manufacturing process. To estimate the required machining allowance (RMA) grade we will use Table B.1 [2]. For the sand-casting process and Grey Iron, the recommended RMA grade is F.

Required machining allowance according to the F grade and the largest dimension of a part 130mm (see drawing) is 2 mm according to the table 2 [2].

To estimate casting tolerance (CT) grade we will use table A1 (for long series) [2]. For the sand-casting process and the Grey Iron, the CT10 could be applied. The results of estimation of casting tolerances are presented in Table.3.

The sketches of RMA and CT location are presented in Fig.5.



R = Raw casting basic dimension

F = Dimension after final machining

RMA = Required machining allowance

CT = Casting tolerance

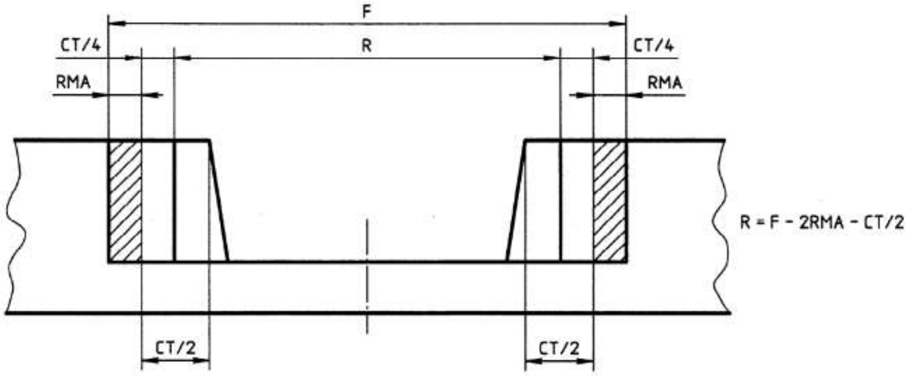


Fig.5. CT an RMA illustration

Table.3. CT and RMA Tolerances

Dimension of a part	RMA	Min limit of size for external features (or max for internal features)	Casting tolerance, mm	Raw casting basic dimension

90	2	94	3,6	95,8±1,8
55	2	54	3,2	55,6±1,6
⌀50	2	⌀46	3,2	44,4±1,6
15	2	19	2,4	20,2±1,2

We considered the following recommendations when designing the Sand Casting blank:

- a workpiece is placed in the way that corresponds to the lowest possible height in the mold;
- the parting line lies within the plane of symmetry;
- the casting does not contain sharp corners; radii of 2-5 mm were applied;
- a draft angle of 2° was applied to all walls perpendicular to parting plane to facilitate removing the part from the mold;
- the RMA should be added only to the surfaces, for which the secondary process (machining) will be applied;
- the 1 center holes and a main pocket of the part will be obtained using cores;
- small features of the part (e.g., small holes) will be obtained by a secondary process.

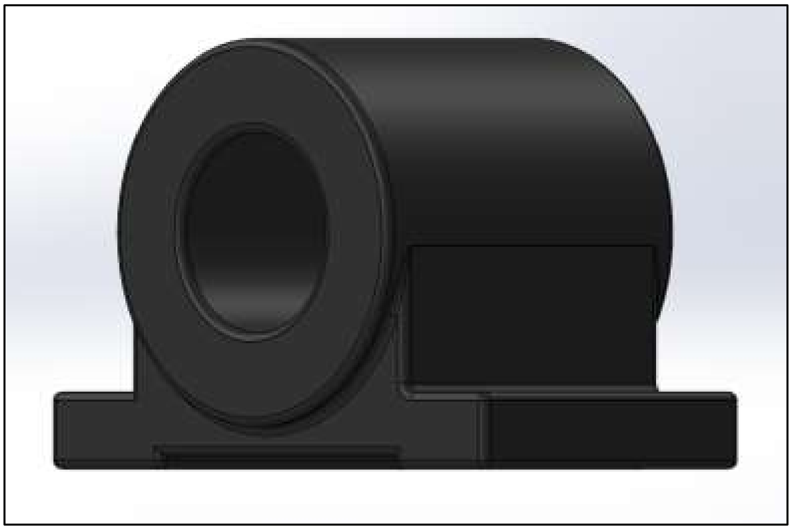


Fig.6. 3D Model of the Casting Blank

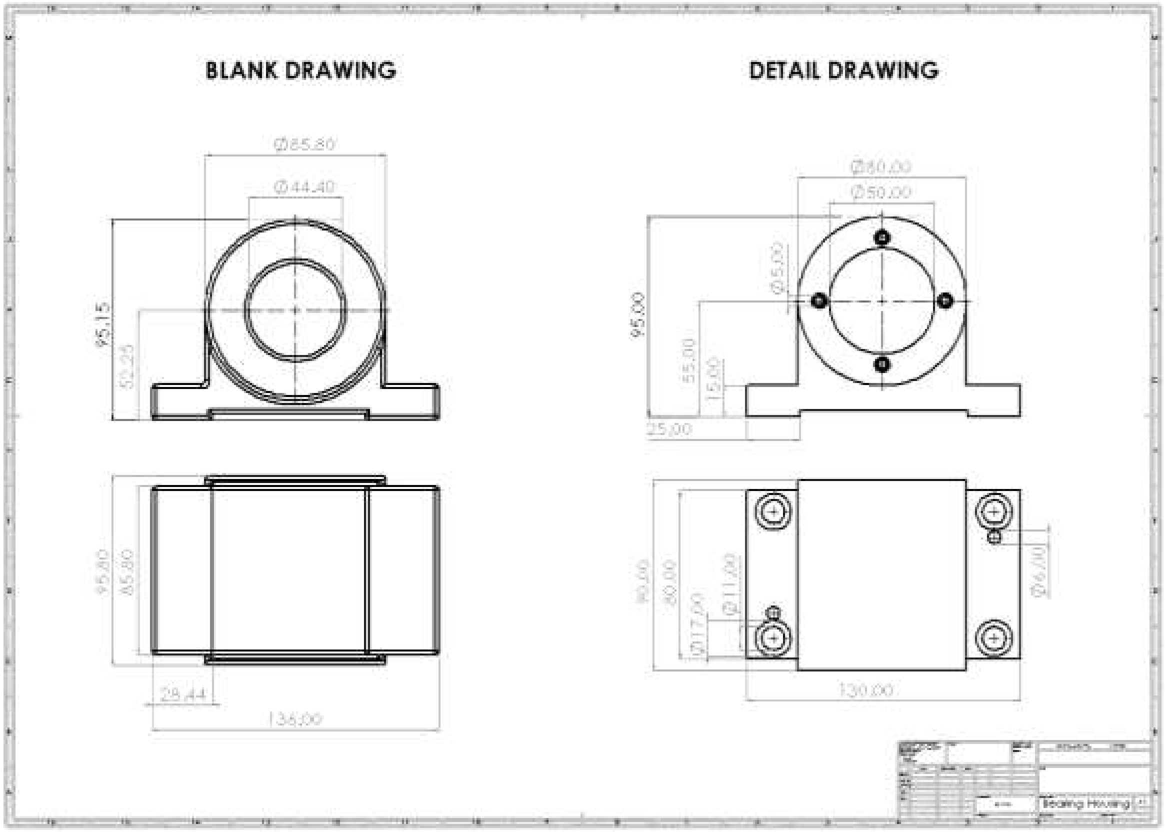


Fig.7. Technical 2D drawing of the Blank

2.3. Locating Scheme selection

The General Manufacturing Data (MD) correction algorithm consists of two stages:

- Rationale for choosing general manufacturing datum (GMD)
- Rationale for choosing a manufacturing datum for the first manufacturing operation.

2.3.1. Choosing General Manufacturing Datum (GMD)

General manufacturing datum (GMD) is a set of datum surfaces that can be used to perform all operations of the manufacturing process or most of it.

The initial data to justify the choice of GMD are the working drawing of the part. To solve the problems of the first stage, it is necessary to classify the surfaces of the part for their intended purpose. The design of any part can be represented as a set of four types of surfaces:

1. Main functional (design) datum
2. Auxiliary functional (design) datum
3. Fastening surfaces
4. Free surfaces

For further analysis let’s classify surfaces of a given part according to their purpose:

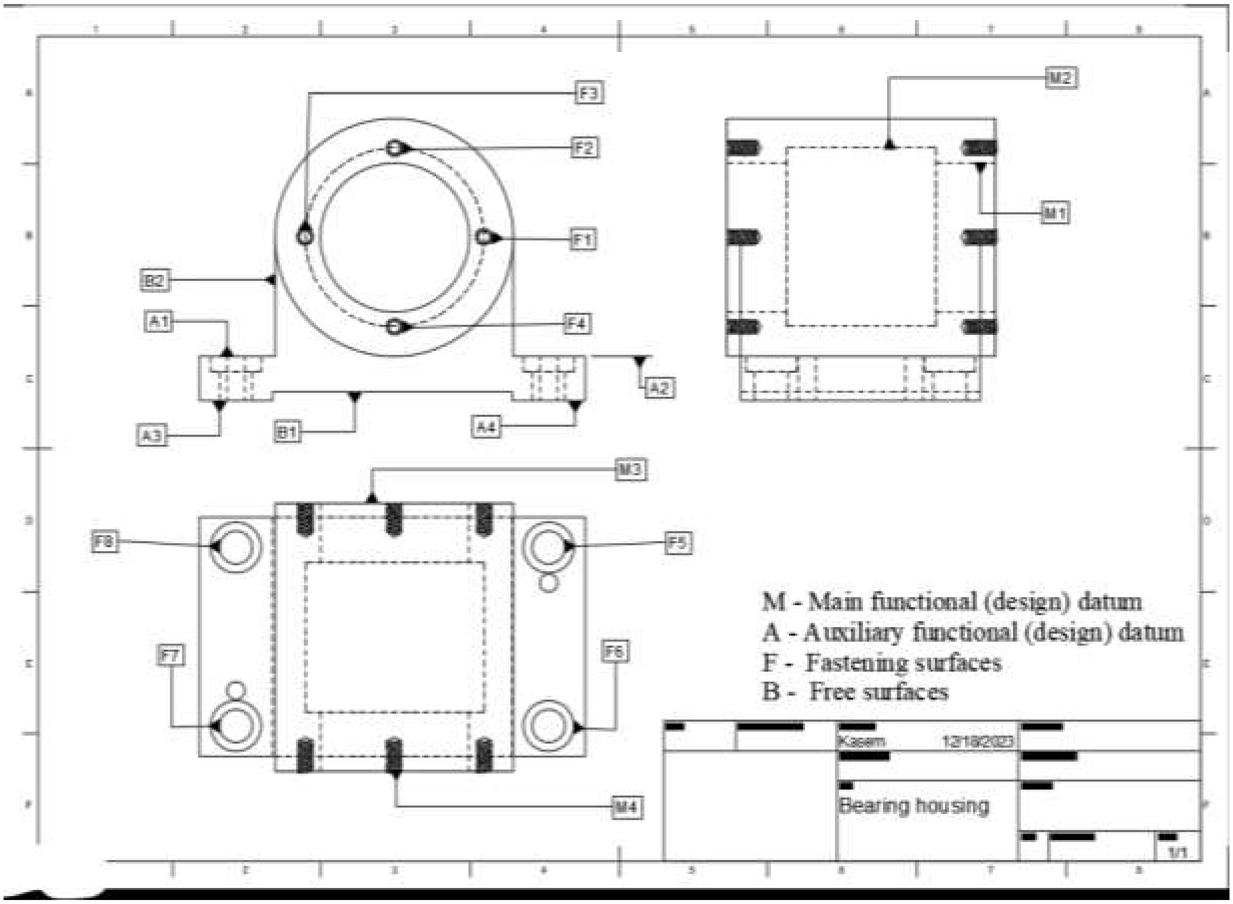


Fig.8. Surfaces Classification

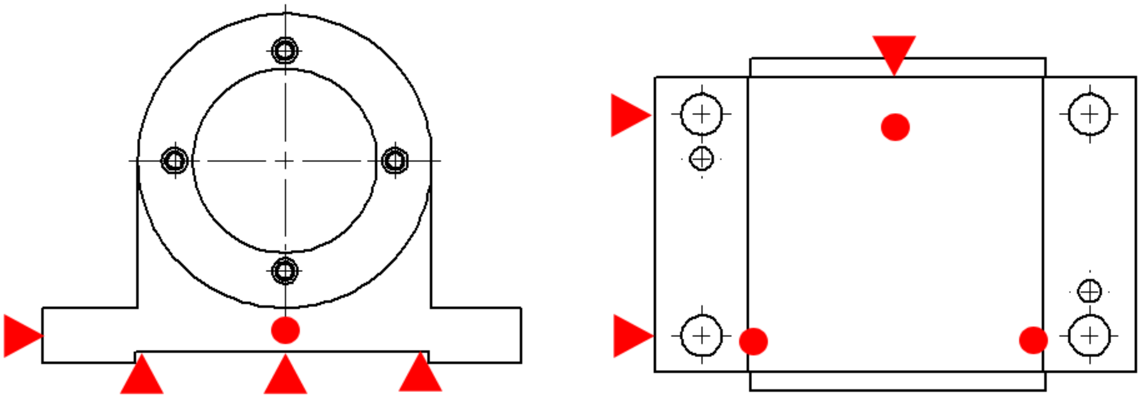


Fig.9. GMD Locating scheme

The formula for the locating scheme presented in Fig.8 is as follows:

$$LS_{GMD} = S(3) + DS(2) + O(1)$$

where S(3) – setting datum, deprives the workpiece 3 degrees of freedom,
DS (2) – double support datum, deprives the workpiece 2 degrees of freedom,
and O(1) – support datum, deprives the workpiece 1 degree of freedom.

This scheme is implemented using: a plane, round head and diamond head locating pins. In this case, the "Bearing Housing" is sufficiently oriented, which allows processing its surfaces with the specified requirements for the spatial position. In our case GMD remains unchanged.

$$GMD = Const$$

2.3.2. Choosing MD for the first manufacturing operation

When choosing datum surfaces for the first manufacturing operations it is necessary to ensure openness for processing of all surfaces of GMD and to choose machines that can carry out consecutive processing of GMD surfaces for achievement of the set quality characteristics. Otherwise, it is necessary to take into account that the full set of the GMD has to be processed during next first technological operations.

Let’s consider possible locating schemes for the first manufacturing operations as well as their advantages and disadvantages. For this purpose, we will use the following recommendations:

- for MD select surfaces that aren’t supposed to be processed according to the drawing
- if all surfaces of the workpiece have to be processed, then as MD we take the surfaces that have the lowest allowance, if the allowances are uniform, it is necessary to choose surfaces on which defects are not allowed;
- choose as MD surfaces for which it is necessary to provide a uniform allowance for the next stages of processing;
- if there are several possible schemes of basing, then as MD we accept the option with the shortest dimensional chain.

The first variant is presented in Fig.9.1.

Advantages:

- Easy to implement.
- Ensures the correct placement of untreated surfaces related to the treated ones.

Disadvantages:

- Blocks processing the workpiece from 2 sides (as it is cylindrical).
- Does not ensure the alignment of the perpendicularity of the main hole.

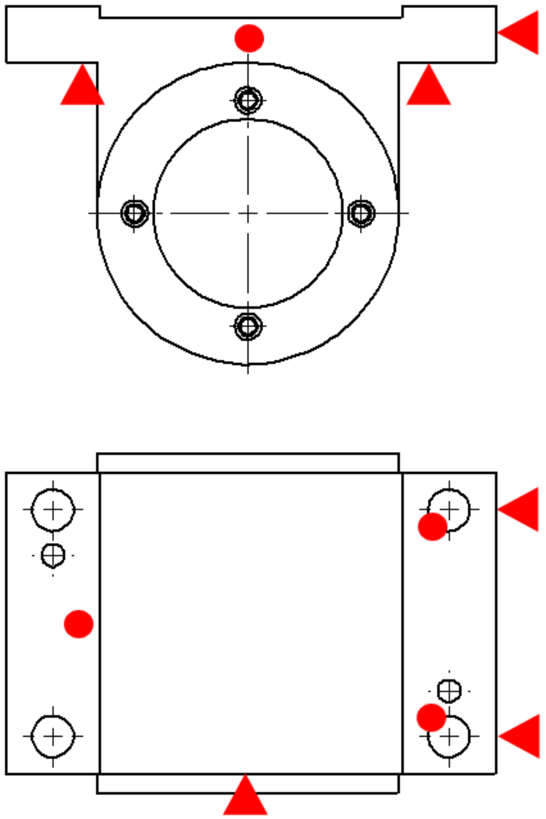


Fig.10. Locating scheme for first manufacturing operation MD

Conclusion: The first locating scheme is easy to implement, and provides the correct spatial position of the untreated surfaces relative to the processed surfaces. The given scheme allows processing several additional surfaces besides the general manufacturing datum during the first manufacturing operation. Therefore, we will use the second variant of locating scheme for processing general manufacturing datum.

2.4. Design of the typical surfaces processing routes

The design of a part can be divided into a set of typical geometric shapes, united by a common service purpose of the part. Typical structural elements are: cylindrical or conical external and internal surfaces, a set of planes, shaped surfaces - screw, involute and others. Depending on the type of surface, different cutting tools can be used to achieve a given surface accuracy and, as a result, there are different sequences of surface treatment.

The development of machining routes for individual surfaces is the first of seven tasks solved in the design of process plan. The manufacturing process thus created, rolled up in time and space, solves the problems of dimensional accuracy, shape and quality of individual surfaces, but does not take into account the accuracy of the relative position at all. This task will be solved later by assigning the locating schemes and dividing the processing stages into modules - rough, finish and final.

When developing a manufacturing process, it is necessary to select one of several possible machining options, which will provide the best economic solution. Therefore, in order to save time, it is necessary to use standard, proven in practice, processes for manufacturing parts and machining their main surfaces.

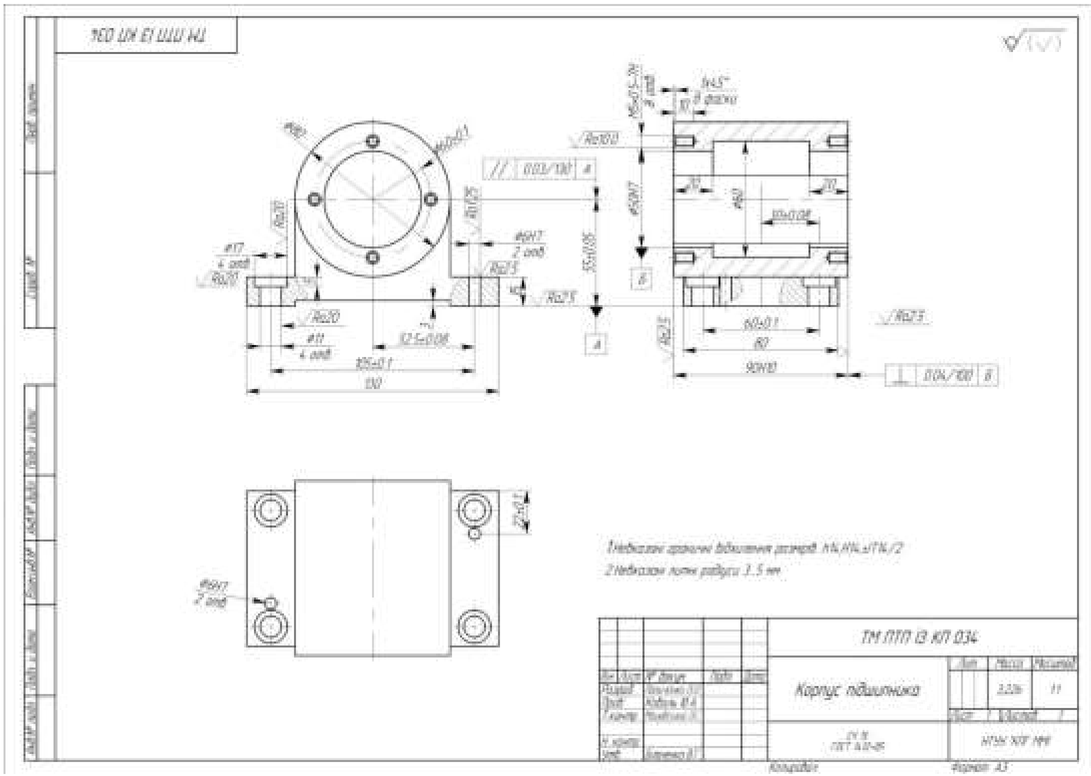


Fig.11. The part “Bearing Housing”

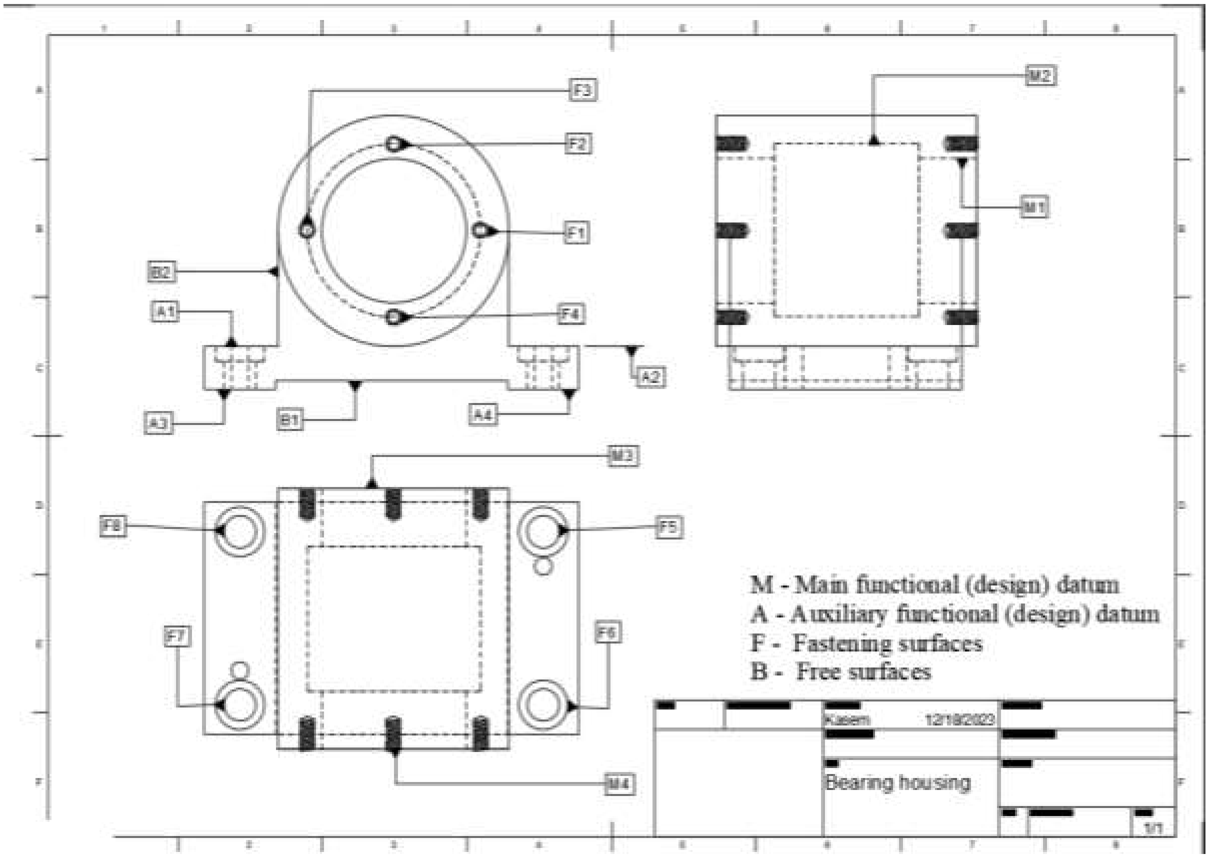


Fig.12. Surfaces Classification

Table.4. Machining Sequences

Surfaces	IT	Ra	Machining Sequence	IT	Ra
	According to the drawing			After Machining	
1	2	3	4	5	6
M1	14	-	Centering Boring	14	-
M2	14	-	Centering Boring	14	-
M3, M4	14	2,5	Rough Milling Finish Milling	14	2,5
(F1, F2, F3, F4)*2	7H	10	Centering Drilling Countersinking Threading	7H	10

A1, A2	14	2,5	Rough Milling Finish Milling	14	2,5
F4, F5 F6, F7	H7	20	Centering Drilling Counterboring	H7	20
A3, A4	14	-	Rough Milling Finish Milling	14	-

2.5. Design of the operational manufacturing process plan

Let’s consider the following recommendations:

1. Surfaces that are the datums for the subsequent stages of processing should be processed first
2. Each subsequent manufacturing step or operation must improve the quality characteristics of the treated surfaces
   - If this requirement is not met, e.g. when implementing heat treatment, then it is necessary to return to the processing of the workpiece surfaces, which are datum for subsequent processing stages.
3. The roughing must be separated from the next stages of processing by a certain period of time, or aging operations should be provided, especially for critical, large-sized and high-value parts.
4. For timely detection of defects on surfaces where they are not allowed, these surfaces should be processed at the early stages of the manufacturing process.
5. During roughing the first should be processed surfaces that have the highest allowance and the most responsible surfaces
6. Finishing of the most responsible surfaces must be performed at the latest manufacturing steps.
7. The surfaces which least reduce the overall stiffness of the workpiece should be processed first
8. Surfaces with a precise relative spatial position should be processed in one installation
9. Do not change the tool when finishing precise responsible surfaces
10. Fastening surfaces must be processed at the 3rd stage of the manufacturing process, after finishing the related surface

005 Multipurpose

Machine: DMG Mori DMU 50 (5 Axis CNC Machine {x, y, z, b, c})

Position 1 (GMD Locating)

005.01 Rough Milling surface M3
005.02 Finish Milling surface M3 to dimension
005.03 Rough Milling surface M4
005.04 Finish Milling surface M4 to dimension

010.01 Rough Milling the surfaces A1 and A2
010.02 Finish Milling the surfaces A1 and A2 to dimension

015.01 Drill holes F4, F5, F6 and F7
015.02 Counterboring the holes F4, F5, F6 and F7

+++ Turn the table 90^o counterclockwise along axis "b" , and 90^o clockwise along axis "c" +++

020.01 Centering the hole M1
020.02 Reaming the hole M1 to dimension ⌀50
020.03 Reaming the core hole M2 to dimension ⌀60 (the inner hole)

025.01 Centering the holes F1, F2, F3 and F4
025.02 Drilling the holes F1, F2, F3 and F4 to dimension, "⌀M5x0.5-7H"
025.03 Countersinking the holes F1, F2, F3 and F4
025.04 Tapping the holes F1, F2, F3 and F4, "⌀M5x0.5-7H"

+++ Turn the table 180^o along axis "c" +++

*** NB: The holes F' are the holes located on the surface M3 (opposite side) **
030.01 Centering the holes F'1, F'2, F'3 and F'4
030.02 Drilling the holes F'1, F'2, F'3 and F'4 to dimension, "⌀M5x0.5-7H"
030.03 Countersinking the holes F'1, F'2, F'3 and F'4
030.04 Tapping the holes F'1, F'2, F'3 and F'4, "⌀M5x0.5-7H"

Position 2 (Remove, and reinstall workpiece according to MD Locating)

035.21 Rough Milling the surfaces A3 and A4
035.22 Finish Milling the surfaces A3 and A4 to dimension

2.6 Machine and Tooling Selection

2.6.1. Machine Selection

Type and size of machine

The types of machines are specified by the already preselected manufacturing processes. For example, if turning is the selected process then a lathe (or turning center) will be the type of machine to be used.

At the first cut selection the only factor considered is the physical size of the machine in relation to the workpiece. E.g. a lathe whose machine bed is shorter than that of the length of the part cannot be used to turn that part.

Power/Force analysis

After having calculated the power requirements for all operations, those machines that cannot meet the maximum power requirement can be discounted.

The exception of this is if there are no other machines available. In this case, reducing feeds and speeds and/or the depth of cut can reduce the power required.

On the other hand, those machines with a far greater power output than required can also be discounted. The only exception of this is if such a machine has a higher spindle speed required by one or more operations.

Capability analysis

The factors considered in the capability analysis are the dimensional and geometric accuracy and the surface finish required.

Operational analysis

The operational factor to be considered by the process planner is that of the batch size. Those machines that do not meet the economic batch quantity should be discounted.

Taking into account all the previously discussed requirements, limitations, and the process plan outlined in the prior chapter, the TAJMAC-ZPS H500 horizontal machining center has been identified as the most suitable machine for this application.

The horizontal machining center in the H 500 version (see general technical data in fig. 8.1) is a highly productive machine for the complex chip machining of parts from the steel, grey cast iron and soft metal alloys clamped on the rotary table. It enables to perform the milling operations in three mutually perpendicular X, Y, Z coordinate axes and in the rotary B axis. It also enables to perform the drilling, boring, reaming and thread cutting operations as well as the usage of the screw die heads without aligning bush in the Z axis.

		DMU 50
Working area		
Travel X / Y / Z	mm | *in*	500 / 450 / 400 | *19.7/17.7/15.7*
Main drive (standard)		
Speed range	rpm	20 – 14.000
Drive power (100 / 40 % DC)	kW | *hp*	14,5/20,3 | *19.4/27.2*
Torque (40 % DC)	Nm | *ft-lbs*	121 | *89.2*
Main drive (optional)		
Speed range	rpm	20 – 18.000
Drive power (100 / 40 % DC)	kW | *hp*	25/35 | *33.5/46.9*
Torque (40 % DC)	Nm | *ft-lbs*	130| *95.9*
Feed		
Rapid traverse X / Y / Z	m/min | *ipm*	30 | *1,181*
Maximum thrust force X / Y / Z	kN | *lbf*	4,8 | *1,079*
Fixed table		
Clamping area	mm | *in*	700 × 500 | *27.6 x 19.7*
Maximum load	kg | *lbs*	500 | *1,102*
Integrated swivelling rotary table*		
Clamping area	mm | *in*	ø 630 × 500 | *ø 24.8 x 19.7*
Maximum load	kg | *lbs*	300 | *661*
Swivel range	Degrees	–5 / +110
Optional pick-up tool change system		
Capacity	pockets	16
Tool weight	kg | *lbs*	6 | *13.2*
Maximum tool length	mm | *in*	300 | *11.8*
Maximum tool diameter	mm | *in*	80 / 130 | *3.1/5.1*
Optional tool change system with chain and double gripper		
Capacity	pockets	30 / 60
Tool weight	kg | *lbs*	6 | *13.2*
Maximum tool length	mm | *in*	300 | *11.8*
Maximum tool diameter	mm | *in*	80 / 130 | *3.1/5.1*
Machine weight / connected values		
Weight	kg | *lbs*	4.480 | *9,880*
Power	kW | *hp*	21 | *28.2*
Maximum current rating	A	31

Fig.13. Technical data of the selected machine

2.6.2. Tooling Selection and Cutting Conditions

Evaluation of process and machine selections – Provided the selection of processes and machines is satisfactory, the range of tools that can be used should be limited to those suitable for the processes and machines selected. Therefore, this limits the initial list of possible suitable tooling.

Analysis of machining operations – A specific machine will carry out every operation required. Each machine tool to be used will have specific tool types to carry our certain operations. This analysis should enable the identification of specific tool types for specific operations.

Analysis of workpiece characteristics – At this step the following should be considered: workpiece material and geometry, dimensional and geometric accuracy, and surface finish. This enables to identify suitable tool materials and geometry.

Tooling analysis – Using the tooling data available, the general tooling specifications generated ant the 3rd stage can be translated into a statement of tooling requirements for the job, that is, a tooling list. This will obviously reflect whatever tooling is actually available for the operations required.

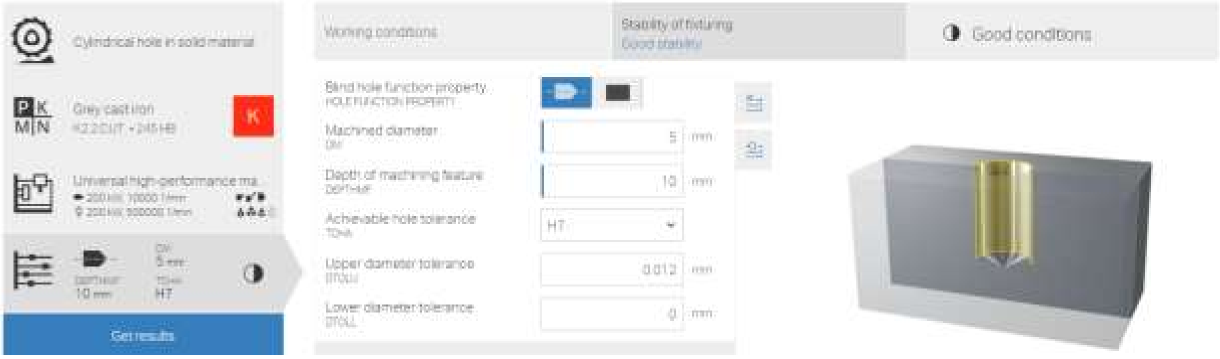
Selection of tooling – If single-piece tooling is being used, then a suitable toolholder should be selected before fully defining the tool geometry and material.

If insert-type tooling is being used then the following steps should be followed:

- Select clamping system;
- Select toolholder type and size;
- Select insert shape;
- Select insert size;
- Determine tool edge radius;
- Select insert type;
- Select tool material.

To select the appropriate cutting tool and cutting conditions we will use CoroPlus® ToolGuide [1] Firstly, enter the initial data, incl. type of surface, depth of cut, radial cutting width and workpiece material.

Tool selection for the manufacturing step
005.10 Drilling the holes F1, F2, F3 and F4 to dimension, ”⌀M5x2.5-7H”



Cylindrical hole in solid material

Grey cast iron
K2.2.C.UT • 245 HB

Universal high-performance ma
200 kW, 10000 1/min
200 kW, 500000 1/min

DEPTHMF 10 mm
DM 5 mm
TCHA H7

Working conditions

Stability of fixturing
Good stability

Good conditions

Blind hole function property
HOLE FUNCTION PROPERTY

Machined diameter
DM: 5 mm

Depth of machining feature
DEPTHMF: 10 mm

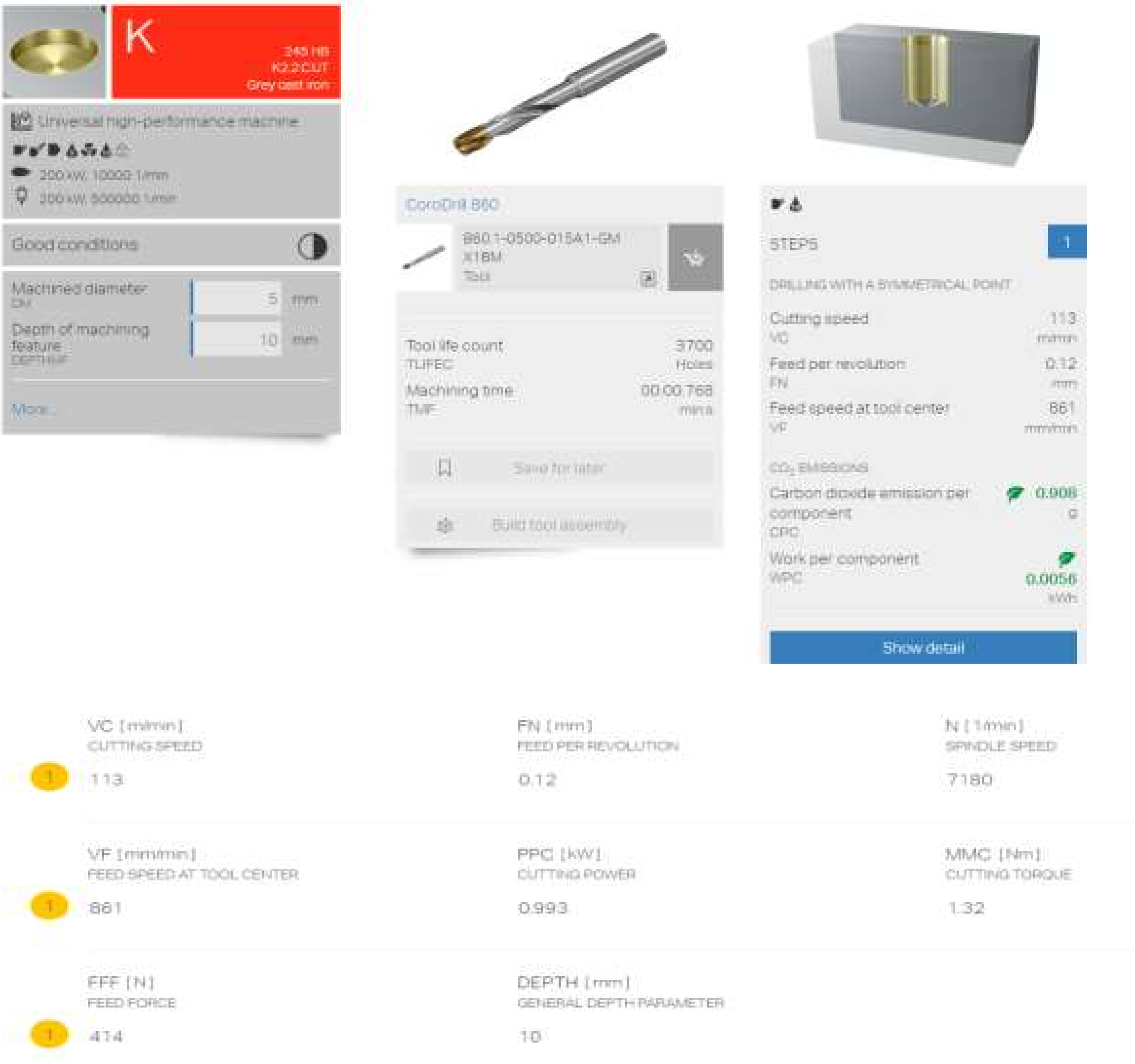
Achievable hole tolerance
TCHA: H7

Upper diameter tolerance
DTOLU: 0.012 mm

Lower diameter tolerance
DTOLL: 0 mm

Fig.14. Initial Data

After applying the initial data, consider the results of analysis: recommended cutting tool and cutting conditions



245 HB
K2.2.C.UT
Grey cast iron

Universal high-performance machine
200 kW, 10000 1/min
200 kW, 500000 1/min

Good conditions

Machined diameter
DM: 5 mm

Depth of machining feature
DEPTHMF: 10 mm

CoroDrill 860

860.1-0500-015A1-GM
X1BM
Tool

Tool life count
TLIFEC: 3700 Holes

Machining time
TMF: 00:00.768 min:s

STEPS: 1

DRILLING WITH A SYMMETRICAL POINT

Cutting speed
VC: 113 m/min

Feed per revolution
FN: 0.12 mm

Feed speed at tool center
VF: 861 mm/min

CO_2 EMISSIONS

Carbon dioxide emission per component
CPC: 0.908 g

Work per component
WPC: 0.0056 kWh

1	VC [m/min] CUTTING SPEED 113	FN [mm] FEED PER REVOLUTION 0.12	N [1/min] SPINDLE SPEED 7180
1	VF [mm/min] FEED SPEED AT TOOL CENTER 861	PPC [kW] CUTTING POWER 0.993	MMC [Nm] CUTTING TORQUE 1.32
1	FFF [N] FEED FORCE 414	DEPTH [mm] GENERAL DEPTH PARAMETER 10	

Fig.15. Recommended Cutting Tool and Cutting Data

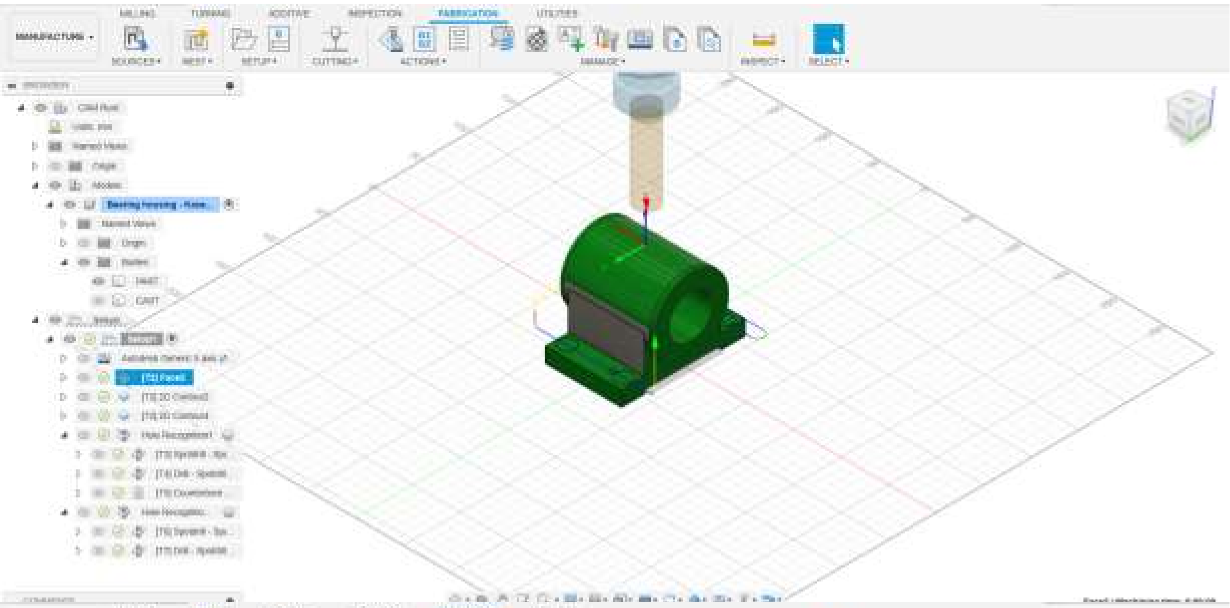
2.7. Cutting Conditions

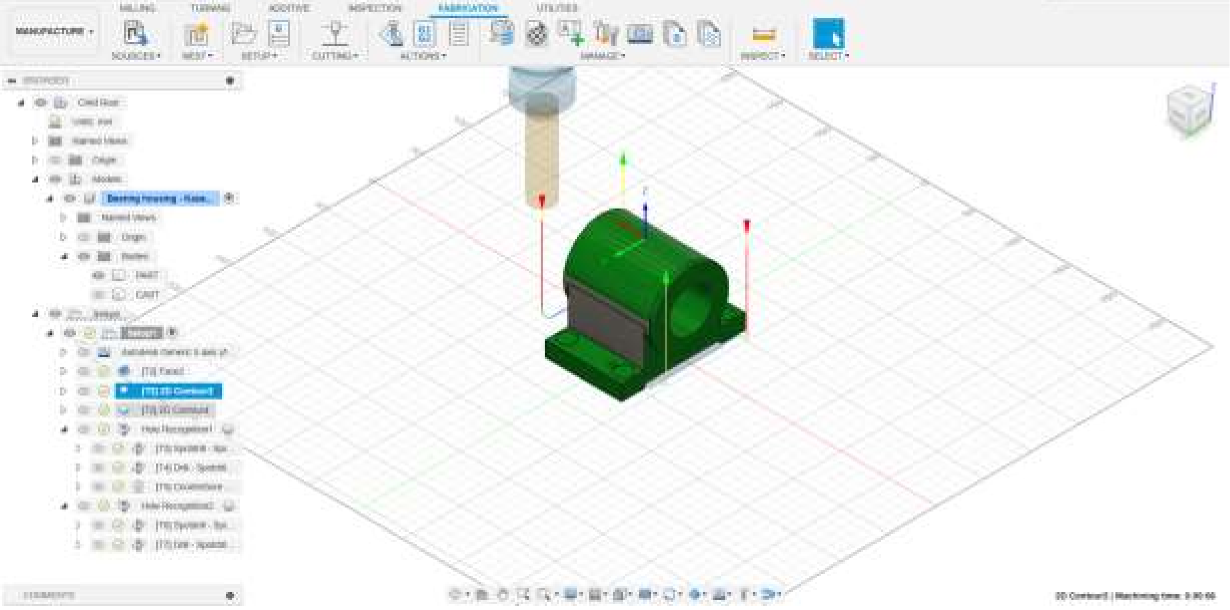
2.8. Simulation and G-Code Generation

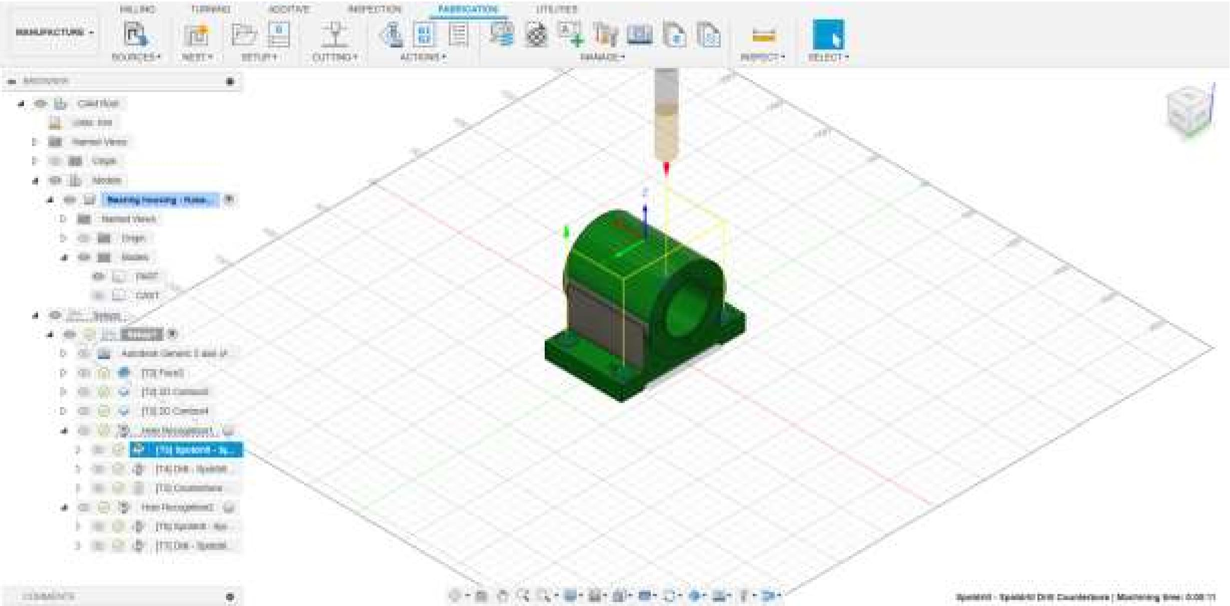
As per the operational Manufacturing Process Plan, we have created a simulation of the machining process on the Software “Autodesk Fusion 360”.

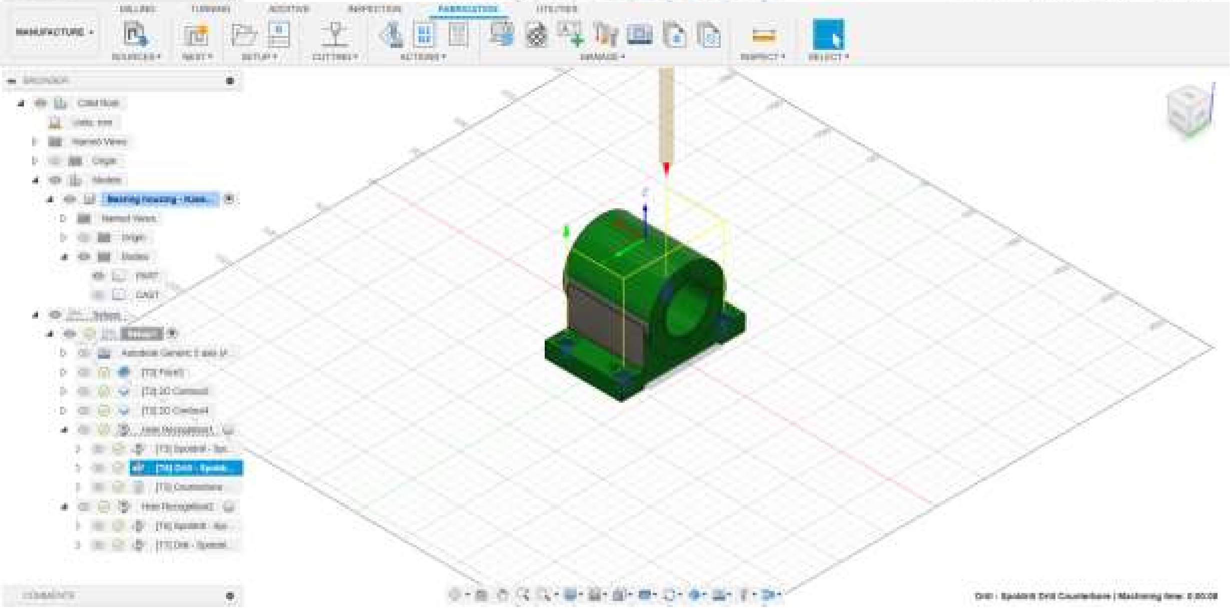
In this simulation, we have recreated the positioning of the workpiece on the Mill table, as well as all the necessary features like Cutting conditions, and used the designed Casting blank from previous section.

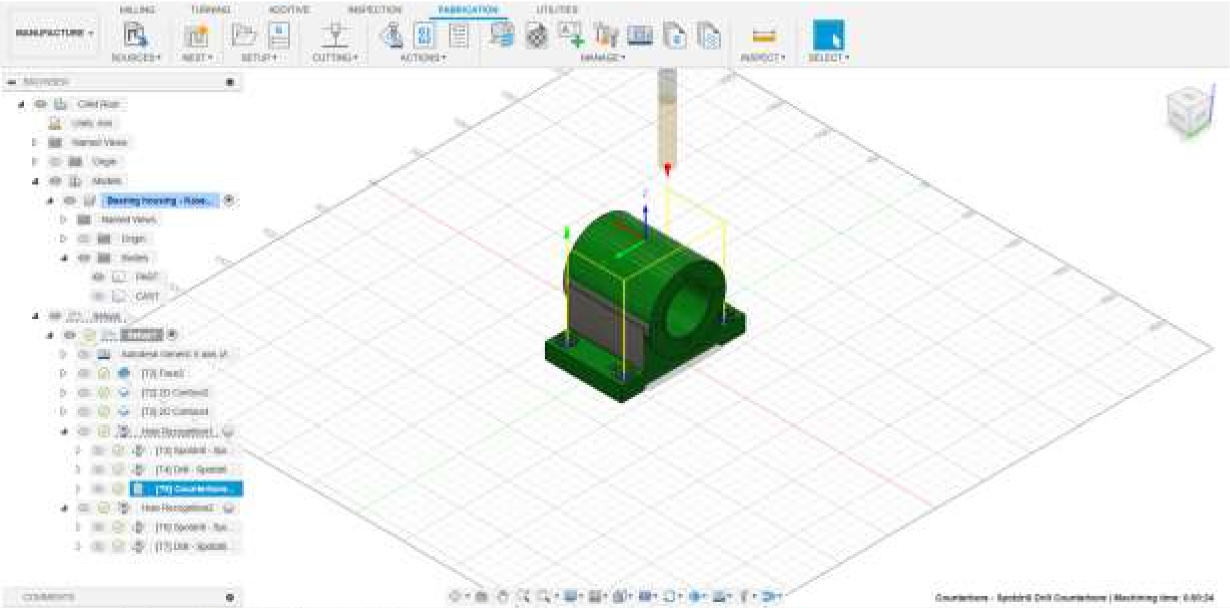
Following are screenshots from the Simulation steps taken on the Software Fusion 360.

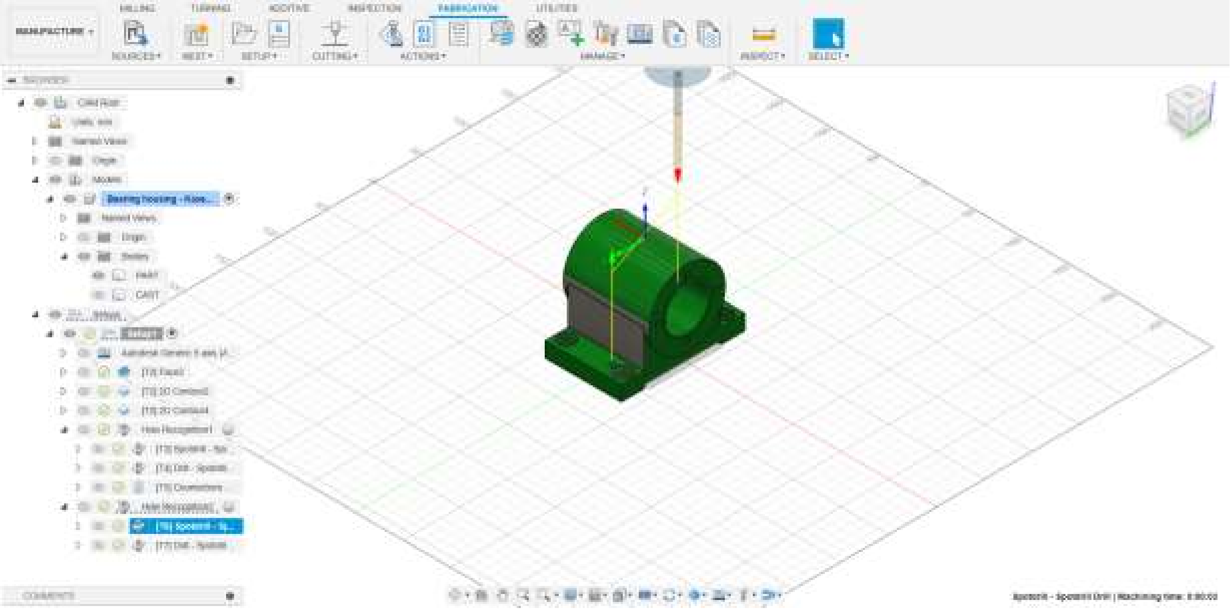

MANUFACTURE
MILLING
TURNING
ADDITIVE
INSPECTION
FABRICATION
UTILITIES
SOURCES
NEST
SETUP
CUTTING
ACTIONS
MANAGE
INSPECT
SELECT

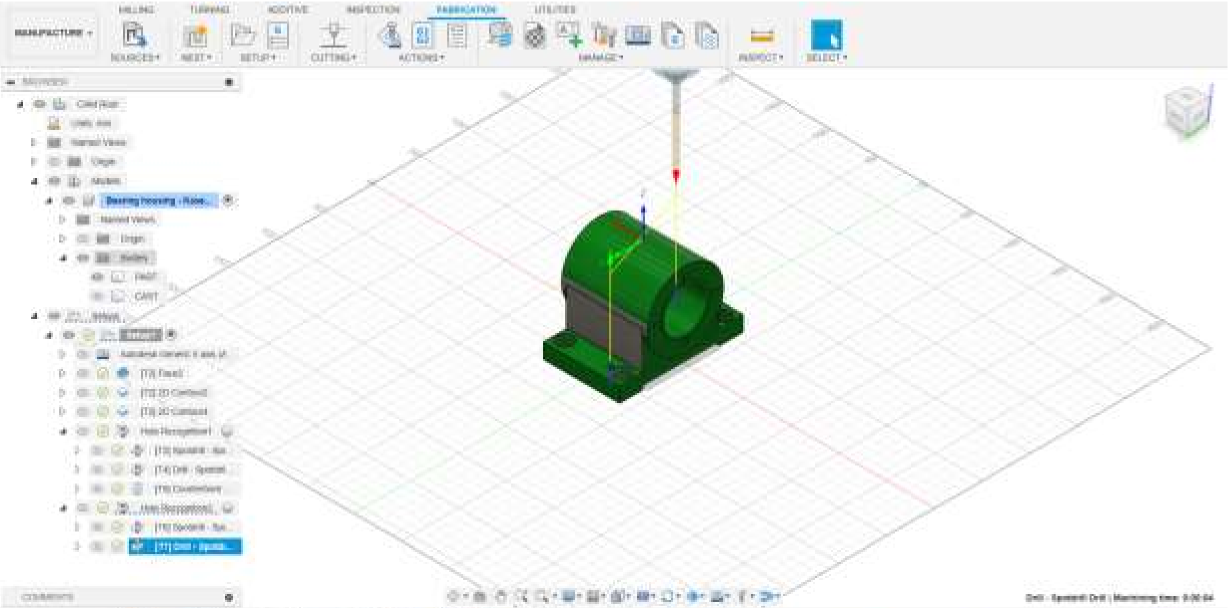

MANUFACTURE
MILLING
TURNING
ADDITIVE
INSPECTION
FABRICATION
UTILITIES
SOURCES
NEST
SETUP
CUTTING
ACTIONS
MANAGE
INSPECT
SELECT

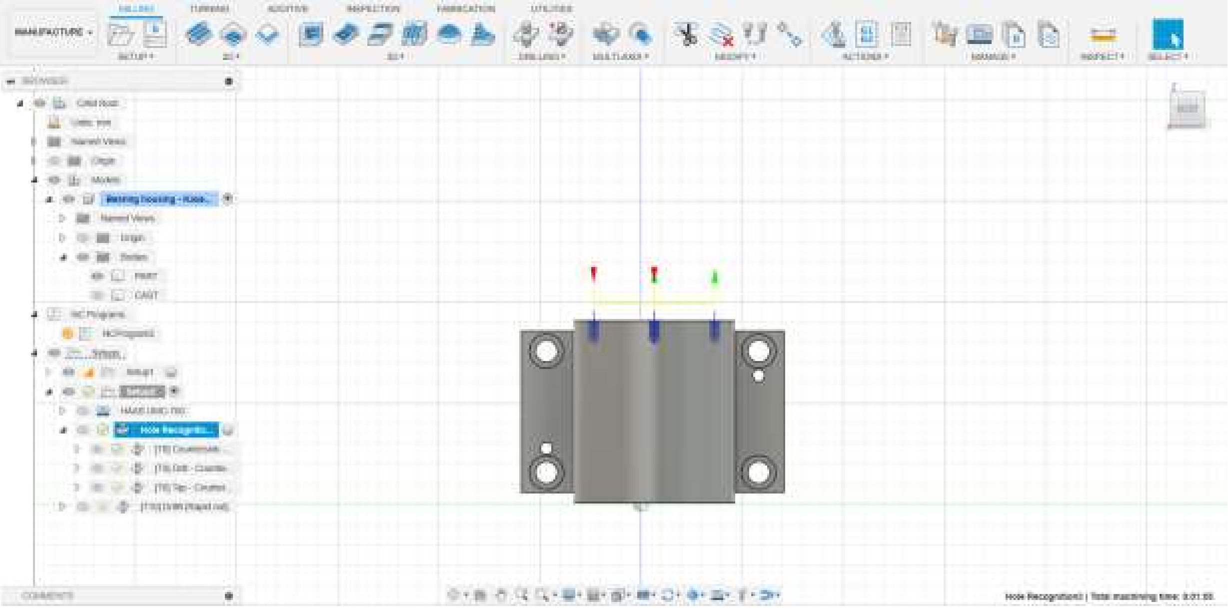


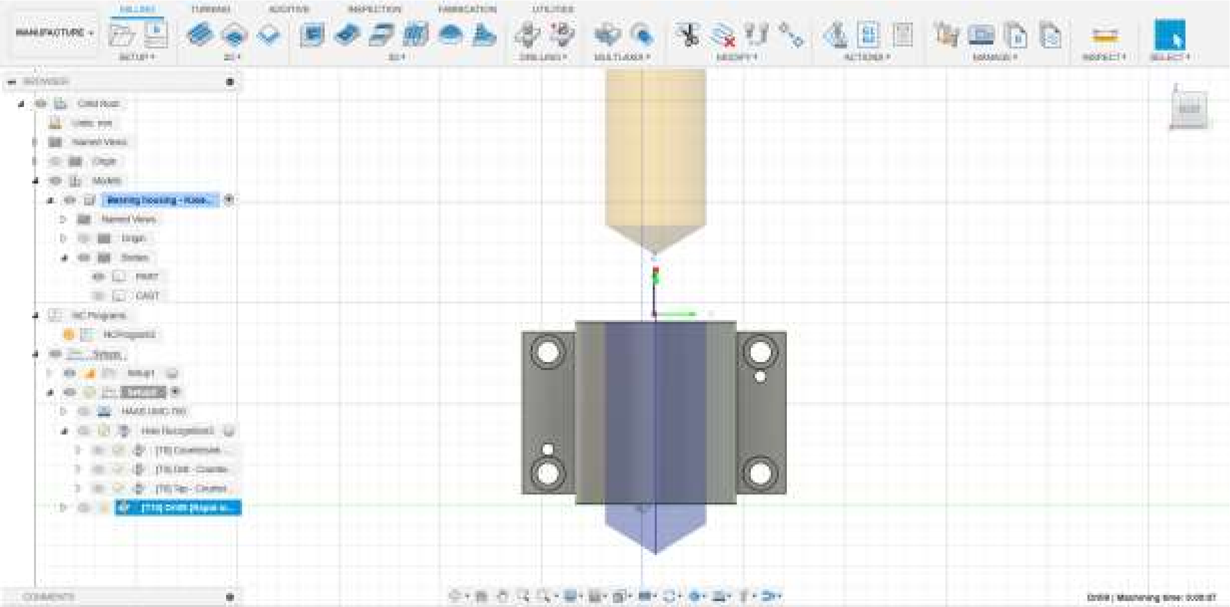


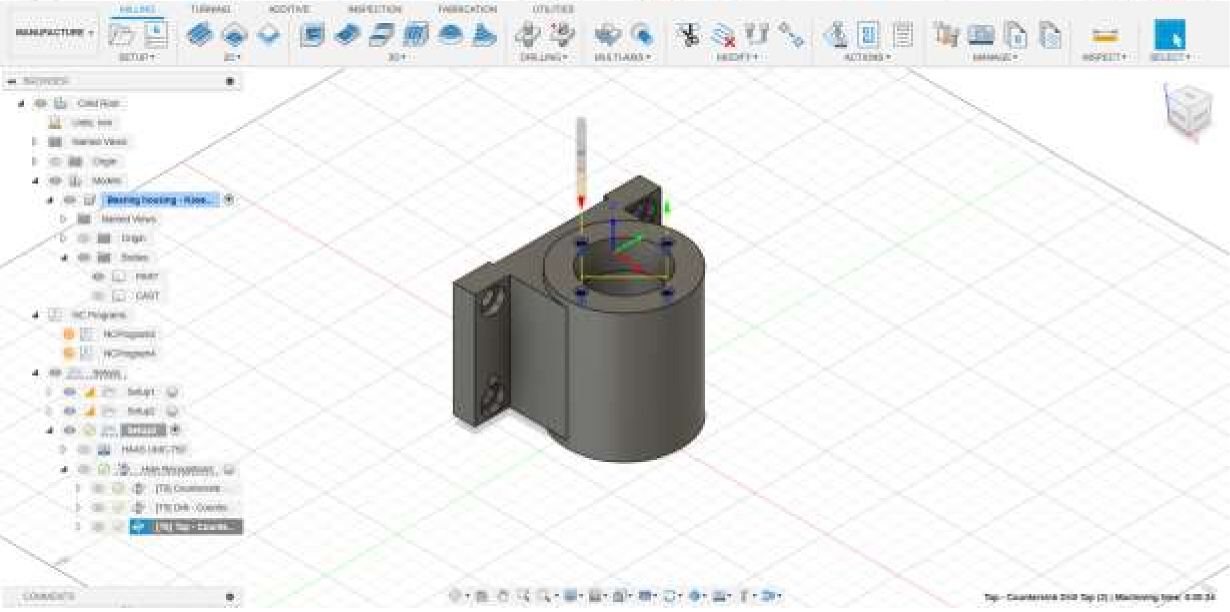





MANUFACTURE
MILLING
TURNING
ADDITIVE
INSPECTION
FABRICATION
UTILITIES
SOURCES
NEST
SETUP
CUTTING
ACTIONS
MANAGE
INSPECT
SELECT
BROWSER
COMMENTS


MANUFACTURE
MILLING
TURNING
ADDITIVE
INSPECTION
FABRICATION
UTILITIES
SETUP
2D
3D
DRILLING
MULTI-AXIS
MODIFY
ACTIONS
MANAGE
INSPECT
SELECT
BROWSER
COMMENTS





After simulating all sequences, we can finally generate a G-Code, or rather CNC Code that we would be able to use to create these machining sequences on a real machine, the CNC Code will be attached at the end of this document as Annex.

CHAPTER 3: Fixture Design

2.1. Purpose of the Fixture

This chapter focuses on the design and analysis of a critical component for achieving successful machining of the bearing housing: the fixture. As repeatability and security are paramount during the machining process, the fixture plays a vital role in ensuring the bearing housing is precisely positioned and held securely throughout all machining operations.

2.2. Possible Fixture designs and solutions

Due to the need for precise and consistent workpiece positioning during machining, a table vise has been selected as the primary fixturing method. Table vises offer several advantages that make them well-suited for this application. They provide rapid and repeatable setup due to their simple design and ease of use. This minimizes setup time and ensures consistent part location for each machining operation. Furthermore, table vises utilize a robust clamping mechanism that guarantees secure holding of the bearing housing throughout all machining steps. This secure hold is critical for maintaining dimensional accuracy and preventing part movement caused by cutting forces or vibrations.

Detailed drawings and images of these fixture designs will be provided in the following figures, illustrating their specific applications in the machining sequence of the Bearing Housing part. Each fixture has been carefully selected to optimize the manufacturing process, ensuring that each operation is conducted with the highest level of precision and efficiency.

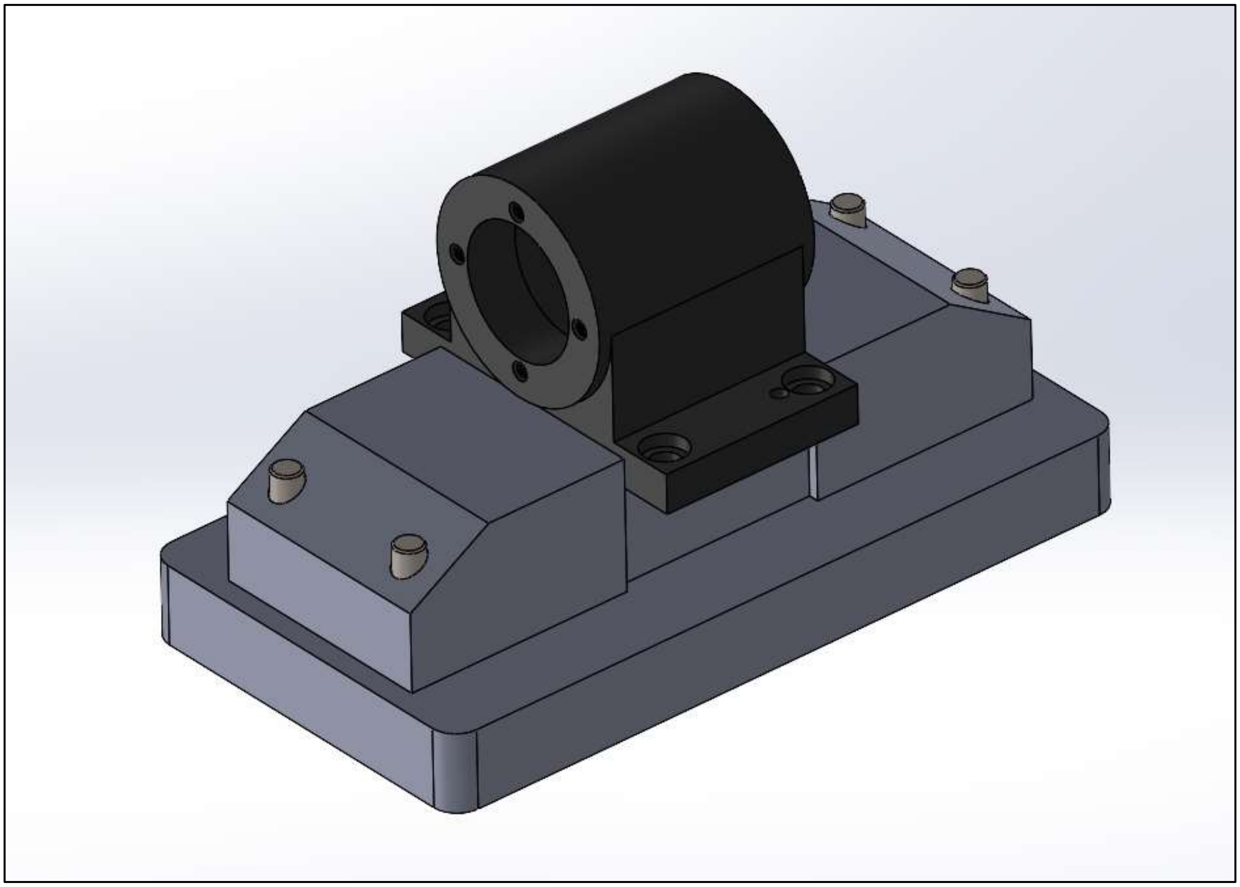


Fig.20. Vise 3D Model

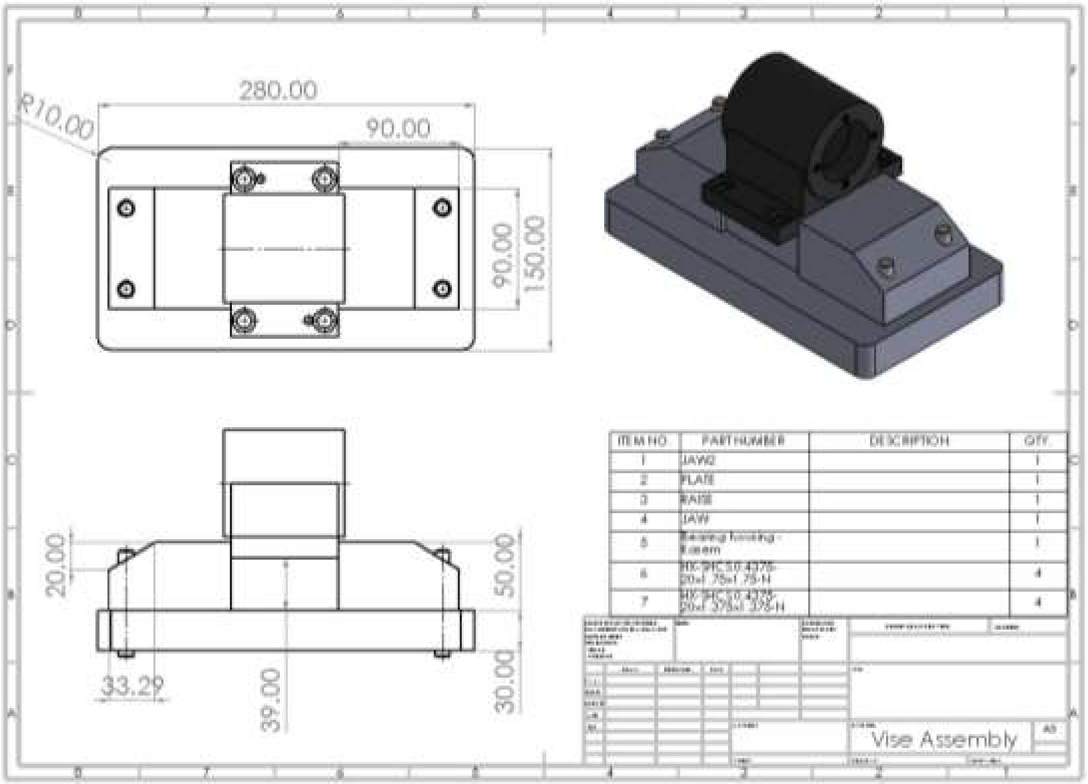


Fig.20. Drawing of the Vise

2.3. Clamping Force calculation

2.3.1. Vise Fixture

Calculating the clamping force is essential for machining operations because theoretical projections often fall short in accounting for real-world variables, such as material inconsistencies and machining dynamics. These calculations ensure that the fixture applies sufficient force to hold the workpiece securely without causing deformation. To achieve accurate and practical results, we employ the method detailed below, which considers the actual cutting forces, the coefficient of friction between the workpiece and the fixture, and other operational parameters.

To calculate the necessary clamping force for a fixture, use the equation below:

$$Clamping\ Force = {}^{Cutting\ Force}/_{Coefficient\ of\ Friction}$$

To determine the cutting force, we use the following formula:

$$Cutting\ Force = {}^{4.5 \times k \times f \times d \times b}/_{Cutting\ Speed}$$

Where:

- k: for the material constant
- f: for the feed rate
- d: for the depth of the cut
- b: for the width of the cut

As previously mentioned on Tool Selection, we have a preliminary Cutting Data to work with which will helps us calculate the required Clamping Force:

$$Cutting\ Force = 414\ N$$

We will assume a Coefficient of Friction close to 0.4, so now we can calculate the required Clamping Force

$$Clamping\ Force = {}^{Cutting\ Force}/_{Coefficient\ of\ Friction}$$

$$Clamping\ Force = {}^{414}/_{0.4} = 1{,}035\ N$$

So the required Clamping Force will tally up to the mentioned 1,035 N

CHAPTER 4: Economical Section

1. Cost Estimation

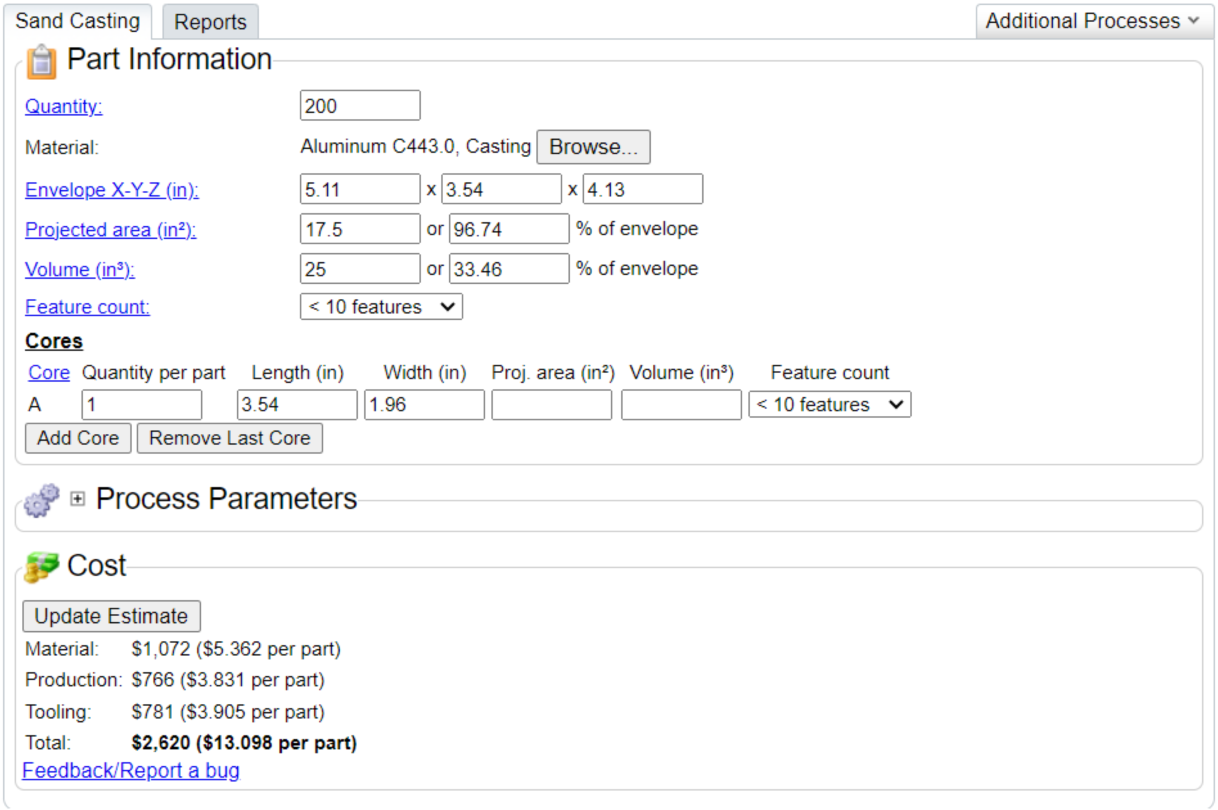
To estimate the cost of casting we will use the on-line application Cost Estimator at the customparnet.com.

Due to an error in the website, there was no option to change material to Grey Iron, in order to go further with our estimation processes we had to choose the closest material to it, in terms of price per kg, as well as density, which plays a crucial role in determining the cost of Casting.

The Estimation process will be split in two different processes:

- Casting Cost Estimation
- Machining Cost Estimation

The latter will be our guide to Time estimation in further steps of this project.



Part Information

Quantity: 200

Material: Aluminum C443.0, Casting

Envelope X-Y-Z (in): 5.11 x 3.54 x 4.13

Projected area (in²): 17.5 or 96.74 % of envelope

Volume (in³): 25 or 33.46 % of envelope

Feature count: < 10 features

Cores

Core	Quantity per part	Length (in)	Width (in)	Proj. area (in²)	Volume (in³)	Feature count
A	1	3.54	1.96			< 10 features

Cost

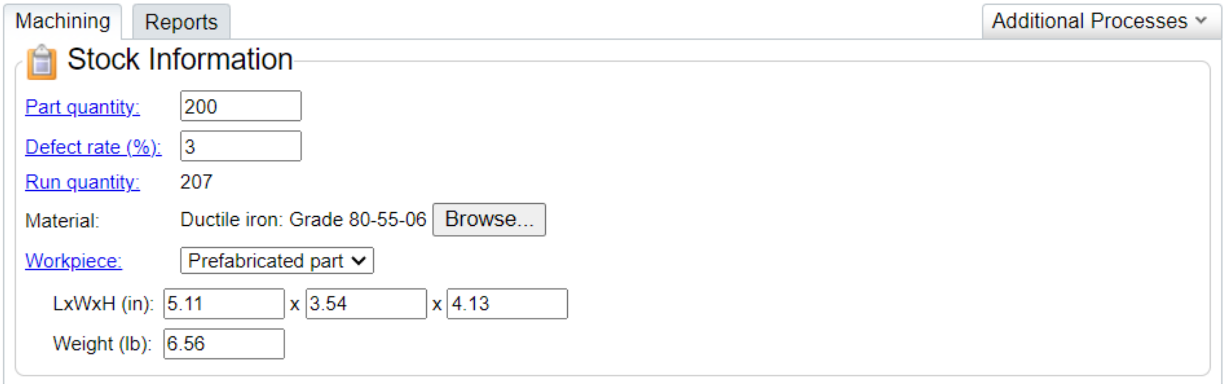
Material: $1,072 ($5.362 per part)

Production: $766 ($3.831 per part)

Tooling: $781 ($3.905 per part)

Total: **$2,620 ($13.098 per part)**

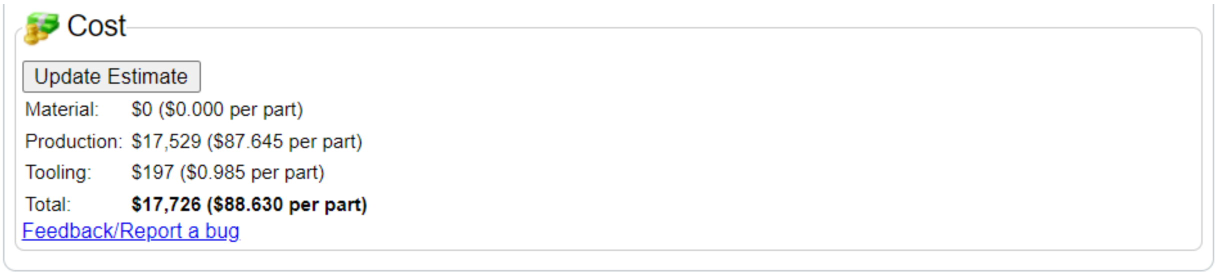
Fig.16. Sand Casting Cost Estimation



Stock Information

Part quantity: 200
Defect rate (%): 3
Run quantity: 207
Material: Ductile iron: Grade 80-55-06
Workpiece: Prefabricated part
LxWxH (in): 5.11 x 3.54 x 4.13
Weight (lb): 6.56

Fig.17. Stock Information for Machining cost estimation



Cost

Material:	$0 ($0.000 per part)
Production:	$17,529 ($87.645 per part)
Tooling:	$197 ($0.985 per part)
Total:	**$17,726 ($88.630 per part)**

Fig.18. Cost Estimation for Machining Sequences

Overall, the tally cost of producing 200 parts will be the sum of sand-casting costs and machining costs, as following:

$$C_T = C_c + C_m = 2{,}620 + 17{,}726 = \$20{,}346$$

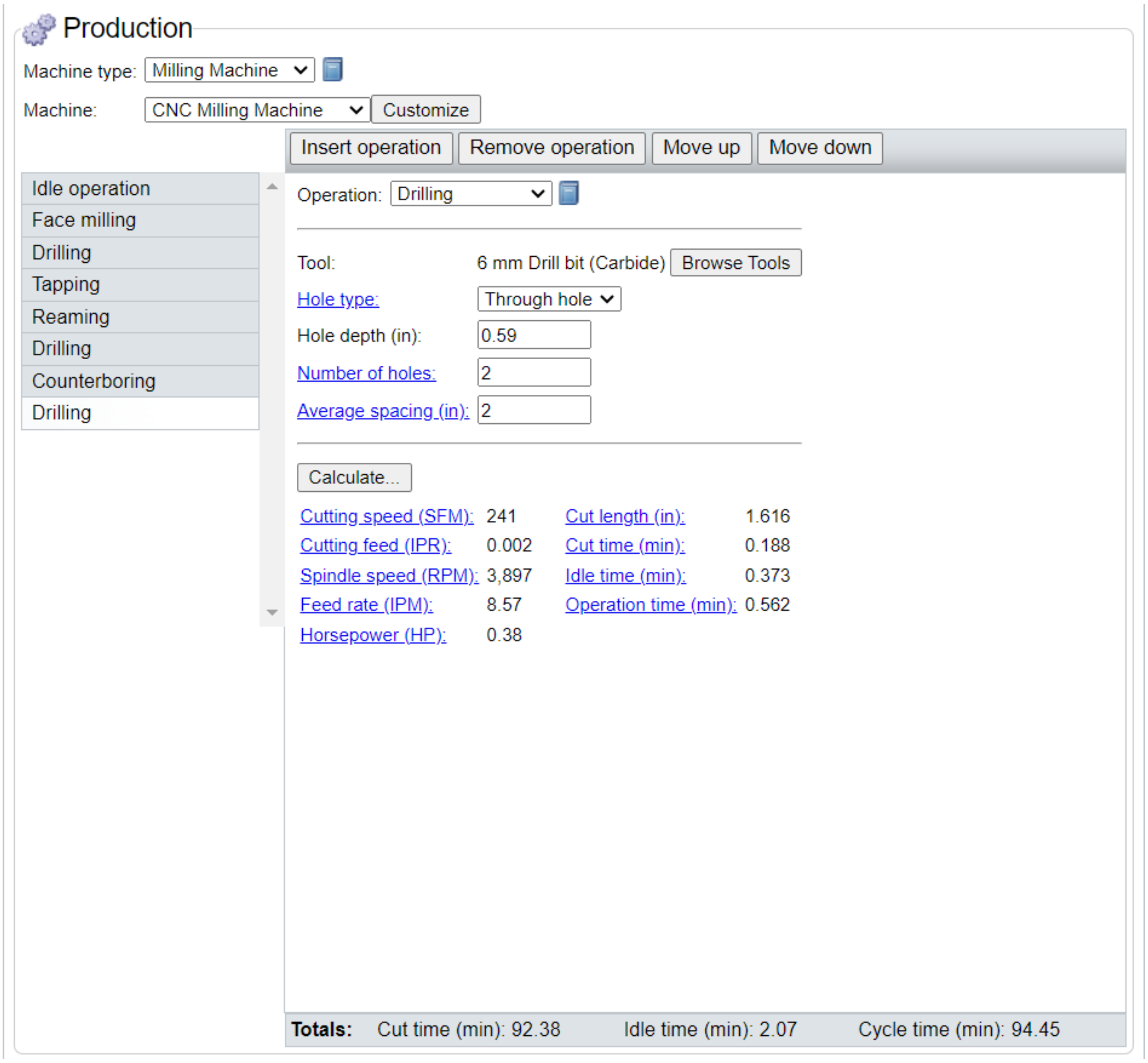
C_T: Total Cost; C_C: Cost of Casting ; C_m: Cost of Machining

Which accords to roughly: ~$101.73 per part.

2. Time Estimation

After simulating and estimating the cost of the Machining, we come up with a rough Time estimation per part, as illustrated below.

Due to Difficulties estimating Casting time we will leave that for a foundry to give us a quote, which can be done at a later date.



Machine type: Milling Machine

Machine: CNC Milling Machine

Operation: Drilling

Tool: 6 mm Drill bit (Carbide)

Hole type: Through hole

Hole depth (in): 0.59

Number of holes: 2

Average spacing (in): 2

Cutting speed (SFM):	241	Cut length (in):	1.616
Cutting feed (IPR):	0.002	Cut time (min):	0.188
Spindle speed (RPM):	3,897	Idle time (min):	0.373
Feed rate (IPM):	8.57	Operation time (min):	0.562
Horsepower (HP):	0.38		

Totals: Cut time (min): 92.38 Idle time (min): 2.07 Cycle time (min): 94.45

Fig.19. Time Estimation

The Time estimation for Machining per part will amount to the following:

The Cut Time for a total of 92.38 min
The Idle Tie for a total of 2.07 min

Therefore, the Cycle Time will be 94.45 min.

REFERENCES

- Custompartnet : https://www.custompartnet.com
- Castings - System of dimensional tolerances and machining allowances ISO 8062 [2]
- Haas CNC : https://www.haascnc.com/index.html
- Sandvik Coromant : https://www.sandvik.coromant.com
- Jig and Fixture Design – Edward G. Hofmann
- Jigs and Fixtures – P.H. Joshi
- ResearchGate
- CAD Websites
    - https://www.mcmaster.com
    - https://grabcad.com
    - https://www.traceparts.com

ANNEX

This annex is reserved for the CNC Code generation from the Simulation previously performed on Fusion 360.

```
%
O01001
(Using high feed G1 F650. instead of G0.)
(Machine)
(  vendor: DMG)
(  model: MORI U50)
(  description: Haas legacy design)
(T2 D=25. CR=0. - ZMIN=-80. - flat end mill)
(T3 D=18. CR=0. TAPER=90deg - ZMIN=-88.5 - spot drill)
(T4 D=11. CR=0. TAPER=118deg - ZMIN=-98.305 - drill)
(T5 D=14. CR=0. - ZMIN=-85. - flat end mill)
(T6 D=6.35 CR=0. TAPER=90deg - ZMIN=-83. - spot drill)
(T7 D=6. CR=0. TAPER=118deg - ZMIN=-96.803 - drill)
N10 G90 G94 G17
N15 G21
N20 G53 G0 Z0.

(Face2)
N25 T2 M6
N30 S3881 M3
N35 G17 G90 G94
N40 G54
N45 G53 G0 X-736.6 Y-203.2
N50 M11
N55 M13
N60 G0 B0. C0.
N65 M10
N70 M12
N75 M8
N80 G1 X63.01 Y-64.289 F650.
N85 G0 G43 Z-35. H2
N90 T3
N95 G0 Z-65.
N100 G1 Z-77.5 F1064.77
N105 G18 G3 X60.51 Z-80. I-2.5 K0. F3194.31
N110 G1 X-60.51
N115 G17 G2 Y-51.327 I0. J6.481
N120 G1 X60.51
N125 G18 G2 X63.01 Z-77.5 I0. K2.5
N130 G0 Z-55.
N135 Y52.899
N140 Z-65.
N145 G1 Z-77.5 F1064.77
N150 G3 X60.51 Z-80. I-2.5 K0. F3194.31
N155 G1 X-60.51
N160 G3 X-63.01 Z-77.5 I0. K2.5
N165 G0 Z-35.

(2D Contour2)
N170 G17
N175 G0 Z15. F650.
N180 G1 X65. Y43.298 F650.
N185 G0 Z15.
N190 Z5.
N195 G1 Z1. F1064.77
N200 Z-77.5
N205 G18 G3 X62.5 Z-80. I-2.5 K0. F3194.31
N210 G1 X60.
N215 G17 G3 X57.5 Y40.798 I0. J-2.5
N220 G1 Y-39.202
N225 G3 X60. Y-41.702 I2.5 J0.
N230 G1 X62.5
N235 G18 G2 X65. Z-77.5 I0. K2.5
N240 G0 Z15.

(2D Contour4)
N245 G17
```

```
N250 G1 X-65. Y-41.702 F650.
N255 G0 Z15.
N260 Z5.
N265 G1 Z1. F1064.77
N270 Z-77.5
N275 G18 G2 X-62.5 Z-80. I2.5 K0. F3194.31
N280 G1 X-60.
N285 G17 G3 X-57.5 Y-39.202 I0. J2.5
N290 G1 Y40.798
N295 G3 X-60. Y43.298 I-2.5 J0.
N300 G1 X-62.5
N305 G18 G3 X-65. Z-77.5 I0. K2.5
N310 G0 Z15.
N315 M9
N320 M5
N325 G53 G0 Z0.

(Spotdrill - Spotdrill Drill Counterbore)
N330 M1
N335 T3 M6
N340 S5000 M3
N345 G17 G90 G94
N350 G54
N355 G53 G0 X-736.6 Y-203.2
N360 M11
N365 M13
N370 G0 B0. C0.
N375 M10
N380 M12
N385 M8
N390 G1 X30. Y-52.5 F650.
N395 G0 G43 Z15. H3
N400 T4
N405 G0 Z5.
N410 G98 G81 X30. Y-52.5 Z-88.5 R-75. F333.33
N415 X-30.
N420 Y52.5
N425 X30.
N430 G80
N435 G0 Z15.
N440 M9
N445 M5
N450 G53 G0 Z0.

(Drill - Spotdrill Drill Counterbore)
N455 M1
N460 T4 M6
N465 S5000 M3
N470 G17 G90 G94
N475 G54
N480 G53 G0 X-736.6 Y-203.2
N485 M11
N490 M13
N495 G0 B0. C0.
N500 M10
N505 M12
N510 M8
N515 G1 X30. Y-52.5 F650.
N520 G0 G43 Z15. H4
N525 T5
N530 G0 Z5.
N535 G98 G73 X30. Y-52.5 Z-98.305 R-75. Q2.75 F1000.
N540 X-30.
N545 Y52.5
N550 X30.
N555 G80
N560 G0 Z15.
N565 M9
N570 M5
N575 G53 G0 Z0.

(Counterbore - Spotdrill Drill Counterbore)
N580 M1
N585 T5 M6
N590 S5000 M3
N595 G17 G90 G94
N600 G54
N605 G53 G0 X-736.6 Y-203.2
N610 M11
N615 M13
N620 G0 B0. C0.
N625 M10
N630 M12
N635 M8
N640 G1 X29.25 Y-52.55 F650.
N645 G0 G43 Z15. H5
N650 T6
N655 G0 Z-79.4
N660 G1 Z-80. F1000.
N665 Y-53.25
N670 G3 X30. Y-54. I0.75 J0.
```

```
N675 X29.869 Y-53.994 Z-80.585 I0. J1.5
N680 X29.74 Y-53.977 Z-81.171 I0.131 J1.494
N685 X29.612 Y-53.949 Z-81.756 I0.26 J1.477
N690 X29.487 Y-53.91 Z-82.342 I0.388 J1.449
N695 X29.366 Y-53.859 Z-82.927 I0.513 J1.41
N700 X29.25 Y-53.799 Z-83.512 I0.634 J1.359
N705 X29.14 Y-53.729 Z-84.098 I0.75 J1.299
N710 X29.036 Y-53.649 Z-84.683 I0.86 J1.229
N715 X30.702 Y-51.175 Z-85. I0.964 J1.149
N720 X29.298 Y-53.825 I-0.702 J-1.325
N725 X30.702 Y-51.175 I0.702 J1.325
N730 X29.689 Y-51.486 I-0.351 J-0.663
N735 G1 X29.361 Y-52.105
N740 G0 Z5.
N745 G1 X-30.75 Y-52.55 F650.
N750 G0 Z-79.4
N755 G1 Z-80. F1000.
N760 Y-53.25
N765 G3 X-30. Y-54. I0.75 J0.
N770 X-30.131 Y-53.994 Z-80.585 I0. J1.5
N775 X-30.26 Y-53.977 Z-81.171 I0.131 J1.494
N780 X-30.388 Y-53.949 Z-81.756 I0.26 J1.477
N785 X-30.513 Y-53.91 Z-82.342 I0.388 J1.449
N790 X-30.634 Y-53.859 Z-82.927 I0.513 J1.41
N795 X-30.75 Y-53.799 Z-83.512 I0.634 J1.359
N800 X-30.86 Y-53.729 Z-84.098 I0.75 J1.299
N805 X-30.964 Y-53.649 Z-84.683 I0.86 J1.229
N810 X-29.298 Y-51.175 Z-85. I0.964 J1.149
N815 X-30.702 Y-53.825 I-0.702 J-1.325
N820 X-29.298 Y-51.175 I0.702 J1.325
N825 X-30.311 Y-51.486 I-0.351 J-0.663
N830 G1 X-30.639 Y-52.105
N835 G0 Z5.
N840 G1 X-30.75 Y52.45 F650.
N845 G0 Z-79.4
N850 G1 Z-80. F1000.
N855 Y51.75
N860 G3 X-30. Y51. I0.75 J0.
N865 X-30.131 Y51.006 Z-80.585 I0. J1.5
N870 X-30.26 Y51.023 Z-81.171 I0.131 J1.494
N875 X-30.388 Y51.051 Z-81.756 I0.26 J1.477
N880 X-30.513 Y51.09 Z-82.342 I0.388 J1.449
N885 X-30.634 Y51.141 Z-82.927 I0.513 J1.41
N890 X-30.75 Y51.201 Z-83.512 I0.634 J1.359
N895 X-30.86 Y51.271 Z-84.098 I0.75 J1.299
N900 X-30.964 Y51.351 Z-84.683 I0.86 J1.229
N905 X-29.298 Y53.825 Z-85. I0.964 J1.149
```

```
N910 X-30.702 Y51.175 I-0.702 J-1.325
N915 X-29.298 Y53.825 I0.702 J1.325
N920 X-30.311 Y53.514 I-0.351 J-0.663
N925 G1 X-30.639 Y52.895
N930 G0 Z5.
N935 G1 X29.25 Y52.45 F650.
N940 G0 Z-79.4
N945 G1 Z-80. F1000.
N950 Y51.75
N955 G3 X30. Y51. I0.75 J0.
N960 X29.869 Y51.006 Z-80.585 I0. J1.5
N965 X29.74 Y51.023 Z-81.171 I0.131 J1.494
N970 X29.612 Y51.051 Z-81.756 I0.26 J1.477
N975 X29.487 Y51.09 Z-82.342 I0.388 J1.449
N980 X29.366 Y51.141 Z-82.927 I0.513 J1.41
N985 X29.25 Y51.201 Z-83.512 I0.634 J1.359
N990 X29.14 Y51.271 Z-84.098 I0.75 J1.299
N995 X29.036 Y51.351 Z-84.683 I0.86 J1.229
N1000 X30.702 Y53.825 Z-85. I0.964 J1.149
N1005 X29.298 Y51.175 I-0.702 J-1.325
N1010 X30.702 Y53.825 I0.702 J1.325
N1015 X29.689 Y53.514 I-0.351 J-0.663
N1020 G1 X29.361 Y52.895
N1025 G0 Z15.
N1030 M9
N1035 M5
N1040 G53 G0 Z0.

(Spotdrill - Spotdrill Drill)
N1045 M1
N1050 T6 M6
N1055 S5000 M3
N1060 G17 G90 G94
N1065 G54
N1070 G53 G0 X-736.6 Y-203.2
N1075 M11
N1080 M13
N1085 G0 B0. C0.
N1090 M10
N1095 M12
N1100 M8
N1105 G1 X18. Y-52.5 F650.
N1110 G0 G43 Z15. H6
N1115 T7
N1120 G0 Z5.
N1125 G98 G81 X18. Y-52.5 Z-83. R-75. F333.33
N1130 X-18. Y52.5
N1135 G80
N1140 G0 Z15.
N1145 M9
N1150 M5
N1155 G53 G0 Z0.

(Drill - Spotdrill Drill)
N1160 M1
N1165 T7 M6
N1170 S5000 M3
N1175 G17 G90 G94
N1180 G54
N1185 G53 G0 X-736.6 Y-203.2
N1190 M11
N1195 M13
N1200 G0 B0. C0.
N1205 M10
N1210 M12
N1215 M8
N1220 G1 X18. Y-52.5 F650.
N1225 G0 G43 Z15. H7
N1230 T2
N1235 G0 Z5.
N1240 Z-75.
N1245 Z-78.
N1250 G1 Z-81.5 F1000.
```

```
N1255 G0 Z-81.38
N1260 G1 Z-83. F1000.
N1265 G0 Z-82.88
N1270 G1 Z-84.5 F1000.
N1275 G0 Z-84.38
N1280 G1 Z-86. F1000.
N1285 G0 Z-85.88
N1290 G1 Z-87.5 F1000.
N1295 G0 Z-87.38
N1300 G1 Z-89. F1000.
N1305 G0 Z-88.88
N1310 G1 Z-90.5 F1000.
N1315 G0 Z-90.38
N1320 G1 Z-92. F1000.
N1325 G0 Z-91.88
N1330 G1 Z-93.5 F1000.
N1335 G0 Z-93.38
N1340 G1 Z-95. F1000.
N1345 G0 Z-94.88
N1350 G1 Z-96.5 F1000.
N1355 G0 Z5.
N1360 Z-94.5
N1365 G1 Z-96.803 F1000.
N1370 G0 Z5.
N1375 G1 X-18. Y52.5 F650.
N1380 G0 Z-75.
N1385 Z-78.
N1390 G1 Z-81.5 F1000.
N1395 G0 Z-81.38
N1400 G1 Z-83. F1000.
N1405 G0 Z-82.88
N1410 G1 Z-84.5 F1000.
N1415 G0 Z-84.38
N1420 G1 Z-86. F1000.
N1425 G0 Z-85.88
N1430 G1 Z-87.5 F1000.
N1435 G0 Z-87.38
N1440 G1 Z-89. F1000.
N1445 G0 Z-88.88
N1450 G1 Z-90.5 F1000.
N1455 G0 Z-90.38
N1460 G1 Z-92. F1000.
N1465 G0 Z-91.88
N1470 G1 Z-93.5 F1000.
N1475 G0 Z-93.38
N1480 G1 Z-95. F1000.
N1485 G0 Z-94.88
N1490 G1 Z-96.5 F1000.
N1495 G0 Z5.
N1500 Z-94.5
N1505 G1 Z-96.803 F1000.
N1510 G0 Z5.
N1515 Z15.

N1520 M5
N1525 M9
N1530 G53 G0 Z0.
N1535 G53 G0 X-736.6 Y-203.2
N1540 M11
N1545 M13
N1550 G0 B0. C0.
N1555 M30

%
O01001
(Using high feed G1 F650. instead of G0.)
(Machine)
(  vendor: DMG)
(  model: MORI U50)
(  description: Haas Classic legacy design)
(T6 D=5. CR=0. - ZMIN=-12.3 - right hand tap)
(T8 D=7.938 CR=0. TAPER=90deg - ZMIN=-7.3 - spot drill)
(T9 D=5.055 CR=0. TAPER=118deg - ZMIN=-15.319 - drill)
(T10 D=50. CR=0. TAPER=118deg - ZMIN=-118.8 - drill)
N10 G90 G94 G17
N15 G21
N20 G53 G0 Z0.
```

```
(Countersink - Countersink Drill Tap)
N25 T8 M6
N30 S5000 M3
N35 G17 G90 G94
N40 G54
N45 G53 G0 X-736.6 Y-203.2
N50 M11
N55 M13
N60 G0 B0. C0.
N65 M10
N70 M12
N75 M8
N80 G1 X8.5 Y-29.202 F650.
N85 G0 G43 Z15. H8
N90 T9
N95 G0 Z5.
N100 G98 G81 X8.5 Y-29.202 Z-7.3 R1.2 F333.33
N105 X38.5 Y0.798
N110 X8.5 Y30.798
N115 X-21.5 Y0.798
N120 G80
N125 G0 Z15.
N130 M9
N135 M5
N140 G53 G0 Z0.

(Drill - Countersink Drill Tap)
N145 M1
N150 T9 M6
N155 S5000 M3
N160 G17 G90 G94
N165 G54
N170 G53 G0 X-736.6 Y-203.2
N175 M11
N180 M13
N185 G0 B0. C0.
N190 M10
N195 M12
N200 M8
N205 G1 X8.5 Y-29.202 F650.
N210 G0 G43 Z15. H9
N215 T6
N220 G0 Z5.
N225 G98 G73 X8.5 Y-29.202 Z-15.319 R1.2 Q1.264 F1000.
N230 X38.5 Y0.798
N235 X8.5 Y30.798
N240 X-21.5 Y0.798
N245 G80
N250 G0 Z15.
N255 M9
N260 M5
N265 G53 G0 Z0.

(Tap - Countersink Drill Tap)
N270 M1
N275 T6 M6
N280 S500 M3
N285 G17 G90 G94
N290 G54
N295 G53 G0 X-736.6 Y-203.2
N300 M11
N305 M13
N310 G0 B0. C0.
N315 M10
N320 M12
N325 M8
N330 G1 X38.5 Y0.798 F650.
N335 G0 G43 Z15. H6
N340 T10
N345 G0 Z5.
N350 G98 G84 X38.5 Y0.798 Z-12.3 R0.2 F250.
N355 X8.5 Y-29.202
N360 X-21.5 Y0.798
N365 X8.5 Y30.798
N370 G80
N375 G0 Z15.
N380 M9
N385 M5
N390 G53 G0 Z0.

(Drill9)
N395 M1
N400 T10 M6
N405 S5000 M3
N410 G17 G90 G94
N415 G54
N420 G53 G0 X-736.6 Y-203.2
N425 M11
N430 M13
N435 G0 B0. C0.
N440 M10
N445 M12
N450 M8
N455 G1 X8.5 Y0.798 F650.
N460 G0 G43 Z15. H10
N465 T8
```

```
N470 G0 Z5.
N475 G98 G81 X8.5 Y0.798 Z-118.8 R1.2 F1000.
N480 G80
N485 G0 Z15.

N490 M5
N495 M9
N500 G53 G0 Z0.
N505 G53 G0 X-736.6 Y-203.2
N510 M11
N515 M13
N520 G0 B0. C0.
N525 M30

%
O01003
(Using high feed G1 F650. instead of G0.)
(Machine)
(  vendor: DMG)
(  model: MORI U50)
(  description: Haas Classic legacy design)
(T6 D=5. CR=0. - ZMIN=-12.3 - right hand tap)
(T8 D=7.938 CR=0. TAPER=90deg - ZMIN=-7.3 - spot drill)
(T9 D=5.055 CR=0. TAPER=118deg - ZMIN=-15.319 - drill)
N10 G90 G94 G17
N15 G21
N20 G53 G0 Z0.

(Countersink - Countersink Drill Tap 2)
N25 T8 M6
N30 S5000 M3
N35 G17 G90 G94
N40 G54
N45 G53 G0 X-736.6 Y-203.2
N50 M11
N55 M13
N60 G0 B0. C0.
N65 M10
N70 M12
N75 M8
N80 G1 X-21.5 Y-0.798 F650.
N85 G0 G43 Z15. H8
N90 T9
N95 G0 Z5.
N100 G98 G81 X-21.5 Y-0.798 Z-7.3 R1.2 F333.33
N105 X8.5 Y-30.798
N110 X38.5 Y-0.798
N115 X8.5 Y29.202
N120 G80
N125 G0 Z15.
N130 M9
N135 M5
N140 G53 G0 Z0.

(Drill - Countersink Drill Tap 2)
N145 M1
N150 T9 M6
N155 S5000 M3
N160 G17 G90 G94
N165 G54
N170 G53 G0 X-736.6 Y-203.2
N175 M11
N180 M13
N185 G0 B0. C0.
N190 M10
N195 M12
N200 M8
N205 G1 X-21.5 Y-0.798 F650.
N210 G0 G43 Z15. H9
N215 T6
N220 G0 Z5.
N225 G98 G73 X-21.5 Y-0.798 Z-15.319 R1.2 Q1.264 F1000.
N230 X8.5 Y-30.798
N235 X38.5 Y-0.798
N240 X8.5 Y29.202
N245 G80
N250 G0 Z15.
N255 M9
N260 M5
N265 G53 G0 Z0.

(Tap - Countersink Drill Tap 2)
N270 M1
N275 T6 M6
N280 S500 M3
N285 G17 G90 G94
N290 G54
N295 G53 G0 X-736.6 Y-203.2
N300 M11
```

```
N305 M13
N310 G0 B0. C0.
N315 M10
N320 M12
N325 M8
N330 G1 X-21.5 Y-
0.798 F650.
N335 G0 G43 Z15. H6
N340 T8
N345 G0 Z5.
N350 G98 G84 X-21.5
Y-0.798 Z-12.3 R0.2
F250.
N355 X8.5 Y-30.798
N360 X38.5 Y-0.798
N365 X8.5 Y29.202
N370 G80
N375 G0 Z15.

N380 M5
N385 M9
N390 G53 G0 Z0.
N395 G53 G0 X-736.6
Y-203.2
N400 M11
N405 M13
N410 G0 B0. C0.
N415 M30

%
```