الجمهورية الجزائرية الديمقراطية الشعبية
وزارة التعليم العالي والبحث العلمي
جامعة باجي مختار- عنابـة

UNIVERSITÉ BADJI MOKHTAR – ANNABA
BADJI MOKHTAR – ANNABA UNIVERSITY

Faculty of technology
Department of mechanical engineering
Field: science and technology
Sector: mechanical engineering
specialty: mechanical manufacturing and production engineering

Master's thesis

Thesis topic:

Development of a CNC machining process plan for a part using CAMWorks software

presented by: MAHAMDIA ZOUHIR ABD ELJALIL

Supervisor: Pr. *BOUSSAID Ouzine*

Before the following examination committee:

Pr LAGRED	**President**
Pr BOUSSAID	**Supervisor**
PrMOKAS	**Member**
PrHAMADACHE	**Member**

2024/2025

DEDICATIONS

TO ***MY PARENTS*** WHOSE UNWAVERING SUPPORT AND GUIDANCEHAVE BEEN THE FOUNDATION OF MY SUCCESS. TO ***MY FAMILY*** AND TO MY ***DEAR FRIEND***

ACKNOWLEDGMENTS

I would like to express my deepest gratitude to:

Almighty god, for granting me the strength, wisdom, and opportunity to complete this thesis.

My parents, for their support and sacrifices throughout my academic journey.

My thesis supervisor ***Boussaid Ouzine*** for his expert guidance and valuable feedback.

The members of jury, for taking the time to review

and evaluate my work.

To my dear ***friends*** for their support.

Table of Contents

Chapter 3:Computer-Aided Manufacturing (CAM) using CAMWORKS

table of figure

Abstract

This graduation project focuses on the development of a CNC machining process plan for a mechanical part using CAMWorks software. The objective of the work is to demonstrate the digital manufacturing workflow from part design to machining simulation. A specific part was selected, then modeled in 3D using SolidWorks. Following the design phase, CAMWorks was used to define the machining operations, select appropriate tools, generate toolpaths, and simulate the CNC machining process. The project highlights the advantages of computer-aided manufacturing in terms of precision, time-saving, and optimization of production. This work emphasizes the integration between design and manufacturing stages using modern CAM tools.

ملخص

يركز مشروع التخرج هذا على تطوير خطة تشغيل على ماكينة التحكم الرقمي بالحاسوب لقطعة ميكانيكية باستخدام برنامج كام وركس. يهدف العمل إلى عرض سير عملية التصنيع الرقمي بدءًا من تصميم القطعة وصولًا إلى محاكاة عملية التشغيل. تم اختيار قطعة معينة ثم تصميمها ثلاثي الأبعاد باستخدام برنامج سوليدووركس. بعد مرحلة التصميم، تم استخدام برنامج كام وركس لتحديد عمليات التشغيل المناسبة، واختيار أدوات القطع، وإنشاء مسارات الأدوات، ثم محاكاة عملية التصنيع على ماكينة التحكم الرقمي بالحاسوب. يبرز المشروع مزايا التصنيع بمساعدة الحاسوب من حيث الدقة وتوفير الوقت وتحسين الإنتاج. ويؤكد هذا العمل على التكامل بين مرحلتي التصميم والتصنيع باستخدام أدوات التصنيع باستخدام الحاسوب الحديثة.

General Introduction:

The machining of complex parts on CNC machines is spreading to all sectors of industry. Not so long ago reserved for certain high-tech industries, CNC machine tools are now used in most mechanical manufacturing plants and small and medium-sized businesses. Nowadays, the acquisition of these machines has become a necessity in industry in general, and remains a condition for technical and economic success. This progress is essentially due to the technological revolution in industrial computing, which has enabled the development of efficient digital solutions with the possibility of implementing more complex algorithms. Most of this technology is based on microprocessors, DSPs (Digital Signal Processors) and PIC microcontrollers. These computers numerically controlled (CNC) machines enable cost-effective production. Cost control remains a major concern.

A presentation of the benefits of simulating the manufacture of a part before deciding whether or not to produce it on the machine. This approach, based on industrial simulation software, enables us to foresee more than one scenario for producing the part, and from there to opt for the scenario that allows us to choose the most cost-effective means, and save time and money. [1]

The objective of our final project is to produce a part shape or complex part shape on a 3-axis and 4-axis numerically controlled machine tool. Our work is divided into three chapters:

In the first chapter, it is necessary to conduct bibliographic research to understand the evolution of numerical control and present new developments in this field. Similarly, we will need to present the advantages of the numerically controlled machine tool, as well as its operation, advantages and disadvantages. Chapter 2 focuses on creating a part using SolidWorks. The third chapter focuses on creating a CNC process plan using Camworks. Finally, a conclusion.

Chapter 1: Literature Review

Introduction:

In the field of manufacturing and engineering, machining processes play a pivotal role in shaping raw materials into intricate and functional parts. These processes involve the precise removal of material from a workpiece to achieve desired geometries, dimensions, and surface finishes. From the simplest shapes to the most complex forms, machining techniques have evolved to cater to the ever-increasing demands of various industries. Understanding the different types of machining processes is essential for optimizing production and achieving desired parts.

Machining is a manufacturing process in which a workpiece (such as a metal or plastic part) is cut, shaped, or finished by removing unwanted material using various cutting tools or abrasives. It is a subtractive manufacturing process, meaning the material is removed from a solid block or stock to create the desired final shape or features. It is a fundamental aspect of metalworking and is essential for the production of a wide range of parts used in various industries, including automotive, aerospace, construction, and consumer goods. [2]

1 The principal Machining Operations:

1.1 Milling:

Milling is a widely used machining process that produces flat and complex shapes based on the cutter's path and design. As an interrupted cutting operation, it exposes the tool to repeated impact and thermal shock, requiring robust materials and optimized cutter geometry. [3]

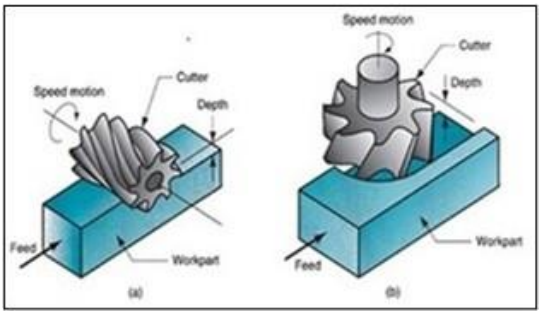


Fig.1.1: two basic forms of milling operation (a) peripheral milling (b) face milling

The principal milling operations:

Peripheral milling:

In peripheral milling, also known as plain milling, the tool's axis runs parallel to the machined surface, and cutting is done by the edges along the tool's outer circumference.

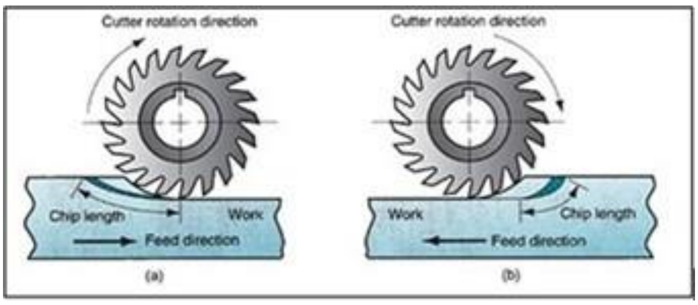


Fig.1.2: two forms of peripheral milling (a) up milling (b) down milling

a) **Slab Milling**: A fundamental type of peripheral milling where the cutter's width exceeds the workpiece width on both sides.
b) **Slotting**: Also known as slot milling, this process uses a cutter narrower than the workpiece to create a slot. When a thin cutter is used, it can produce narrow slots or divide the part entirely—referred to as saw milling.
c) **Side Milling**: A method where the cutter machines along the side surface of the workpiece.
d) **Straddle Milling**: Similar to side milling, but cutting occurs on both sides of the workpiece simultaneously.
e) **Form Milling**: Involves a cutter with specially shaped teeth that define the profile of the machined slot. As the tool shape determines the final form, it is considered a forming process [3].

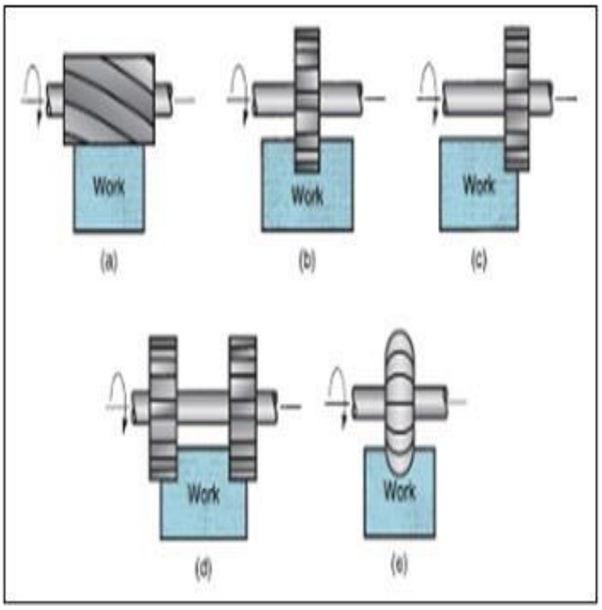


Fig.1.3: milling operations

1.2 Turning:

Turning is a machining process where a single-point cutting tool removes material from the surface of a rotating workpiece. The tool moves parallel to the axis of rotation to produce a cylindrical shape. This operation is typically performed on a lathe, which rotates the workpiece and controls the tool's feed rate and depth of cut.

The principal turning operations:

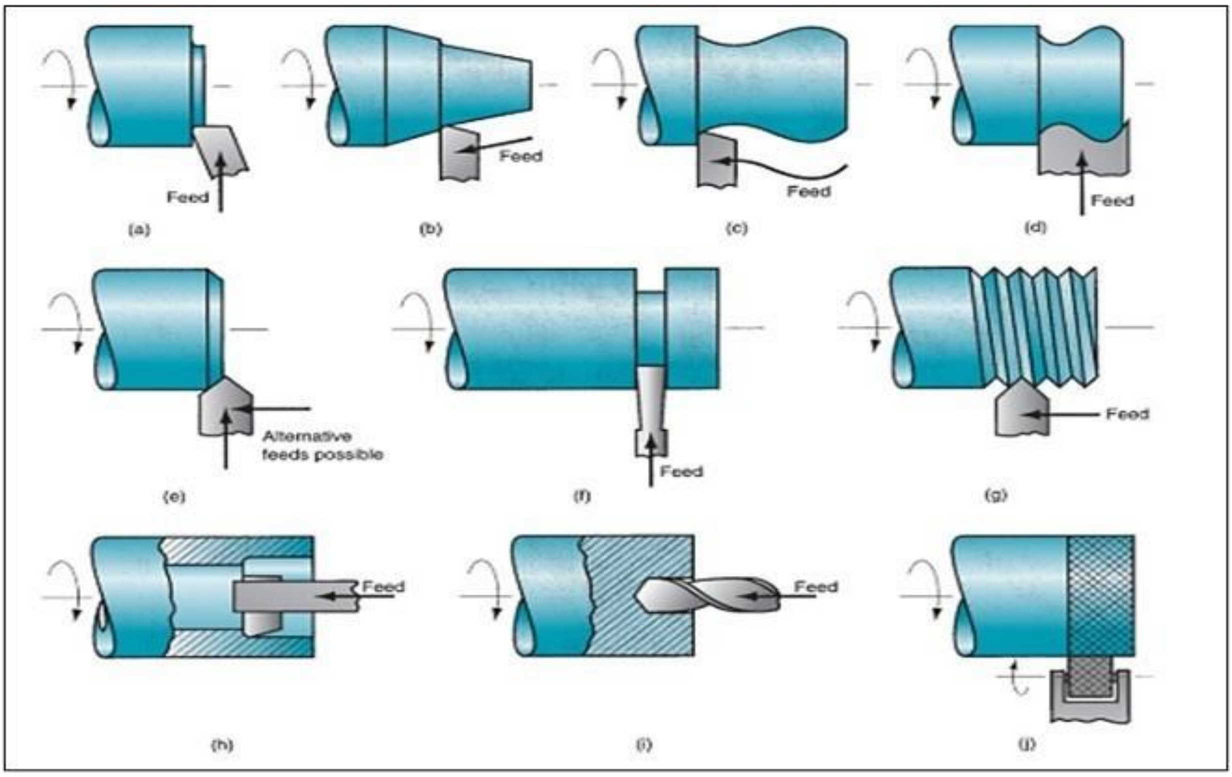


Fig.1.4: machining operations other than turning that are performed on a lathe

a) Facing
The tool moves radially into the end of the rotating workpiece to produce a flat surface.

b) Taper Turning
By feeding the tool at an angle to the workpiece axis, a tapered or conical surface is generated.

c) **Contour Turning**
The tool follows a non-linear path along the workpiece, creating a curved or complex profile instead of a straight surface.

d) **Form Turning**
Also known as forming, this process uses a tool with a specific shape that is pressed radially into the workpiece to transfer its form.

e) **Chamfering**
A tool is used to cut a beveled edge on the corner of the cylinder, producing a chamfer.

f) **Cutoff (Parting)**
The tool advances radially into the workpiece at a specific location to separate the part.

g) **Threading**
A pointed tool moves linearly along the outer surface of the rotating workpiece to cut helical threads.

h) **Boring**
A single-point tool is fed along the axis of rotation inside a pre-existing hole to enlarge or finish it.

i) **Drilling**
Performed by feeding a drill bit axially into the rotating workpiece; similar tools can be used for reaming.

j) **Knurling**
A non-cutting process that forms a patterned texture on the work surface using hardened rollers [3].

1.3 Drilling:

Drilling is a machining process used to create round holes in a workpiece using a rotating tool called a drill bit, typically a twist drill. The drill feeds into the stationary workpiece, forming a hole equal in diameter to the bit. It is commonly performed on a drill press or similar machines.

Principal drilling operations:

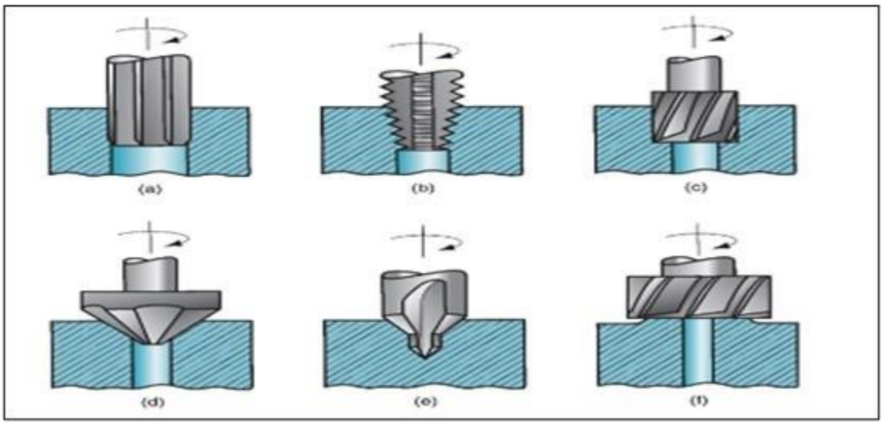


Fig.1.5: Machining operations related to drilling [3]

a) **Reaming:**
Reaming is used to slightly enlarge an existing hole, improving diameter accuracy and surface finish. The tool, known as a reamer, typically features straight flutes.

b) **Tapping:**
Performed with a tap, this process creates internal screw threads within a pre-drilled hole. Further details are provided in a dedicated section.

c) **Counterboring:**
Counterboring produces a stepped hole, where a larger diameter follows a smaller one partway down. This allows bolt heads to sit flush with the surface.

d) **Countersinking:**
Similar to counterboring, but the step is conical, designed to accommodate flat-head screws and bolts.

e) **Centering:**
Also called center drilling, this operation creates a small, precise starting hole to guide subsequent drilling. The tool used is a center drill.

f) **Spot Facing:**
Spot facing resembles milling and is used to create a flat, localized surface on a workpiece [3].

2 Computer Numerical Control (CNC) machine tools:

Computer Numerical Control (CNC) machining is the automated control of machine tools through a computer, representing an advancement over earlier Numerical Control (NC) systems that relied on punched cards or tape for input. CNC machines use motorized tools and platforms, guided by computer-generated instructions—typically in the form of G-code or M-code—to perform precise operations. These instructions can be manually programmed or, more commonly, generated through CAD (Computer-Aided Design) or CAM (Computer-Aided Manufacturing) software. CNC technology offers greater flexibility, easier programming, and real-time adjustments, which has led to its widespread adoption as computing costs have decreased. Even in 3D printing, similar principles apply, where the model is first sliced before generating the machine instructions, often also in G-code.[4]

2.1 Types of CNC machines

The main advantage of CNC machines lies in their ability to work with a wide range of materials while maintaining high precision. Unlike conventional machines, CNC systems efficiently handle thick and tough materials, easily penetrating them without loss of accuracy. They also excel at producing intricate details, such as cutting precise shapes like the letter "V," which can be challenging for other machines. This capability makes CNC machines highly versatile, able to process materials including wood and metals. Due to this versatility, multiple types of CNC machines have been developed to suit different manufacturing needs and applications.

2.1.1 The Lathes:

One common type of CNC machine used in metal shaping and manufacturing is the CNC lathe. This machine works by rotating the workpiece, allowing it to be formed into the desired shape. Figure 1-6 illustrates the shape and components of a CNC lathe . [5]

fig.1.6: lathe machine

2.1.2 Milling Machine:

This CNC machine uses rotary cutters to shape different materials. It performs milling operations that can also be done manually. Figure 1-7 shows the typical shape of a milling machine [5]

Fig.1.7: milling machine

2.1.3 Plasma Cutters

This type of CNC machine cuts steel and other materials using a high-temperature plasma arc. The cutting process is carried out with plasma torches. Figure 1-8 shows the CNC plasma cutters

fig.1.8: CNC plasma cutters [5].

- **The elements of the lathe machine**

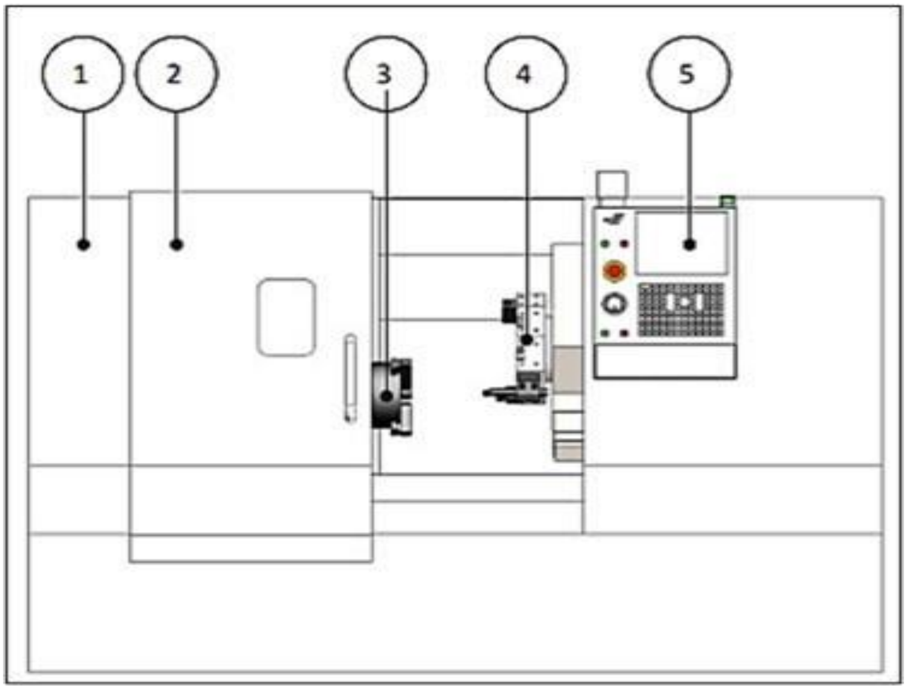


Fig.1.9: CNC lathes

1. Sheetmetal

Protective housing that contains cutting chips and capture coolant for recycling

2 – Door

The door is closed during operation. Lathes can be dangerous if the part is thrown or a tool breaks during machining. The window is made from a special high impact glass. The lathe should not be operated if this glass is cracked.

3–Spindle

The spindle is attached at one end the machine drive system. The other end attaches the chuck, which grips the part.

4–Turret

The turret holds and moves the tools. Tools are bolted to the turret using a variety of specialized holders, depending on the type of tool. The turret indexes to present the tool to the work piece.

5–Control

The CNC control used to operate the machine. [6]

Turn Machine axes :

CNC lathes are commonly categorized based on the number of programmable axes they possess. Most vertical CNC lathes have two axes, while horizontal CNC lathes—more widely used—typically feature two axes but can be equipped with three, four, or even six axes to accommodate more complex part manufacturing. The slant bed lathe is especially popular for general machining, as its design directs chips away from the operator and safely directs parts downward toward the chip conveyor in case of accidents. Besides flat bed, slant bed, front and rear, horizontal, and vertical designs, lathes are often classified simply by their number of axes, which remains the most straightforward and common way to identify different types of CNC lathes.

Axis Orientation – Turning :

Most CNC lathes feature two primary axes, X and Z, although additional axes exist but are less common. A specialized third axis, known as the C axis, is available as an option for milling tasks (live tooling) on many CNC lathes. In industry, CNC lathes often use a dual orientation of the X and Z axes, distinguishing between front and rear lathes. Front lathes resemble traditional engine lathes, while all slant bed lathes fall under the rear category. Axis identification does not always follow strict mathematical conventions. Another type, the vertical CNC lathe, is essentially a horizontal lathe rotated by 90°. Figure 1-10 illustrates typical axis configurations for both horizontal and vertical turning machines. [7]

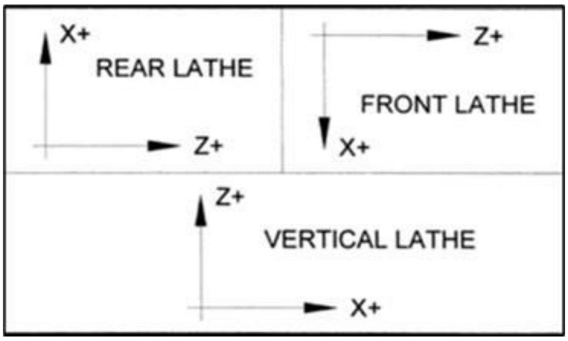


Fig.1.10: Typical machine axes of a CNC lathe (turning center)

Mill machine Axes :

Milling machines and machining centers typically have at least three axes: X, Y, and Z. Adding a fourth axis, usually a rotary or indexing axis (A for vertical or B for horizontal models), increases their flexibility. Machines with five or more axes offer even greater versatility. For instance, a basic five-axis machine like a boring mill includes three main axes plus a rotary (B) and a parallel (W) axis. True complex five-axis machines, often used in the aerospace industry, enable simultaneous multi-axis cutting to handle intricate shapes and hard-to-reach areas.

Terms like “two and a half” or “three and a half” axis machines describe setups where simultaneous axis movement is limited. For example, a four-axis vertical machine with an X, Y, Z, and indexing A axis allows positioning but not simultaneous rotation with other axes, often called a three and a half axis machine. In contrast, a machine with a fully rotating table that moves simultaneously with primary axes is considered a true four-axis machine, demonstrating full multi-axis capability.

Axis Orientation – Milling

A standard 3-axis CNC machine operates using three controlled axes: X, Y, and Z. The X axis runs parallel to the machine table’s longest side, the Y axis follows the shortest side, and the Z axis refers to the spindle's vertical movement. In vertical machining centers, the X axis corresponds to the table’s longitudinal movement, the Y axis to the saddle’s cross movement, and the Z axis to the vertical spindle motion. In horizontal machining centers, the axis definitions shift due to the machine’s orientation: the X axis is still the table’s longitudinal direction, but the Y axis becomes the column’s vertical direction, and the Z axis follows the spindle’s horizontal movement. A horizontal machine can essentially be seen as a vertical machine rotated 90 degrees. Additionally, horizontal machines often include a B axis, which allows indexing rotation. The typical axis arrangement for CNC vertical machines is shown in Figure 1-11. [7].

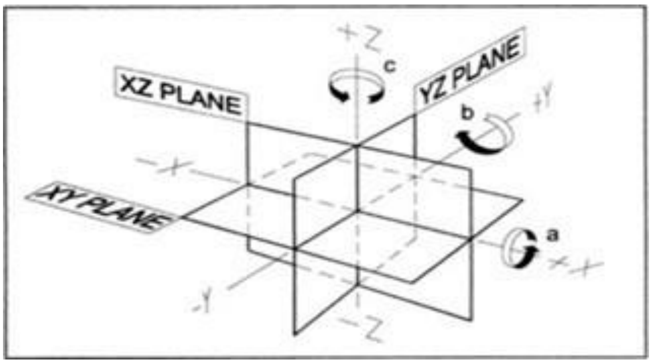


Fig.1.11: standard orientation of planes and CNC machine tool axes

2.2 Different Axes

Any CNC machine can be equipped with additional axes, usually referred to as secondary axes and labeled U, V, and W. These axes run parallel to the primary X, Y, and Z axes, respectively. For rotary or indexing purposes, the extra axes are named A, B, and C, corresponding to rotations around the X, Y, and Z axes. The positive direction of a rotary or indexing axis follows the same direction as advancing a right-handed screw along the positive X, Y, or Z axis. The relationship between the primary and supplementary axes is illustrated in Figure 1-12. [7]

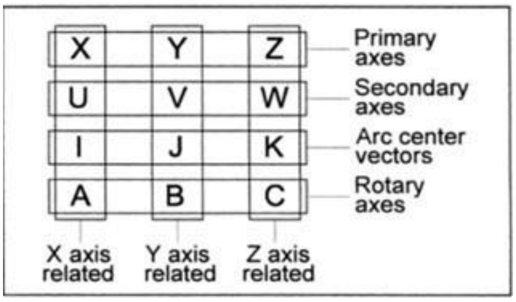


Fig.1.12: relationship of the primary and the secondary machine axes

2.3 CNC coordinate System:

Cartesian Coordinate System

CNC machine motion is based on the Cartesian coordinate system. To operate a CNC machine effectively, it's essential to understand how coordinate systems are defined in both CAM software and CNC machines, and how they interact. This topic begins with an overview of the Cartesian system, followed by a detailed explanation of the relationship between CAM and CNC coordinate systems. It also covers the setup of the Work Coordinate System (WCS) on the machine, which defines the part's location within the workspace. The section concludes with an explanation of tool length and diameter offsets—length offsets compensate for varying tool projections from the holder, while diameter offsets ensure precision in machining, often within .005 inches of tolerance. The Cartesian coordinate system consists of three number lines, labeled X, Y and Z, set at 90-degree angles to each other as shown in Figure 1-13 below. The origin, or Datum, is where the three axes cross each other. The labels, orientations, and directions of the Cartesian coordinate system in Figure 1-13 are typical of most Vertical Machining Center (VMC).

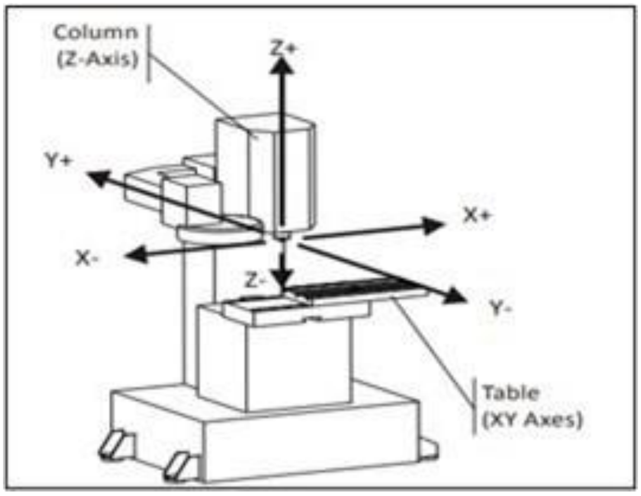


Fig.1.13: VMC Machine Coordinate System (At Home Position)

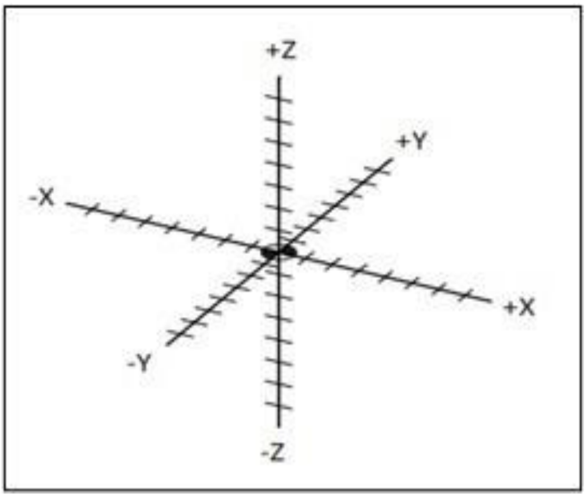


Fig.1.14: 3D cartesian coordinate system

Table 1: Classification of Machine Tools [8]

Number of Axes	Movements	Type of Machining and Possible Operations
1	Z	Broaching, Press
2	X, Z	Turning: all shapes obtained have the same axis of symmetry
2	X, Y	Milling: surfacing, drilling, pocket milling, slot milling, and machining of complex surfaces. The tool axis remains parallel to a fixed direction relative to the workpiece.
3	X, Y, Z	Turning, with spindle control, general turning, milling with rotating tools, off-axis drilling. Spindle movement is controlled.
4	X, Y, Z, B	Milling: surfacing, drilling, pocket milling, slot milling, and machining of complex surfaces. The tool axis remains contained within a fixed plane relative to the workpiece.
4	X, Y, Z, C	Milling (see X, Y, Z, B)
4	X, Y, Z, C	Turning
4	2×(X, Z)	Two-turret turning
5	2×(X, Z, C)	Two-turret turning with spindle control (see X, Z, C)
5	X, Y, Z, A, C	Milling of complex shapes: flank milling, milling with indexing, drilling in all directions
5	X, Y, Z, B, C	Milling of complex shapes (see X, Y, Z, A, C)
5	X, Y, Z, A, B	Milling of complex shapes (see X, Y, Z, A, C)

2.4 Diagram of the CNC machine tool

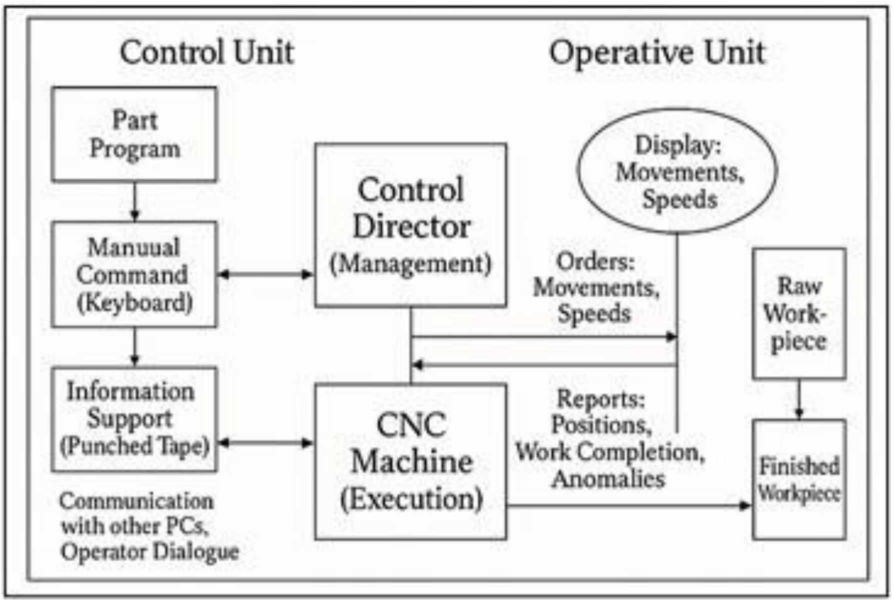


Fig.1.15: Diagram of the CNC machine tool [8]

A) The operative unit:

It represents the machine itself. It is identical to conventional machines in the "machining" function but has superior performance. It can have several motorized machining heads according to one or several independent axis systems, and it also includes additional elements to control the clamping and lubrication devices.

B) The control unit:

It is the brain of the machine, called the Numerical Control Unit (NCU). The NCU sends rotation and movement commands to the machine's mobile parts based on the setpoint values described in the program of the part to be produced.[8]

2.5 The concept of CNC

CNC (Computer Numerical Control) machines are machining tools whose operations are controlled using a series of letters, symbols, and numbers. These instructions are processed by a computer system, which is integrated with the machine via an interface device. This interface acts as a translator, since the machine itself cannot directly interpret the computer's language. The term "CNC" reflects this integration of a computer, machine, and control unit working together to perform automated tasks [9]

2.6 CNC Motion Control

Most CNC machines can accurately position each axis within ±0.0002 inches across the entire working area. This high precision is made possible by a closed-loop servo system, as shown in Figure 1-16. The control unit sends motion commands through a controller board to servomotors, which drive ball screws connected to the machine's axes. These ball screws move the table or column as needed. A servo transmitter monitors the actual axis position and compares it to the commanded position, allowing for continuous feedback. Because ball screws have minimal backlash, reversing direction results in almost immediate response. Additionally, CNC systems use electronic compensation to correct any slight backlash that may remain.

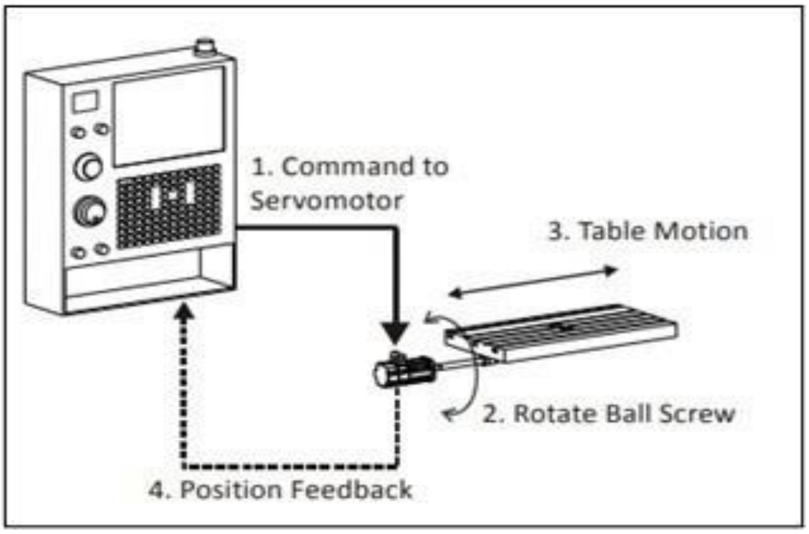


Fig.1.16: Closed Loop Servo Mechanism

2.7 Work Coordinate System

Using machine coordinates for CNC programming would be impractical, as the machine's home position is far from the work area, resulting in large, non-intuitive values. To simplify programming and setup, a Work Coordinate System (WCS) is defined for each CNC program. The WCS is a reference point chosen by the programmer on the part, stock, or fixture. Although it may align with the CAD model's origin, it doesn't have to. Its placement within the machine's working space requires thoughtful selection.

- It must be located using tools like an edge finder, probe, or indicator.
- It should offer high precision—typically within ±0.001 inches.
- It must be repeatable, ensuring consistent part positioning.
- It should consider how the part will be rotated or repositioned during machining.

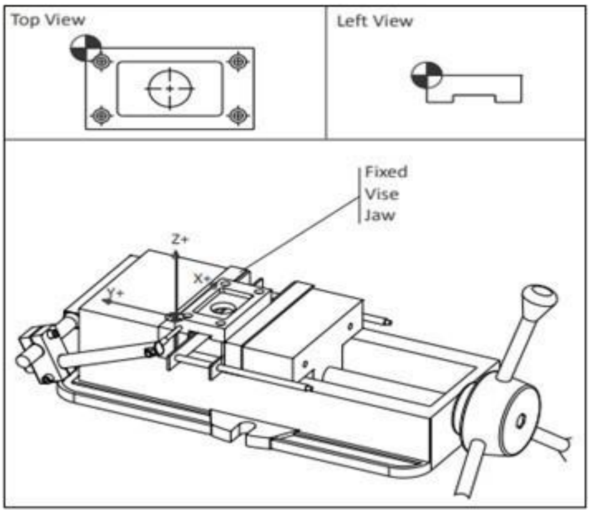


Fig.1.17 Work Coordinate System (WCS) milling

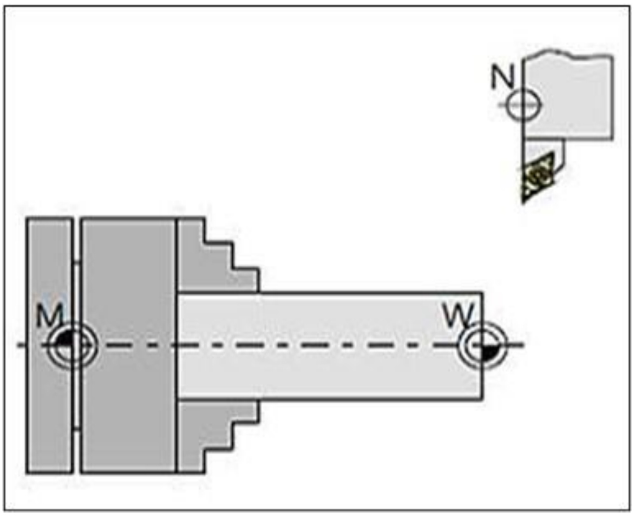


Fig.1.18: Work Coordinate System (WCS) turning

2.8 The advantages of CNC machines

Industry is the backbone of any productive nation, and its success often depends on the extent to which it utilizes advanced machinery. The more accurate and efficient these machines are, the more competitive the industry becomes. With technological advancement, scientists began integrating electronic systems into mechanical equipment, leading to increased control through computer connectivity—ushering in CNC (Computer Numerical Control) systems. advantages of CNC Machines:

- Programs can be created directly on the machine and saved to its internal memory.
- Programs are easy to edit and review.
- Less input data is required, and output production is significantly faster.
- Operators benefit from improved safety.
- CNC systems allow operators to perform other tasks simultaneously, increasing overall efficiency.
- High precision is ensured across all components by using the same program repeatedly.
- Capable of producing complex parts that are difficult or impossible to machine traditionally.
- Reduces total production time, as machine setup is only required during the first operation.
- Lowers production costs by minimizing waste and optimizing workflow.
- Saves time compared to traditional machining methods.
- Enables mass production with uniform accuracy.
- Allows precise control over cutting conditions.
- Facilitates smooth transitions between different product geometries.

2.9 Disadvantage of CNC machines

There is no perfect invention in the world — when something has advantages, it also means it has disadvantages. However, a good invention is one where the advantages outweigh the disadvantages. CNC Disadvantages:

- High cost of the machine.
- Operators must be highly trained to work with this type of machine.
- Shortage of specialists qualified to maintain these machines.
- Requires special tools for remote control and continuous operation.

3 Stages of CNC work

Data is transferred between input and output through the computer, which compares the data within its memory or device. It reads sensor feedback and sends it back to the computer to compare input with sensor readings. This process continues until the lowest possible error is achieved, resulting in the highest accuracy and quality in CNC operations. The main stages of CNC work are:

- CAD
- CAM
- Machine control and operation

3.1 Computer-Aided Design (CAD)

The computer-aided design (CAD) system is closely linked to the evolution of computer graphics, but it extends beyond simple graphics by including advanced analysis and modeling capabilities. Interactive computer graphics (ICG) serves as the essential technical foundation for CAD systems. This software enables the creation of geometric shapes in all dimensions, allowing users to draw 2D designs and convert them into 3D models when needed. It also provides simulations for movement, stress distribution, and plots force and moment curves affecting the engineering part. The files are then saved in formats compatible with CAM software. Popular CAD programs include:

- Autodesk Inventor
- SolidWorks
- CATIA

Figure 1-19 illustrates the CAD software workflow: the process starts with sketching and adjusting settings. For 2D designs, the 2D icon is selected, while 3D models use the 3D icon. Designs can be converted from 2D to 3D within the software. After completing the design, it is exported into programming languages readable by CAM systems.

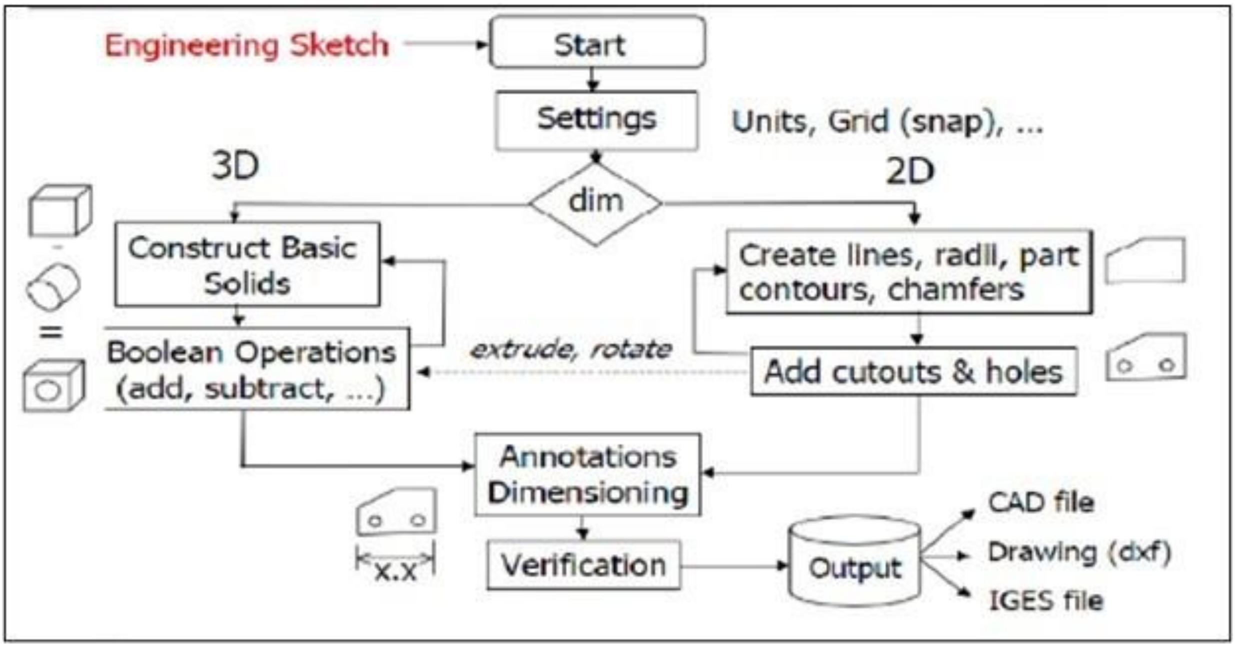


Fig.1.19: Generic CAD Process

3.2 Computer-Aided Manufacturing (CAM)

Computer Aided Manufacturing (CAM) refers to the use of computer systems to plan, control, and manage manufacturing processes by directly or indirectly linking computers with the factory production environment. CAM software converts engineering designs created in CAD into G-code, which CNC machines use to interpret the design. This G-code is transferred to the CNC machine either directly through a network cable or indirectly via flash memory. Once received, the CNC machine sets the necessary parameters to begin the manufacturing process. [10] Common CAM software includes:

- Surfcam
- PowerMill
- HSM
- SolidCAM

Figure 1-20 illustrates a typical CAM system workflow. While the order of steps may vary among different CAM platforms, the core data requirements remain consistent to ensure proper operation. Milling is the primary application of CAM systems, and the following discussion will focus on this area. [11]

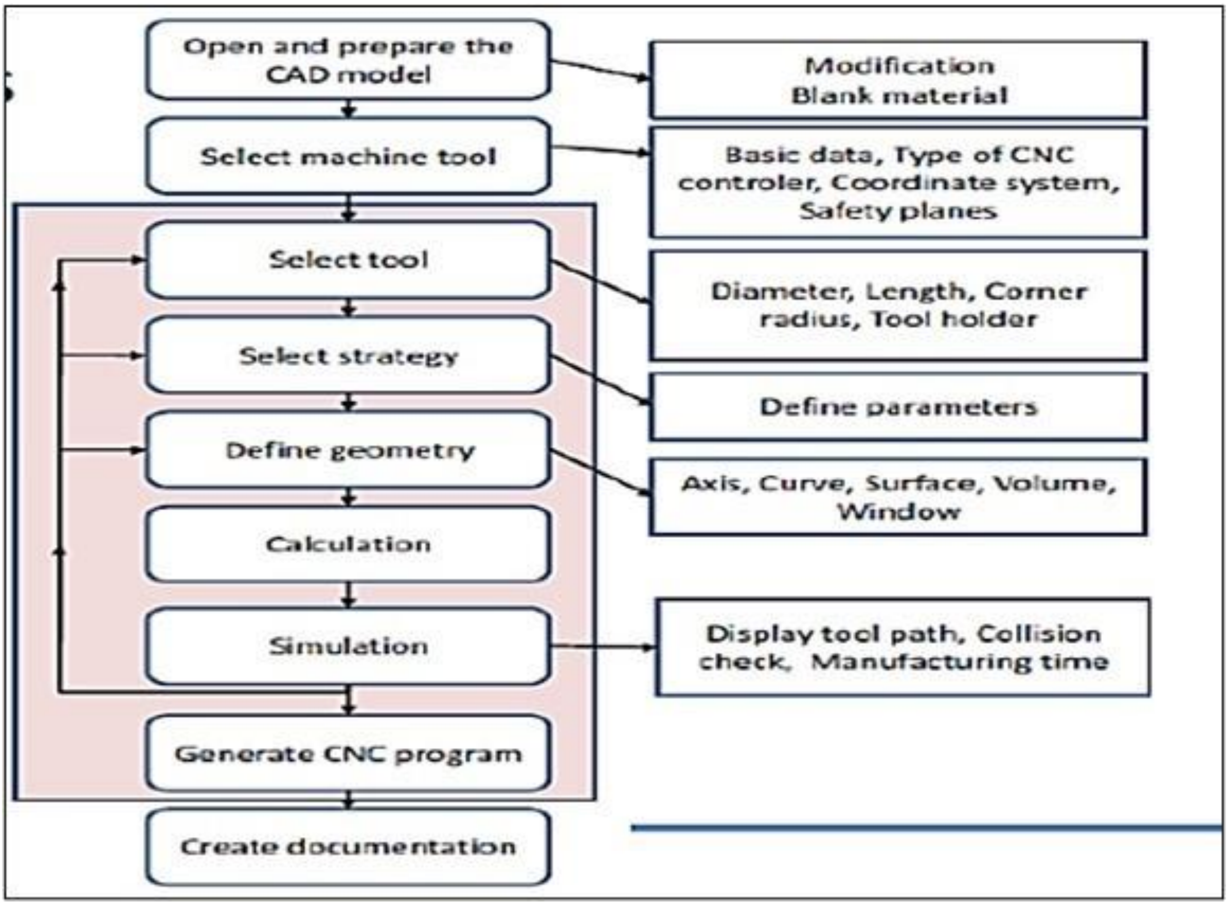


Fig.1.20: CAM Process

3.3 Machine control and operation

CNC machine tools operate by performing linear sliding and rotary movements. These movements vary by machine design, but the system automatically interprets axis commands (such as moving the cutting tool along X+ or Y+) by moving the tool or table accordingly, often in the opposite direction (-X, -Y). The key is that the cutting tool reaches the programmed position, regardless of whether the table or tool moves. To ensure safety, all control systems require a “return to zero” operation before automation begins. The zero point is set at the junction of the positive ends of the movement axes. After loading the program into the CNC control unit, the operator inputs additional data, especially the reference or origin point (zero position) of the part to be machined. For CNC operation, the main steps include:

- Ensuring a proper working environment.
- Providing safe and suitable machine feeding.
- Selecting cutting tools with appropriate length adjustments.
- Choosing the correct metal material.
- Operating the machine.
- Downloading the part program from CAM in the machine’s compatible language.
- Copying the program into machine memory.
- Setting the zero-reference point.
- Choosing suitable cutting parameters.
- Monitoring and controlling the work. [12]

Chapter 2:

Computer-Aided design (CAD) using SOLIDWORKS

Introduction:

The **SOLIDWORKS** Software is a computer-aided design (CAD) software used for mechanical design automation. It enables designers to quickly create and test ideas by sketching, modifying features and dimensions, and producing 3D models and detailed 2D drawings. This document introduces essential concepts and terminology used across the SOLIDWORKS interface and helps users become familiar with its most commonly used functions.

1 Design Method

Before beginning the actual model design, it's useful to plan the modeling approach. Once the design requirements are clear and the right concepts are chosen, you can proceed with model development:

- **Sketches**: Create the necessary sketches, apply dimensions, and define geometric relations appropriately.
- **Features**: Choose suitable features like extrusions and fillets, determine the optimal feature sequence, and apply them in the correct order.
- **Assemblies**: Identify which components to assemble and select the appropriate mate types to ensure proper alignment and movement.

1.1 Sketches

A **sketch** is the foundation of most 3D models, and modeling typically starts with a sketch. From the sketch, you can generate features, combine features to form parts, and then assemble those parts using mates to build a full assembly. Drawings can be created from either parts or assemblies.

A sketch represents a 2D profile or cross-section. It is drawn on a plane or a planar face. While 2D sketches use the X and Y axes, 3D sketches include the Z axis as well.

Regardless of the method, all sketches share the following basic elements:

1.2 Origin

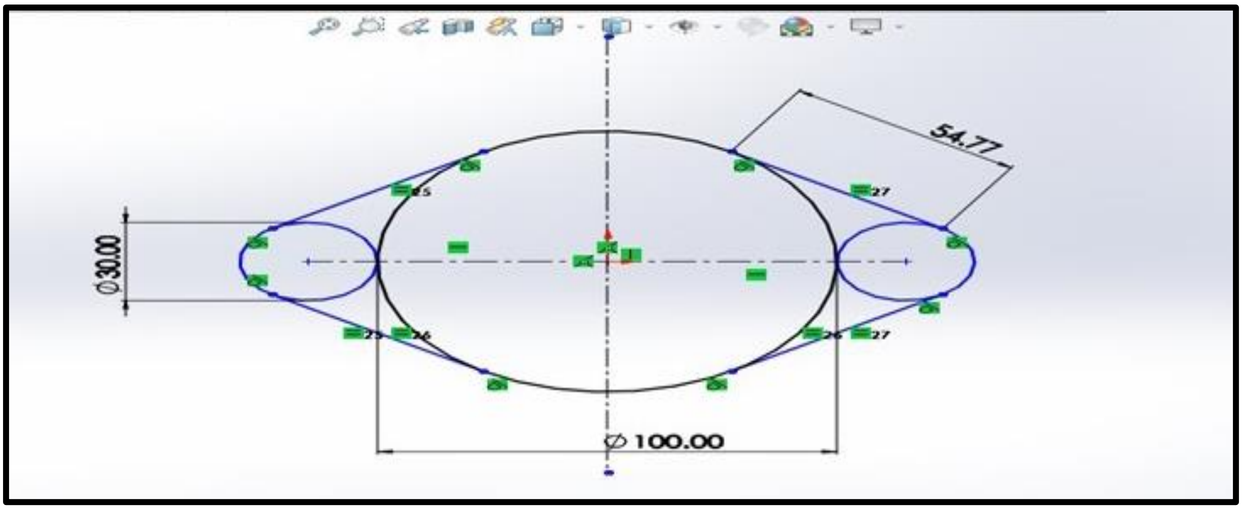
In many cases, a sketch begins at the origin, which serves as a stable reference point or anchor. In the example below, a centerline is also included. This centerline passes through the origin and is used as the axis for creating a revolved feature.

1.3 Dimensions

You can define dimensions between elements like lengths, angles, and radii. Modifying these dimensions alters the size and shape of the part. By strategically applying dimensions, you ensure the design intent is maintained throughout changes. [13]

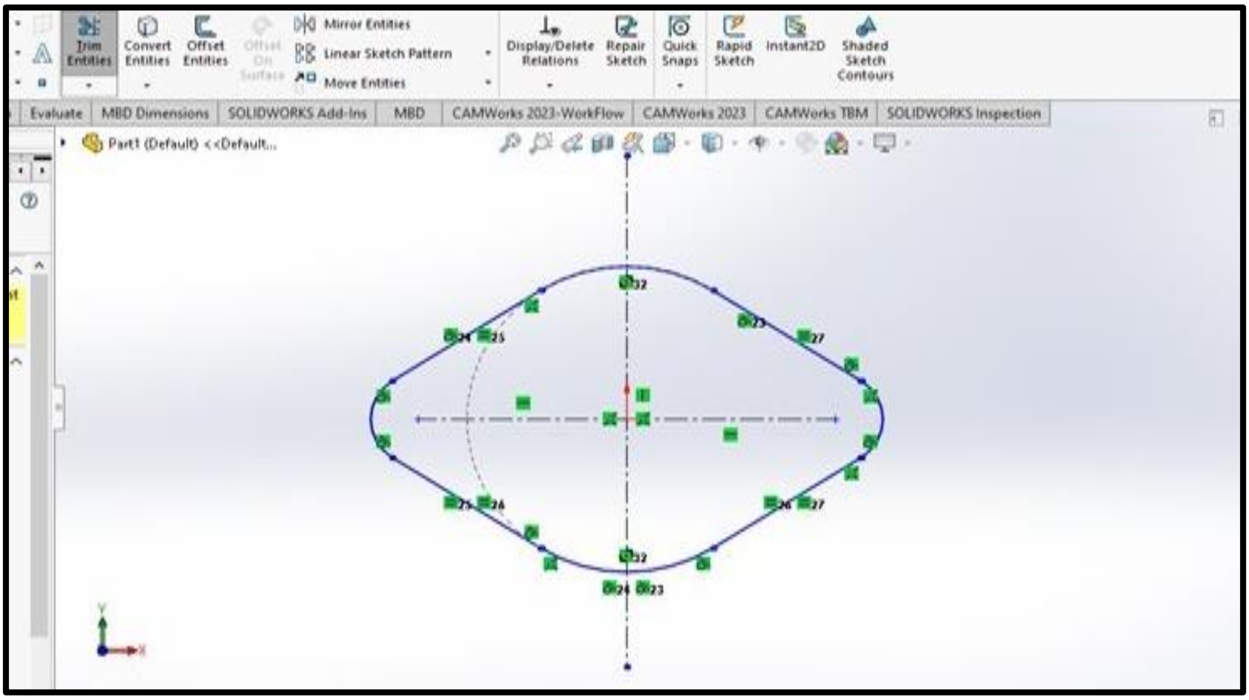
2 Flange bearing housing design:

2.1 Sketches and dimensions:



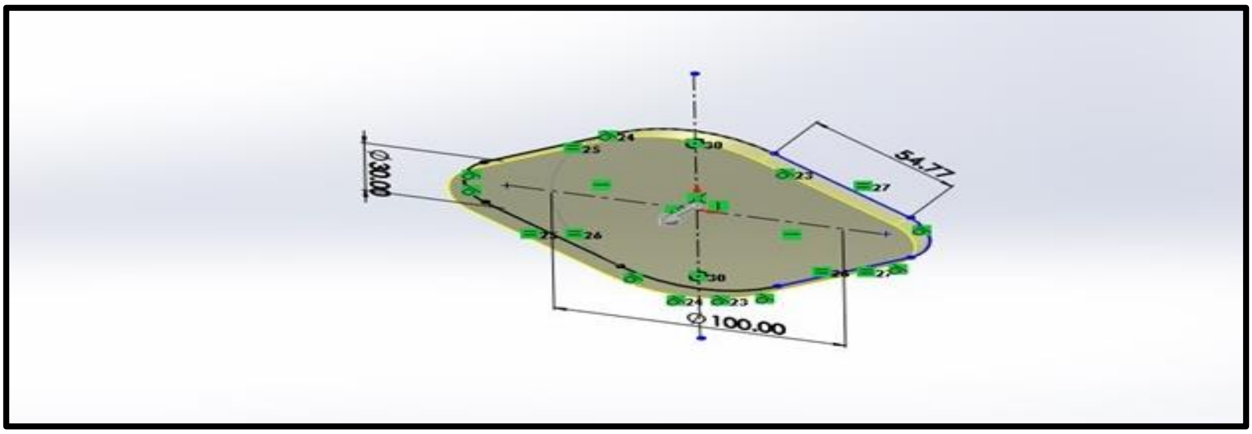
2.2 Trimming

-Trimming unnecessary internal lines using **trim entities**

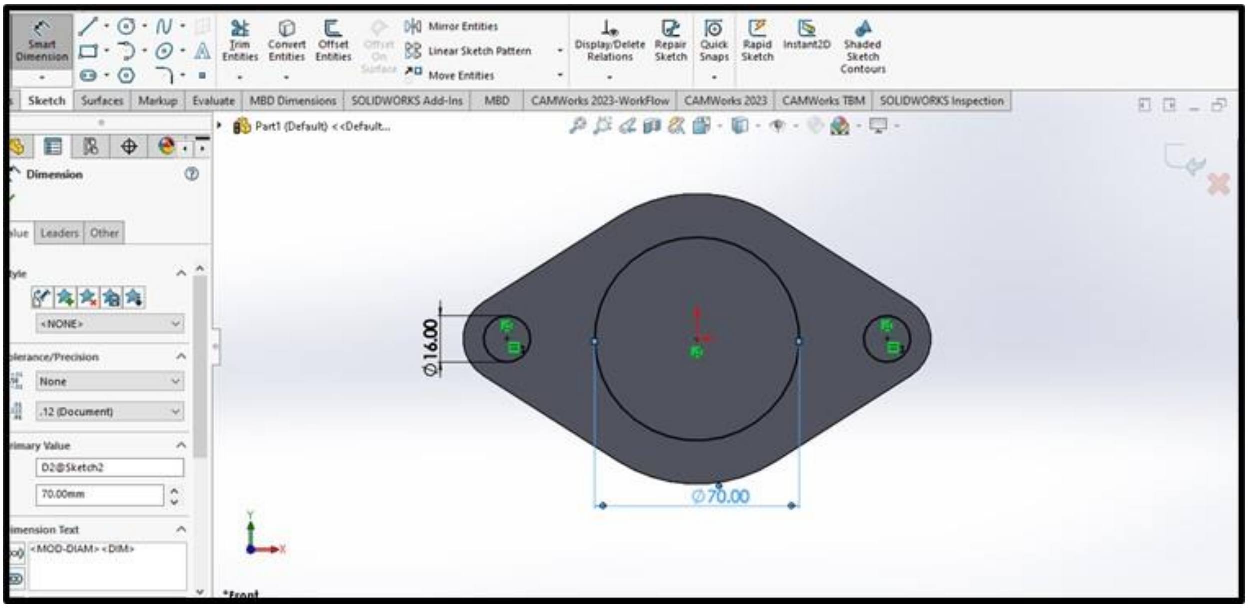


2.3 Features

Once you complete the sketch, you can create a 3D model using features such as an extrude (the base of the faucet) or a revolve (the faucet handle). [13]

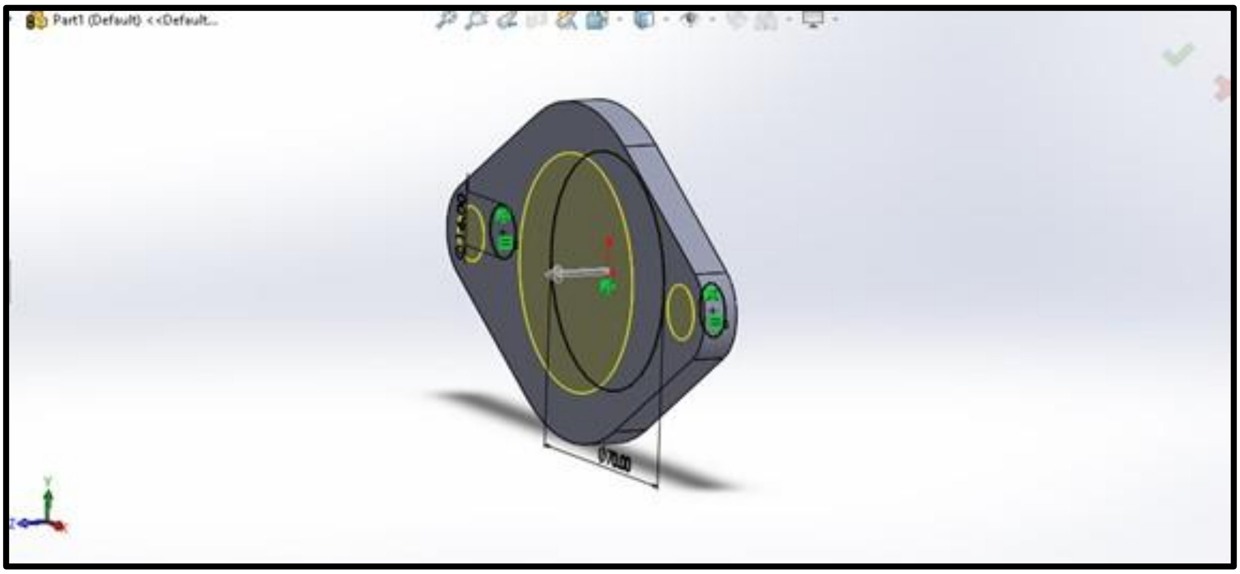


Sketches and dimensions:

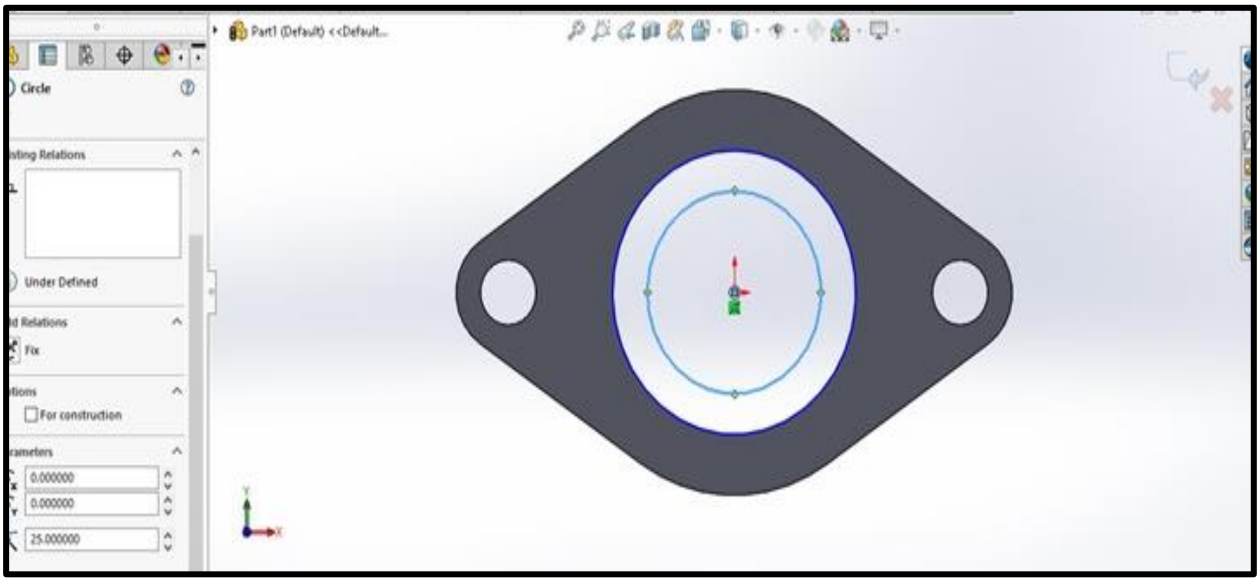


2.4 Remove Material with the Cut-Extrude

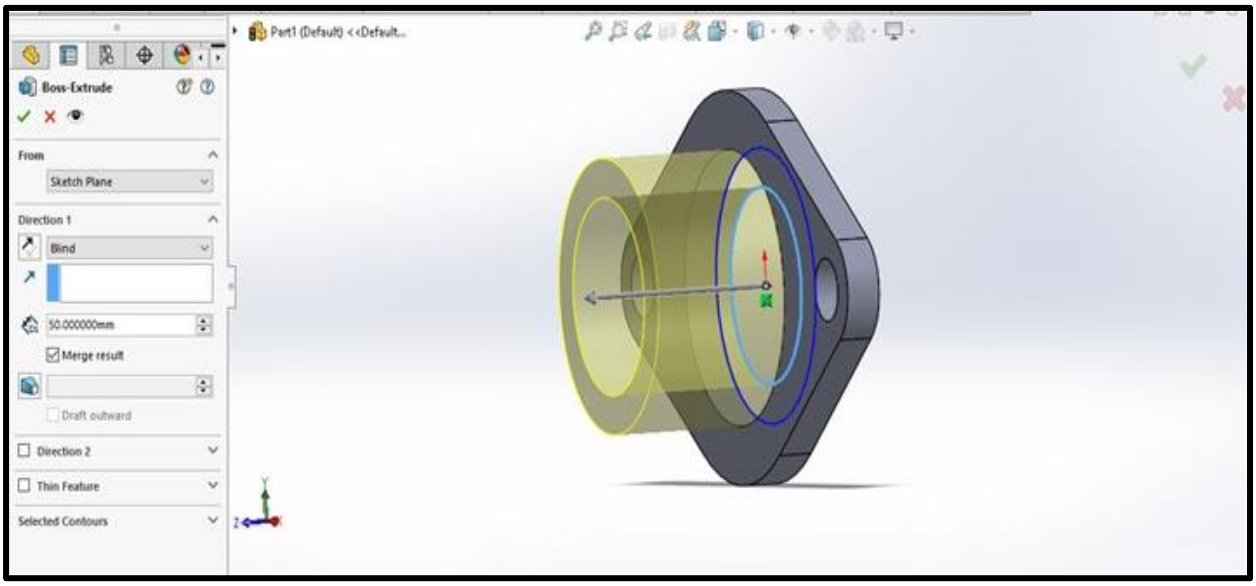
The Cut-Extrude tool is similar to an extrude feature, except that it removes material from the model instead of adding material.[13]



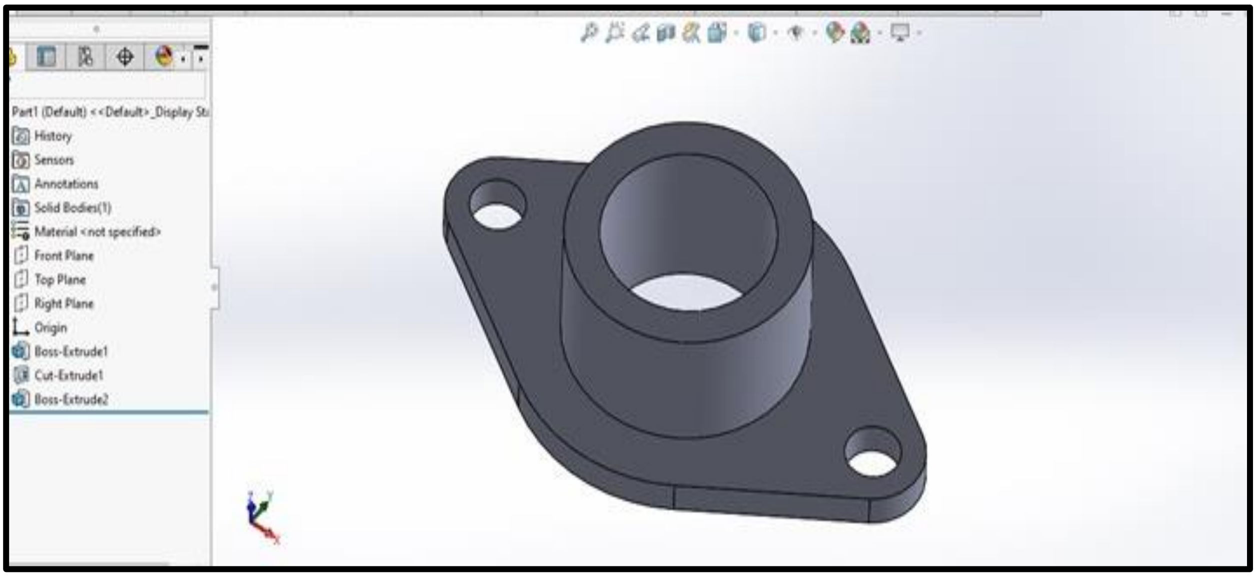
scketsh and dimension:



3D feature :

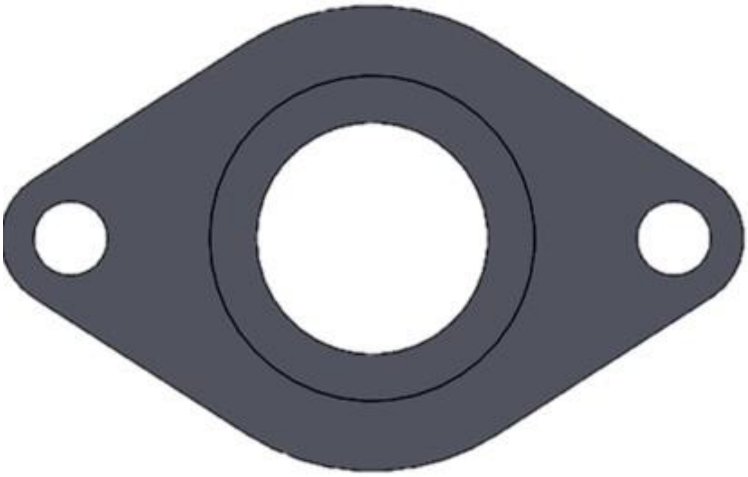


final result:



2.5 Design Views:

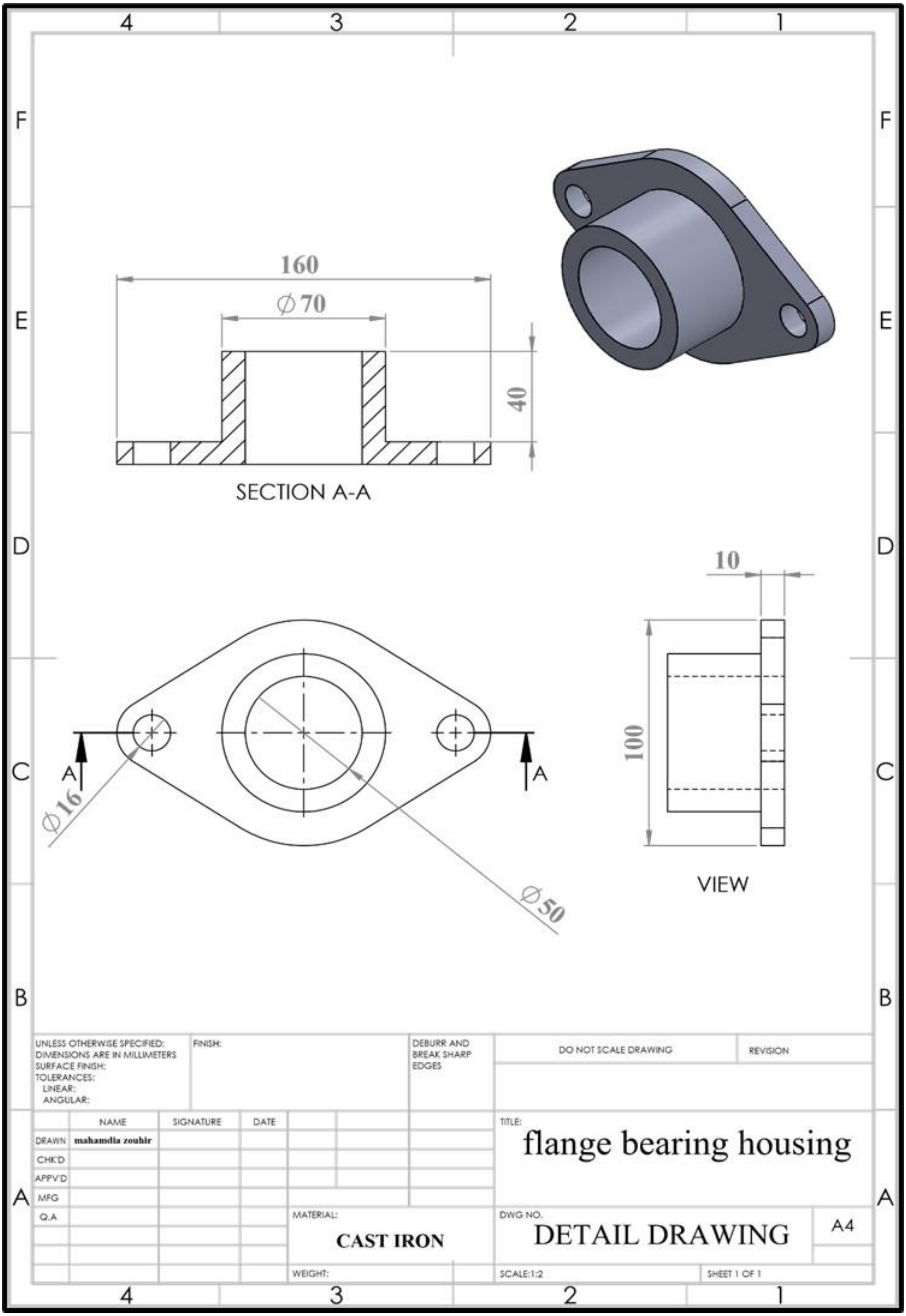
bottom view



front view

Left view

3 Details drawing:



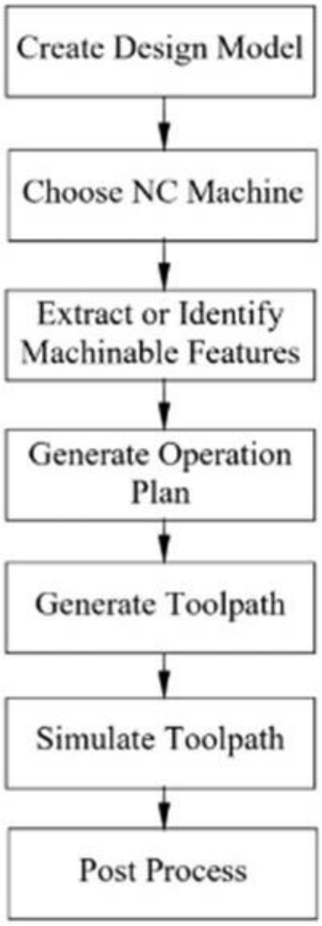
Chapter 3:
Computer-Aided Manufacturing (CAM) using CAMWORKS

1 Introduction:

CAMWorks software, developed by HCL Technologies, is a powerful parametric, feature-based virtual machining software that enhances CNC toolpath creation by using intelligent, automated methods. Instead of traditional manual programming, CAMWorks identifies machinable areas as features, allowing for faster and more intuitive CNC programming. This feature-based approach aligns closely with modern CAD systems, especially SOLIDWORKS, Solid Edge, and CAMWorks Solids, with which CAMWorks is fully integrated. Thanks to this tight integration, users can work within a single interface using the original solid models for both design and machining, eliminating the need to export files in formats like IGES, STEP, SAT, or Parasolid. Toolpaths are generated directly on the SOLIDWORKS model, ensuring accuracy and avoiding approximations. Moreover, these toolpaths are fully associative, meaning any changes to the model automatically update the machining instructions without requiring manual reprogramming. This greatly improves efficiency and reduces the risk of errors. Additionally, CAMWorks is available as a standalone CAD/CAM package that includes an embedded solid modeler (CAMWorks Solids), providing flexibility for users who may not use SOLIDWORKS but still require an integrated and intelligent CAM solution.

2 Virtual Machining

Virtual machining is a simulation-driven technology that enables engineers to define, simulate, and visualize machining processes within a digital environment using computer-aided manufacturing (CAM) tools such as CAMWorks. This virtual setup provides several key advantages, including the ability to easily make adjustments, detect and correct errors early in the process, and gain a clear understanding of machining operations through real-time visualization of simulations. Once the machining process is finalized in the virtual space, the toolpath can be translated into G-code and transferred to a CNC machine on the shop floor to physically produce the part. The workflow of using CAMWorks for virtual machining includes multiple steps: beginning with the creation of the design model (typically a solid model developed in SOLIDWORKS as a part or assembly), followed by the selection of the NC machine and the creation of stock material. Before extracting machinable features, users must define the machine type—whether it’s a mill, lathe, or mill-turn—select the appropriate tool cribs, and choose a compatible post processor. Once these foundational elements are set, machinable features can be identified and an operation plan is generated. This plan outlines the sequence of NC operations required, determines the part setup origin (where the G-code zero point is located), and includes decisions such as tool selection and the definition of machining parameters like feed rate, step-over, and depth of cut. Following this, the software generates the toolpath, simulates the operation to ensure accuracy and efficiency, and finally converts the toolpath into G-code for production.



- Create a solid model, part or assembly, in SOLIDWORKS.

- Choose an NC machine, mill, lathe, or mill-turn, intended for the machining operations, choose a tool crib, select a coordinate system, and select a suitable post Processor

- Create stock, including shape, size and material

- Use AFR to extract machinable features embedded in the solid model, such as holes, pockets, bosses, etc.; or manually identify machinable features

- Generate machining operation plan, including specific machining strategies, cutting tools, and machining parameters, such as feedrate, spindle speed, stepover, depth of cut, etc.

- CAMWorks generates toolpath.

- Simulate machining operations or step through toolpath
Record machining time

- Convert toolpath to G-code
- Verify G-code

Fig.3.1: process of conducting virtual machining using Camworks

3 CAMWorks Machining Modules

The machining modules included in CAMWorks represent a fairly complete set of capabilities in support of virtual machining and toolpath generation. These modules include:

• 2.5 axis mill: includes roughing, finishing, thread milling, face milling, and single point cycles (drilling, boring, reaming, tapping) to machine prismatic features;

• 3 axis mill: includes 2.5 axis capabilities plus strategies to machine complex, contoured surfaces encountered in mold making and aerospace applications;

• 4 axis turning: includes roughing, finishing, grooving, threading, cutoff, and single point cycles (drilling, boring, reaming, tapping);

• Mill-turn: includes milling and turning capabilities for multitasking machine centers;

• Multiaxis machining: 4 axis and 5 axis machining, including high-performance automotive part finishing, impellers, turbine blades, cutting tools, 5 axis trimming, and undercut machining in mold and die making;

• Wire EDM: 2.5 axis and 4 axis cutting operations automate the creation of rough, skim, and tab cuts.

4 CAMWORKS command buttons:

Button Symbol	Name	Function
Define Machine	Define Machine	Allows you to define the machine tool that the part will be machined on, such as 3-axis mill.
Coordinate System	Define Coordinate System	Allows you to define a coordinate system and assign it as the Fixture Coordinate System for the active machine.
Stock Manager	Stock Manger	Allows you to define the raw stock from a bounding box, an extruded sketch or an STL file.
Setup	Part Setup	Allows you to create Part Setups that defines (1) tool orientation (or feed direction), (2) G-code program zero, and (3) the X direction of tool motion.
Extract Machinable Features	Extract Machinable Features	Initiates automatic feature recognition (AFR) to automatically extract solid features that correspond to the machinable features defined in the technology database (TechDB™). The types of machinable features recognized for mill and turn are different. CAMWorks determines the types of features to recognize based on the NC machine selected. The machinable features extracted are listed in the feature manager window under the CAMWorks feature tree tab .
Generate Operation Plan	Generate Operation Plan	Generates operation plans automatically for the selected machinable features. The operation plans and associated machining strategy and machining parameters are selected based on rules defined in TechDB™. An operation plan contains information on how the machinable features are to be machined. The operations generated are listed in the feature manager window under the CAMWorks operation tree tab .
Generate Toolpath	Generate Toolpath	Creates toolpath for the selected operation plans and displays the toolpath on the part. A toolpath is a cutting entity (line, circle, arc, etc.) created by a cutting cycle that defines tool motion.
Simulate Toolpath	Simulate Toolpath	Provides a visual verification of the machining process for the current part by simulating the tool motion and the material removal process.
Step Thru Toolpath	Step Through Toolpath	Allows you to view toolpath movements either one movement at a time, a specified number of movements or all movements.
Save CL File	Save CL File	Allows you to save the current operation and associated parameters in the technology database as CL (cutter location) data for future use.
Post Process	Post Process	Translates toolpath and operation information into G-code for a specific machine tool controller.

Table 2: the major command buttons in CAMWORKS

5 CAMWorks Machine Simulation

CAMWorks Machine Simulation provides a realistic virtual environment that replicates the physical setup of an NC machine, enabling users to conduct machining simulations with high fidelity. This simulation includes detailed computer models of the machine itself, tilt rotary tables, fixtures, tools, tool holders, stock, and parts, accurately representing the actual machining environment. Beyond simulating machining operations, the Machine Simulation module offers advanced tool collision detection within this realistic context, enhancing safety and efficiency. Although offered as a separately licensed module, some legacy machines, such as mills and mill-turn centers, come integrated with CAMWorks. For example, Figure 3-1 illustrates a machining simulation using a legacy mill called Mill Tutorial, where components like the tool, tool holder, tilt and rotary tables, stock, and the machine coordinate system (XYZ) are clearly shown (as detailed in Figure 3-2). The right side of the interface displays the machining operations along with the corresponding G-code in the upper and lower sections of the Move List. At the top, users select the default virtual machine (simulator), Mill Tutorial, and beneath it are controls for running and managing the machining simulation. [14]

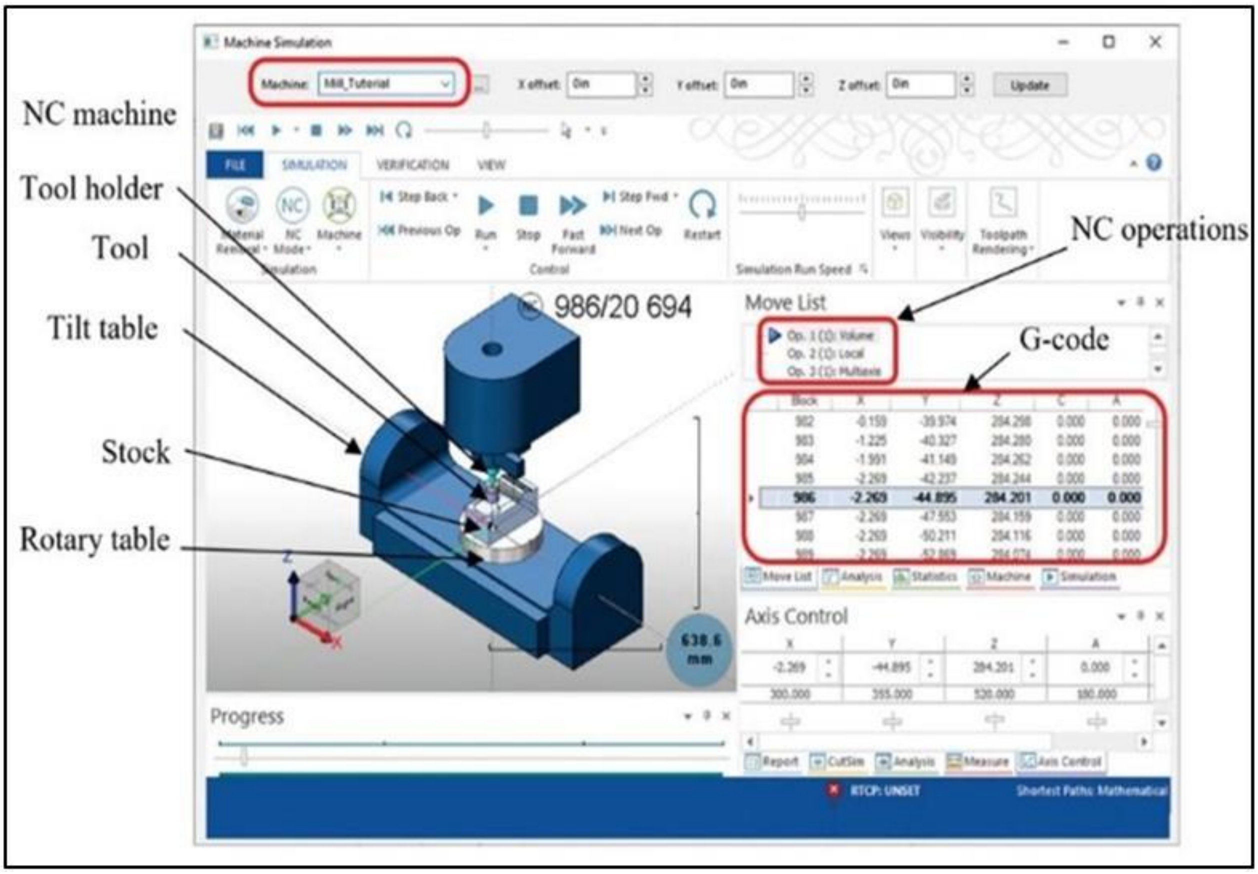


Fig.3.2:CAMWorks Machine Simulation

6 The process plan guide:

we will do two process plans for this piece the first with 3axis mill machine and the second with 4 axis mill-turn machine.

- the 3axes mill:

Mill setup one		**Mill setup two**	
features	**operations**	**features**	**Operations**
Face mill	Face mill	Open pocket	Rough mill, contour mill
Open pocket	Rough mill, contour mill	drill	Center drill, drill
drill	Center drill, drill, rough mill, finish mill		
Operations parameters			
Tools, strategies, spindle speed, surface speed, feed……			

- the 4axes mill-turn:

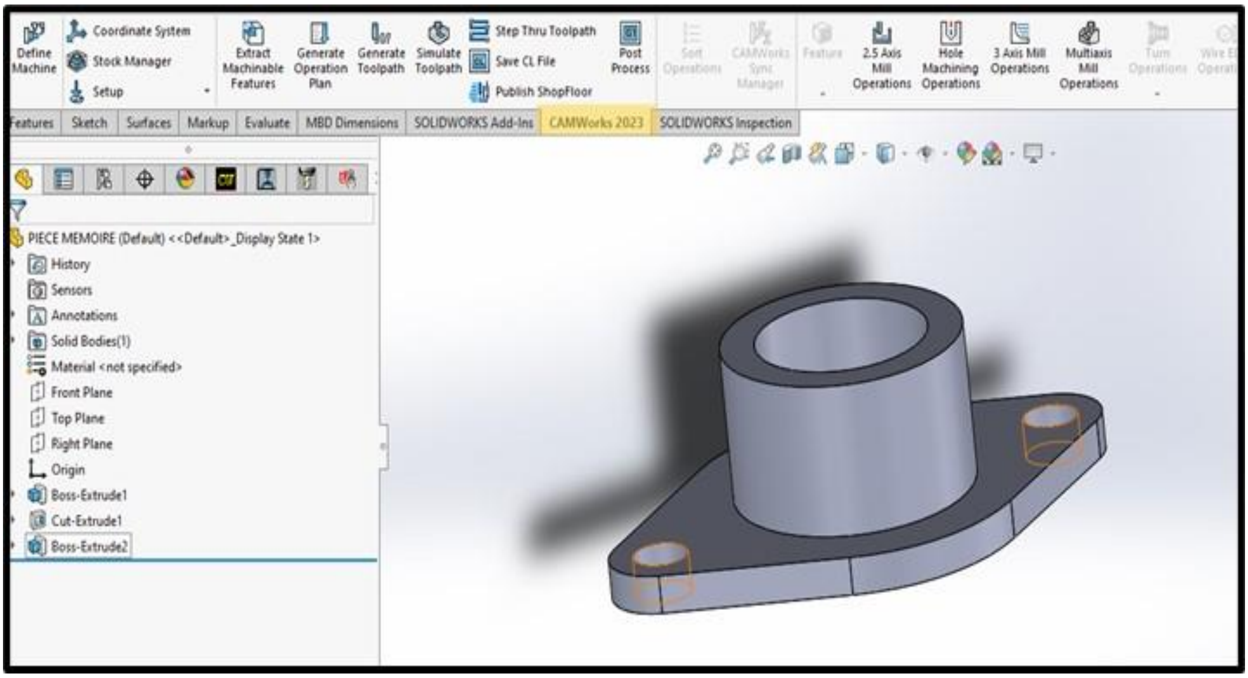
Turn setup one		**Mill setup**		**Turn setup two**	
features	**operations**	**features**	**Operations**	**features**	**operations**
Face feature	Face rough, face finish	Open pocket	Rough mill, contour mill	Cut off	Cut off
Outside Diameter Features	Turn rough, turn finish	drill	Center drill, drill		
Inside Diameter Features	Center drill, drill, bore rough, bore finish				
Operations parameters					
Tools, strategies, spindle speed, surface speed, feed……					

7 THE 3 AXES MILL:

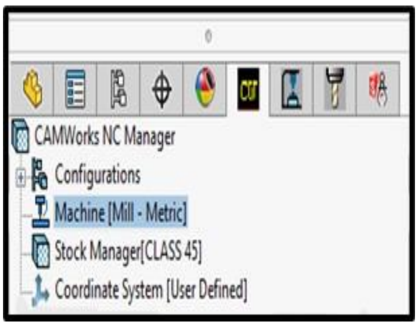
7.1 The principal steps:

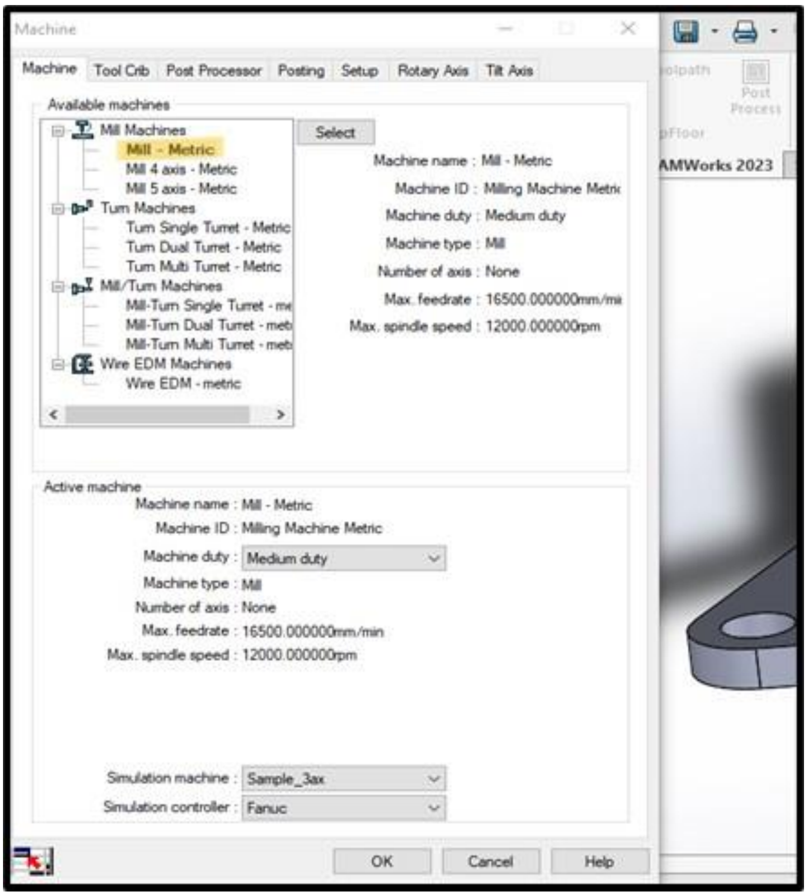
Those first steps: Define machine (machine, tool crib, post processor), stock manager, coordinate system and setup they are the principal steps we must do.

- select camworks from the command manager.

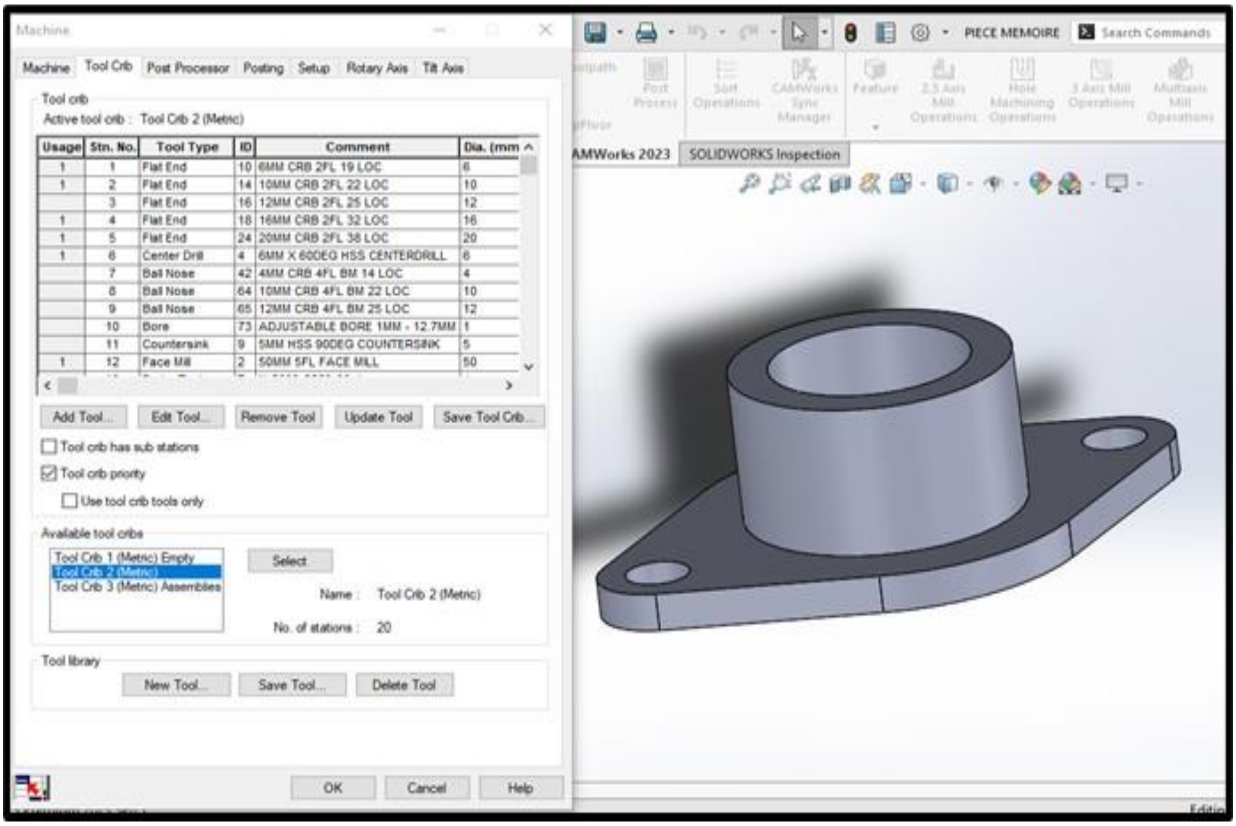


- Choose a type of machine.

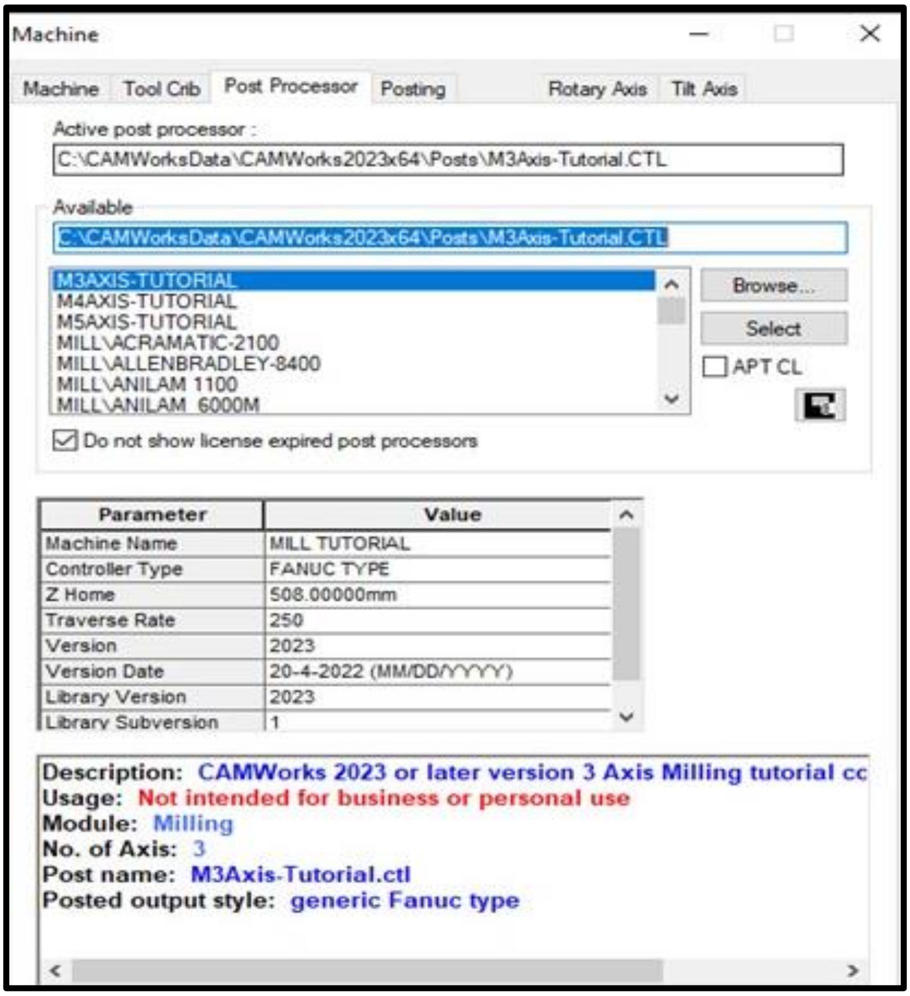




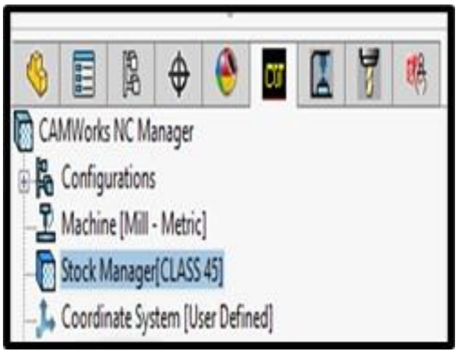
- Choose a tool crib.

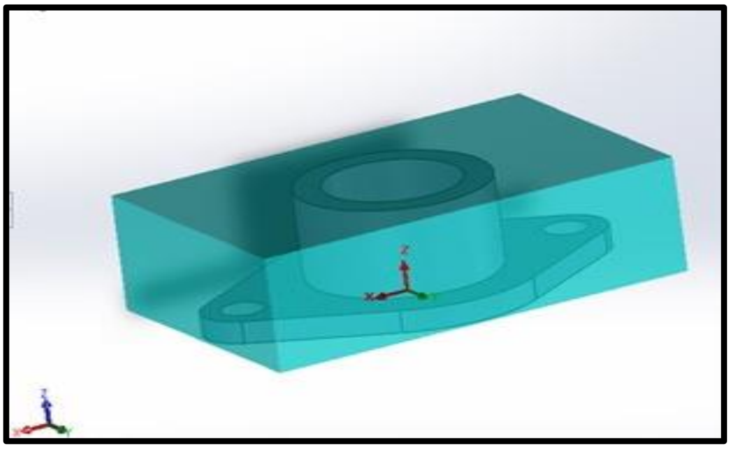


- Select a post processor.

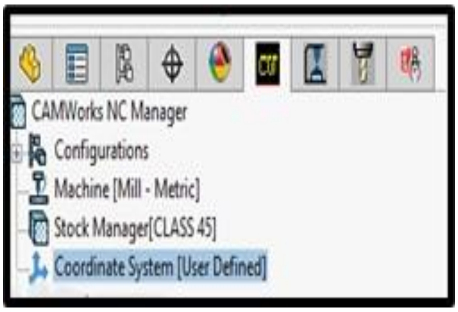


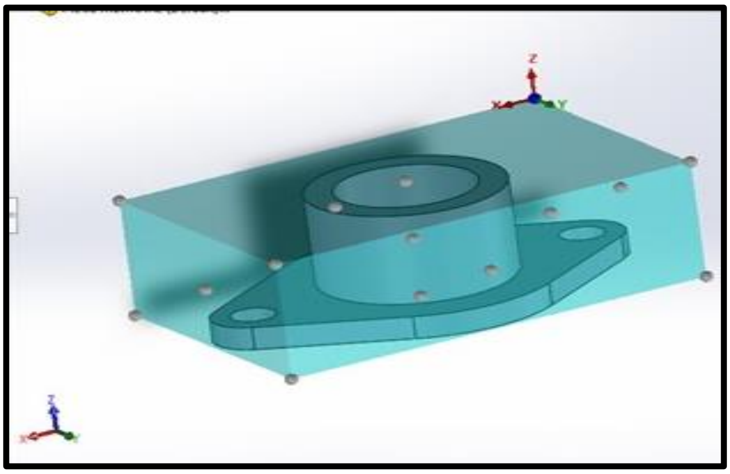
- Define a stock manager or raw material



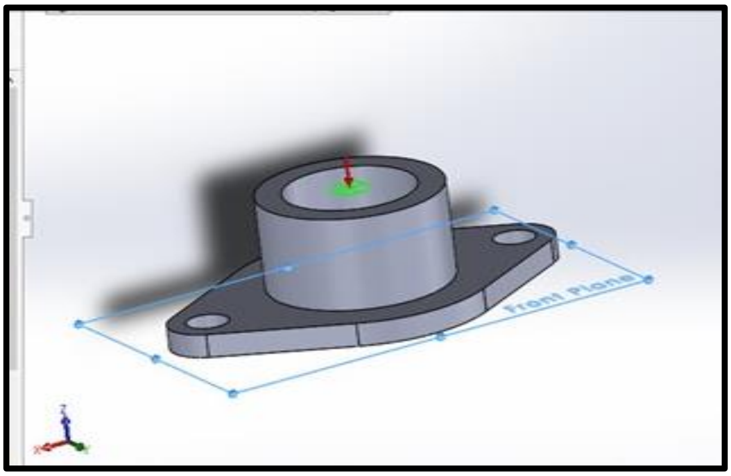


- Define a coordinate system



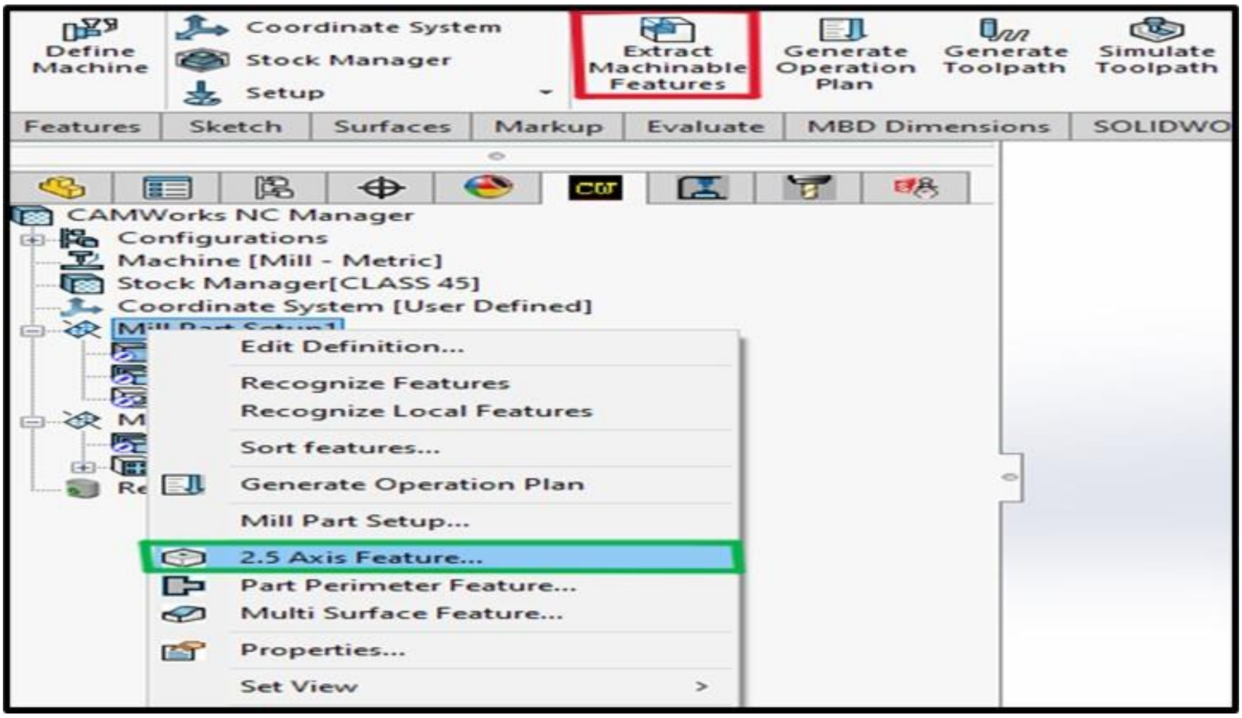


- Define a mill part setup



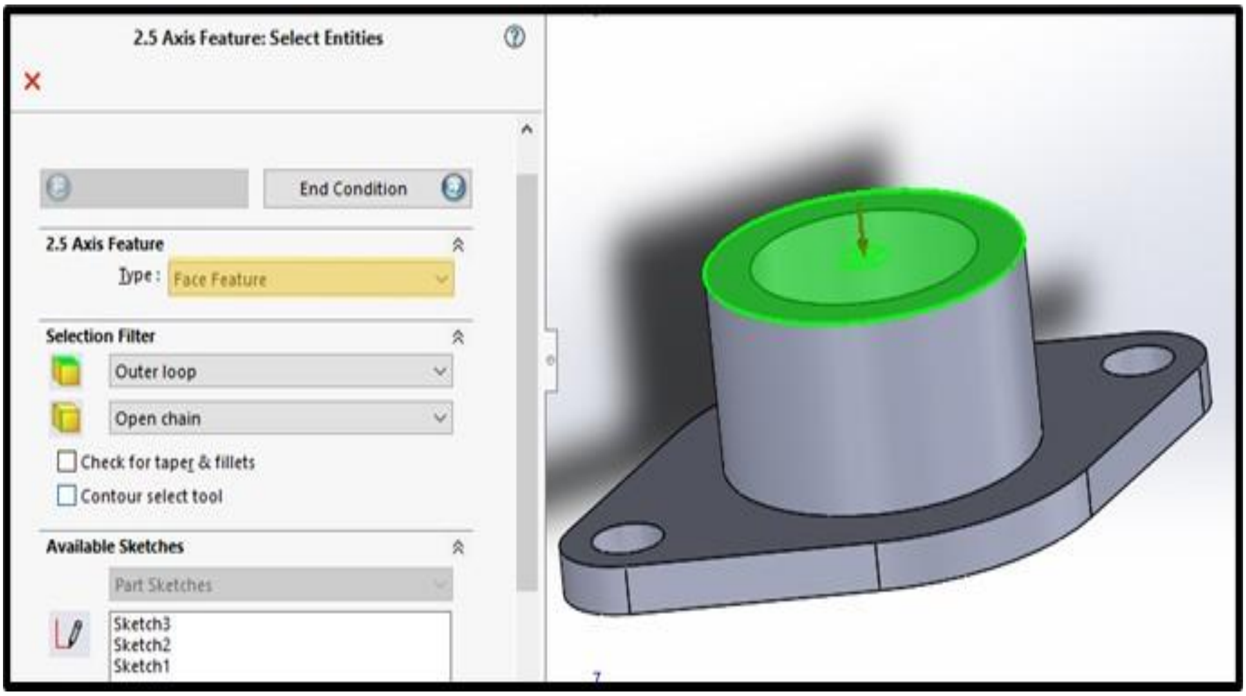
7.2 Mill operations:

We will not use **Extract machinable features** button because it will give us random tools and don't follow our tool crib collection and post processor. instead, we will use **2.5 Axis features**.

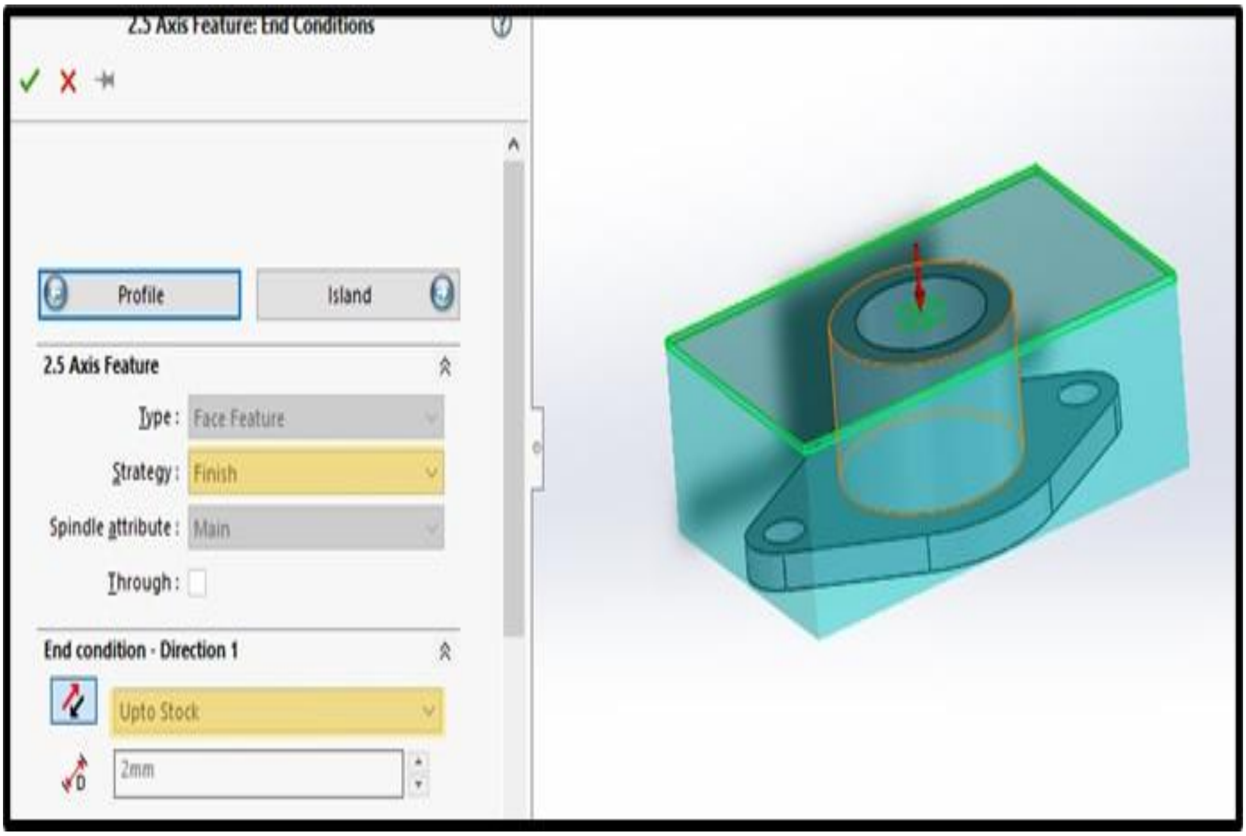


7.2.1 Face mill

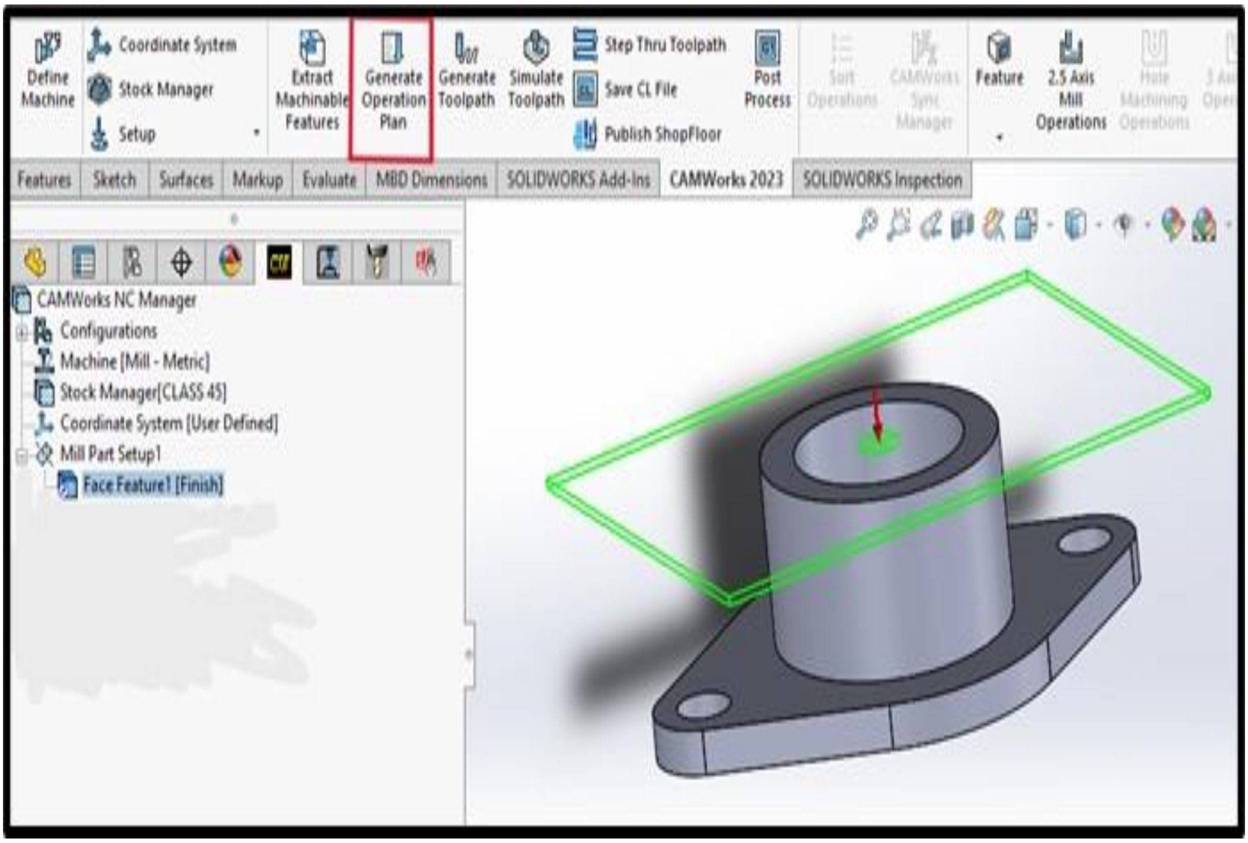
Step 1: choosing face feature as operation type and select the surface that you want to do facing operation on it.



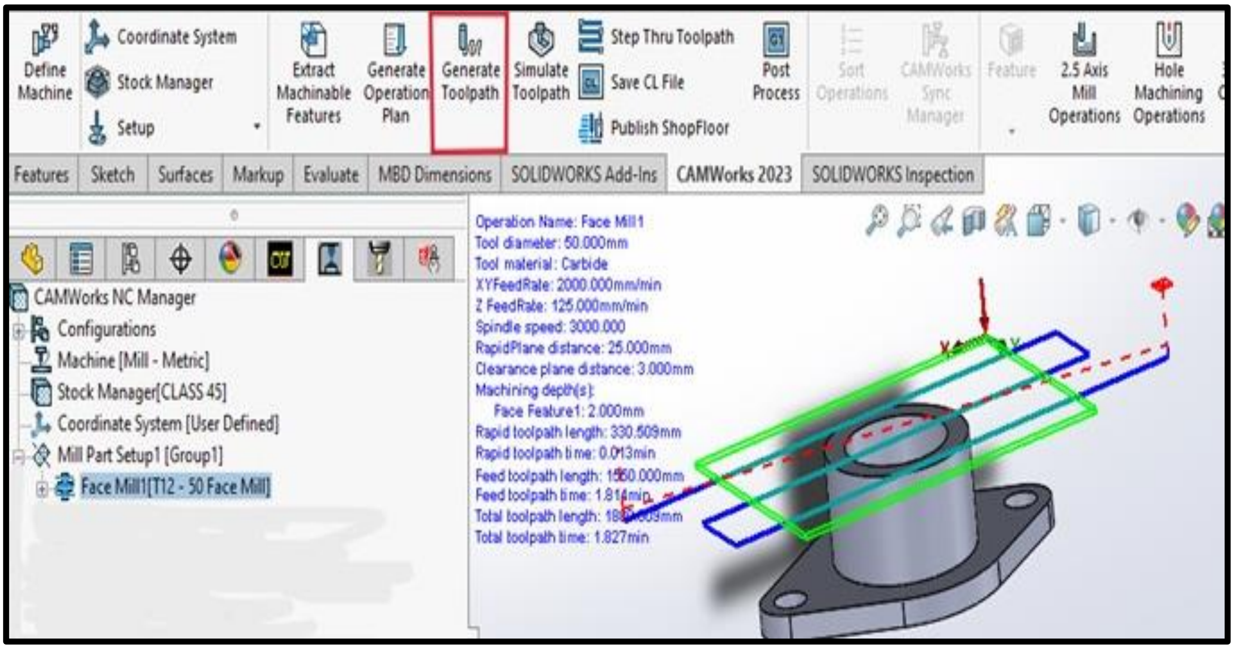
Step 2 : select a strategy and confirm with the green check .



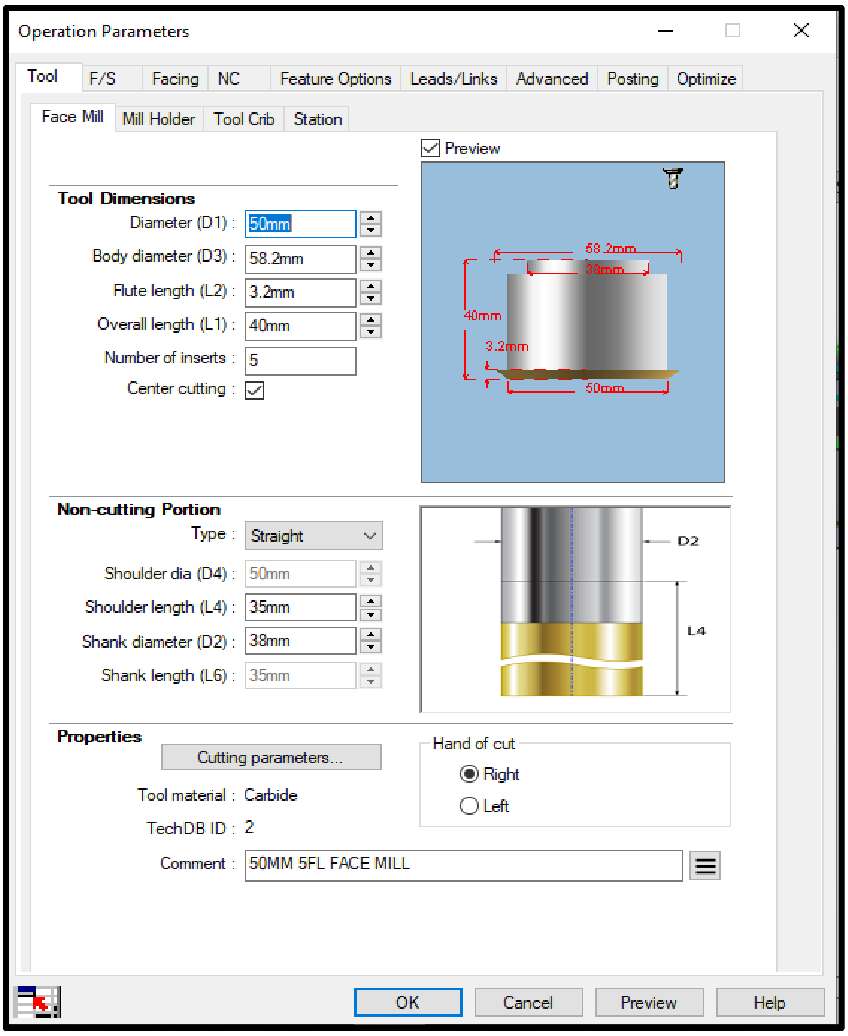
Step 3 : press **generate operation plan** that will give the necessery operations

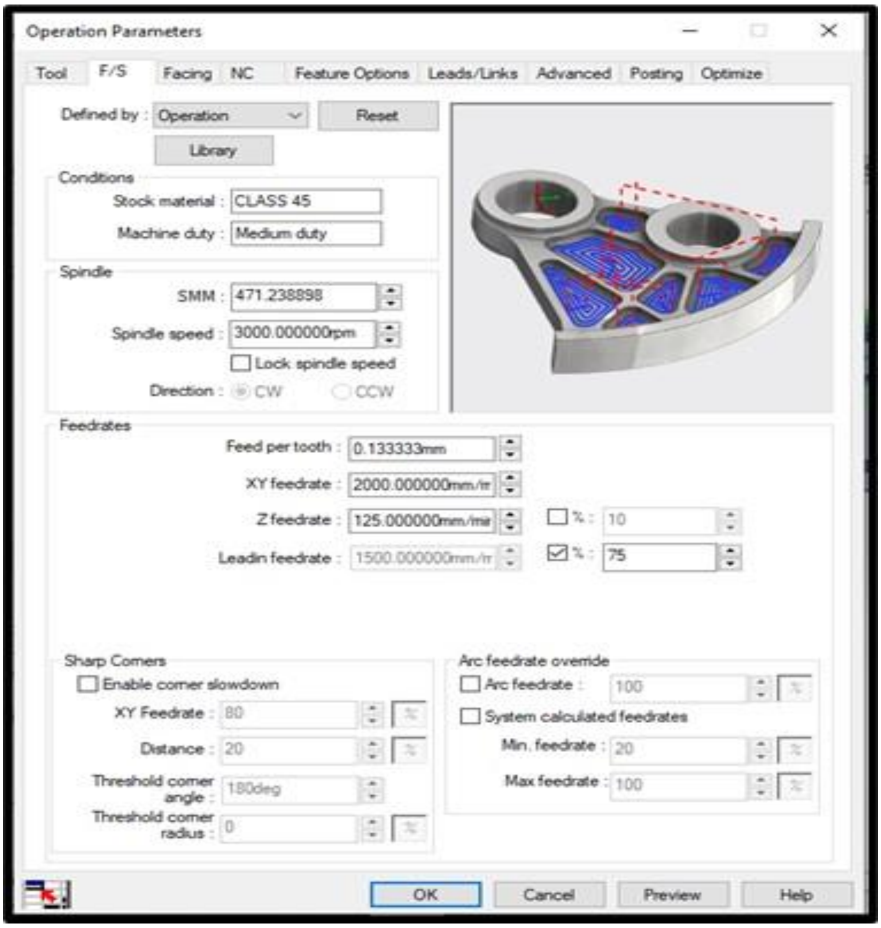


Step 4: press **generate toolpath**.



Face mill:

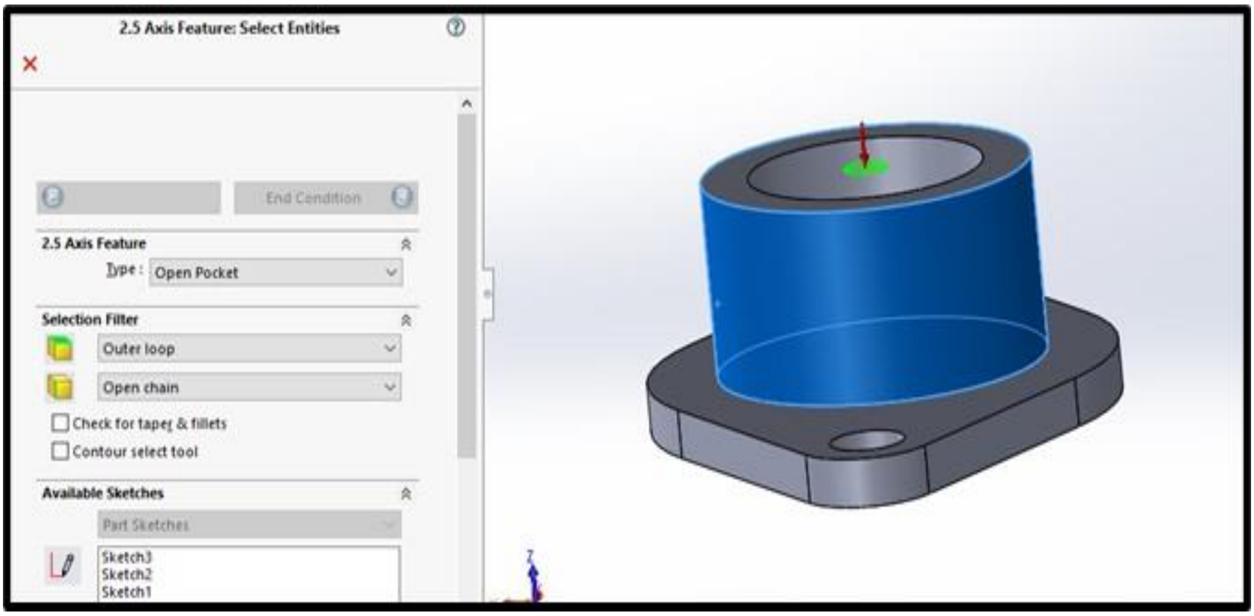




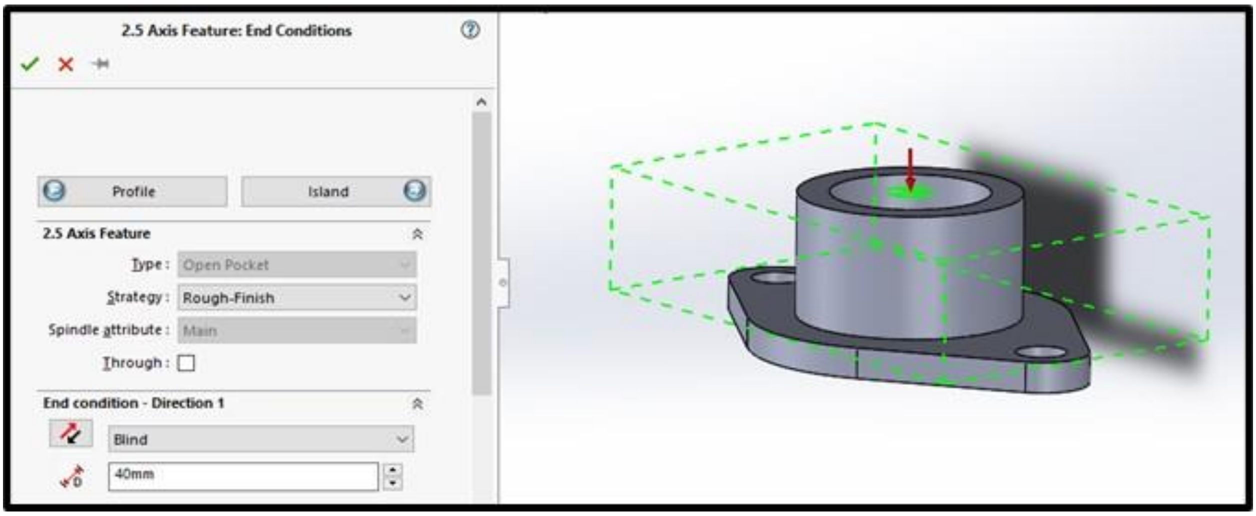
7.2.2 Mill open pocket:

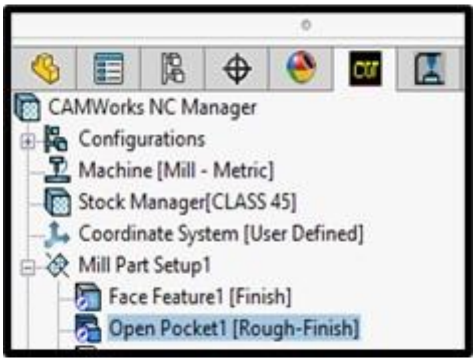
We will take the same steps in the first operation.

Type: open pocket

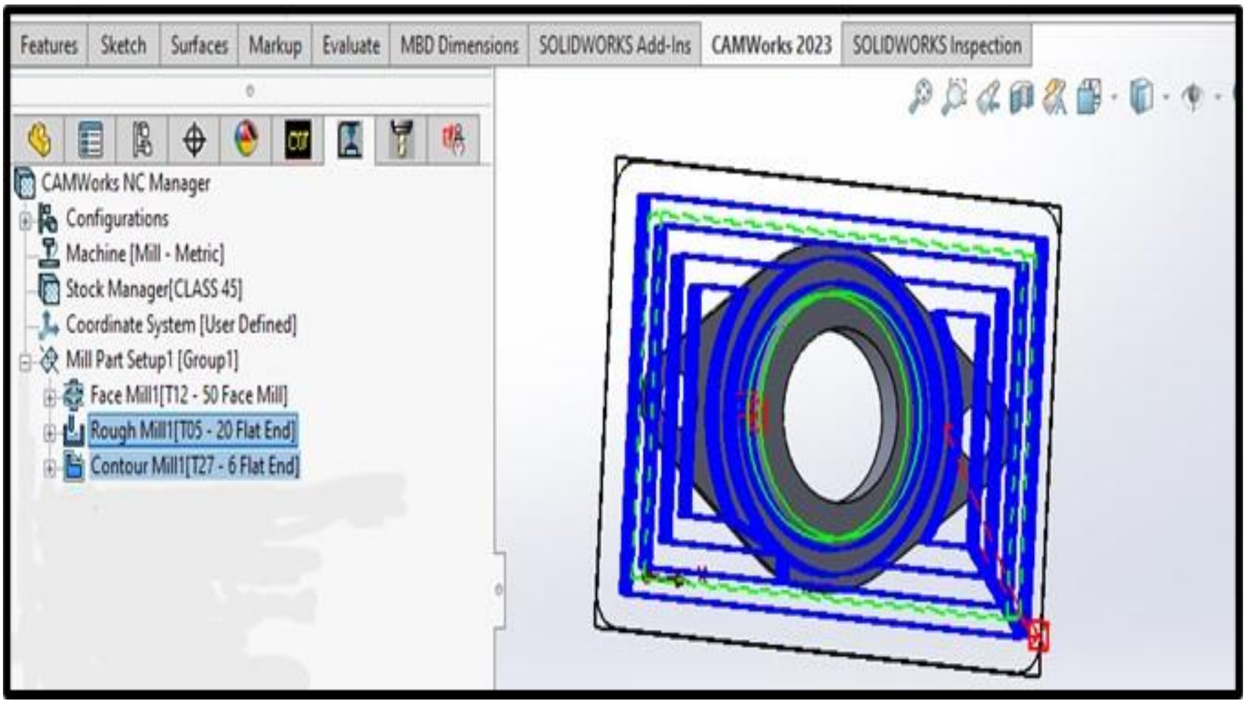


Strategy: rough-finish

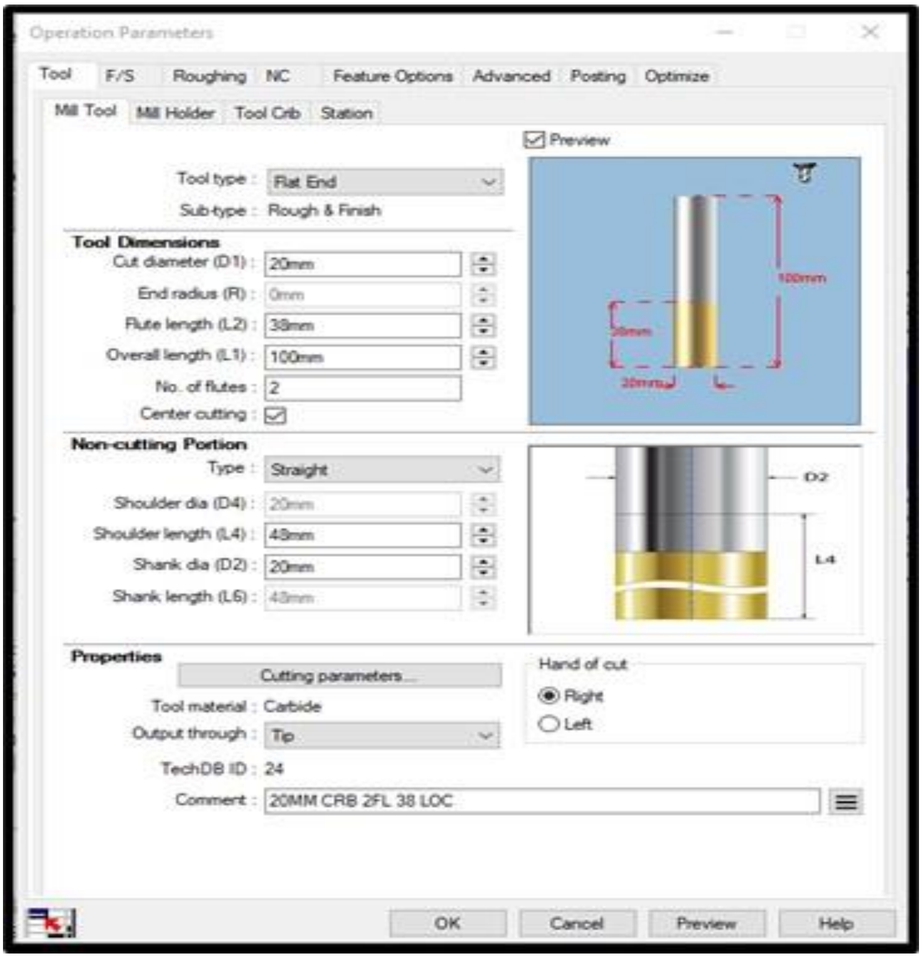


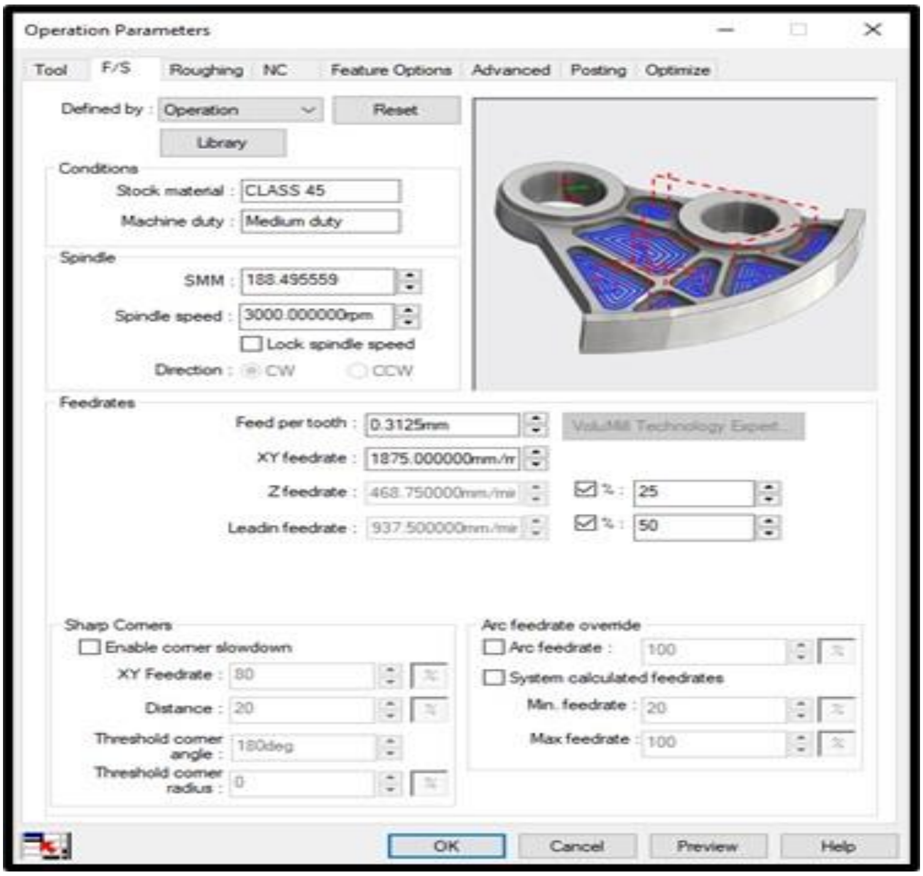


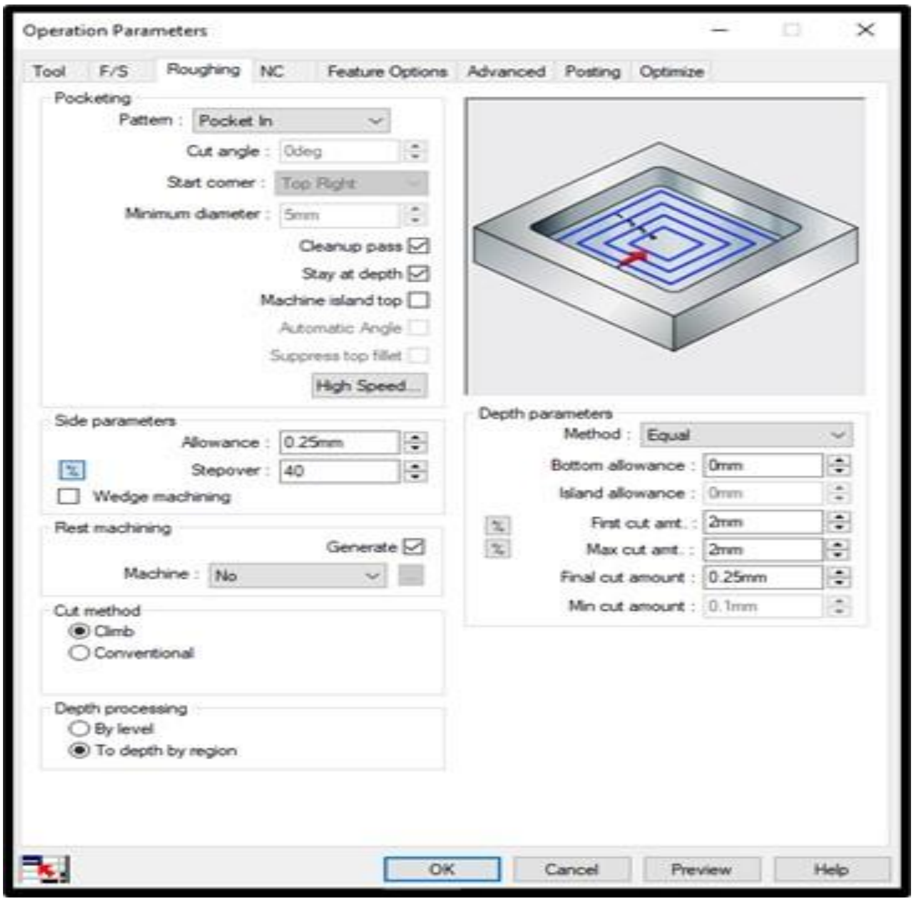
Generate operation plan and tool path.

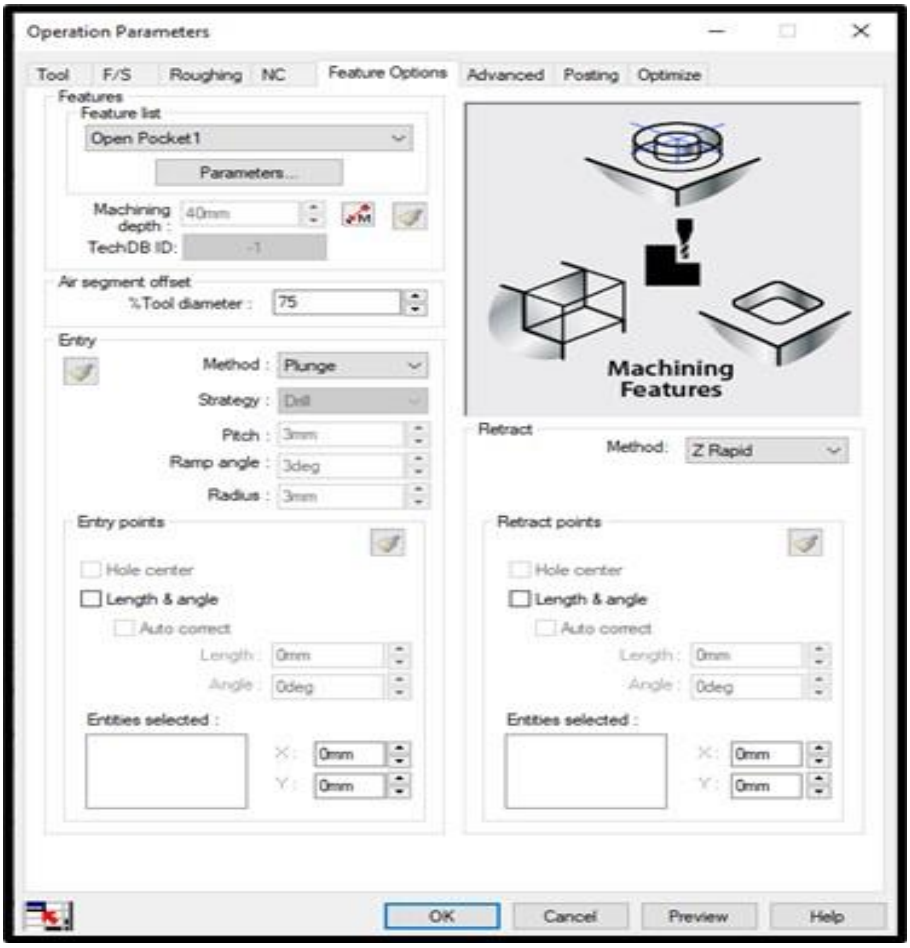


ROUGH MILL:

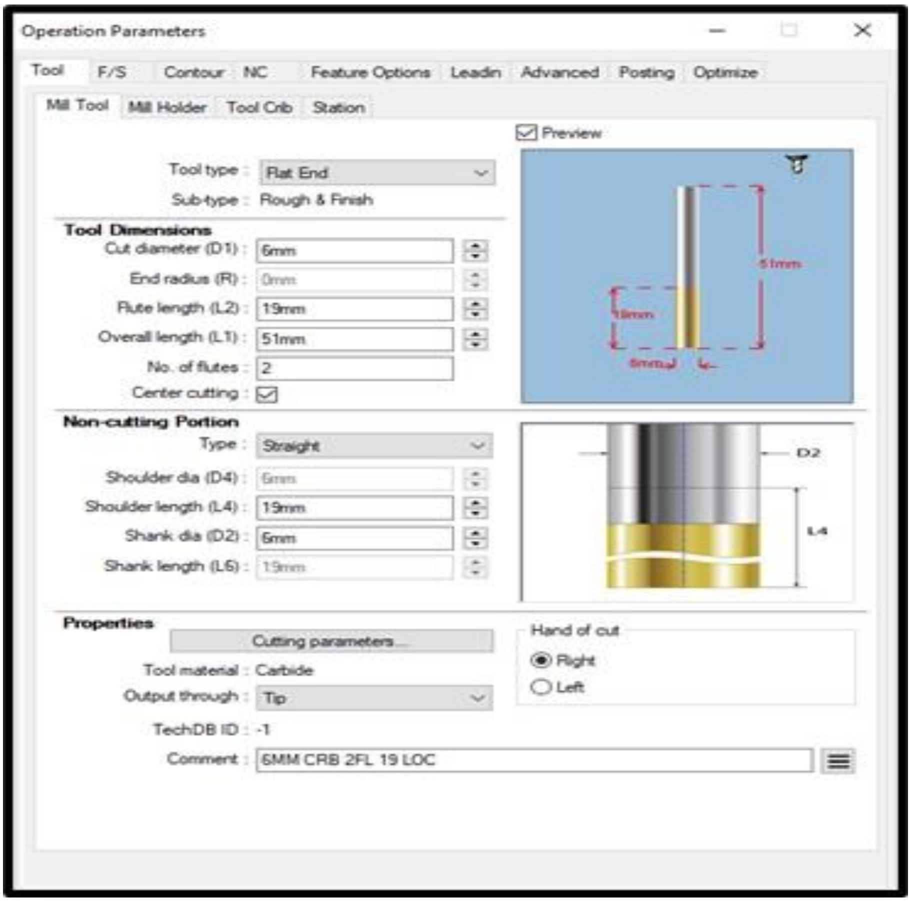

Operation Parameters
Tool
F/S
Roughing
NC
Feature Options
Advanced
Posting
Optimize
Mill Tool
Mill Holder
Tool Crib
Station
Preview
Tool type : Flat End
Sub-type : Rough & Finish
Tool Dimensions
Cut diameter (D1) : 20mm
End radius (R) : 0mm
Flute length (L2) : 38mm
Overall length (L1) : 100mm
No. of flutes : 2
Center cutting :
Non-cutting Portion
Type : Straight
Shoulder dia (D4) : 20mm
Shoulder length (L4) : 48mm
Shank dia (D2) : 20mm
Shank length (L6) : 48mm
D2
L4
Properties
Cutting parameters...
Tool material : Carbide
Output through : Tip
TechDB ID : 24
Comment : 20MM CRB 2FL 38 LOC
Hand of cut
Right
Left
OK
Cancel
Preview
Help

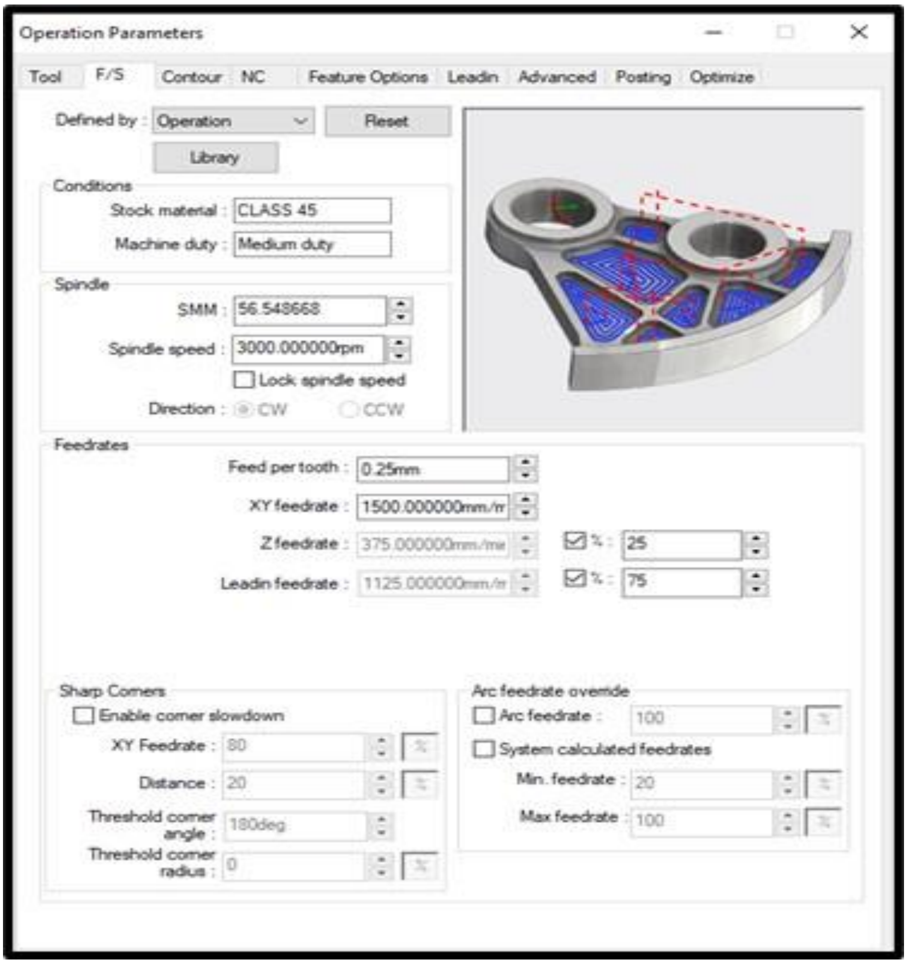

Operation Parameters
Tool
F/S
Roughing
NC
Feature Options
Advanced
Posting
Optimize
Defined by : Operation
Reset
Library
Conditions
Stock material : CLASS 45
Machine duty : Medium duty
Spindle
SMM : 188.495559
Spindle speed : 3000.000000rpm
Lock spindle speed
Direction : CW CCW
Feedrates
Feed per tooth : 0.3125mm
XY feedrate : 1875.000000mm/m
Z feedrate : 468.750000mm/mi
Leadin feedrate : 937.500000mm/mi
% : 25
% : 50
Sharp Corners
Enable corner slowdown
XY Feedrate : 80
Distance : 20
Threshold corner angle : 180deg
Threshold corner radius : 0
Arc feedrate override
Arc feedrate : 100
System calculated feedrates
Min. feedrate : 20
Max feedrate : 100
OK
Cancel
Preview
Help

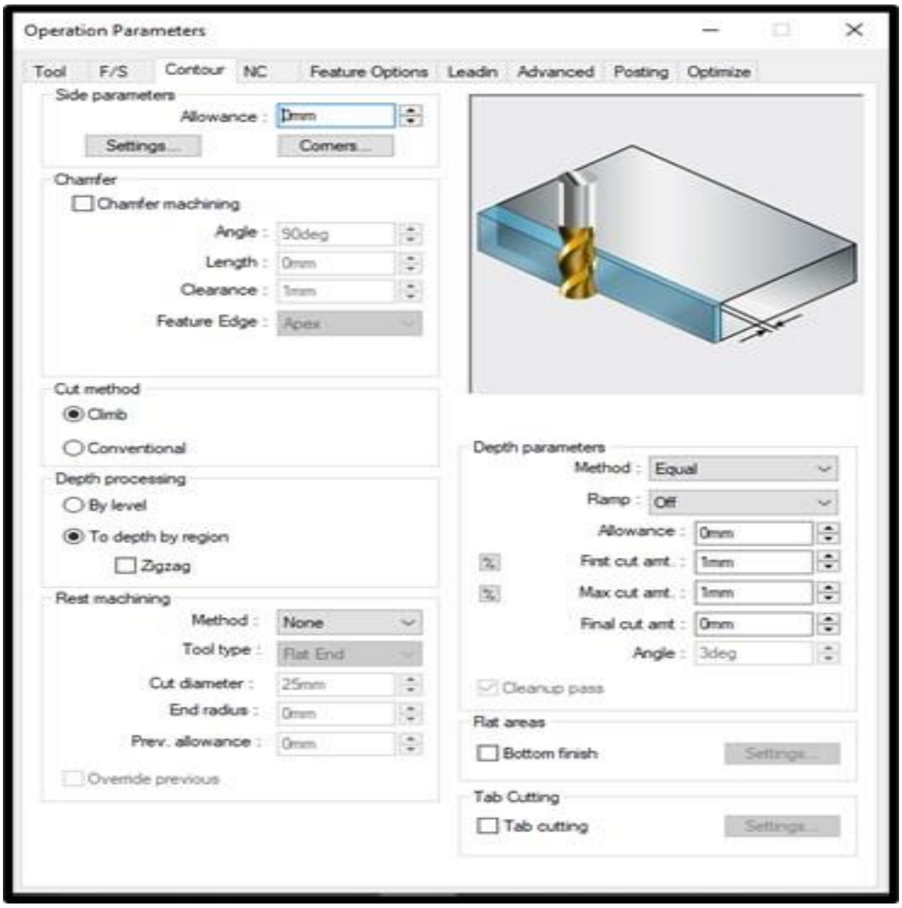

Operation Parameters
Tool
F/S
Roughing
NC
Feature Options
Advanced
Posting
Optimize
Pocketing
Pattern : Pocket In
Cut angle : 0deg
Start corner : Top Right
Minimum diameter : 5mm
Cleanup pass
Stay at depth
Machine island top
Automatic Angle
Suppress top fillet
High Speed...
Side parameters
Allowance : 0.25mm
Stepover : 40
Wedge machining
Rest machining
Generate
Machine : No
Cut method
Climb
Conventional
Depth processing
By level
To depth by region
Depth parameters
Method : Equal
Bottom allowance : 0mm
Island allowance : 0mm
First cut amt. : 2mm
Max cut amt. : 2mm
Final cut amount : 0.25mm
Min cut amount : 0.1mm
OK
Cancel
Preview
Help

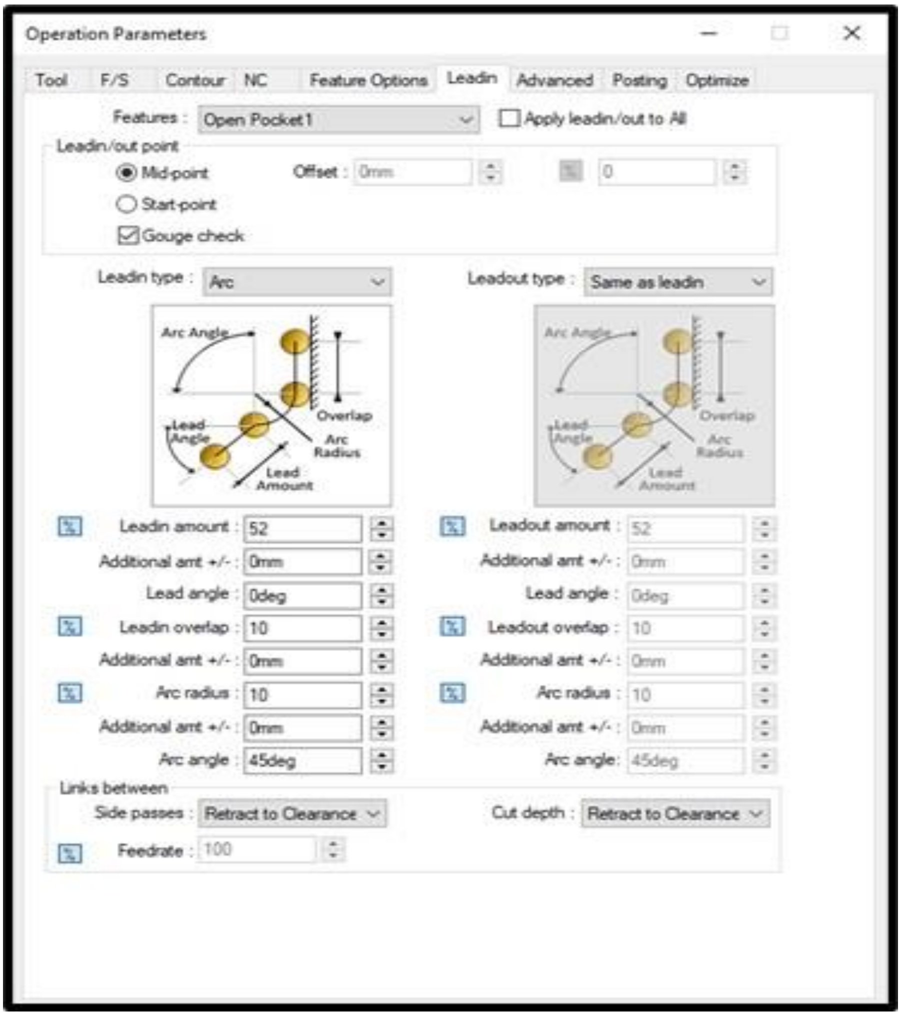

Operation Parameters
Tool
F/S
Roughing
NC
Feature Options
Advanced
Posting
Optimize
Features
Feature list
Open Pocket1
Parameters...
Machining depth : 40mm
TechDB ID: -1
Air segment offset
%Tool diameter : 75
Entry
Method : Plunge
Strategy : Drill
Pitch : 3mm
Ramp angle : 3deg
Radius : 3mm
Entry points
Hole center
Length & angle
Auto correct
Length : 0mm
Angle : 0deg
Entities selected :
X : 0mm
Y : 0mm
Machining Features
Retract
Method: Z Rapid
Retract points
Hole center
Length & angle
Auto correct
Length : 0mm
Angle : 0deg
Entities selected :
X : 0mm
Y : 0mm
OK
Cancel
Preview
Help

CONTOUR MILL:



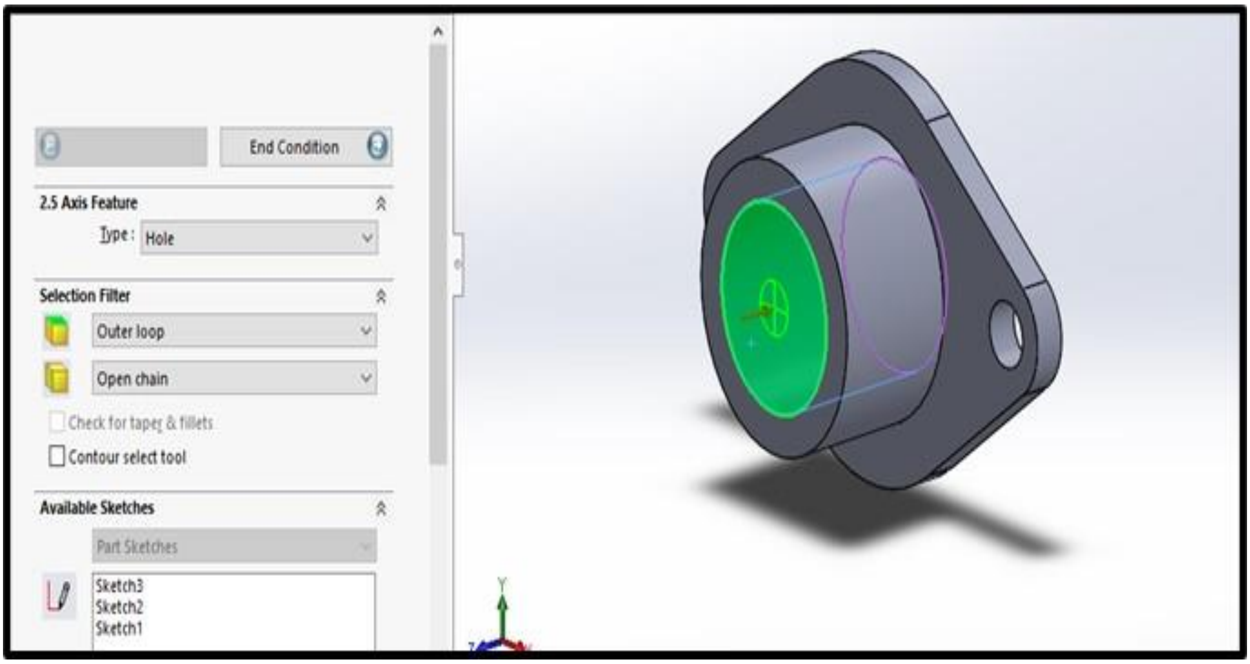



Operation Parameters
Tool
F/S
Contour
NC
Feature Options
Leadin
Advanced
Posting
Optimize
Side parameters
Allowance : 0mm
Settings...
Corners...
Chamfer
Chamfer machining
Angle : 90deg
Length : 0mm
Clearance : 1mm
Feature Edge : Apex
Cut method
Climb
Conventional
Depth processing
By level
To depth by region
Zigzag
Rest machining
Method : None
Tool type : Flat End
Cut diameter : 25mm
End radius : 0mm
Prev. allowance : 0mm
Override previous
Depth parameters
Method : Equal
Ramp : Off
Allowance : 0mm
First cut amt. : 1mm
Max cut amt. : 1mm
Final cut amt : 0mm
Angle : 3deg
Cleanup pass
Flat areas
Bottom finish
Settings...
Tab Cutting
Tab cutting
Settings...

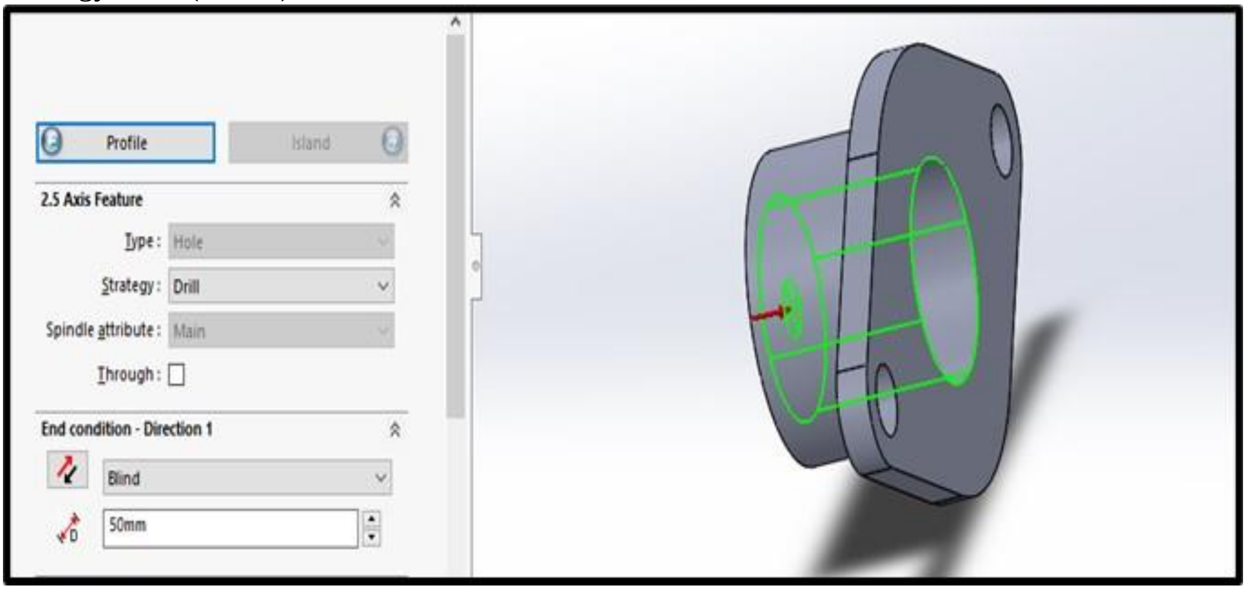

Operation Parameters
Tool
F/S
Contour
NC
Feature Options
Leadin
Advanced
Posting
Optimize
Features : Open Pocket1
Apply leadin/out to All
Leadin/out point
Mid-point
Offset : 0mm
0
Start-point
Gouge check
Leadin type : Arc
Leadout type : Same as leadin
Arc Angle
Lead Angle
Overlap
Arc Radius
Lead Amount
Leadin amount : 52
Additional amt +/- : 0mm
Lead angle : 0deg
Leadin overlap : 10
Additional amt +/- : 0mm
Arc radius : 10
Additional amt +/- : 0mm
Arc angle : 45deg
Leadout amount : 52
Additional amt +/- : 0mm
Lead angle : 0deg
Leadout overlap : 10
Additional amt +/- : 0mm
Arc radius : 10
Additional amt +/- : 0mm
Arc angle : 45deg
Links between
Side passes : Retract to Clearance
Cut depth : Retract to Clearance
Feedrate : 100

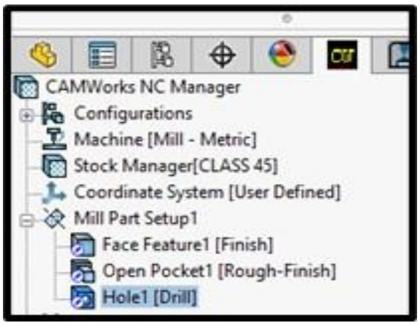
7.2.3 Hole operation:

Type: hole.

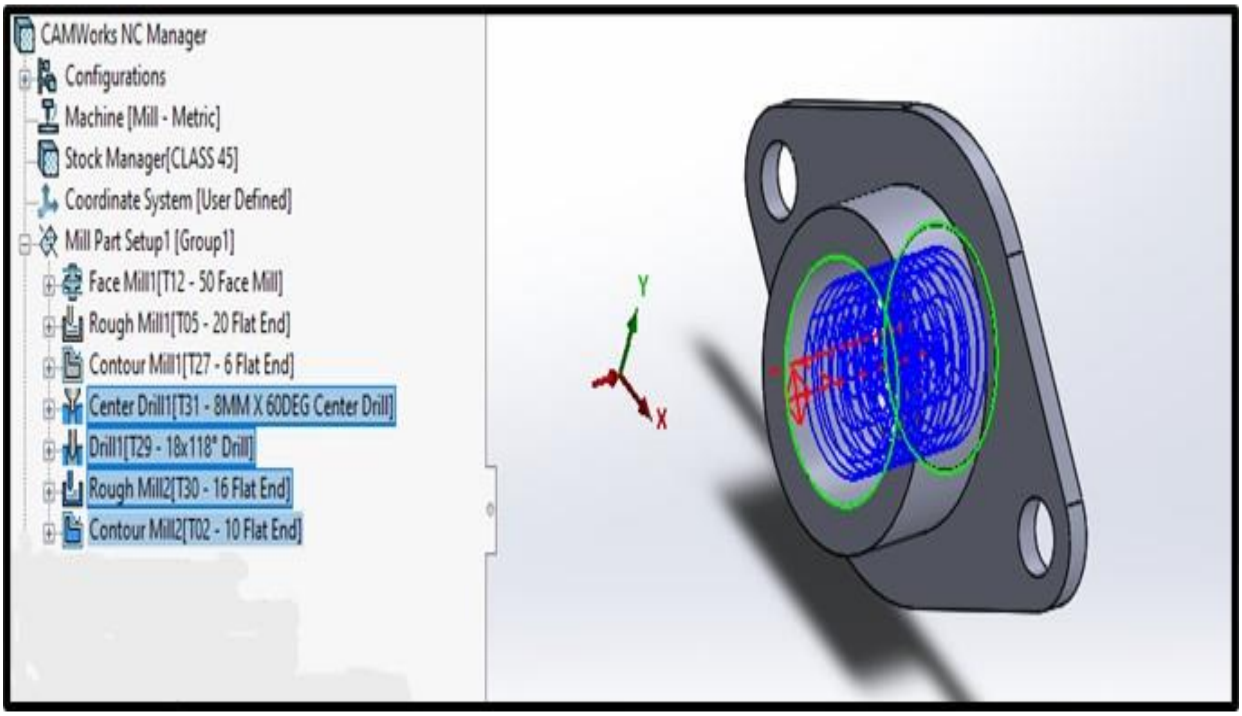


Strategy: drill (50mm)

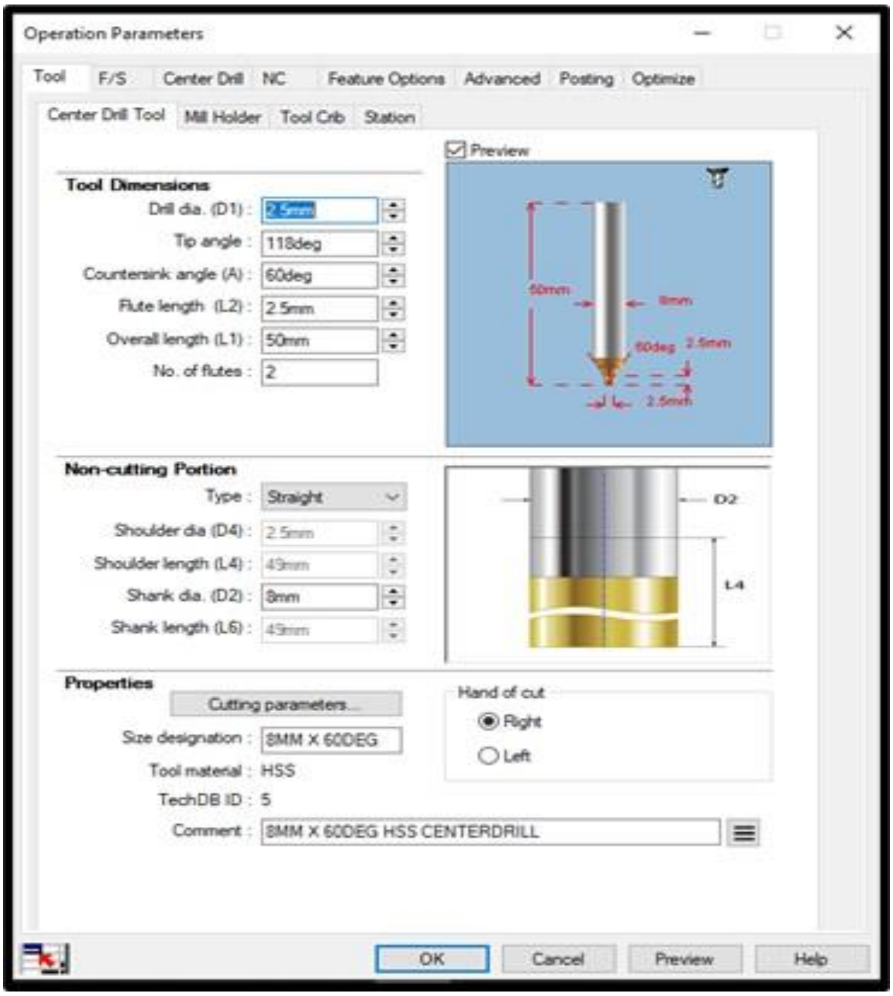


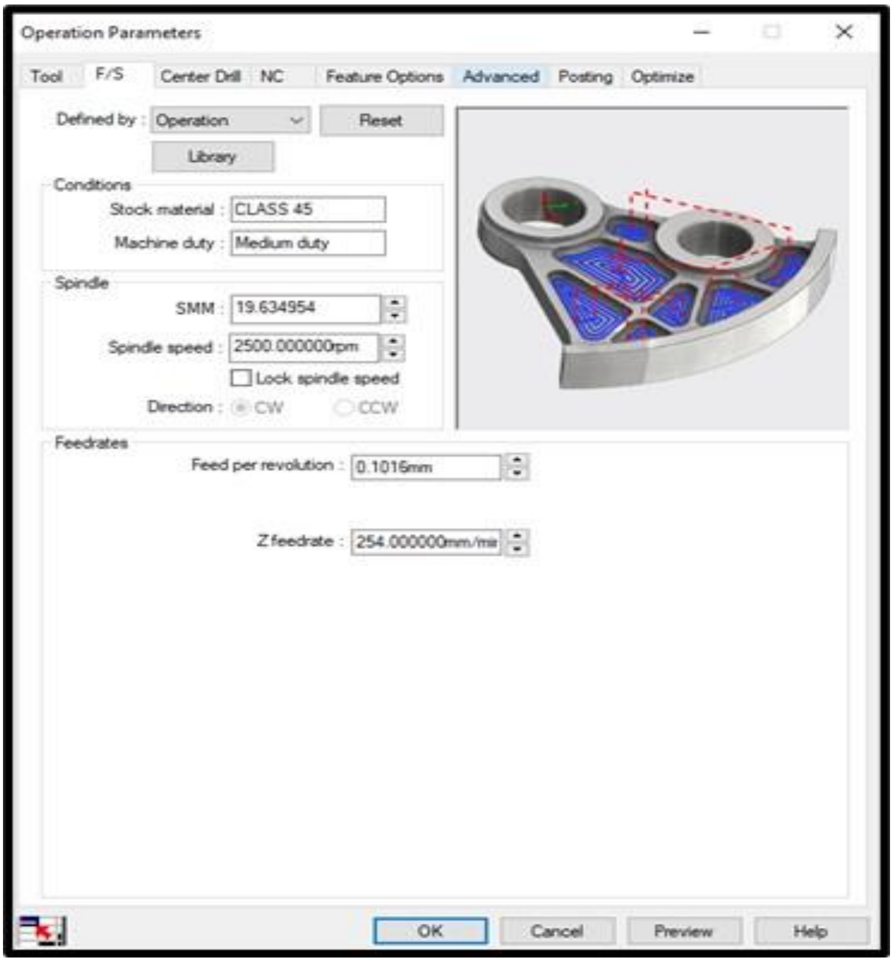


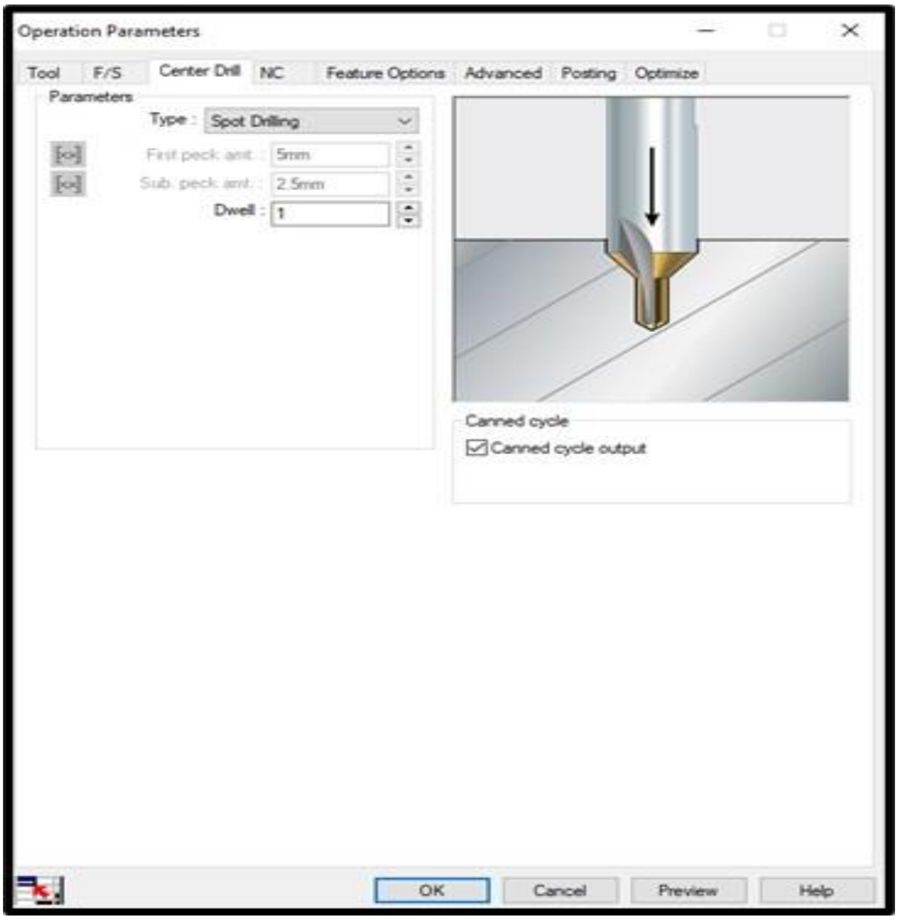
Generate operation plan and tool path.

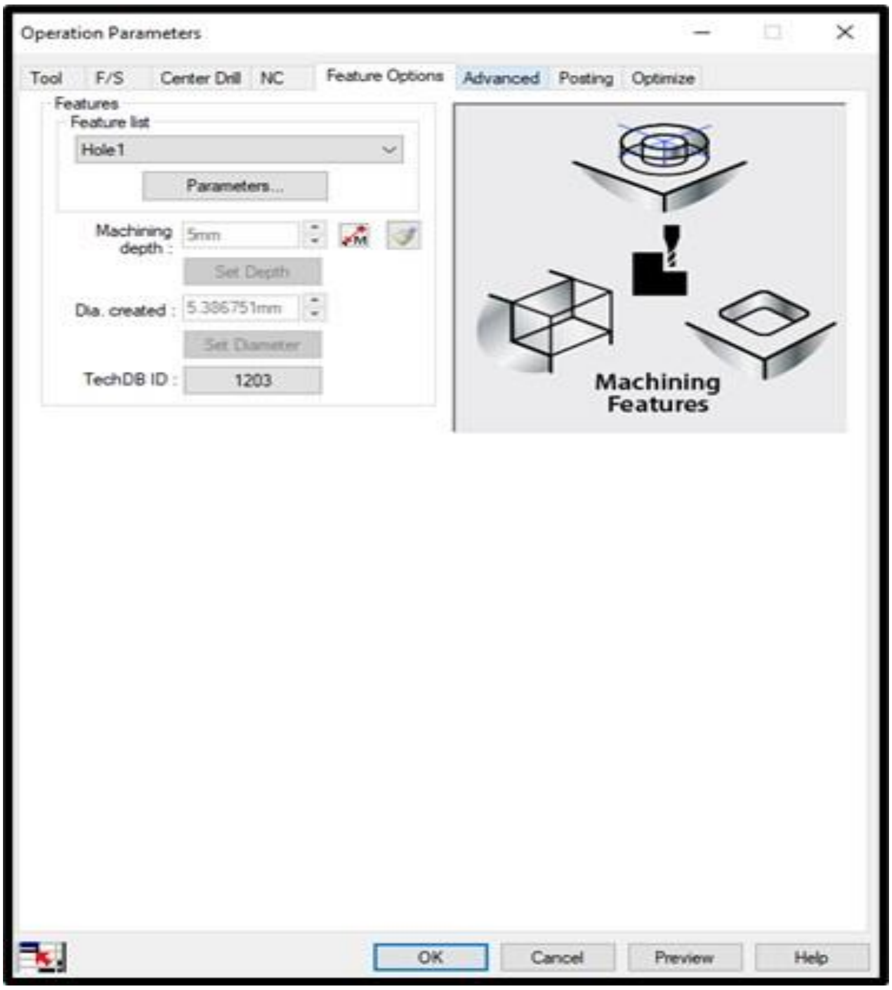


CENTER DRILL

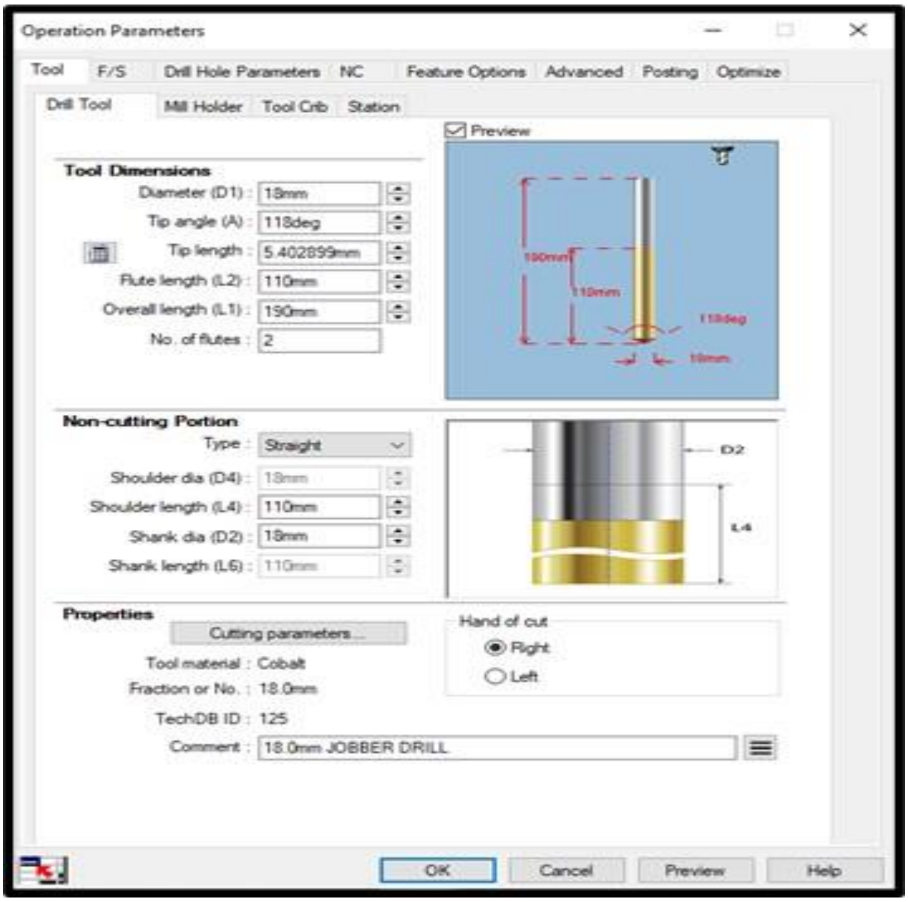


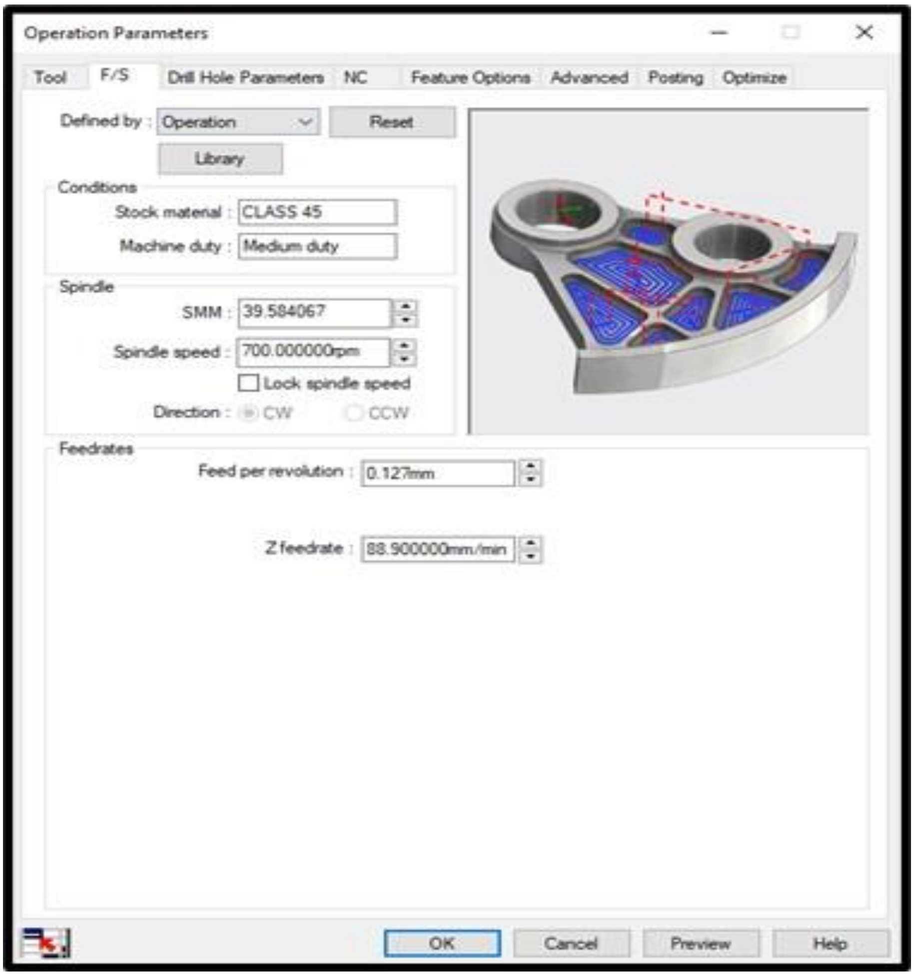

Operation Parameters
Tool
F/S
Center Drill
NC
Feature Options
Advanced
Posting
Optimize
Defined by : Operation
Reset
Library
Conditions
Stock material : CLASS 45
Machine duty : Medium duty
Spindle
SMM : 19.634954
Spindle speed : 2500.000000rpm
Lock spindle speed
Direction : CW CCW
Feedrates
Feed per revolution : 0.1016mm
Z feedrate : 254.000000mm/min
OK
Cancel
Preview
Help

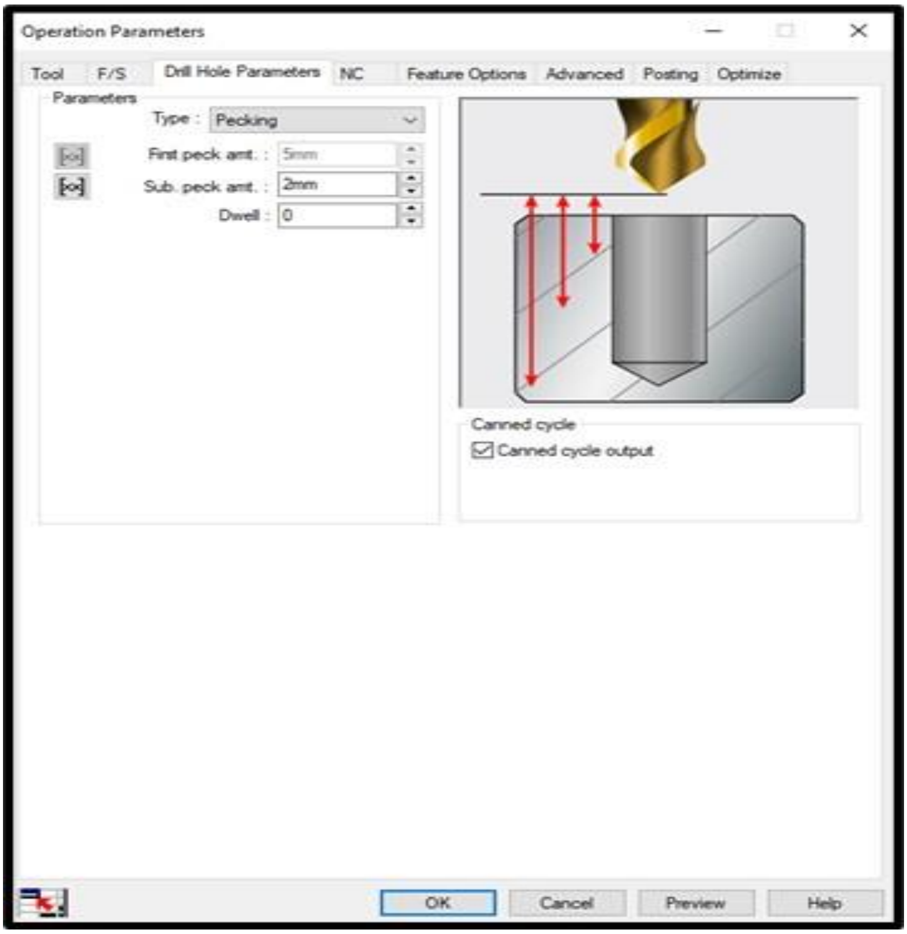

Operation Parameters
Tool
F/S
Center Drill
NC
Feature Options
Advanced
Posting
Optimize
Parameters
Type : Spot Drilling
First peck amt. : 5mm
Sub. peck amt. : 2.5mm
Dwell : 1
Canned cycle
Canned cycle output
OK
Cancel
Preview
Help

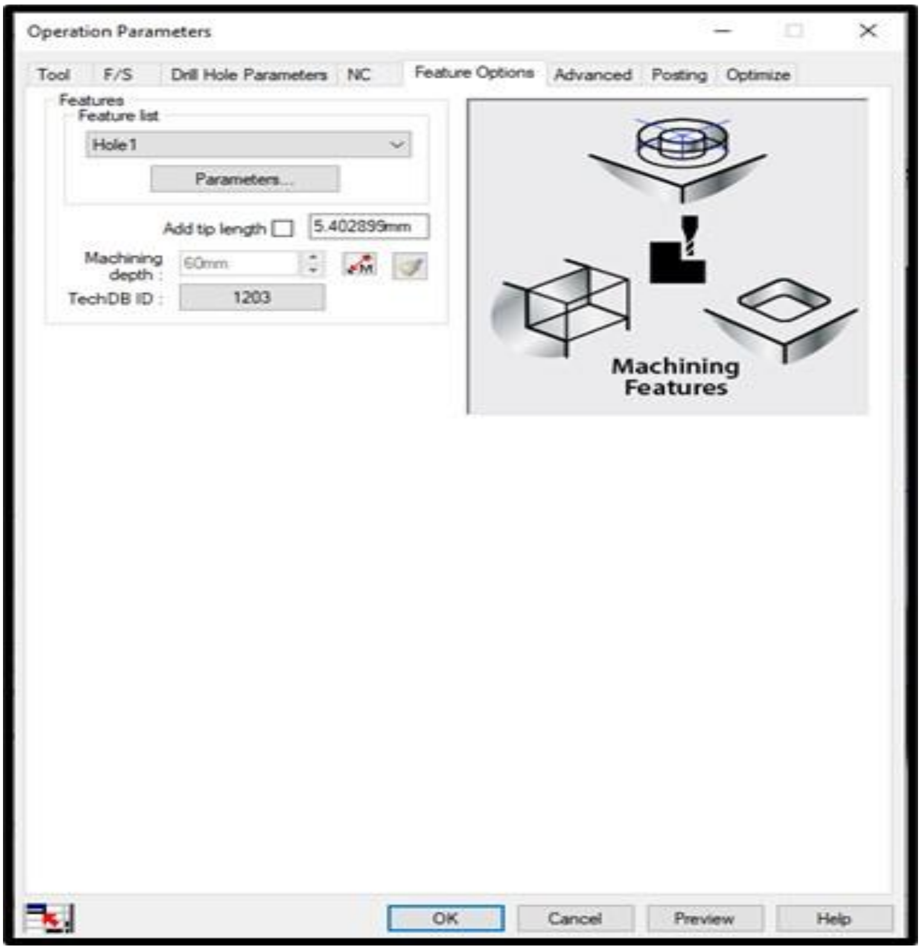


DRILL OPERATION:

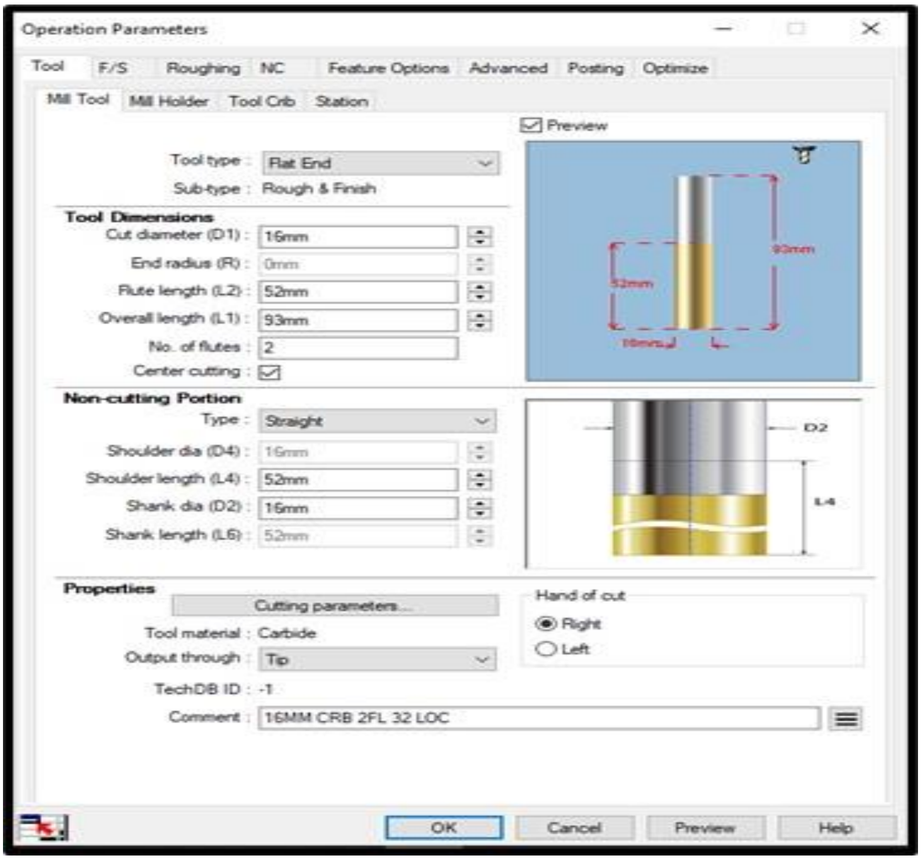


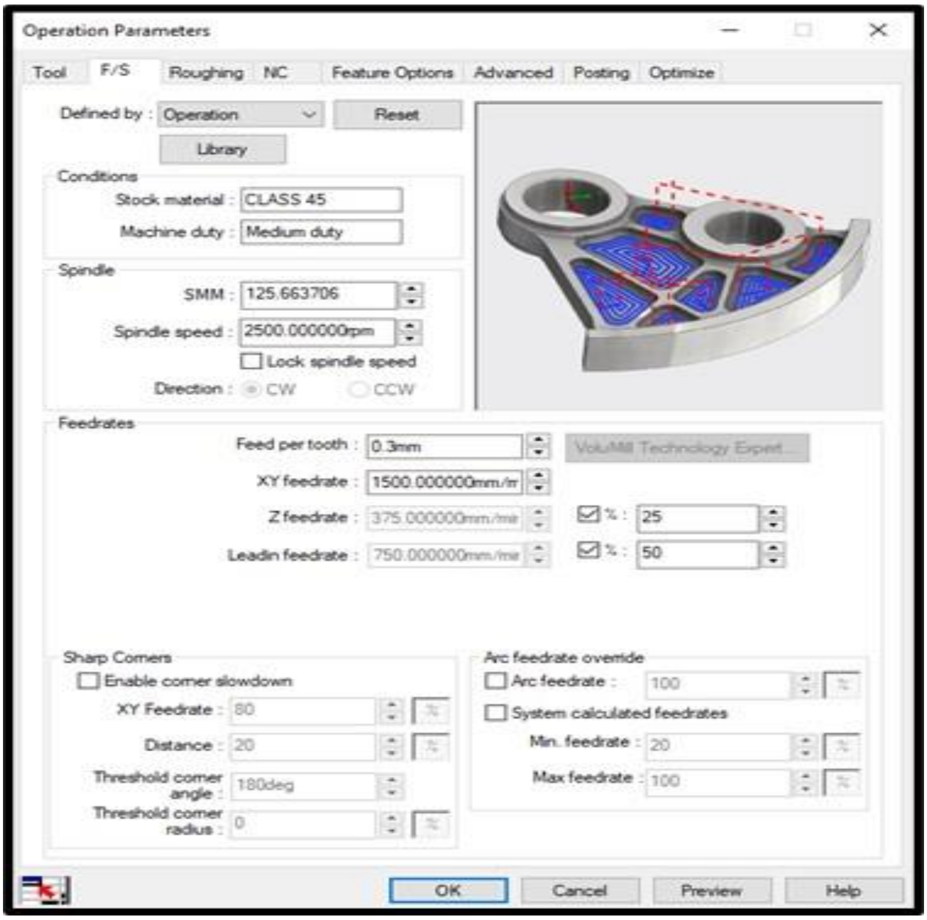

Operation Parameters
Tool
F/S
Drill Hole Parameters
NC
Feature Options
Advanced
Posting
Optimize
Defined by : Operation
Reset
Library
Conditions
Stock material : CLASS 45
Machine duty : Medium duty
Spindle
SMM : 39.584067
Spindle speed : 700.000000rpm
Lock spindle speed
Direction : CW CCW
Feedrates
Feed per revolution : 0.127mm
Z feedrate : 88.900000mm/min
OK
Cancel
Preview
Help

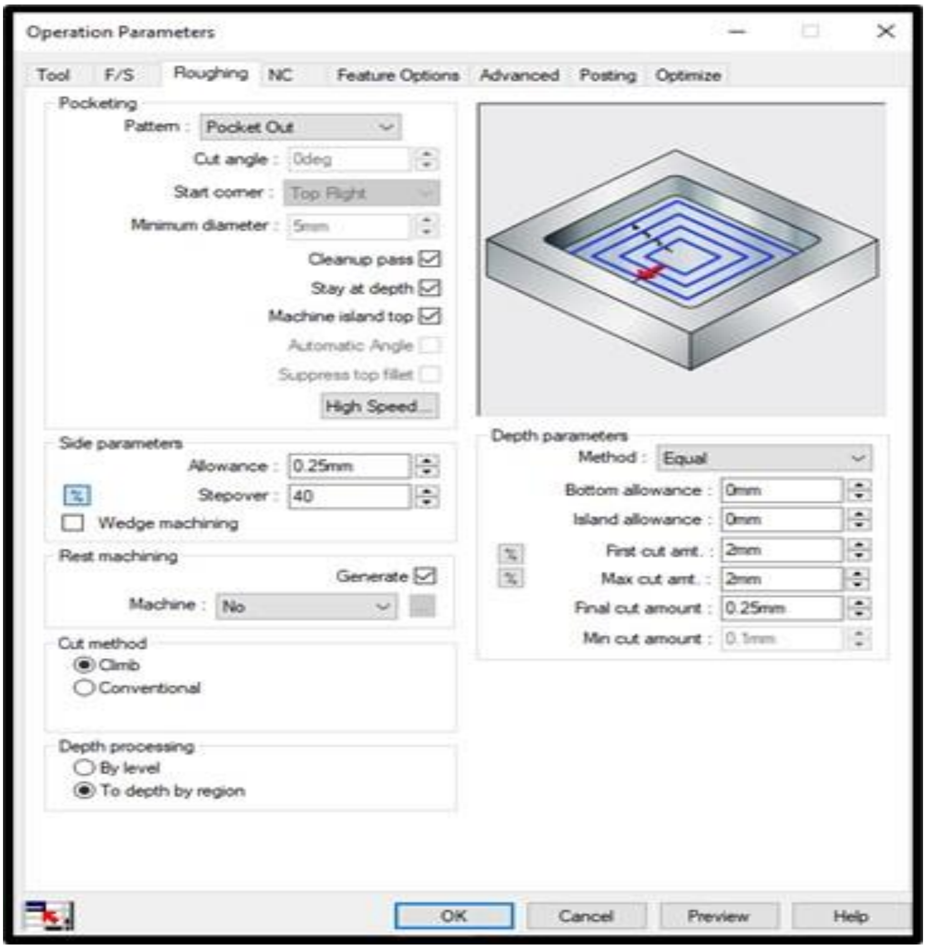

Operation Parameters
Tool
F/S
Drill Hole Parameters
NC
Feature Options
Advanced
Posting
Optimize
Parameters
Type : Pecking
First peck amt. : 5mm
Sub. peck amt. : 2mm
Dwell : 0
Canned cycle
Canned cycle output
OK
Cancel
Preview
Help

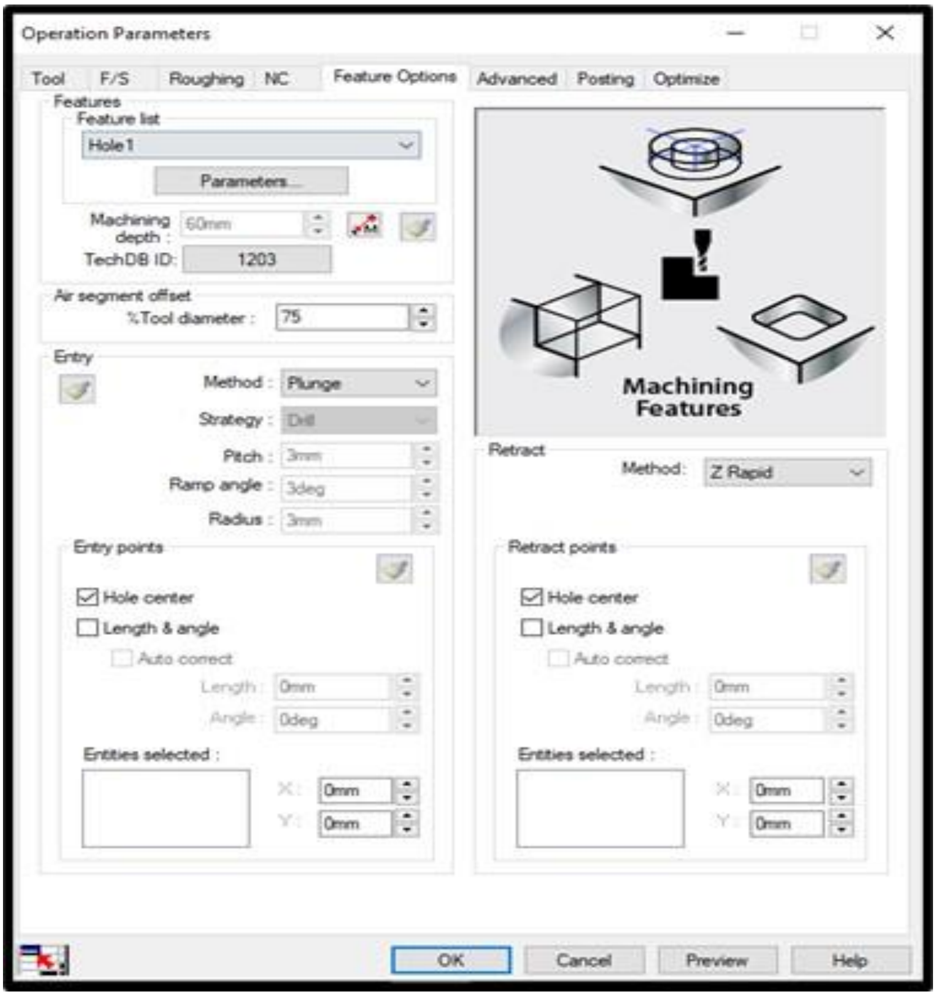

Operation Parameters
Tool
F/S
Drill Hole Parameters
NC
Feature Options
Advanced
Posting
Optimize
Features
Feature list
Hole1
Parameters...
Add tip length
5.402899mm
Machining depth :
60mm
TechDB ID :
1203
Machining Features
OK
Cancel
Preview
Help

ROUGH MILL

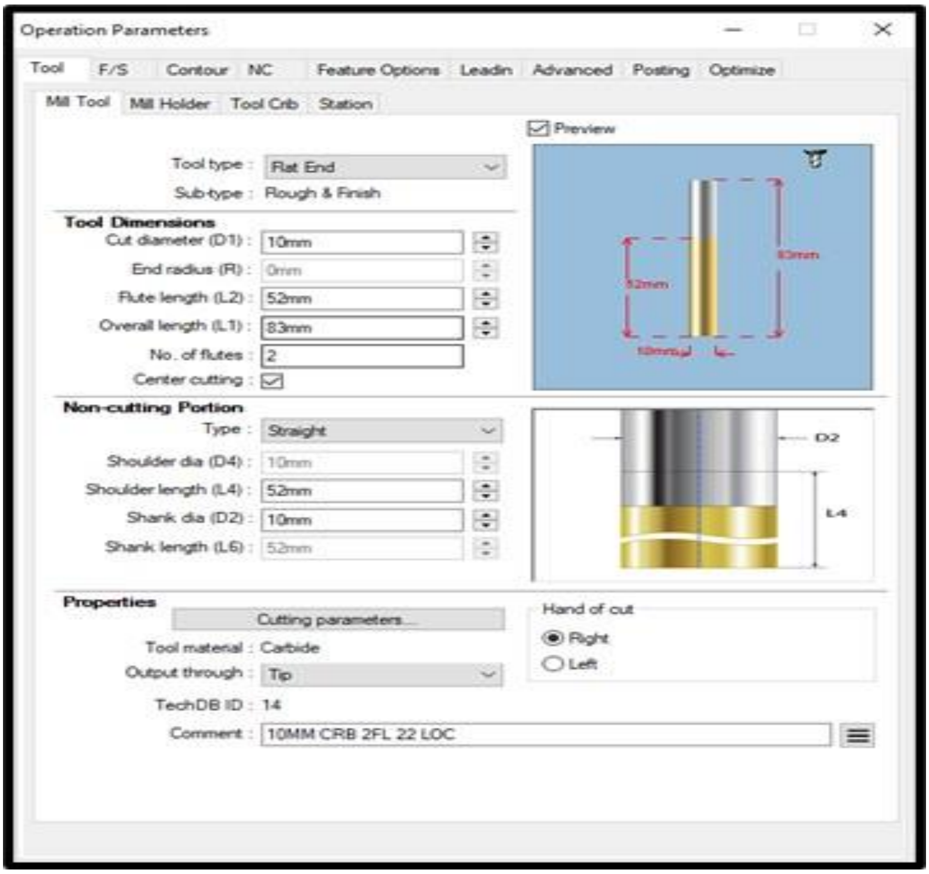

Operation Parameters
Tool
F/S
Roughing
NC
Feature Options
Advanced
Posting
Optimize
Mill Tool
Mill Holder
Tool Crib
Station
Preview
Tool type : Flat End
Sub-type : Rough & Finish
Tool Dimensions
Cut diameter (D1) : 16mm
End radius (R) : 0mm
Flute length (L2) : 52mm
Overall length (L1) : 93mm
No. of flutes : 2
Center cutting :
Non-cutting Portion
Type : Straight
Shoulder dia (D4) : 16mm
Shoulder length (L4) : 52mm
Shank dia (D2) : 16mm
Shank length (L6) : 52mm
D2
L4
Properties
Cutting parameters...
Tool material : Carbide
Output through : Tip
TechDB ID : -1
Comment : 16MM CRB 2FL 32 LOC
Hand of cut
Right
Left
OK
Cancel
Preview
Help

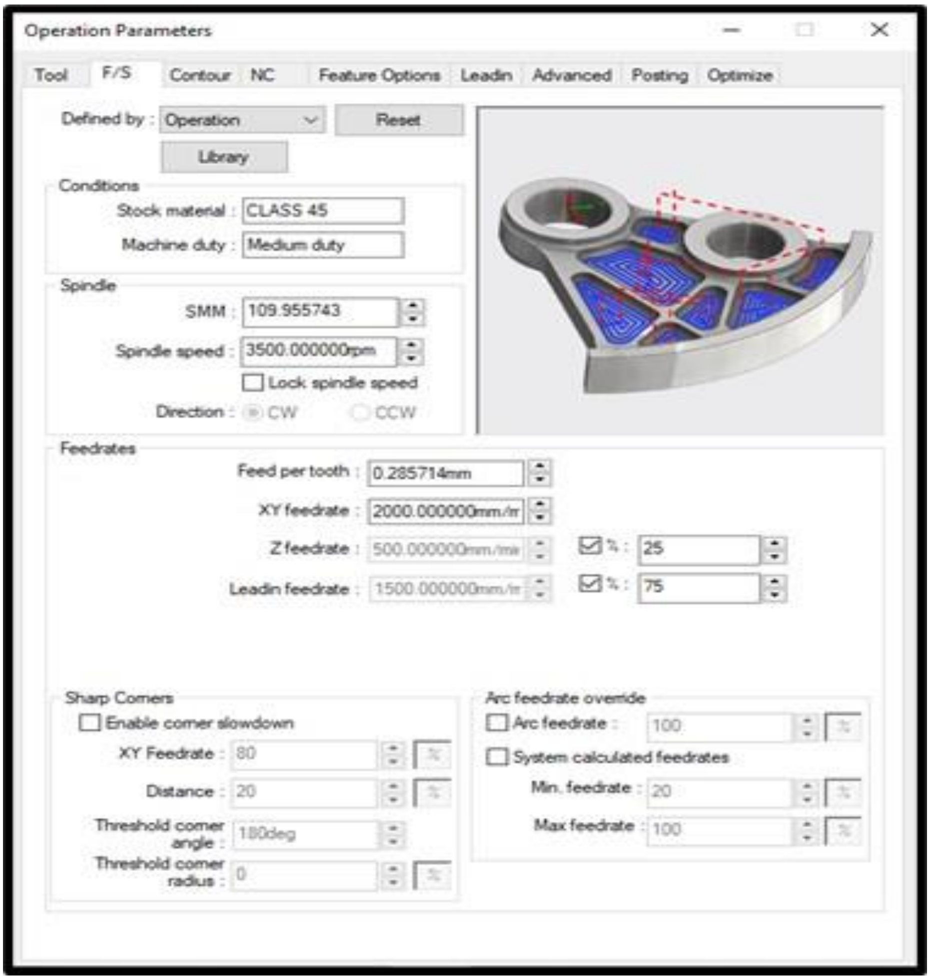

Operation Parameters
Tool
F/S
Roughing
NC
Feature Options
Advanced
Posting
Optimize
Defined by : Operation
Reset
Library
Conditions
Stock material : CLASS 45
Machine duty : Medium duty
Spindle
SMM : 125.663706
Spindle speed : 2500.000000rpm
Lock spindle speed
Direction : CW CCW
Feedrates
Feed per tooth : 0.3mm
XY feedrate : 1500.000000mm/m
Z feedrate : 375.000000mm/mi
Leadin feedrate : 750.000000mm/mi
% : 25
% : 50
Sharp Corners
Enable corner slowdown
XY Feedrate : 80
Distance : 20
Threshold corner angle : 180deg
Threshold corner radius : 0
Arc feedrate override
Arc feedrate : 100
System calculated feedrates
Min. feedrate : 20
Max feedrate : 100
OK
Cancel
Preview
Help

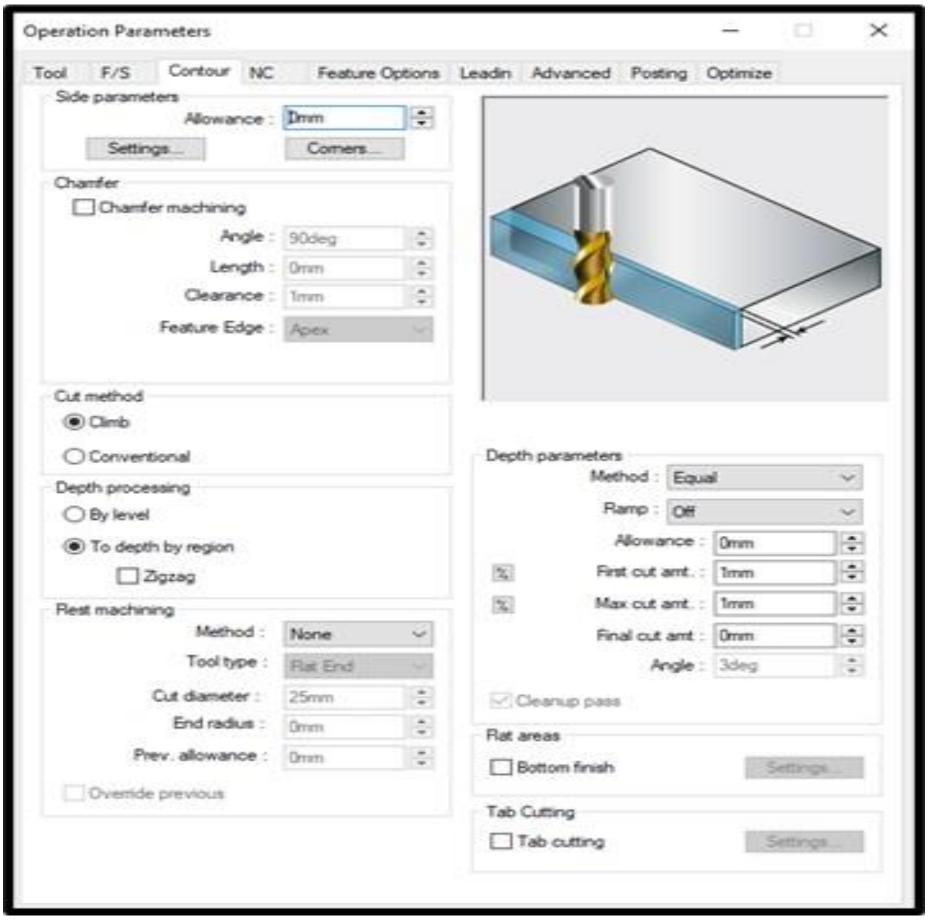

Operation Parameters
Tool
F/S
Roughing
NC
Feature Options
Advanced
Posting
Optimize
Pocketing
Pattern : Pocket Out
Cut angle : 0deg
Start corner : Top Right
Minimum diameter : 5mm
Cleanup pass
Stay at depth
Machine island top
Automatic Angle
Suppress top fillet
High Speed...
Side parameters
Allowance : 0.25mm
Stepover : 40
Wedge machining
Rest machining
Generate
Machine : No
Cut method
Climb
Conventional
Depth processing
By level
To depth by region
Depth parameters
Method : Equal
Bottom allowance : 0mm
Island allowance : 0mm
First cut amt. : 2mm
Max cut amt. : 2mm
Final cut amount : 0.25mm
Min cut amount : 0.1mm
OK
Cancel
Preview
Help

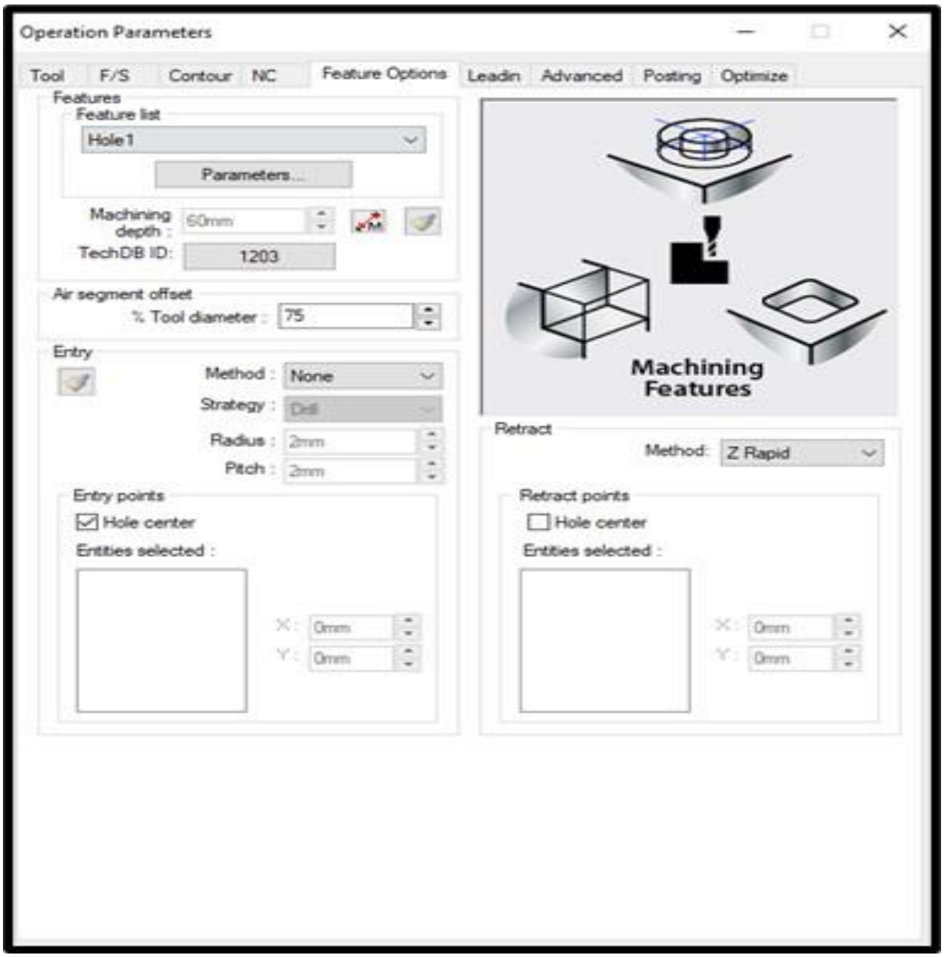


CONTOUR MILL:

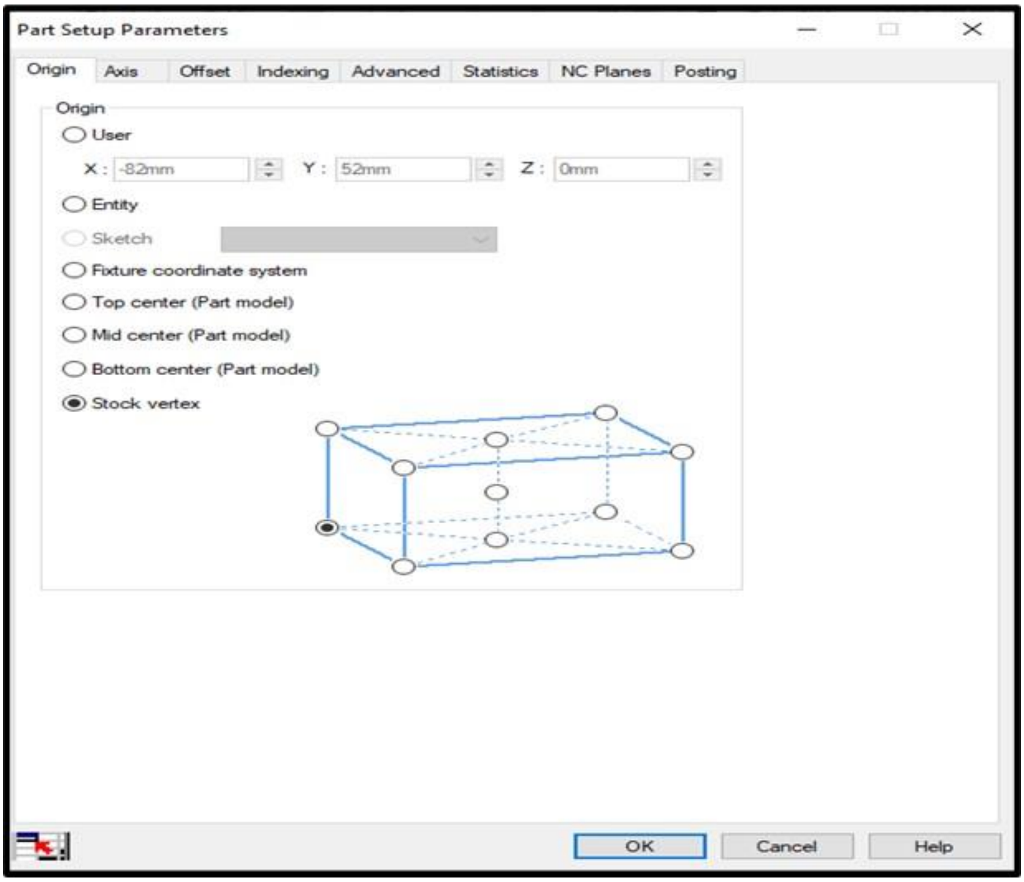


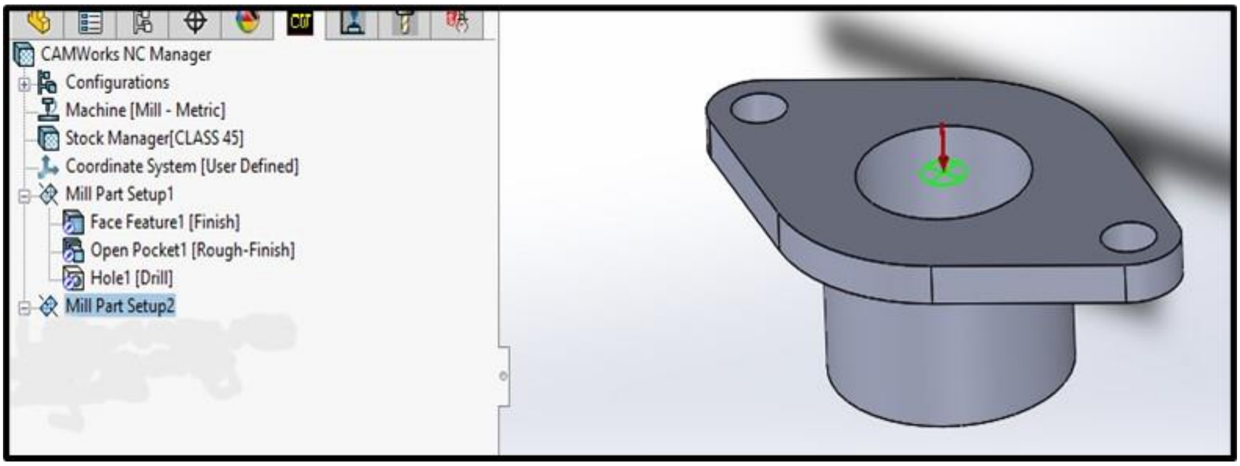

Operation Parameters
Tool
F/S
Contour
NC
Feature Options
Leadin
Advanced
Posting
Optimize
Defined by : Operation
Reset
Library
Conditions
Stock material : CLASS 45
Machine duty : Medium duty
Spindle
SMM : 109.955743
Spindle speed : 3500.000000rpm
Lock spindle speed
Direction : CW CCW
Feedrates
Feed per tooth : 0.285714mm
XY feedrate : 2000.000000mm/m
Z feedrate : 500.000000mm/mi
% : 25
Leadin feedrate : 1500.000000mm/m
% : 75
Sharp Corners
Enable corner slowdown
XY Feedrate : 80
Distance : 20
Threshold corner angle : 180deg
Threshold corner radius : 0
Arc feedrate override
Arc feedrate : 100
System calculated feedrates
Min. feedrate : 20
Max feedrate : 100


Operation Parameters
Tool
F/S
Contour
NC
Feature Options
Leadin
Advanced
Posting
Optimize
Side parameters
Allowance : 0mm
Settings...
Corners...
Chamfer
Chamfer machining
Angle : 90deg
Length : 0mm
Clearance : 1mm
Feature Edge : Apex
Cut method
Climb
Conventional
Depth processing
By level
To depth by region
Zigzag
Rest machining
Method : None
Tool type : Flat End
Cut diameter : 25mm
End radius : 0mm
Prev. allowance : 0mm
Override previous
Depth parameters
Method : Equal
Ramp : Off
Allowance : 0mm
First cut amt. : 1mm
Max cut amt. : 1mm
Final cut amt : 0mm
Angle : 3deg
Cleanup pass
Flat areas
Bottom finish
Settings...
Tab Cutting
Tab cutting
Settings...



7.3 second mill part setup & coordinate system:

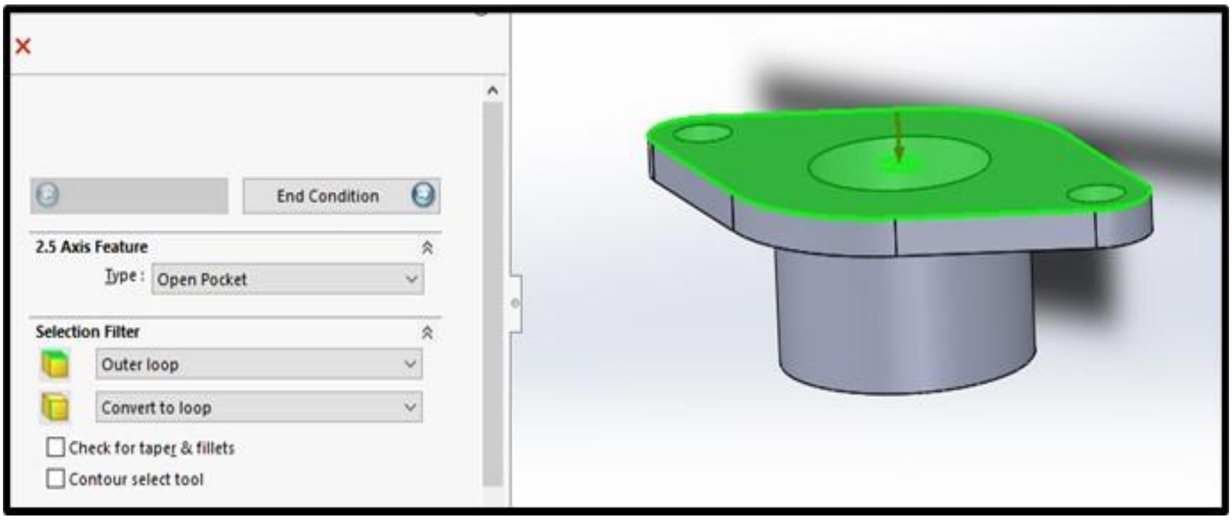




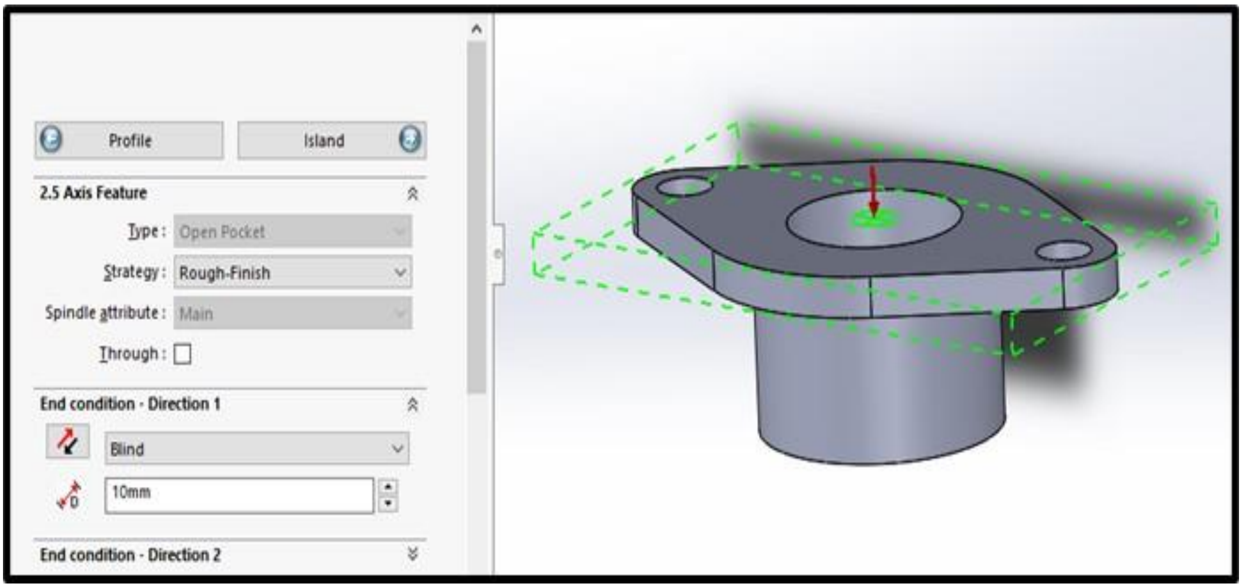
7.4 Mill operations:

7.4.1 Mill pocket

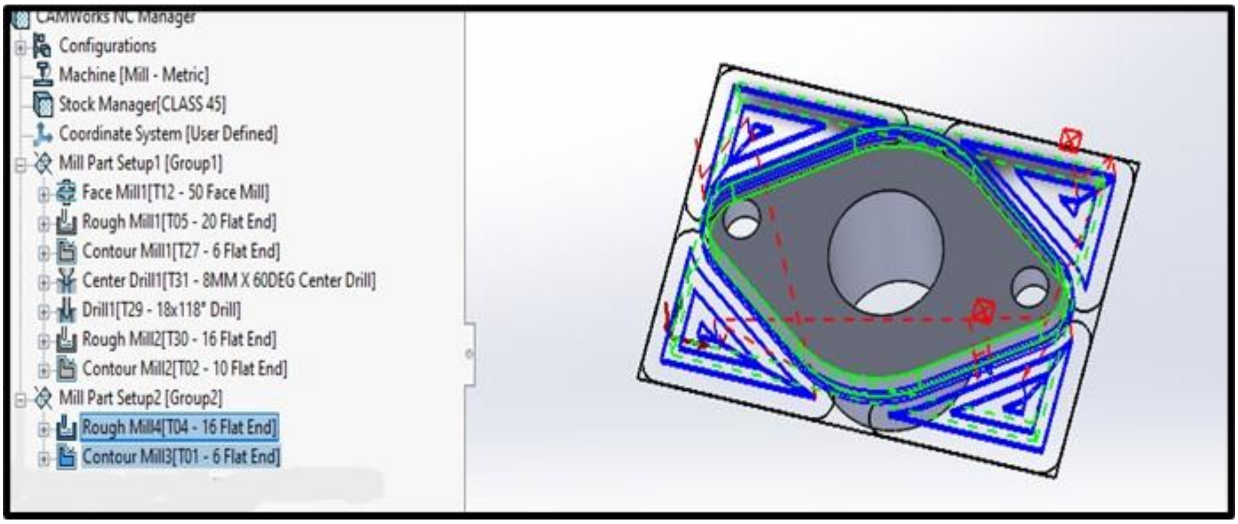
Type: open pocket



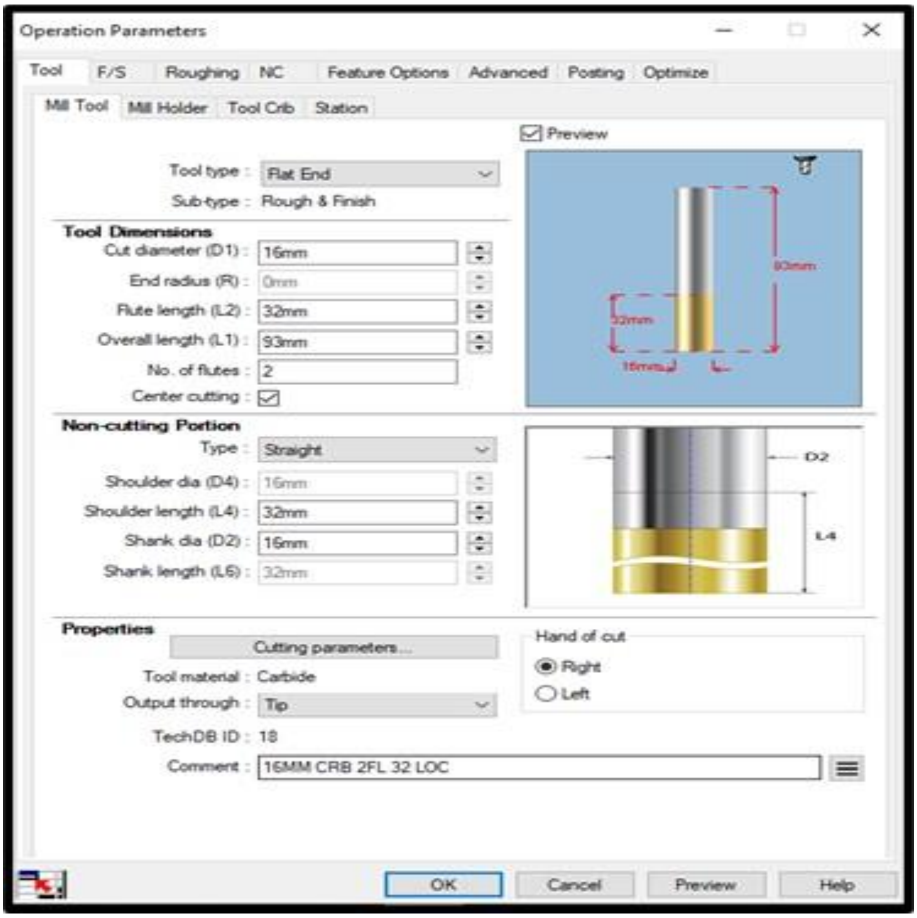
Strategy: rough-finish

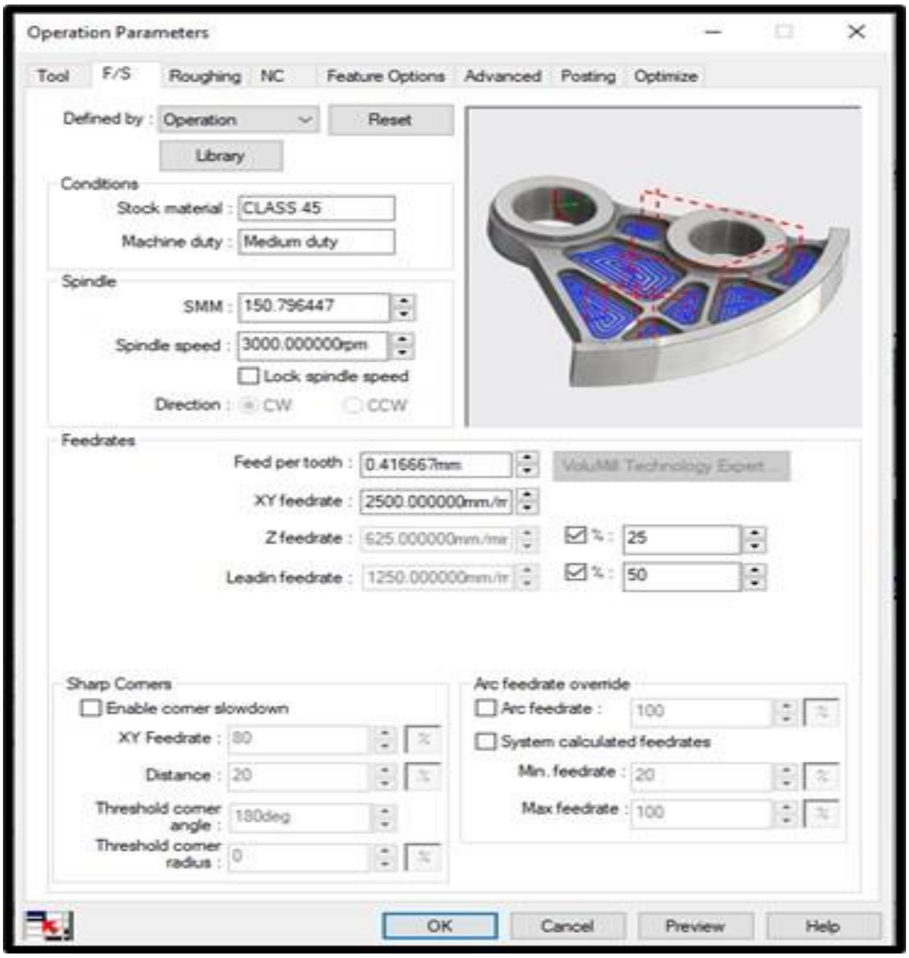


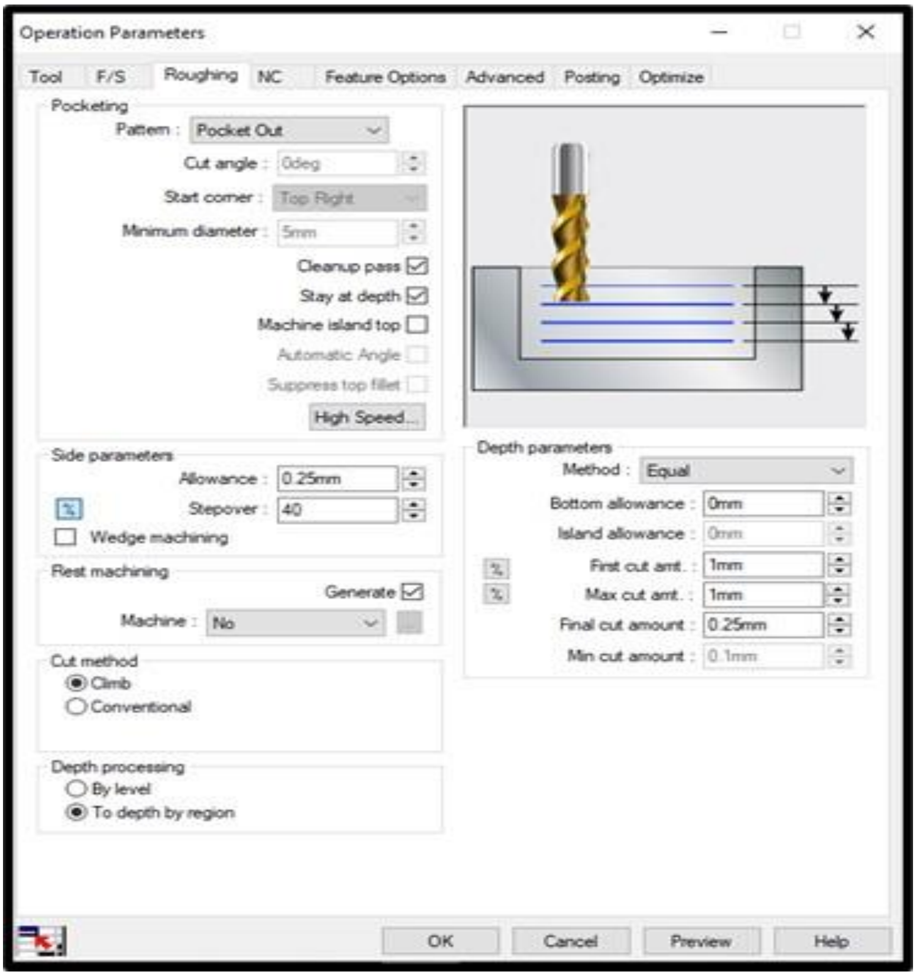
Generate operation plan and tool path.

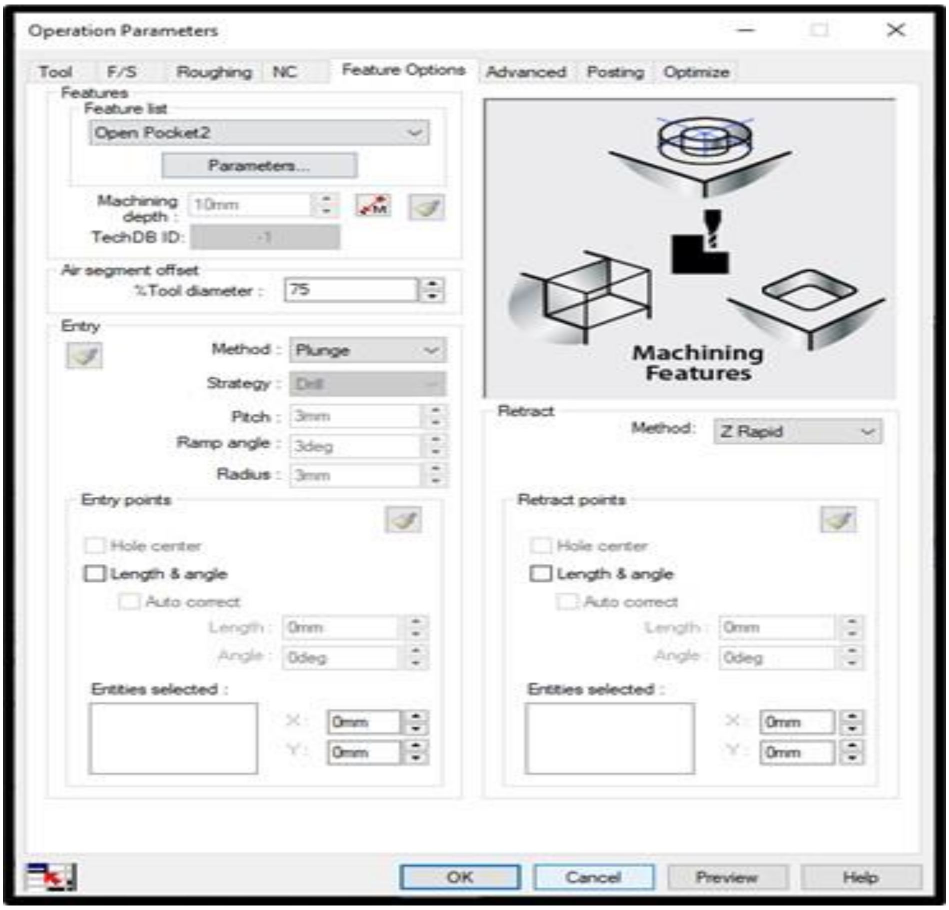


ROUGH-MILL:

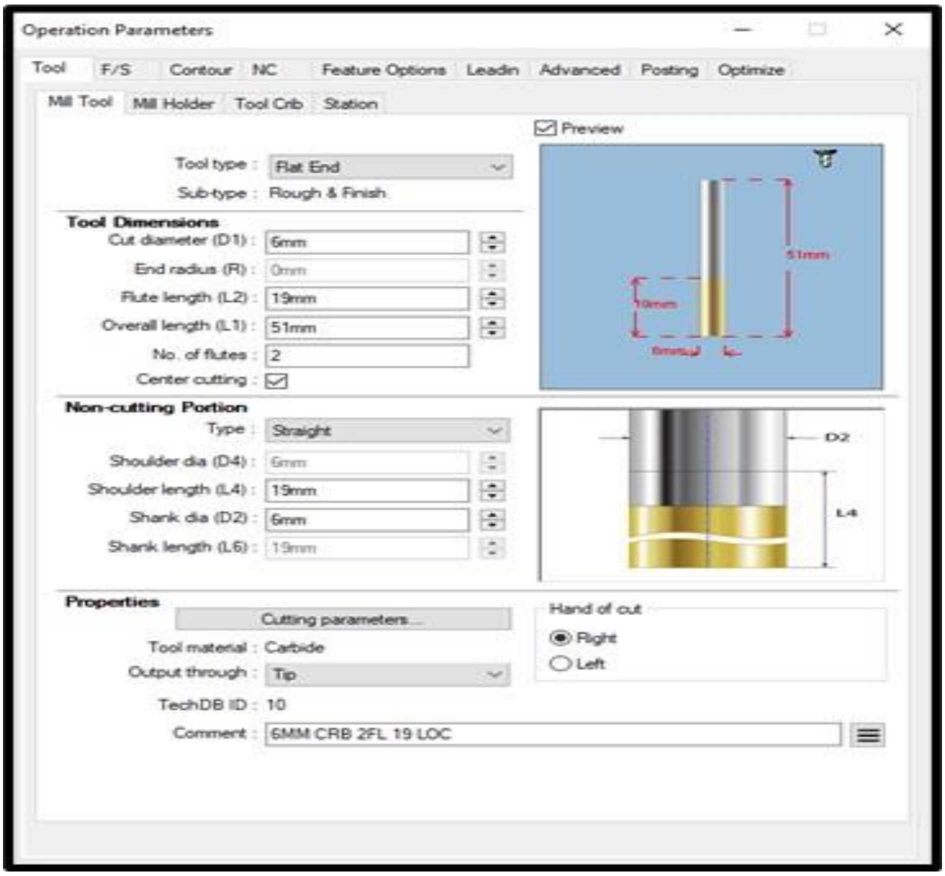


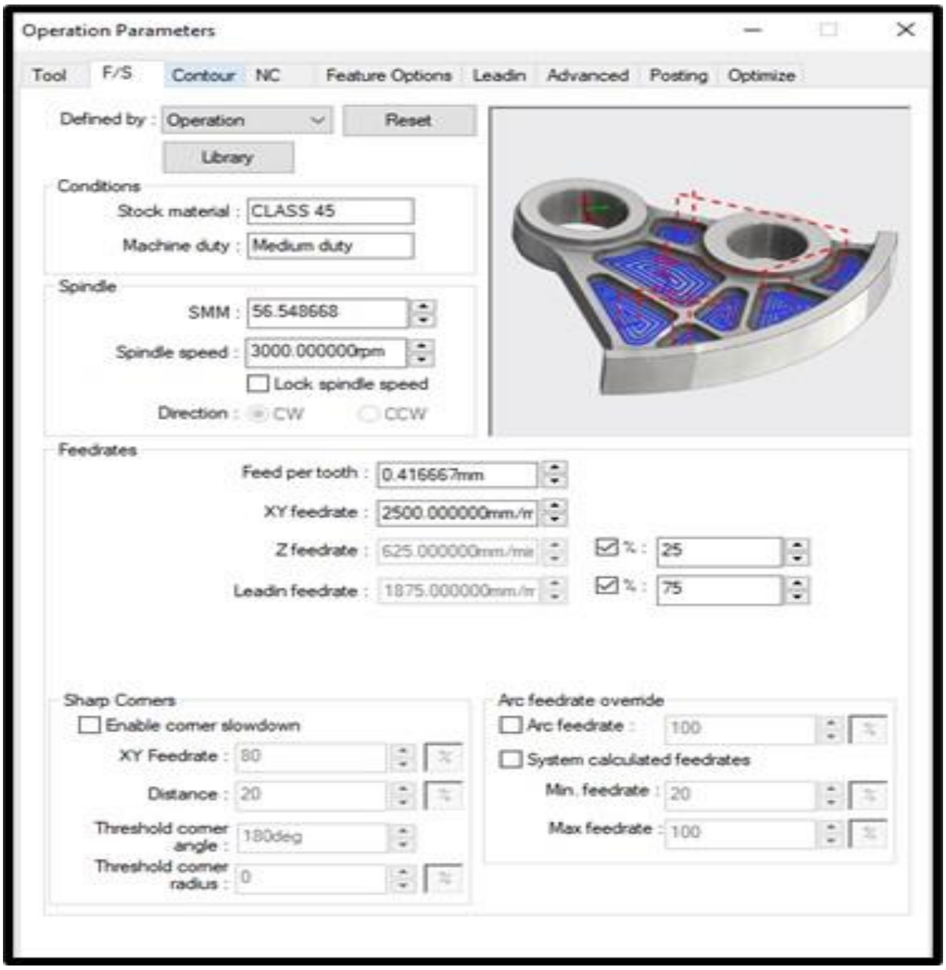

Operation Parameters
Tool
F/S
Roughing
NC
Feature Options
Advanced
Posting
Optimize
Defined by : Operation
Reset
Library
Conditions
Stock material : CLASS 45
Machine duty : Medium duty
Spindle
SMM : 150.796447
Spindle speed : 3000.000000rpm
Lock spindle speed
Direction : CW CCW
Feedrates
Feed per tooth : 0.416667mm
XY feedrate : 2500.000000mm/m
Z feedrate : 625.000000mm/mi
% : 25
Leadin feedrate : 1250.000000mm/m
% : 50
Sharp Corners
Enable corner slowdown
XY Feedrate : 80
Distance : 20
Threshold corner angle : 180deg
Threshold corner radius : 0
Arc feedrate override
Arc feedrate : 100
System calculated feedrates
Min. feedrate : 20
Max feedrate : 100
OK
Cancel
Preview
Help

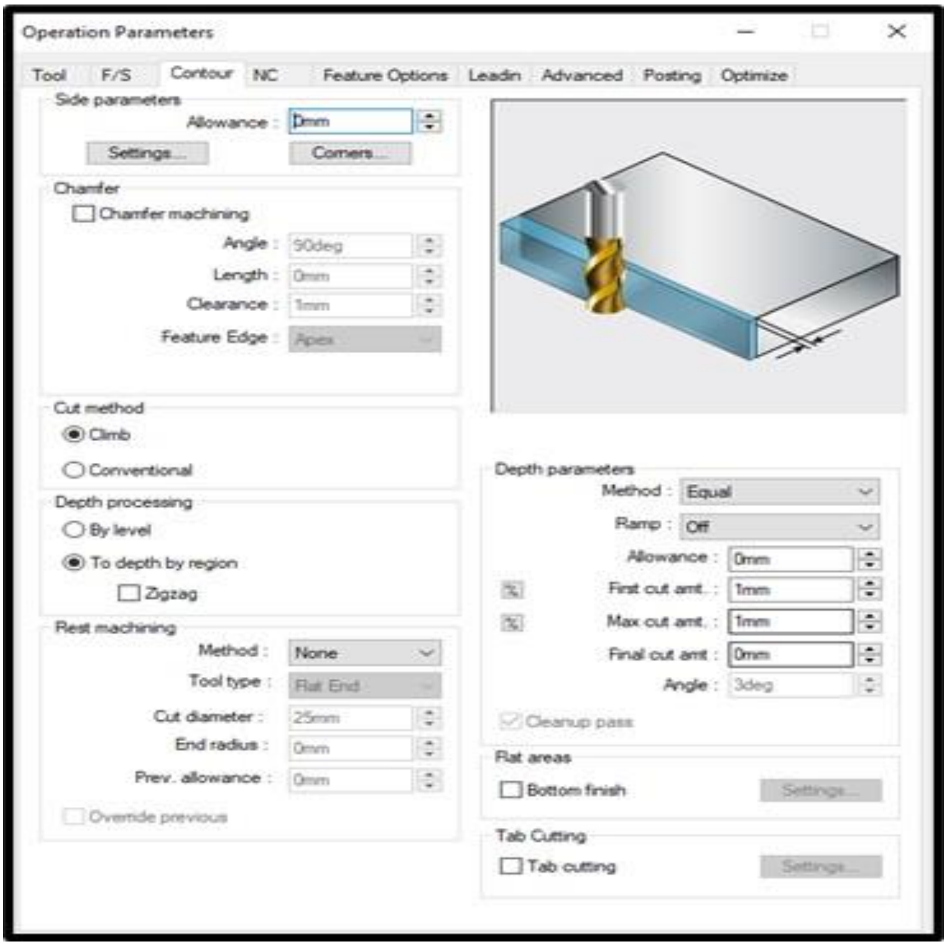

Operation Parameters
Tool
F/S
Roughing
NC
Feature Options
Advanced
Posting
Optimize
Pocketing
Pattern : Pocket Out
Cut angle : 0deg
Start corner : Top Right
Minimum diameter : 5mm
Cleanup pass
Stay at depth
Machine island top
Automatic Angle
Suppress top fillet
High Speed...
Side parameters
Allowance : 0.25mm
Stepover : 40
Wedge machining
Rest machining
Generate
Machine : No
Cut method
Climb
Conventional
Depth processing
By level
To depth by region
Depth parameters
Method : Equal
Bottom allowance : 0mm
Island allowance : 0mm
First cut amt. : 1mm
Max cut amt. : 1mm
Final cut amount : 0.25mm
Min cut amount : 0.1mm
OK
Cancel
Preview
Help

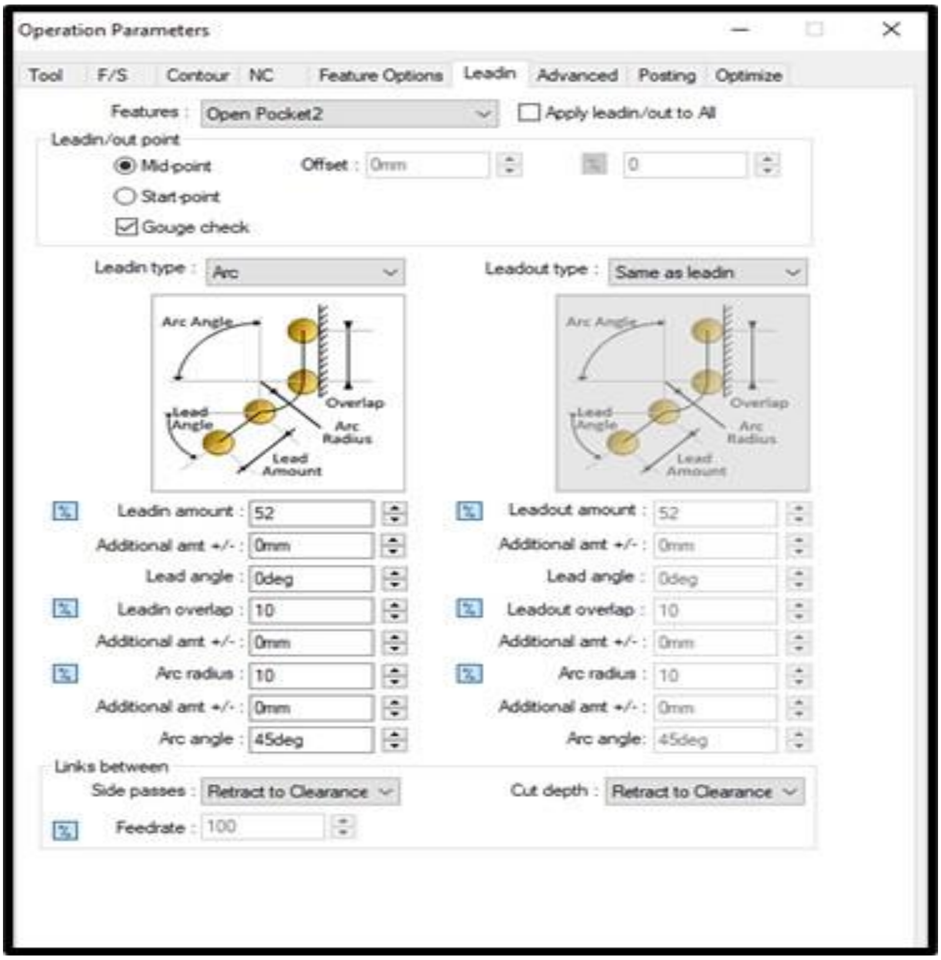


CONTOUR MILL:



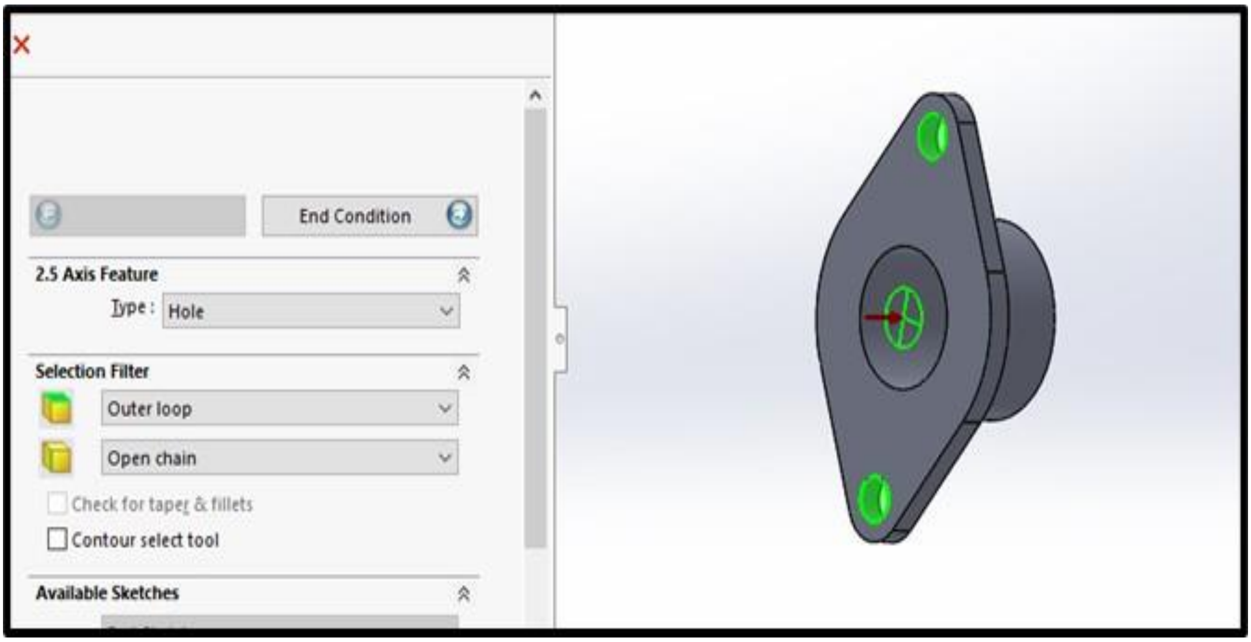

Operation Parameters
Tool
F/S
Contour
NC
Feature Options
Leadin
Advanced
Posting
Optimize
Defined by :
Operation
Reset
Library
Conditions
Stock material :
CLASS 45
Machine duty :
Medium duty
Spindle
SMM :
56.548668
Spindle speed :
3000.000000rpm
Lock spindle speed
Direction :
CW
CCW
Feedrates
Feed per tooth :
0.416667mm
XY feedrate :
Z feedrate :
25
Leadin feedrate :
75
Sharp Corners
Enable corner slowdown
XY Feedrate :
80
Distance :
20
Threshold corner angle :
180deg
Threshold corner radius :
0
Arc feedrate override
Arc feedrate :
100
System calculated feedrates
Min. feedrate :
20
Max feedrate :
100


Operation Parameters
Tool
F/S
Contour
NC
Feature Options
Leadin
Advanced
Posting
Optimize
Side parameters
Allowance :
0mm
Settings...
Corners...
Chamfer
Chamfer machining
Angle :
90deg
Length :
0mm
Clearance :
1mm
Feature Edge :
Apex
Cut method
Climb
Conventional
Depth processing
By level
To depth by region
Zigzag
Rest machining
Method :
None
Tool type :
Flat End
Cut diameter :
25mm
End radius :
0mm
Prev. allowance :
0mm
Override previous
Depth parameters
Method :
Equal
Ramp :
Off
Allowance :
0mm
First cut amt. :
1mm
Max cut amt. :
1mm
Final cut amt :
0mm
Angle :
3deg
Cleanup pass
Flat areas
Bottom finish
Settings...
Tab Cutting
Tab cutting
Settings...

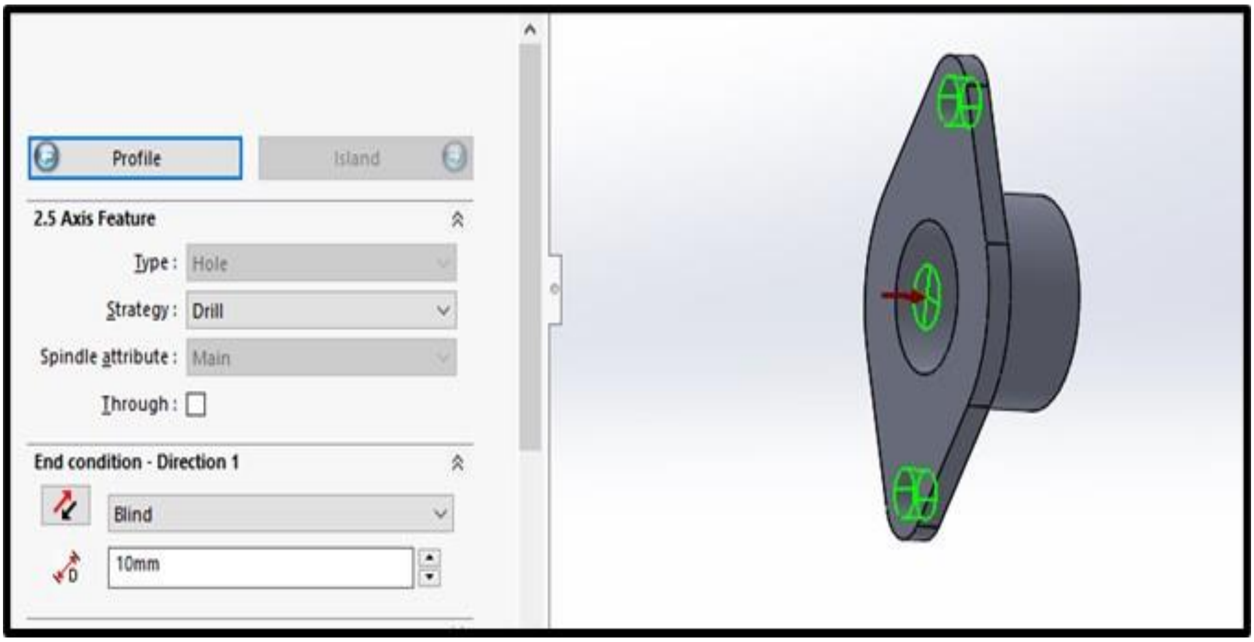


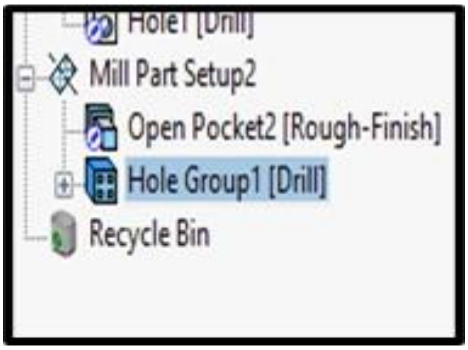
7.4.2 Drill group operation:

Type: hole

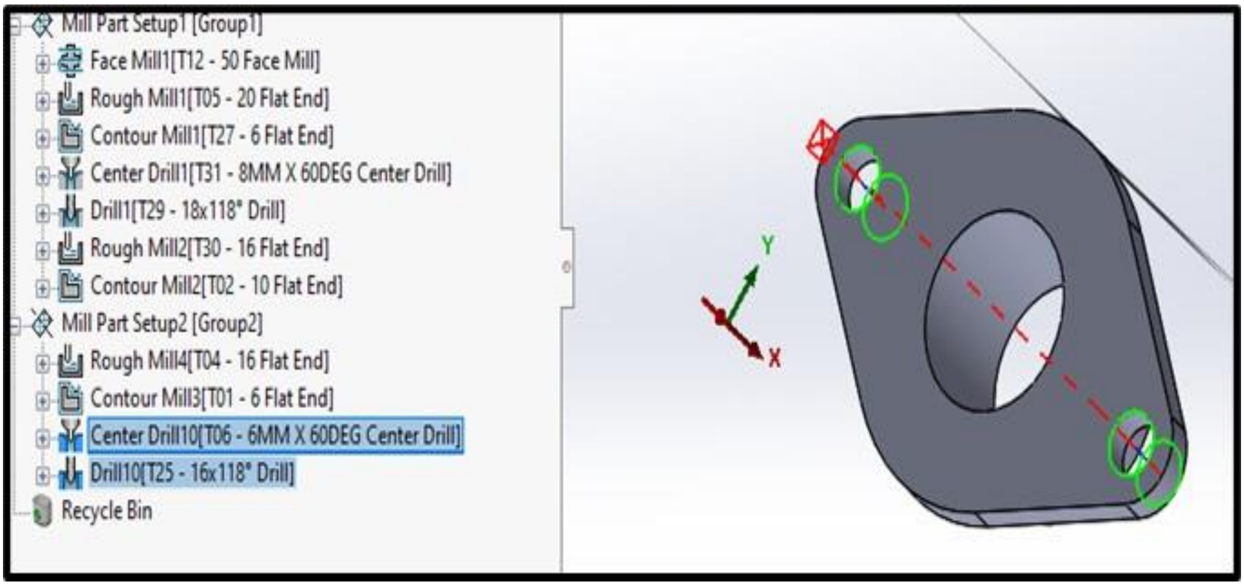


Strategy: drill (10mm)

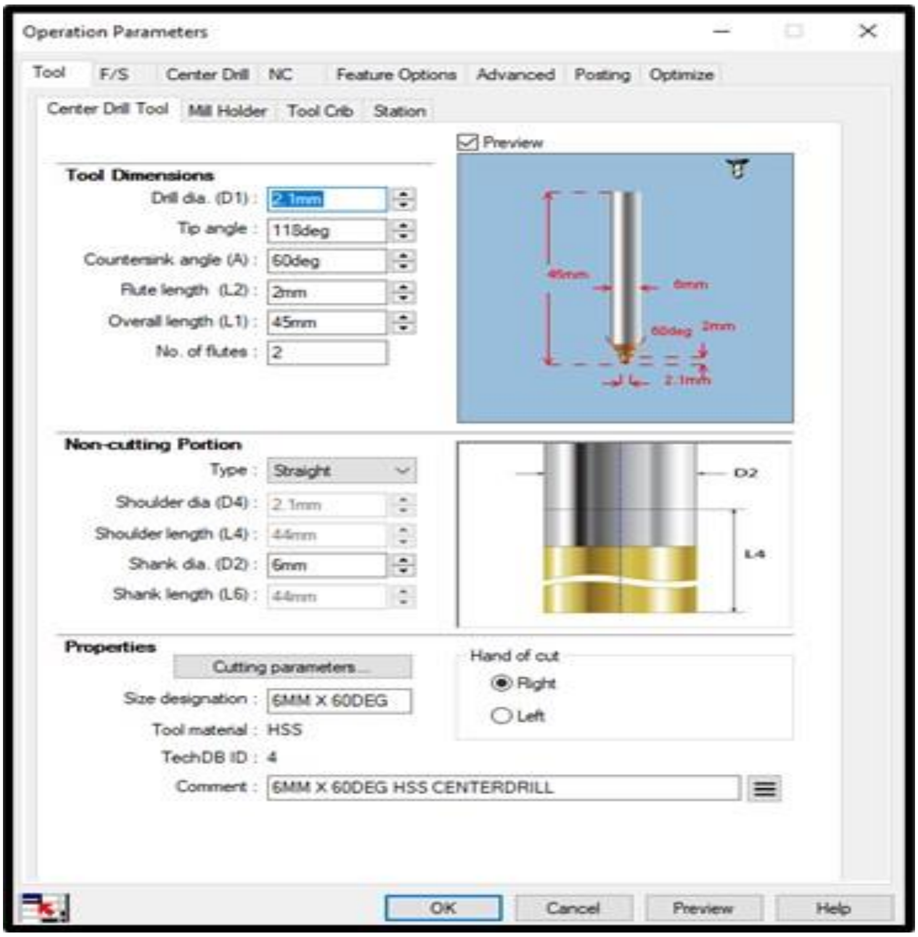


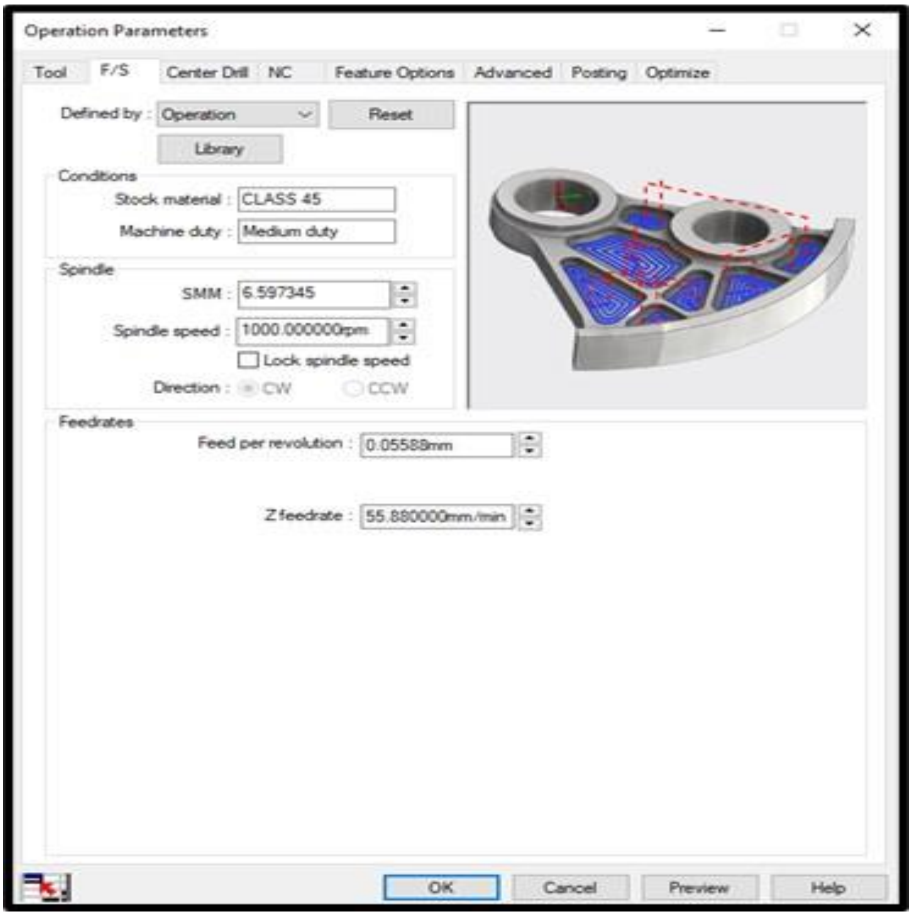


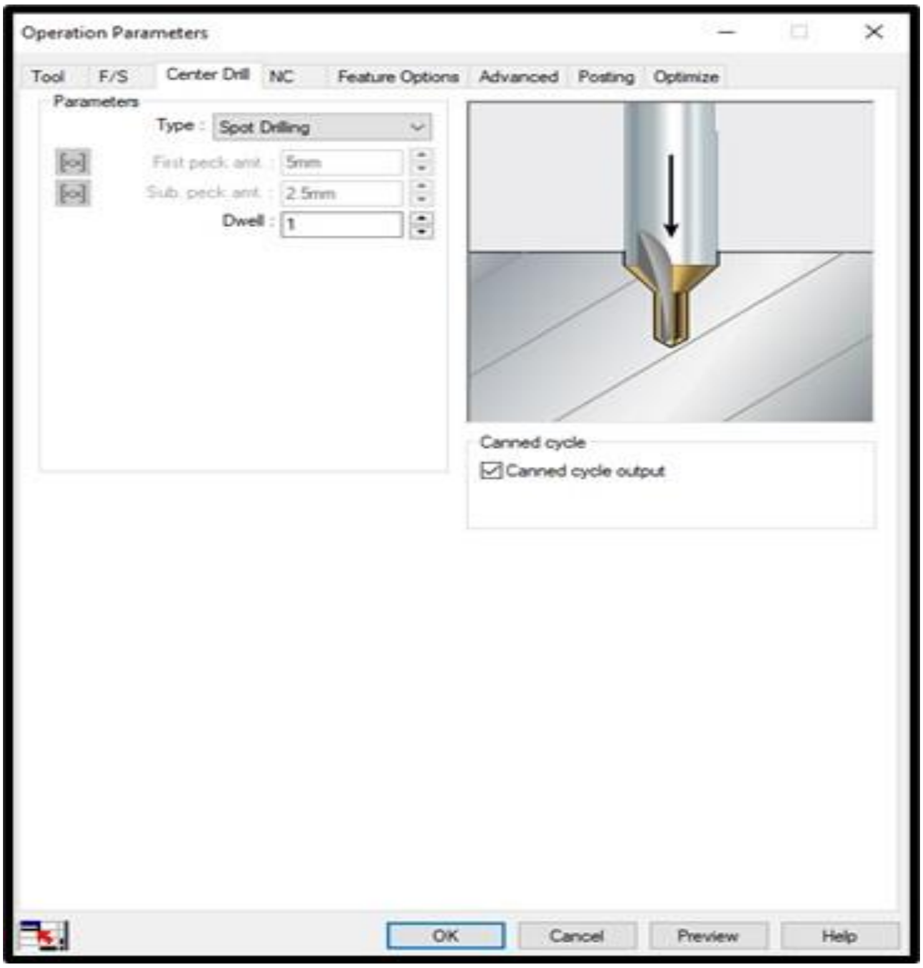
Generate operation plan and tool path.

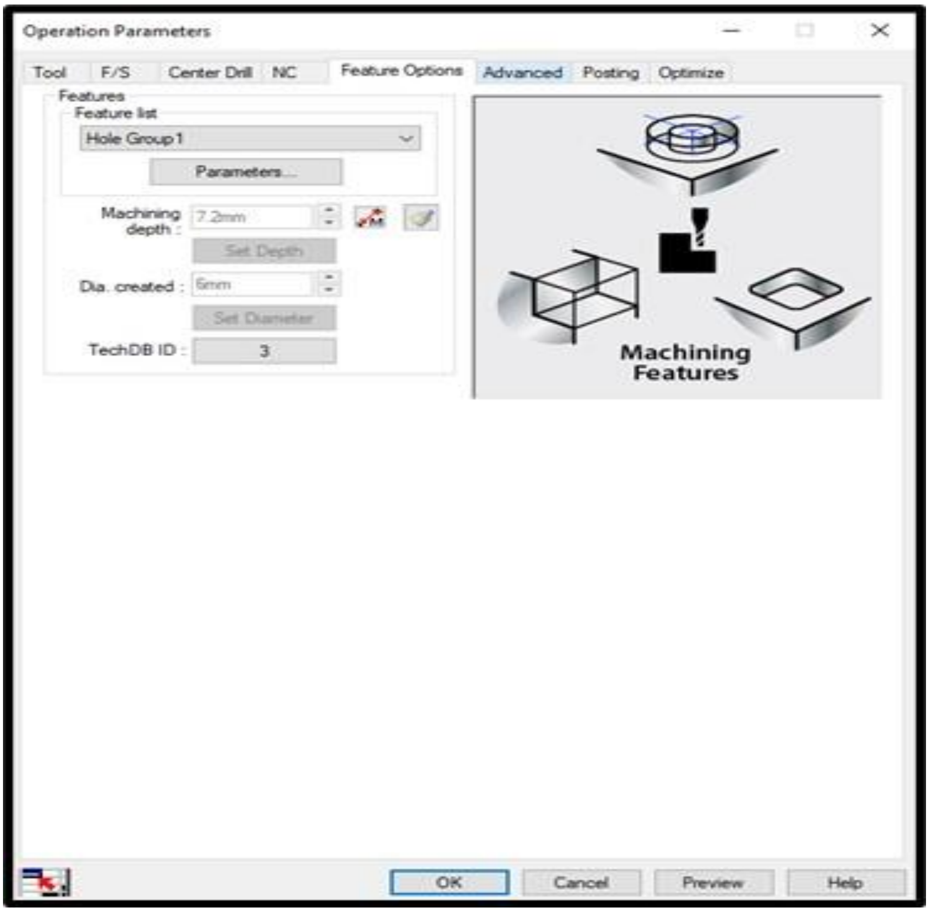


CENTER DRILL:

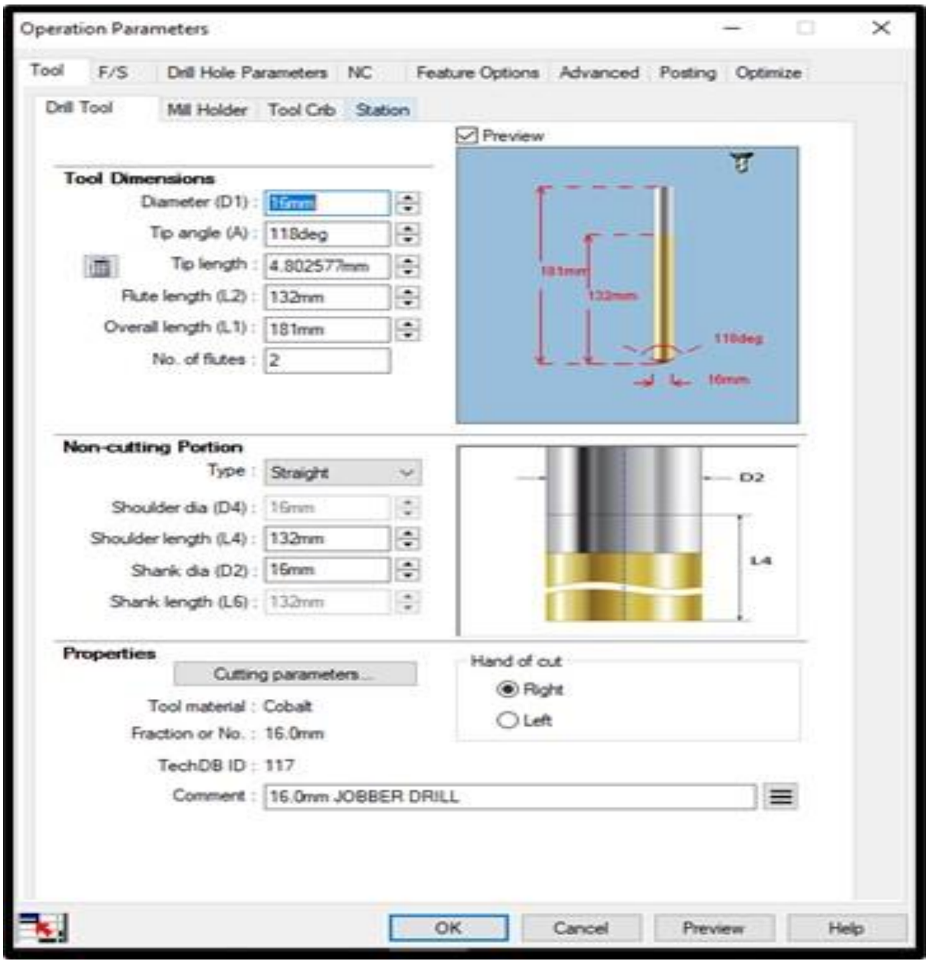


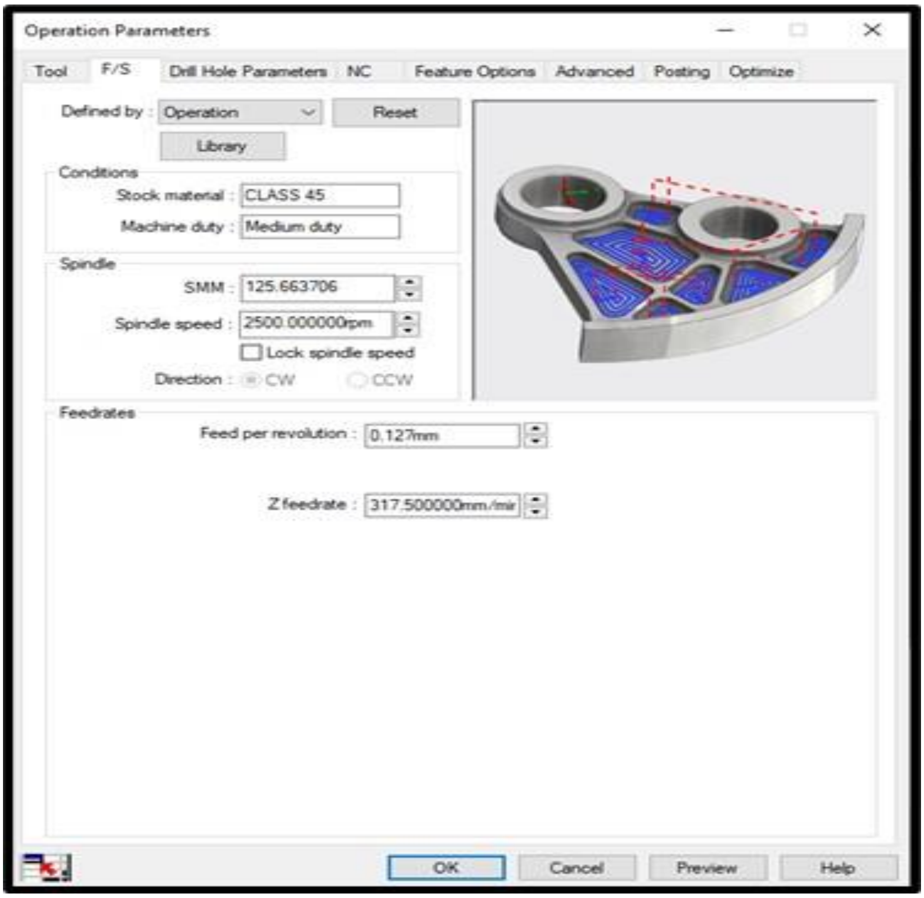


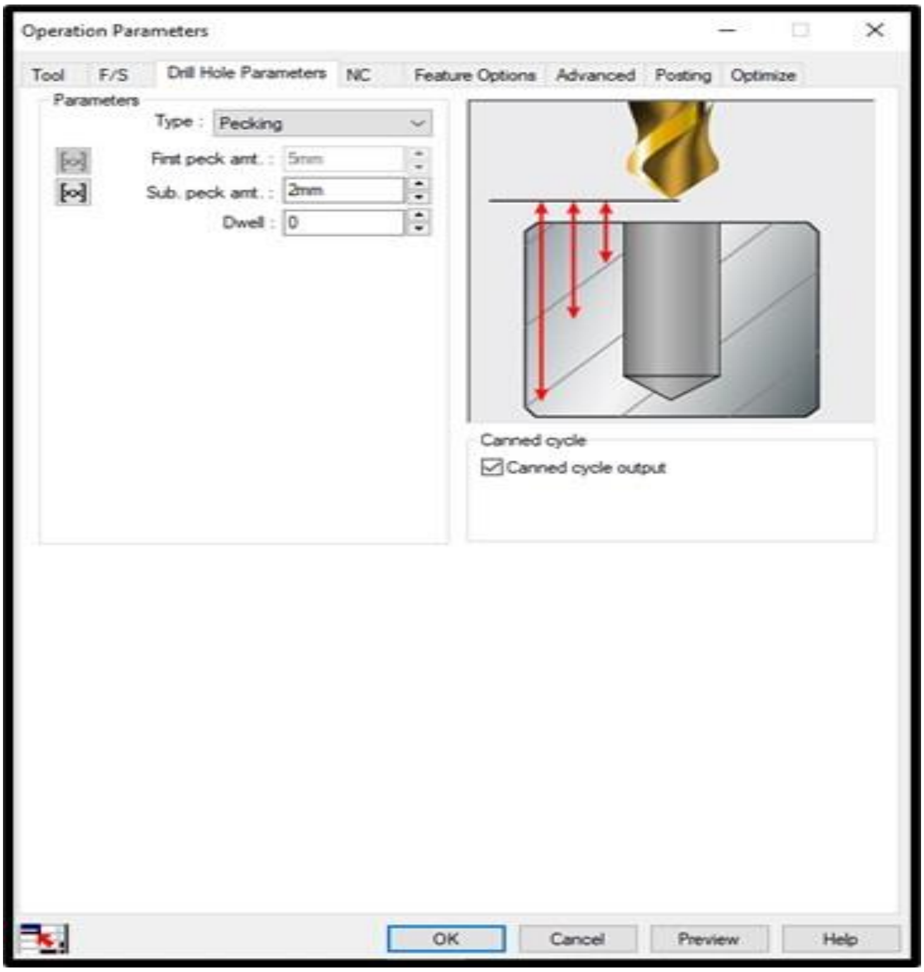

Operation Parameters
Tool
F/S
Center Drill
NC
Feature Options
Advanced
Posting
Optimize
Parameters
Type : Spot Drilling
First peck amt. : 5mm
Sub. peck amt. : 2.5mm
Dwell : 1
Canned cycle
Canned cycle output
OK
Cancel
Preview
Help

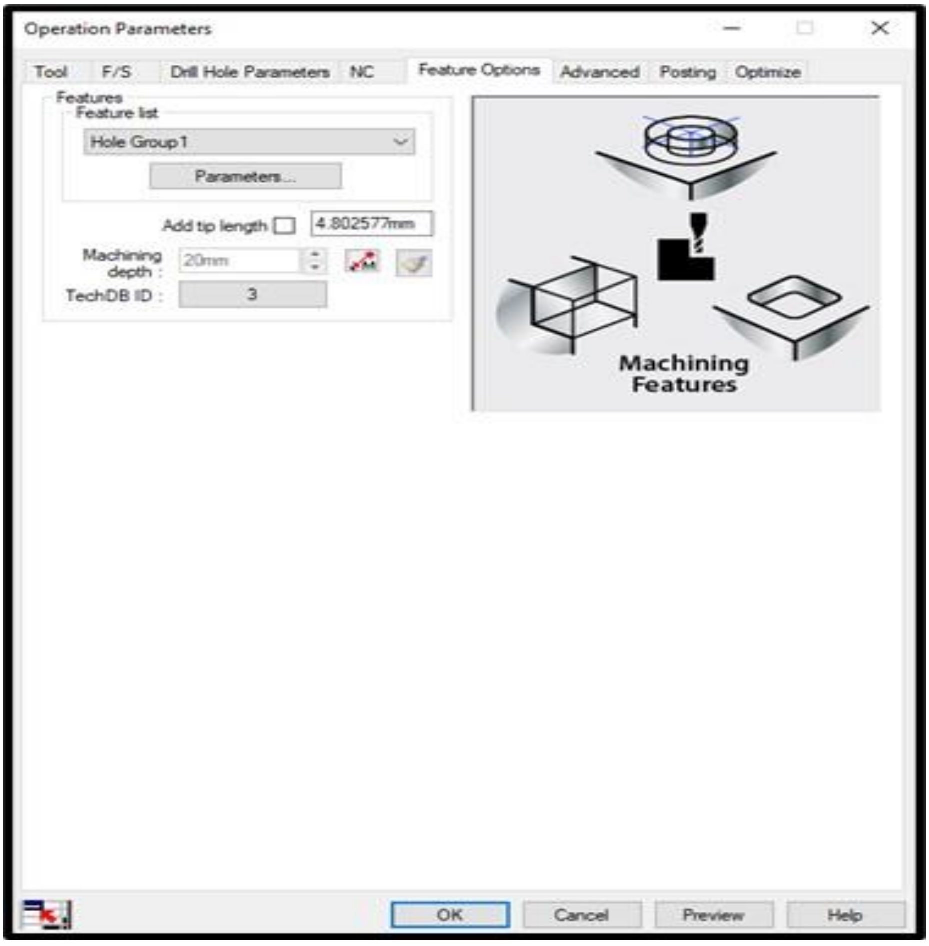

Operation Parameters
Tool
F/S
Center Drill
NC
Feature Options
Advanced
Posting
Optimize
Features
Feature list
Hole Group1
Parameters...
Machining depth : 7.2mm
Set Depth
Dia. created : 6mm
Set Diameter
TechDB ID : 3
Machining Features
OK
Cancel
Preview
Help

DRILL OPERATION:



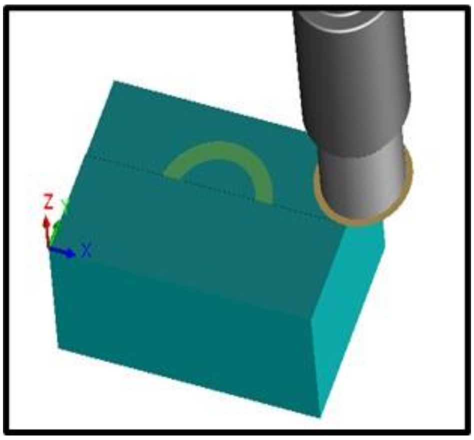


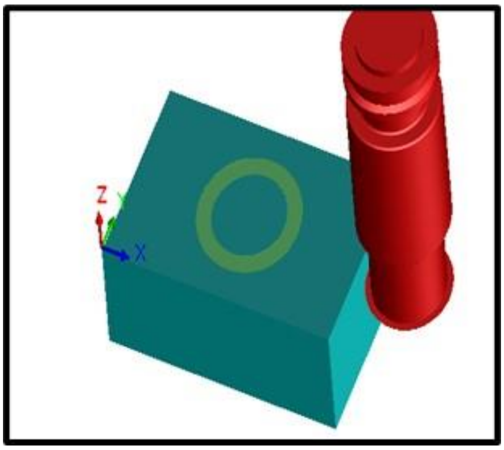

Operation Parameters
Tool
F/S
Drill Hole Parameters
NC
Feature Options
Advanced
Posting
Optimize
Parameters
Type : Pecking
First peck amt. : 5mm
Sub. peck amt. : 2mm
Dwell : 0
Canned cycle
Canned cycle output
OK
Cancel
Preview
Help


Operation Parameters
Tool
F/S
Drill Hole Parameters
NC
Feature Options
Advanced
Posting
Optimize
Features
Feature list
Hole Group1
Parameters...
Add tip length
4.802577mm
Machining depth :
20mm
TechDB ID :
3
Machining Features
OK
Cancel
Preview
Help

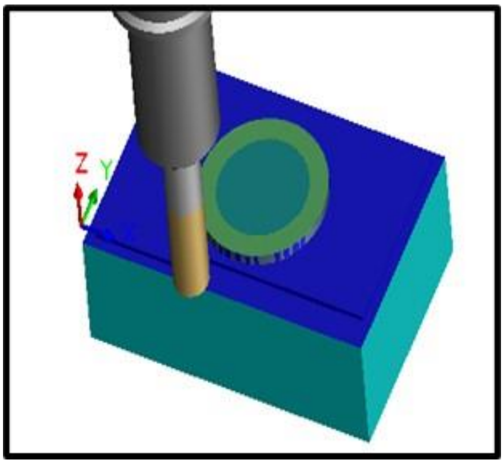
7.4.3 Simulation:

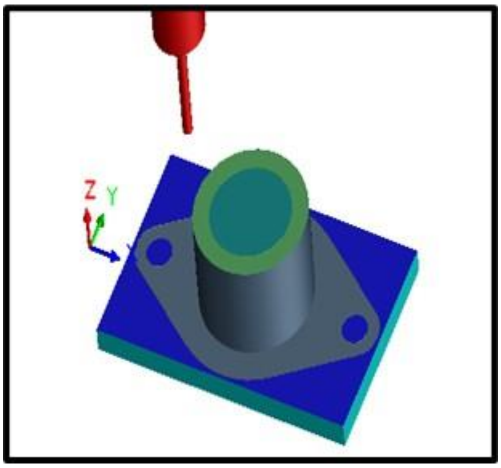
face mill operation:



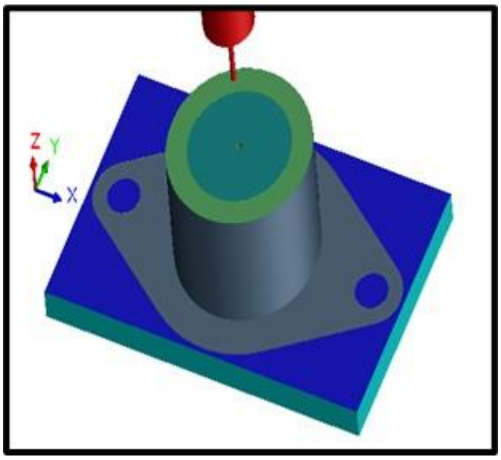


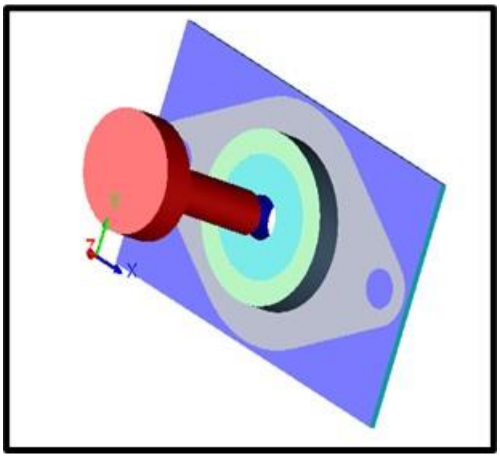
open pocket operation:



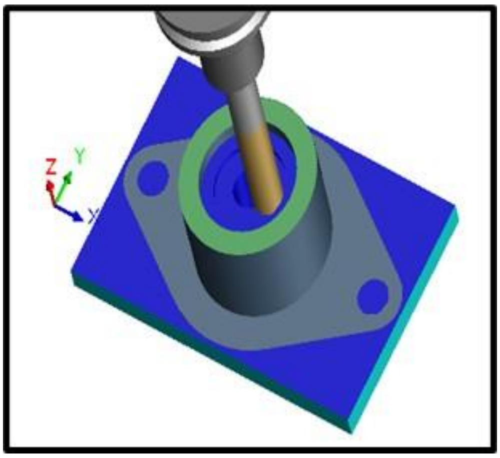


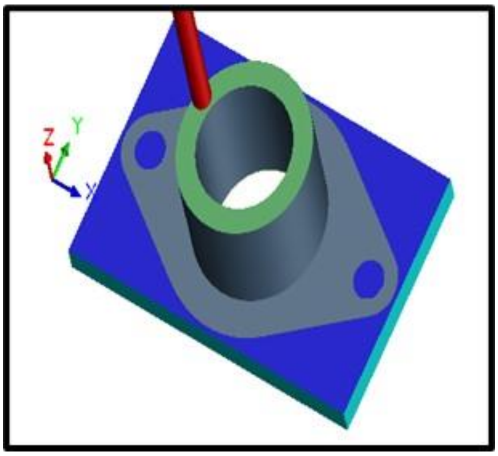
Center drill and drill



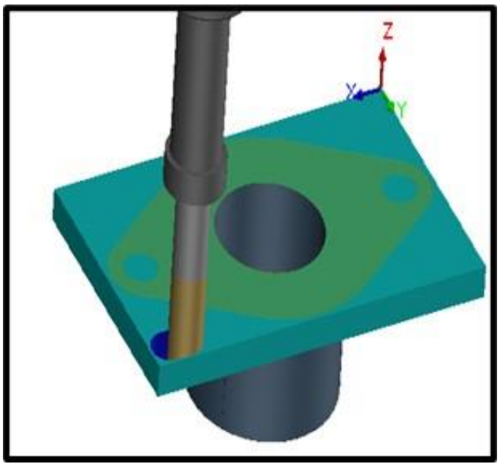


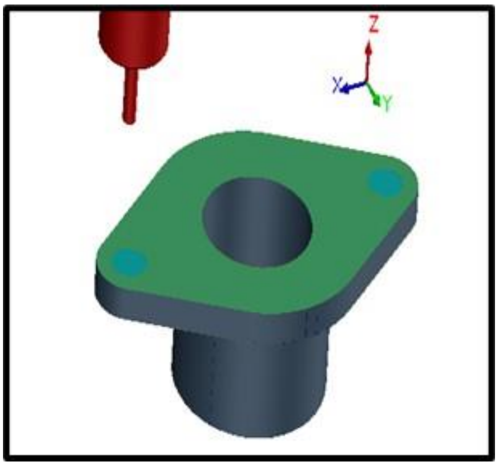
pocket operation:



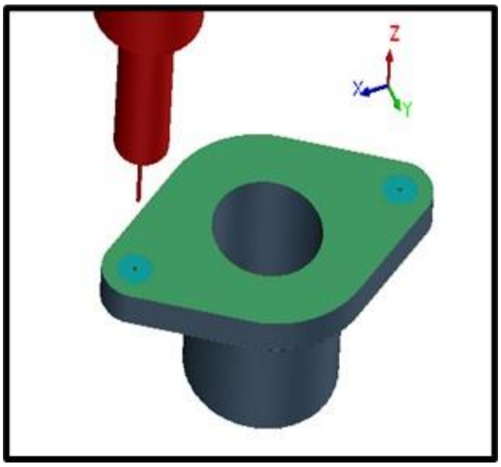


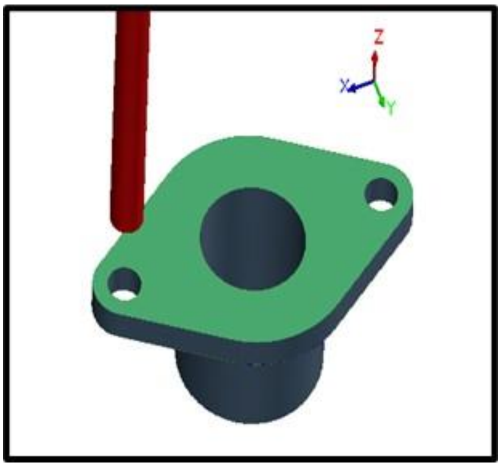
open pocket 2:





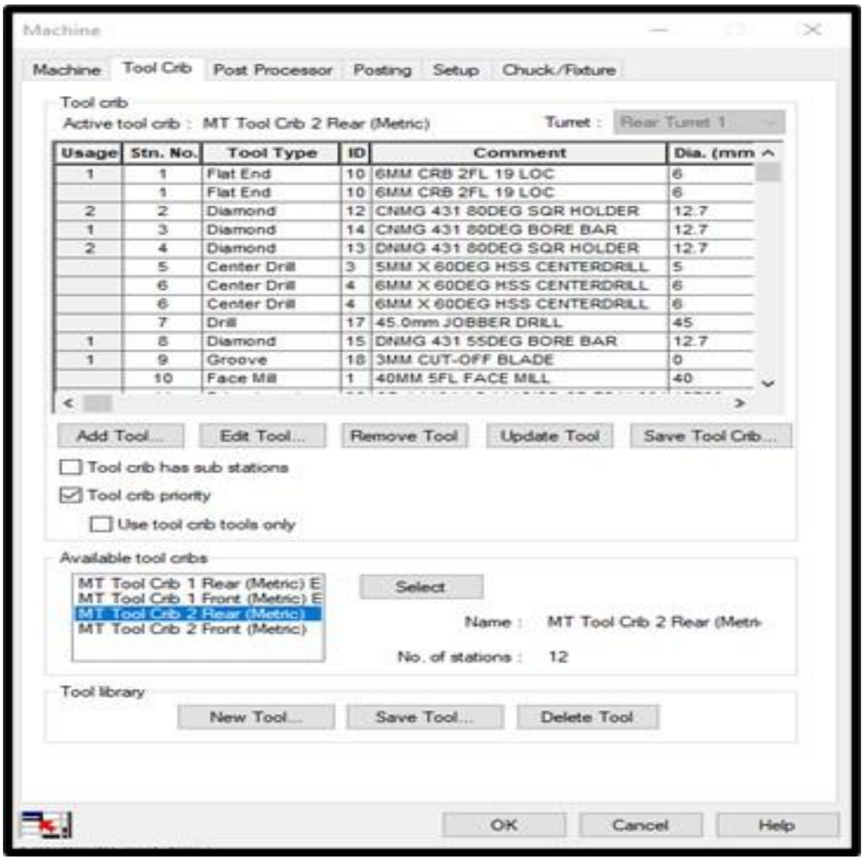
center drill and drill:

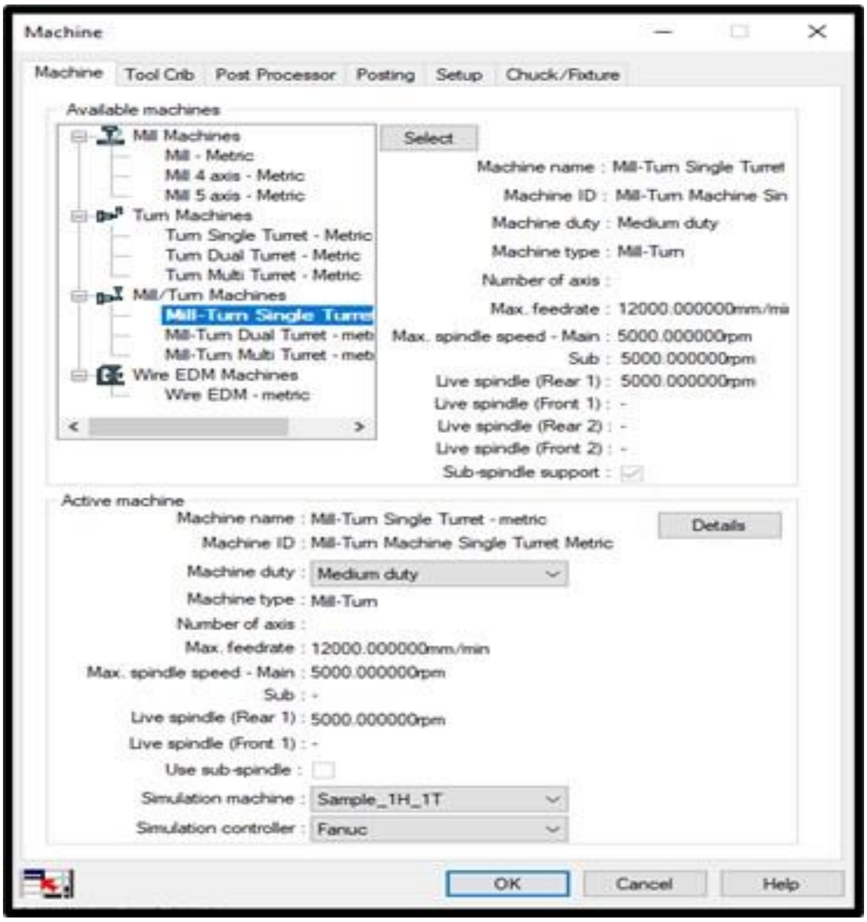




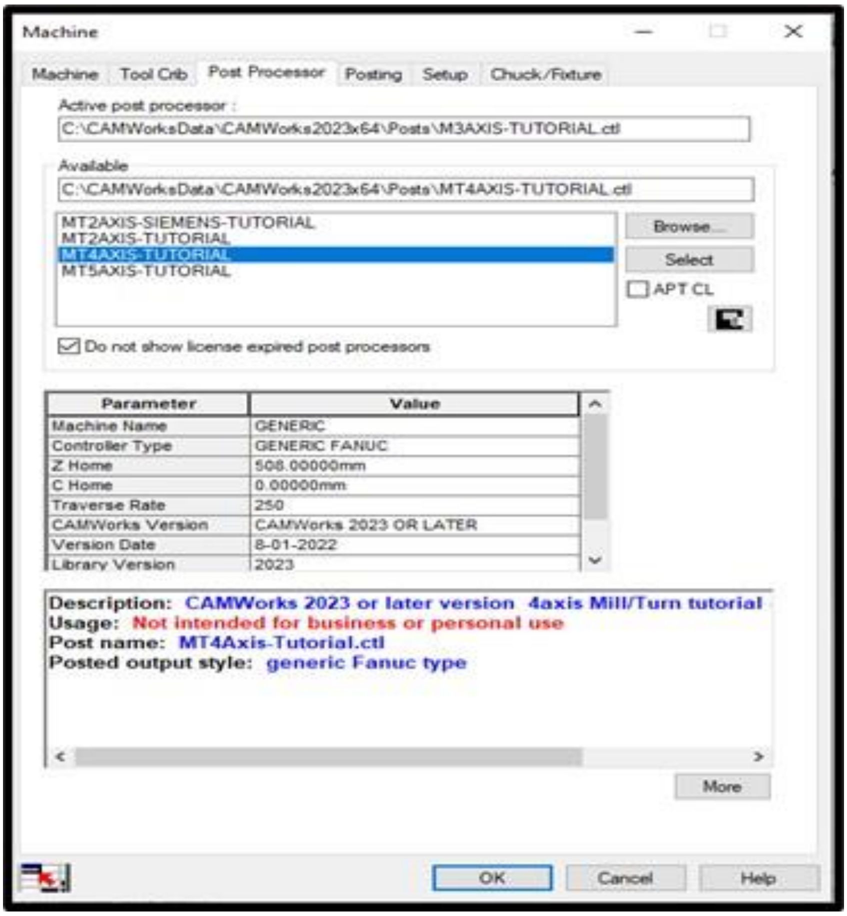
8 The 4 Axis mill-turn machine:

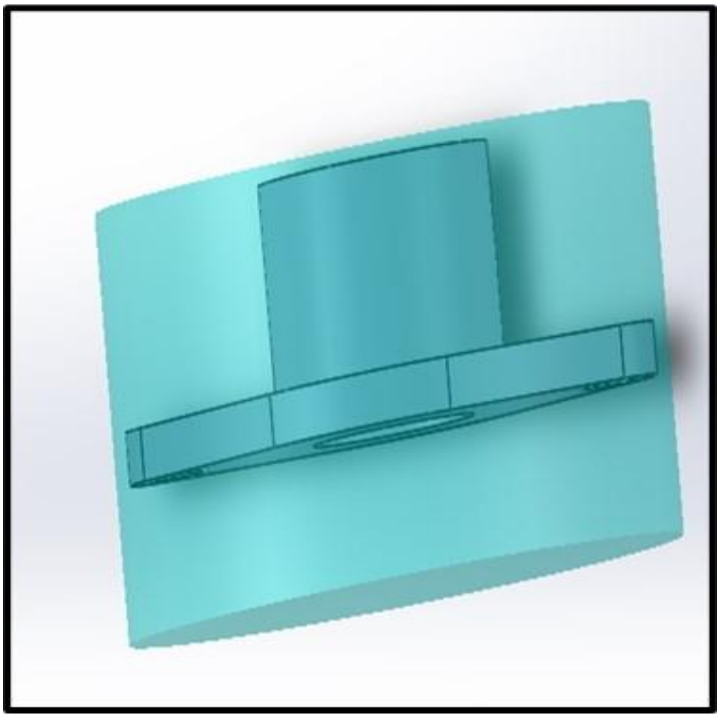
define machine: mill-turn single turret and tool crib





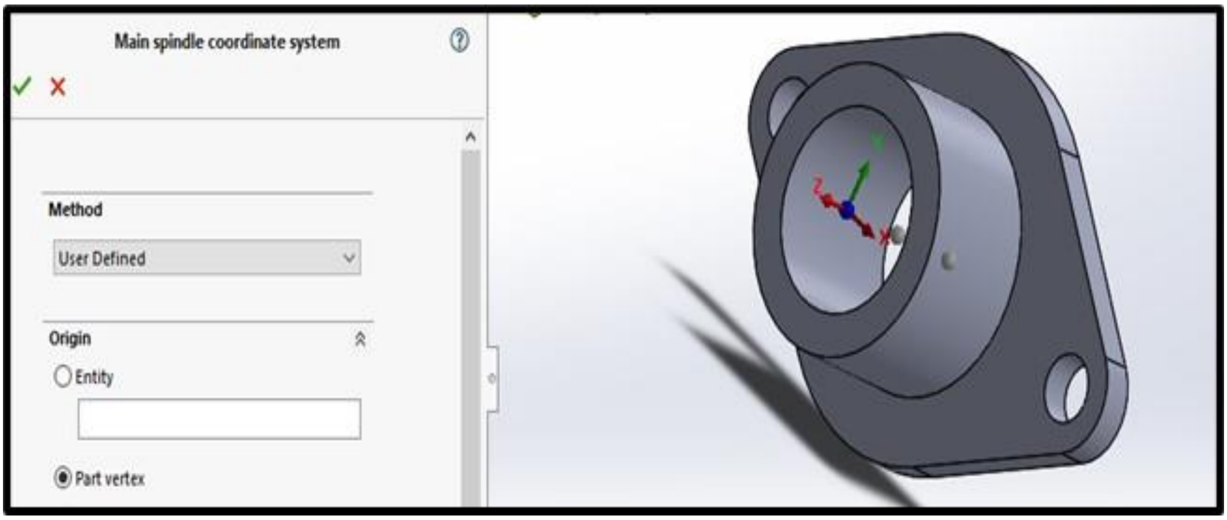
select post processor and raw material:



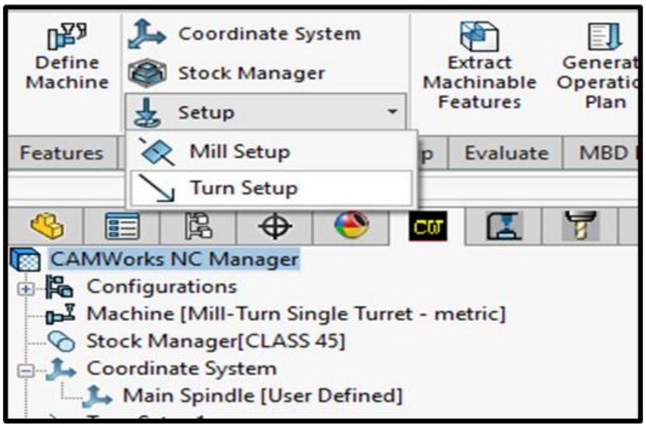


cylindric raw material

choosing coordinate system

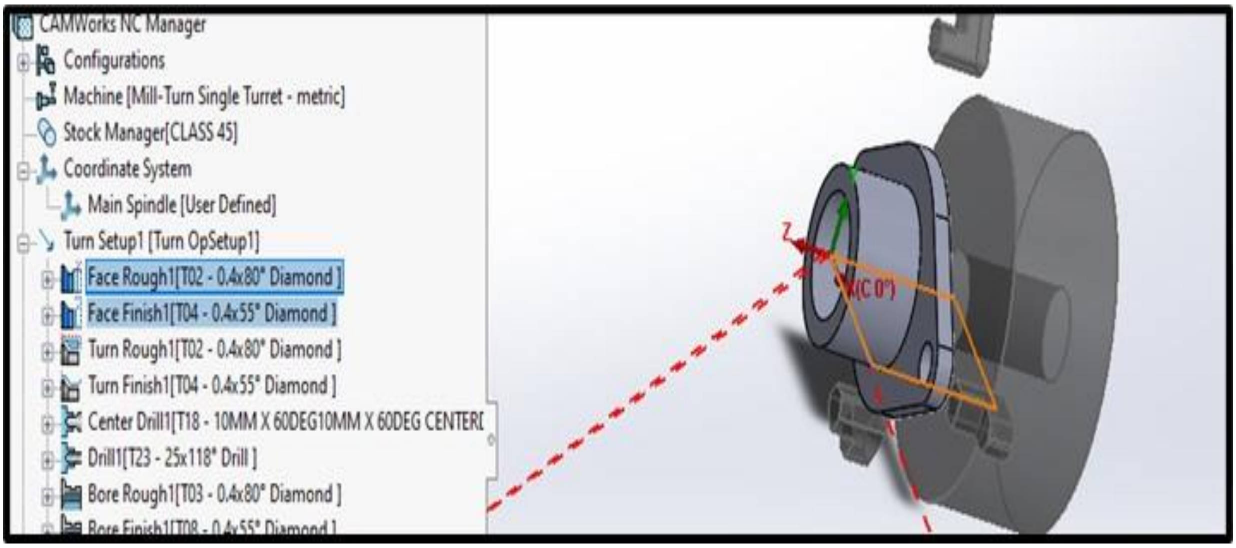


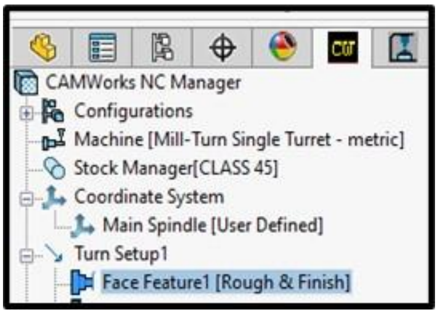
define turn setup:



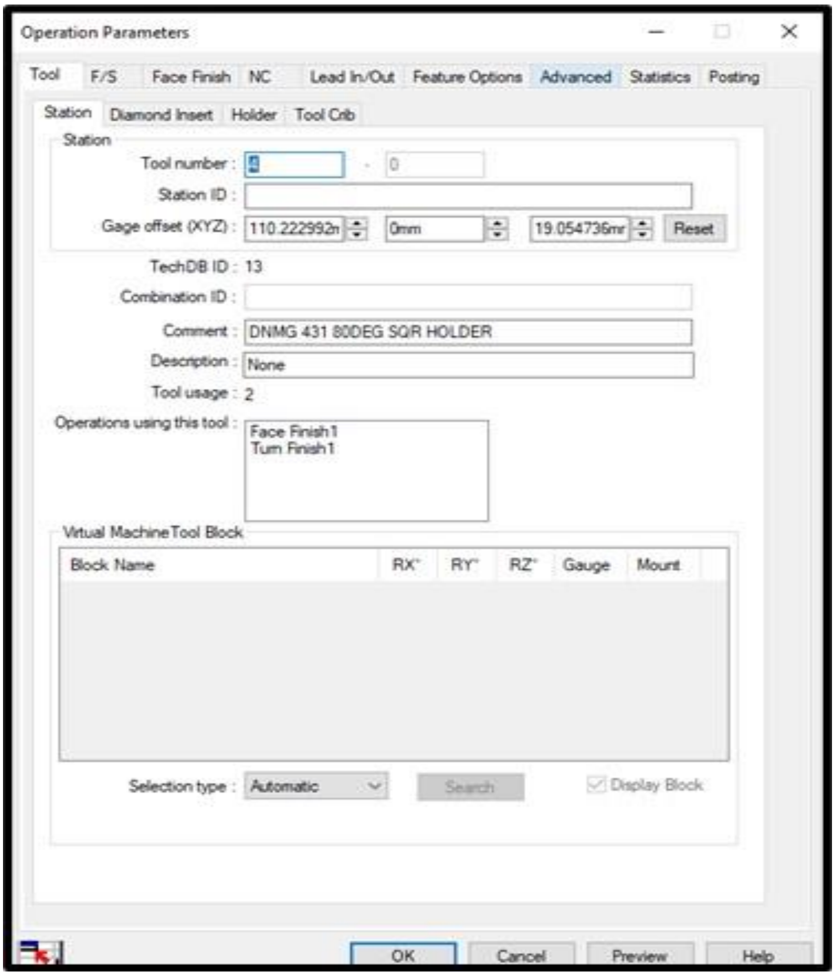
8.1 Turn operations:

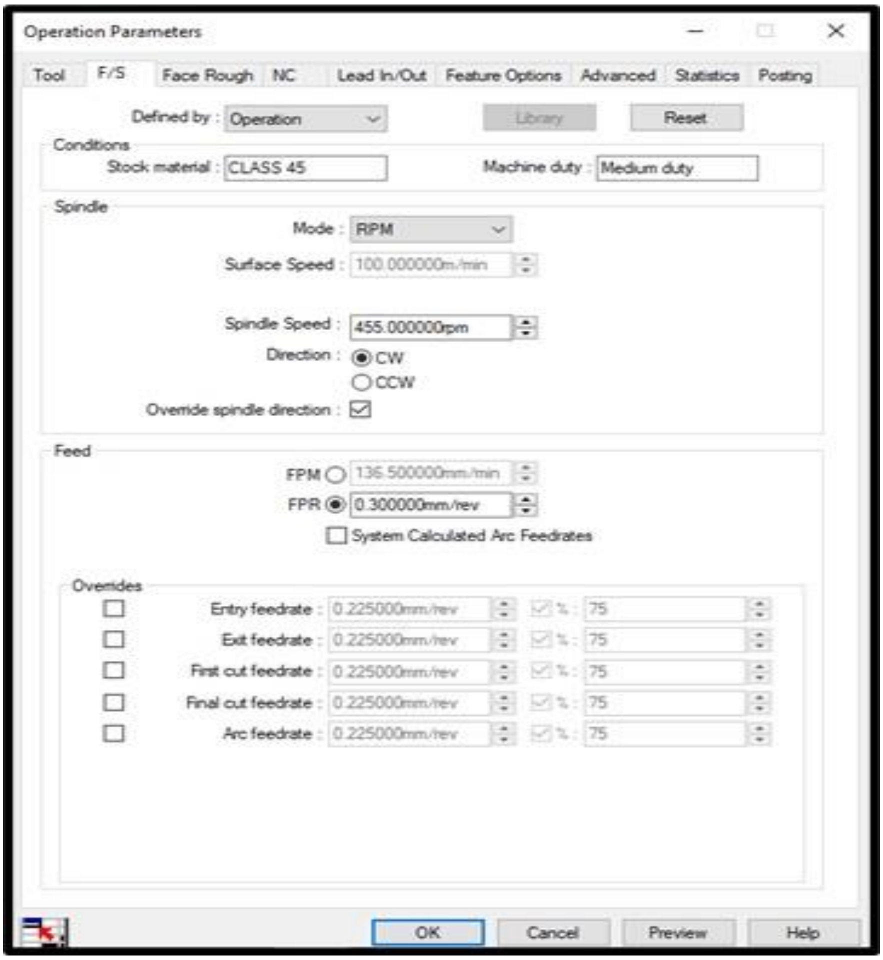
8.1.1 Face feature:

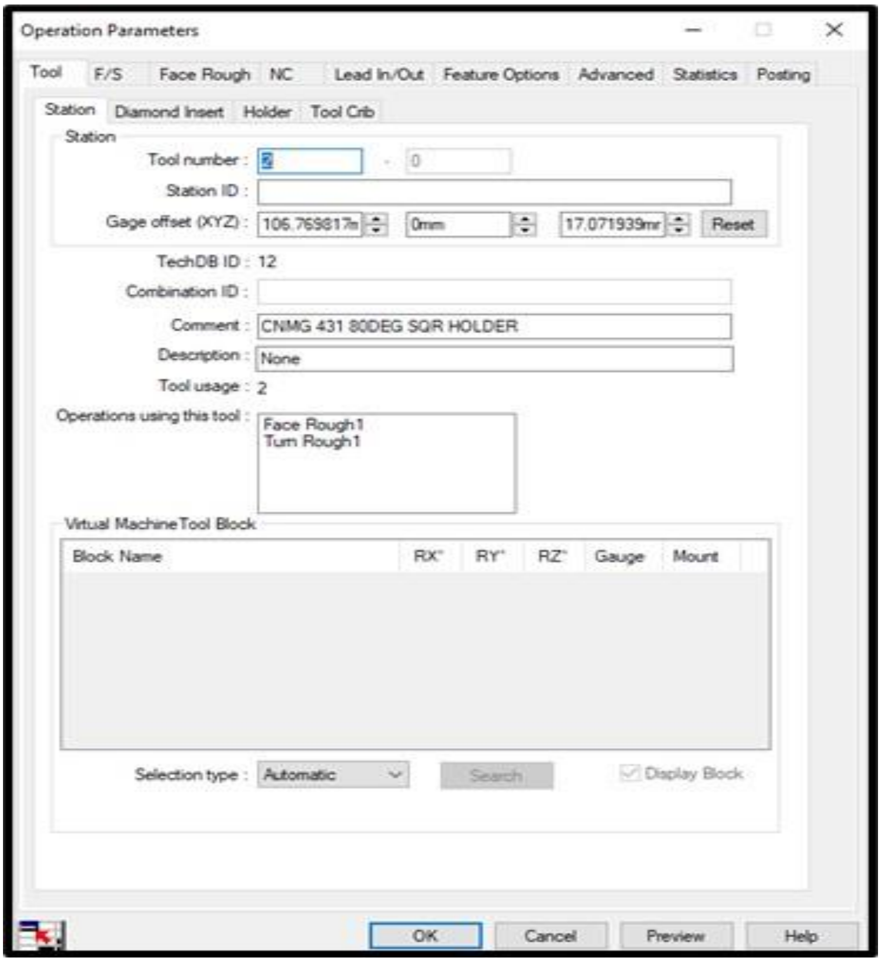


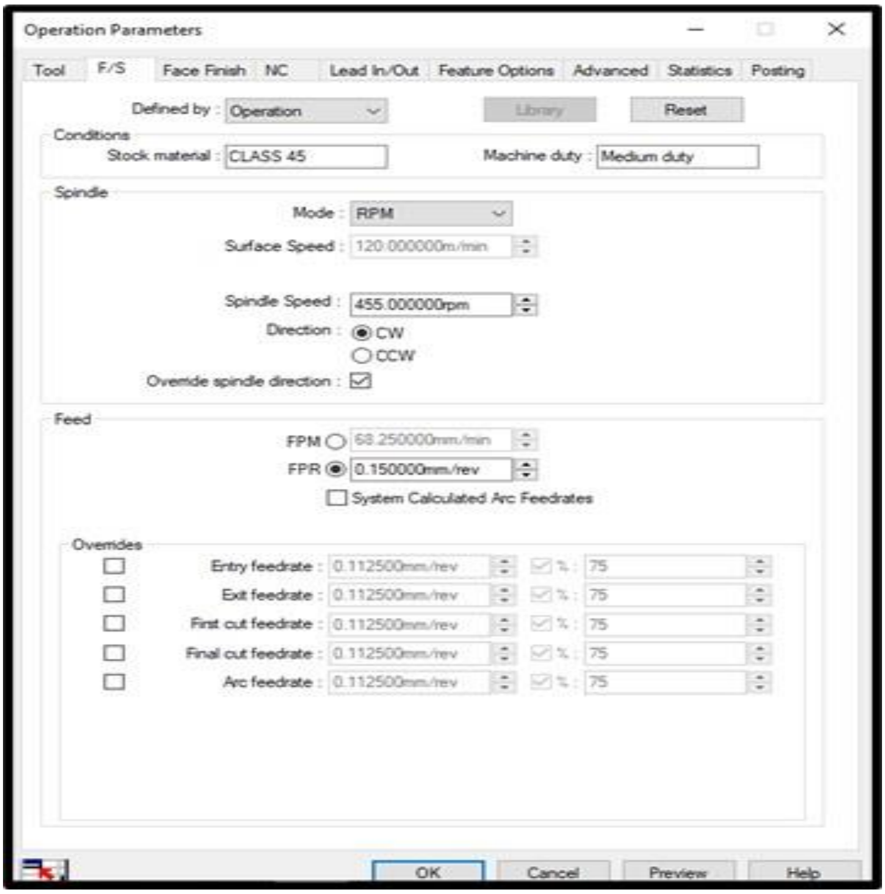


Face rough and face finish:

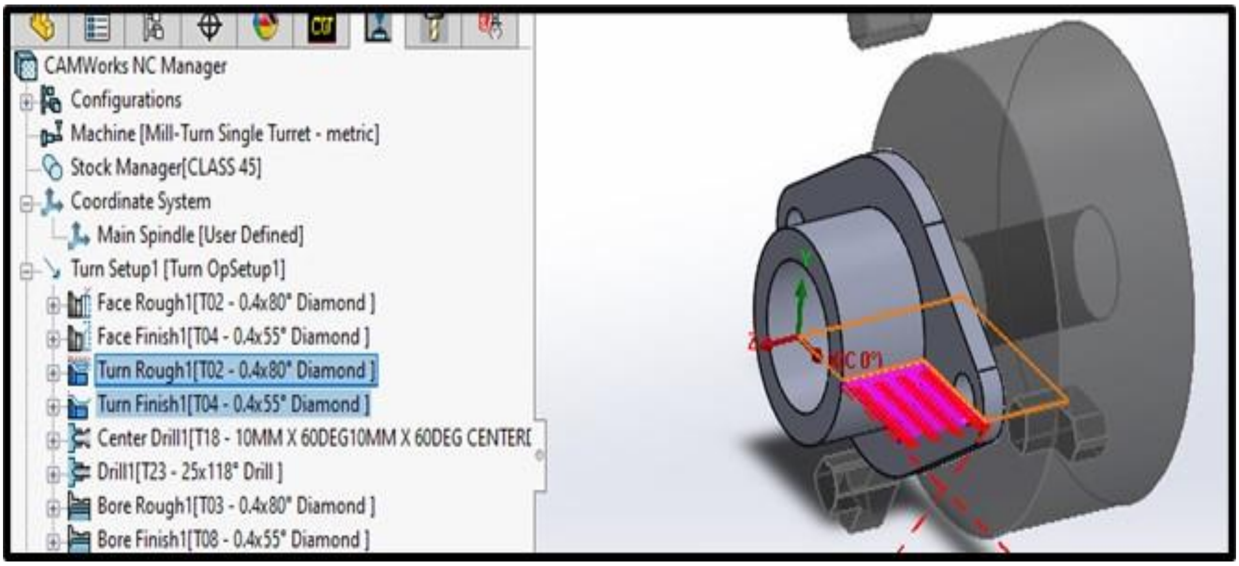


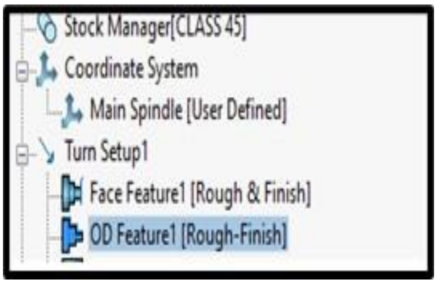

Operation Parameters
Tool
F/S
Face Rough
NC
Lead In/Out
Feature Options
Advanced
Statistics
Posting
Defined by : Operation
Library
Reset
Conditions
Stock material : CLASS 45
Machine duty : Medium duty
Spindle
Mode : RPM
Surface Speed : 100.000000m/min
Spindle Speed : 455.000000rpm
Direction : CW
CCW
Override spindle direction :
Feed
FPM 136.500000mm/min
FPR 0.300000mm/rev
System Calculated Arc Feedrates
Overrides
Entry feedrate : 0.225000mm/rev % : 75
Exit feedrate : 0.225000mm/rev % : 75
First cut feedrate : 0.225000mm/rev % : 75
Final cut feedrate : 0.225000mm/rev % : 75
Arc feedrate : 0.225000mm/rev % : 75
OK
Cancel
Preview
Help


Operation Parameters
Tool
F/S
Face Rough
NC
Lead In/Out
Feature Options
Advanced
Statistics
Posting
Station
Diamond Insert
Holder
Tool Crib
Station
Tool number : 2 - 0
Station ID :
Gage offset (XYZ) : 106.769817m 0mm 17.071939mr Reset
TechDB ID : 12
Combination ID :
Comment : CNMG 431 80DEG SQR HOLDER
Description : None
Tool usage : 2
Operations using this tool : Face Rough1
Turn Rough1
Virtual Machine Tool Block
Block Name RX° RY° RZ° Gauge Mount
Selection type : Automatic
Search
Display Block
OK
Cancel
Preview
Help

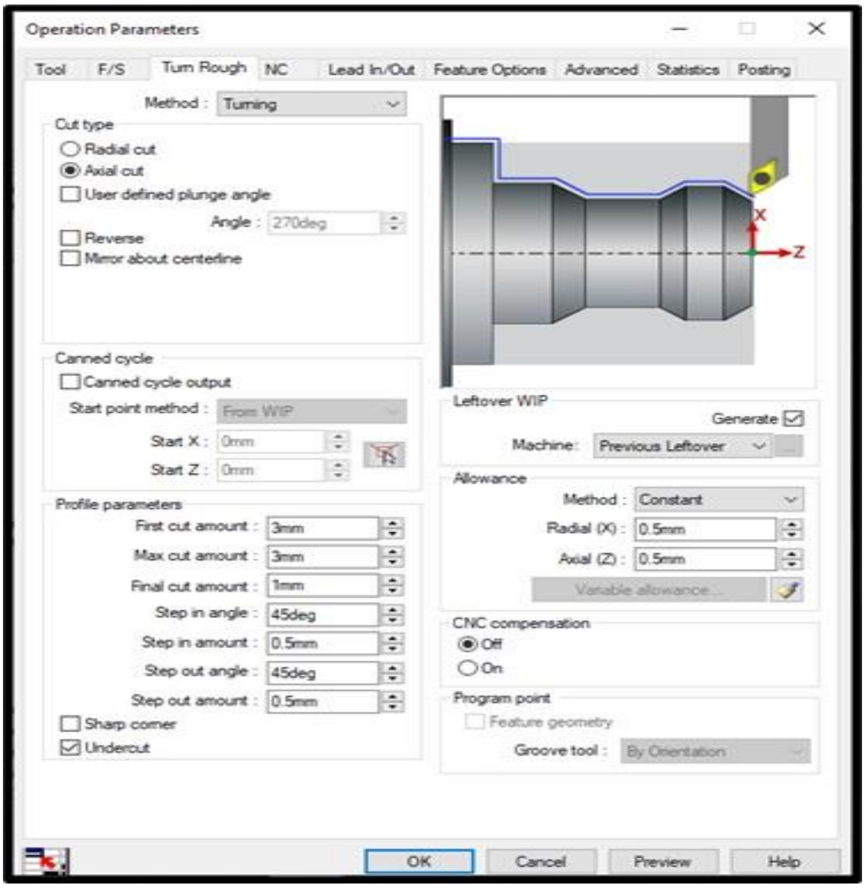


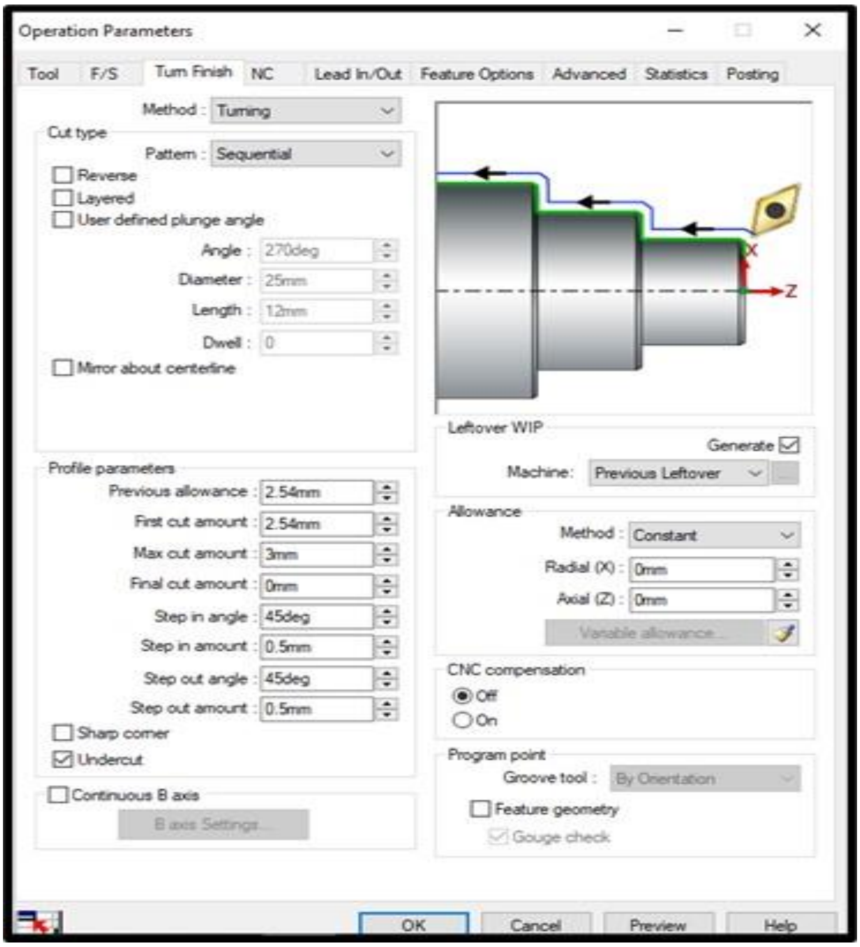
8.1.2 Outside diameter feature:



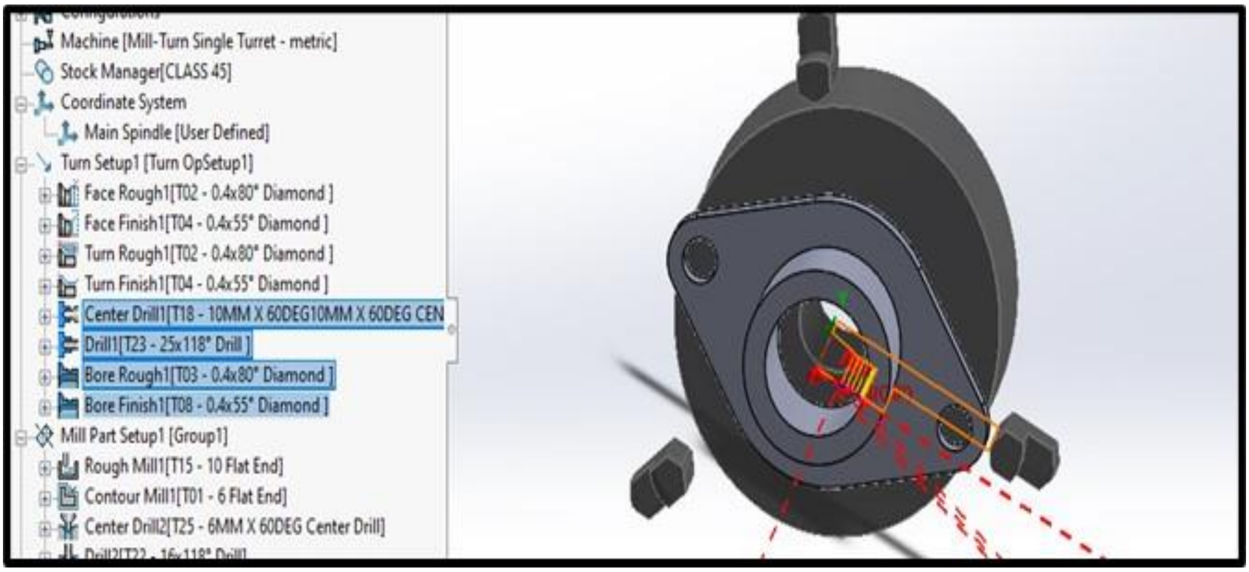


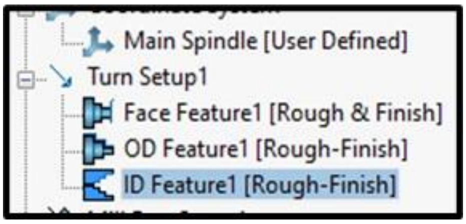
the tool and speeds are the same on the first operation.



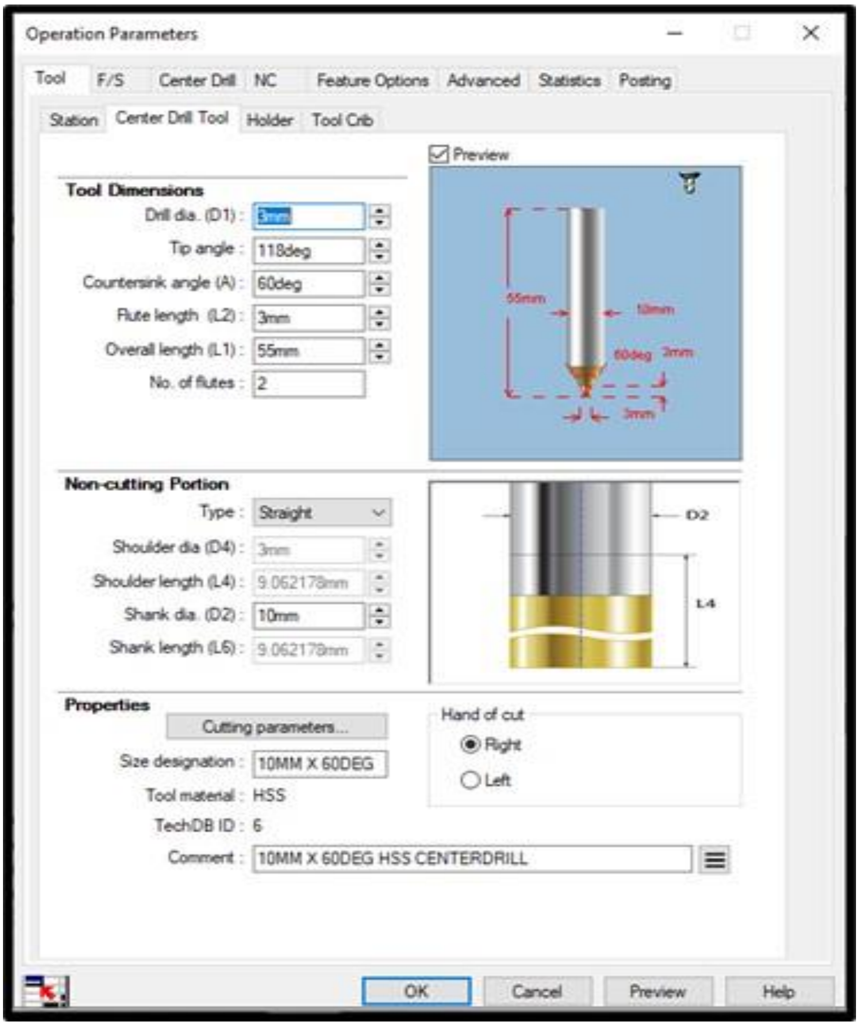


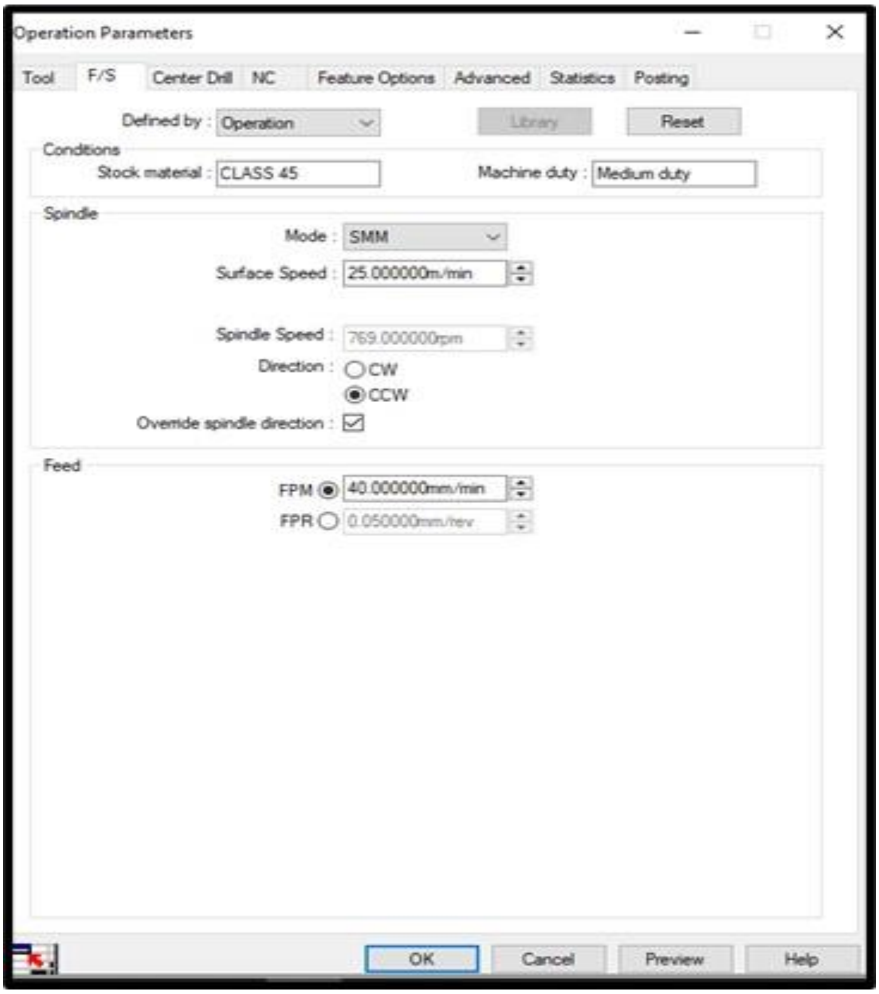
8.1.3 Inside diameter feature:

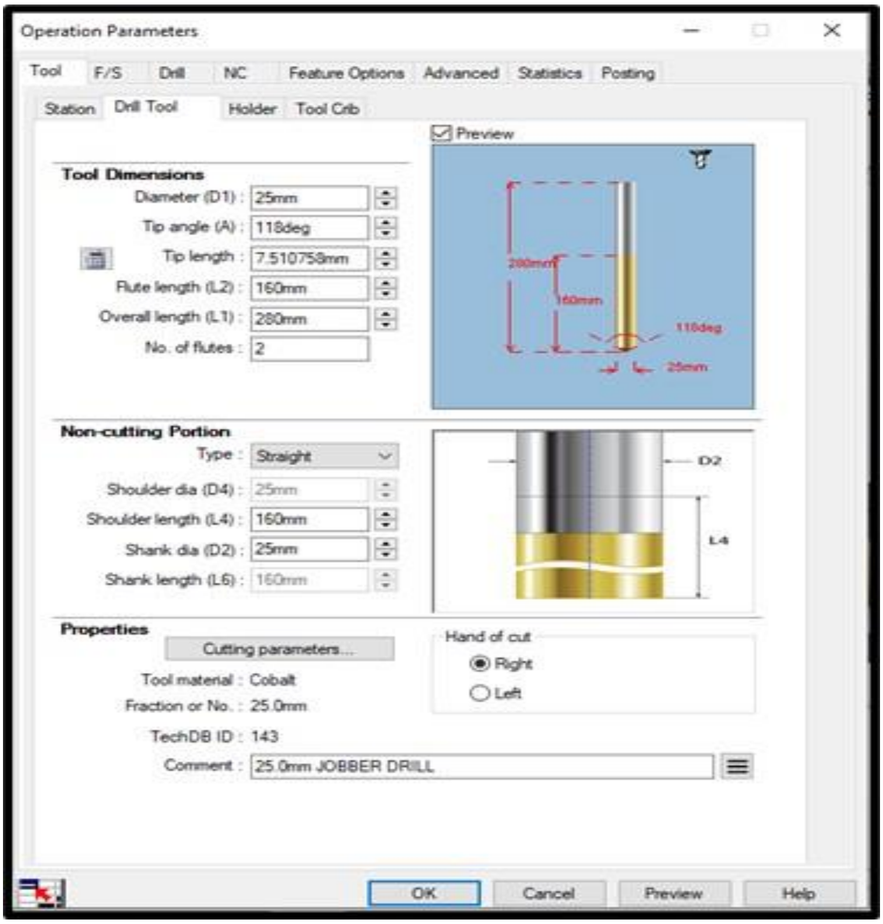


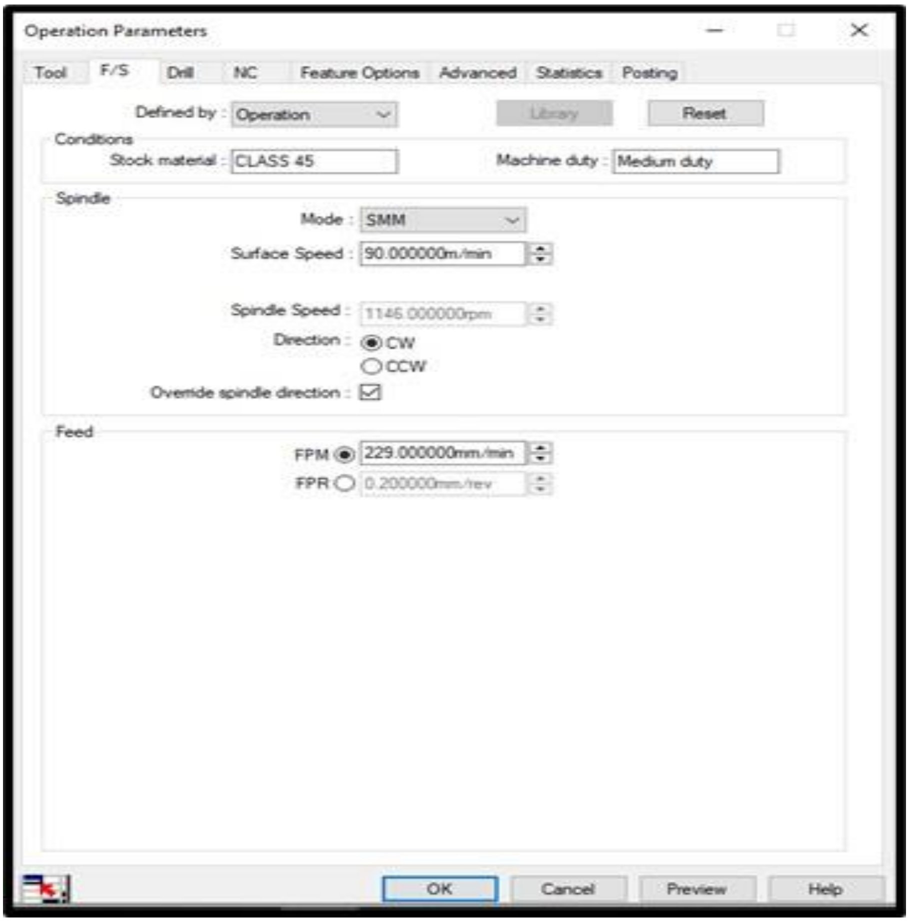


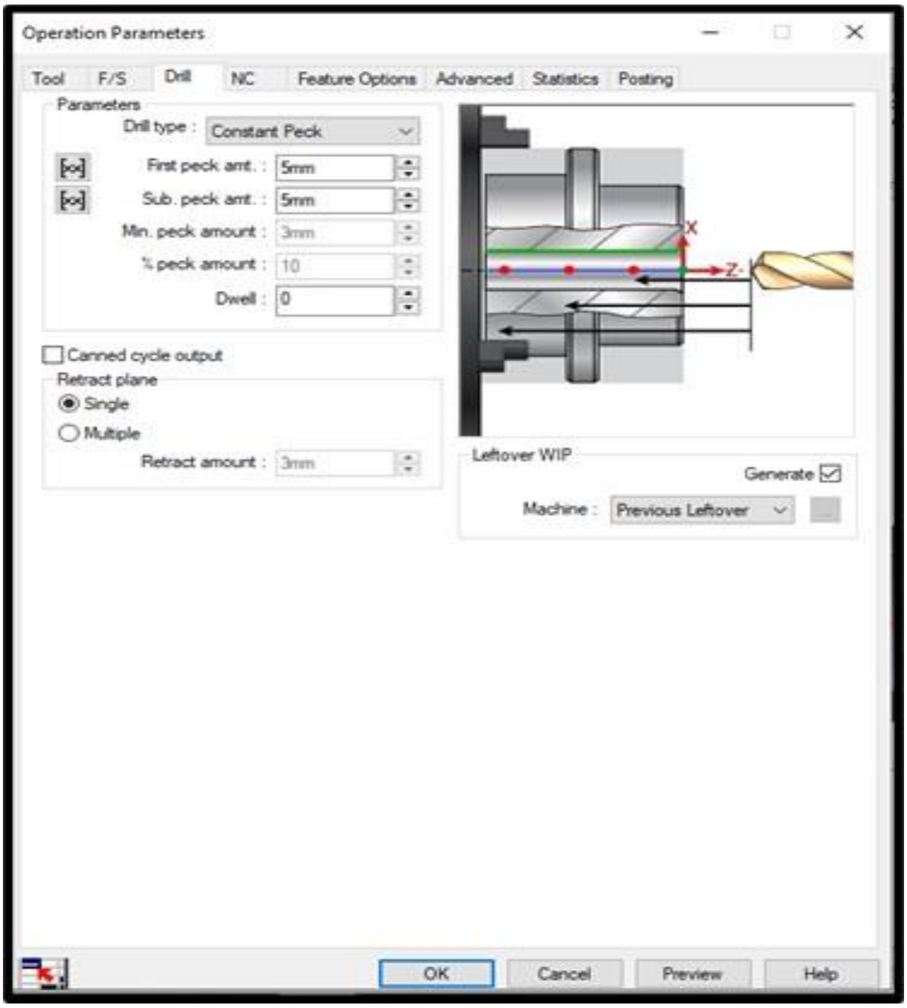
center drill and drill:

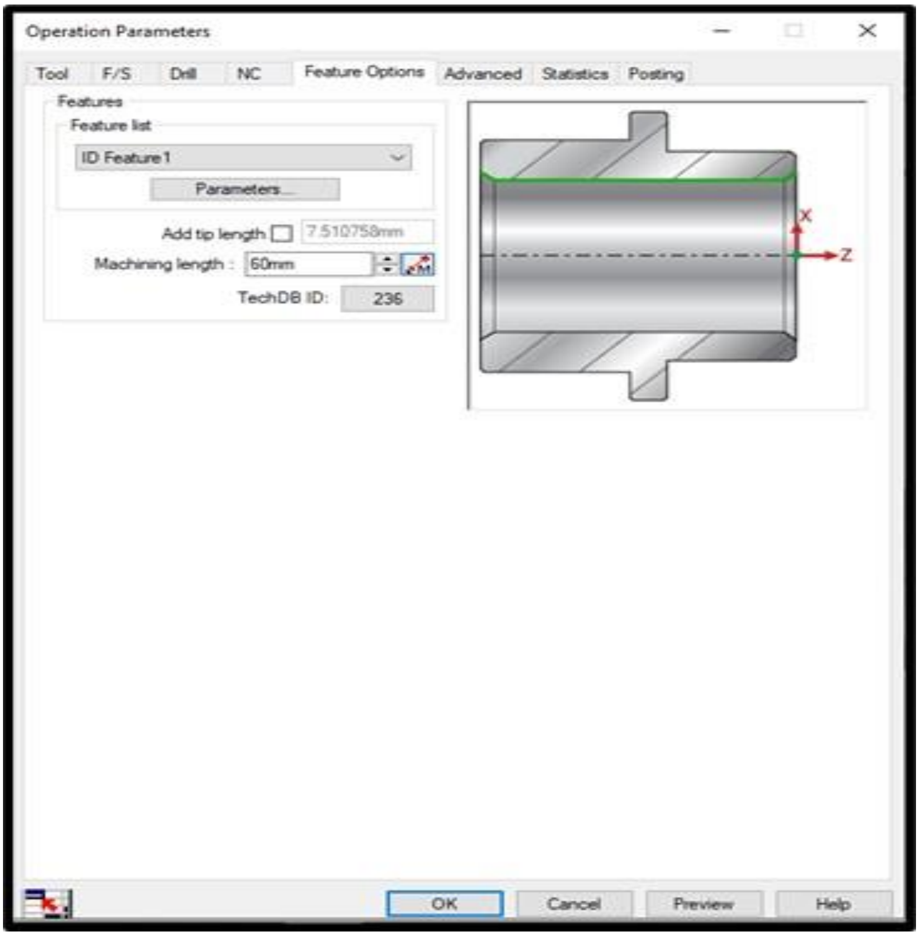



Operation Parameters
Tool F/S Center Drill NC Feature Options Advanced Statistics Posting
Defined by : Operation
Library Reset
Conditions
Stock material : CLASS 45
Machine duty : Medium duty
Spindle
Mode : SMM
Surface Speed : 25.000000m/min
Spindle Speed : 769.000000rpm
Direction : CW CCW
Override spindle direction :
Feed
FPM 40.000000mm/min
FPR 0.050000mm/rev
OK Cancel Preview Help

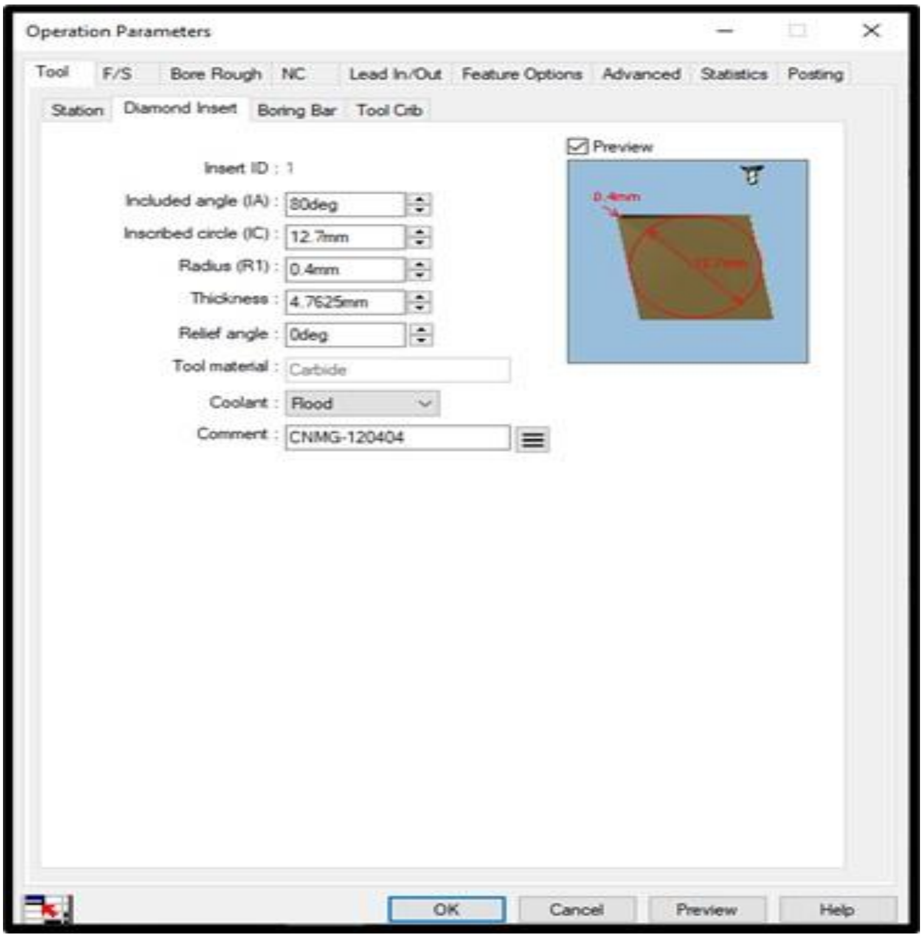

Operation Parameters
Tool F/S Drill NC Feature Options Advanced Statistics Posting
Station Drill Tool Holder Tool Crib
Preview
Tool Dimensions
Diameter (D1) : 25mm
Tip angle (A) : 118deg
Tip length : 7.510758mm
Flute length (L2) : 160mm
Overall length (L1) : 280mm
No. of flutes : 2
Non-cutting Portion
Type : Straight
Shoulder dia (D4) : 25mm
Shoulder length (L4) : 160mm
Shank dia (D2) : 25mm
Shank length (L6) : 160mm
D2
L4
Properties
Cutting parameters...
Tool material : Cobalt
Fraction or No. : 25.0mm
TechDB ID : 143
Comment : 25.0mm JOBBER DRILL
Hand of cut
Right
Left
OK Cancel Preview Help

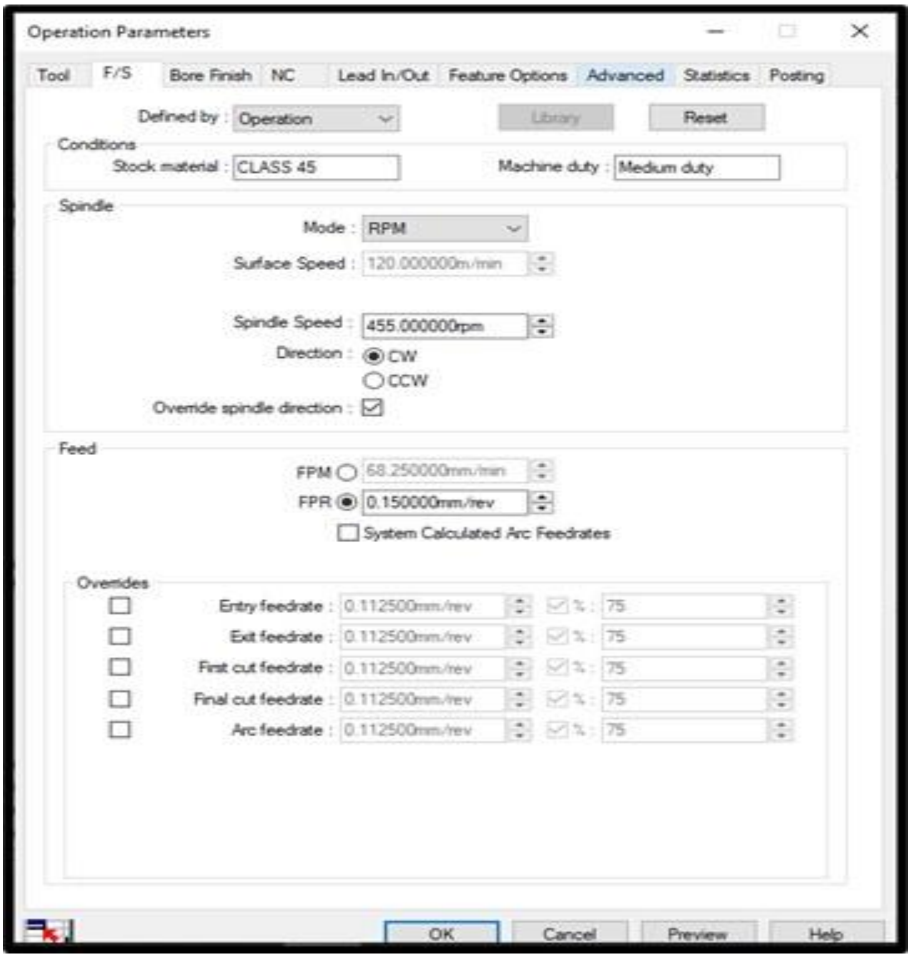

Operation Parameters
Tool
F/S
Drill
NC
Feature Options
Advanced
Statistics
Posting
Defined by : Operation
Library
Reset
Conditions
Stock material : CLASS 45
Machine duty : Medium duty
Spindle
Mode : SMM
Surface Speed : 90.000000m/min
Spindle Speed : 1146.000000rpm
Direction : CW
CCW
Override spindle direction :
Feed
FPM 229.000000mm/min
FPR 0.200000mm/rev
OK
Cancel
Preview
Help


Operation Parameters
Tool
F/S
Drill
NC
Feature Options
Advanced
Statistics
Posting
Parameters
Drill type : Constant Peck
First peck amt. : 5mm
Sub. peck amt. : 5mm
Min. peck amount : 3mm
% peck amount : 10
Dwell : 0
Canned cycle output
Retract plane
Single
Multiple
Retract amount : 3mm
X
Z
Leftover WIP
Generate
Machine : Previous Leftover
OK
Cancel
Preview
Help

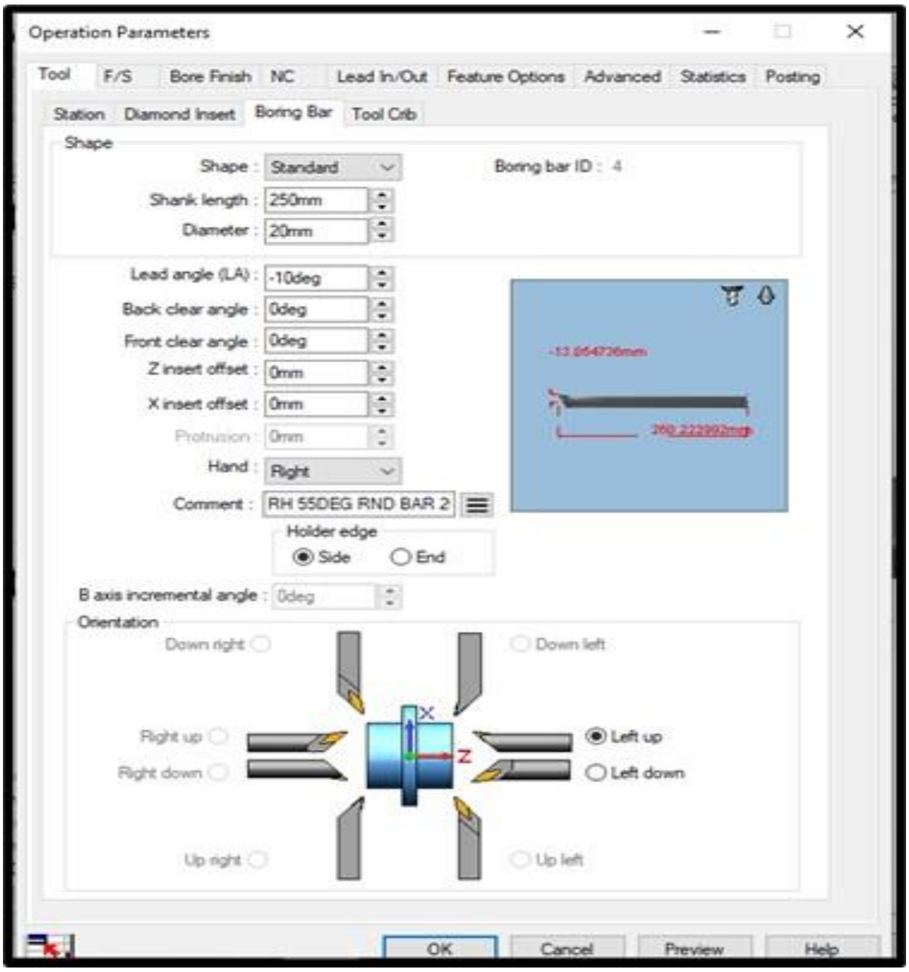


bore rough and finish:

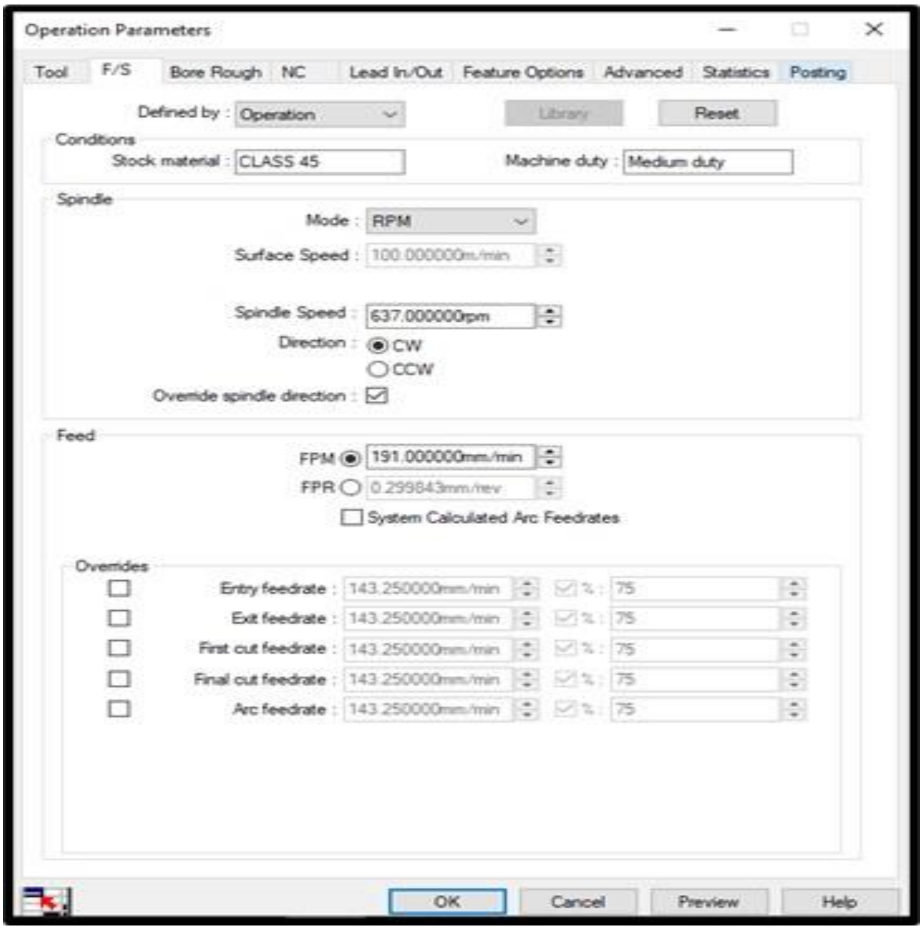




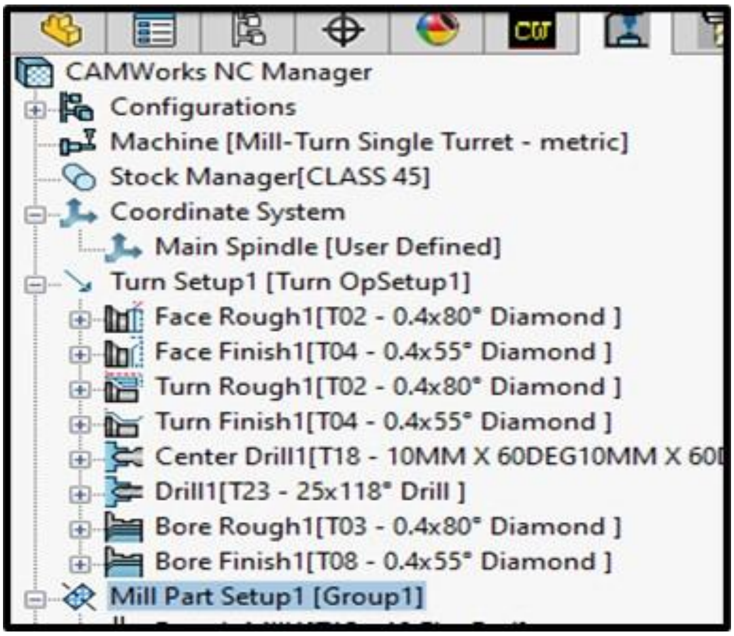
Operation Parameters
Tool
F/S
Bore Finish
NC
Lead In/Out
Feature Options
Advanced
Statistics
Posting
Defined by : Operation
Library
Reset
Conditions
Stock material : CLASS 45
Machine duty : Medium duty
Spindle
Mode : RPM
Surface Speed : 120.000000m/min
Spindle Speed : 455.000000rpm
Direction : CW
CCW
Override spindle direction :
Feed
FPM 68.250000mm/min
FPR 0.150000mm/rev
System Calculated Arc Feedrates
Overrides
Entry feedrate : 0.112500mm/rev
Exit feedrate : 0.112500mm/rev
First cut feedrate : 0.112500mm/rev
Final cut feedrate : 0.112500mm/rev
Arc feedrate : 0.112500mm/rev
75
OK
Cancel
Preview
Help



Operation Parameters
Tool
F/S
Bore Finish
NC
Lead In/Out
Feature Options
Advanced
Statistics
Posting
Station
Diamond Insert
Boring Bar
Tool Crib
Shape
Shape : Standard
Boring bar ID : 4
Shank length : 250mm
Diameter : 20mm
Lead angle (LA) : -10deg
Back clear angle : 0deg
Front clear angle : 0deg
Z insert offset : 0mm
X insert offset : 0mm
Protrusion : 0mm
Hand : Right
Comment : RH 55DEG RND BAR 2
Holder edge
Side
End
B axis incremental angle : 0deg
Orientation
Down right
Down left
Right up
Right down
Left up
Left down
Up right
Up left
X
Z
OK
Cancel
Preview
Help

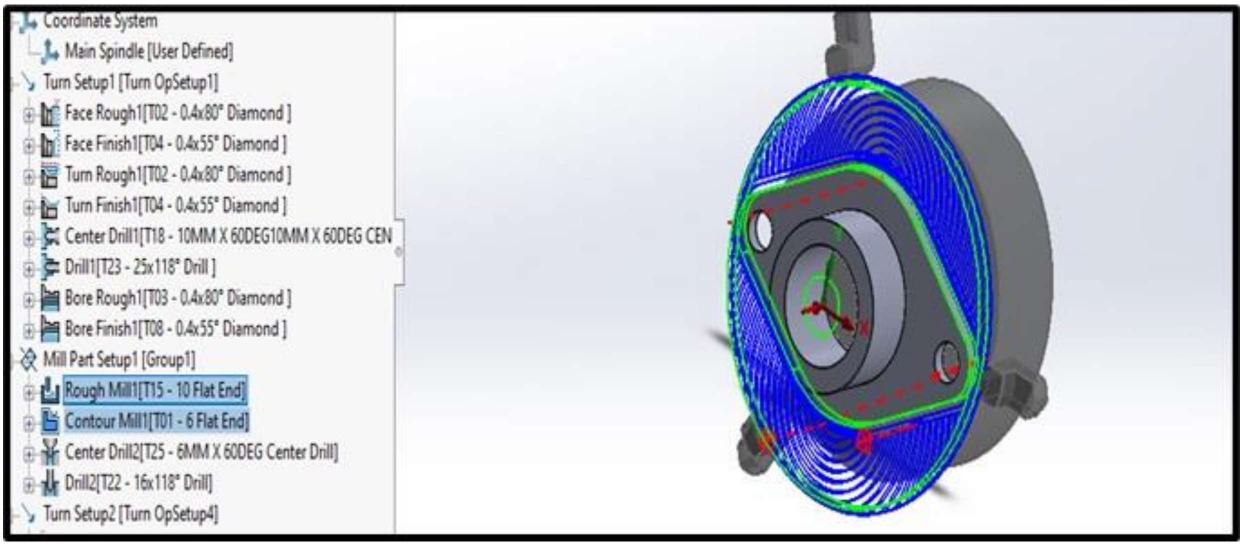


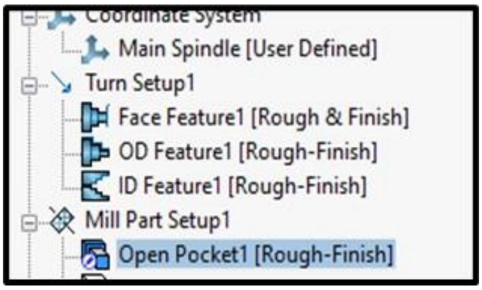
- **mill part setup:**



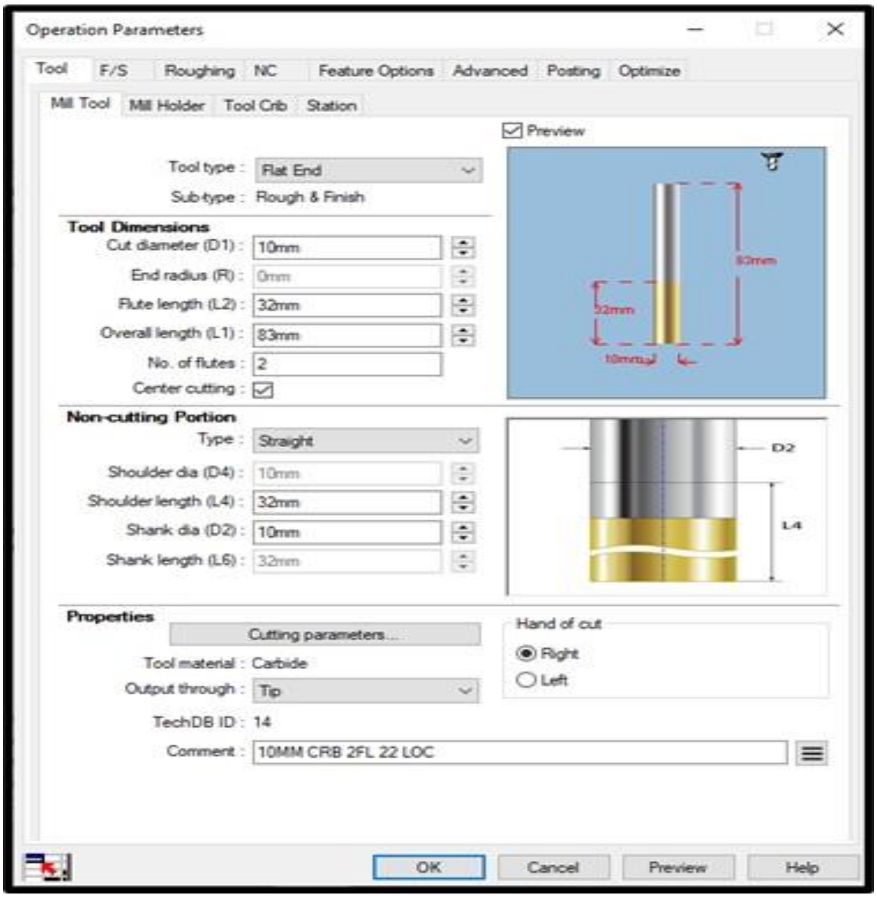
8.2 Mill operations:

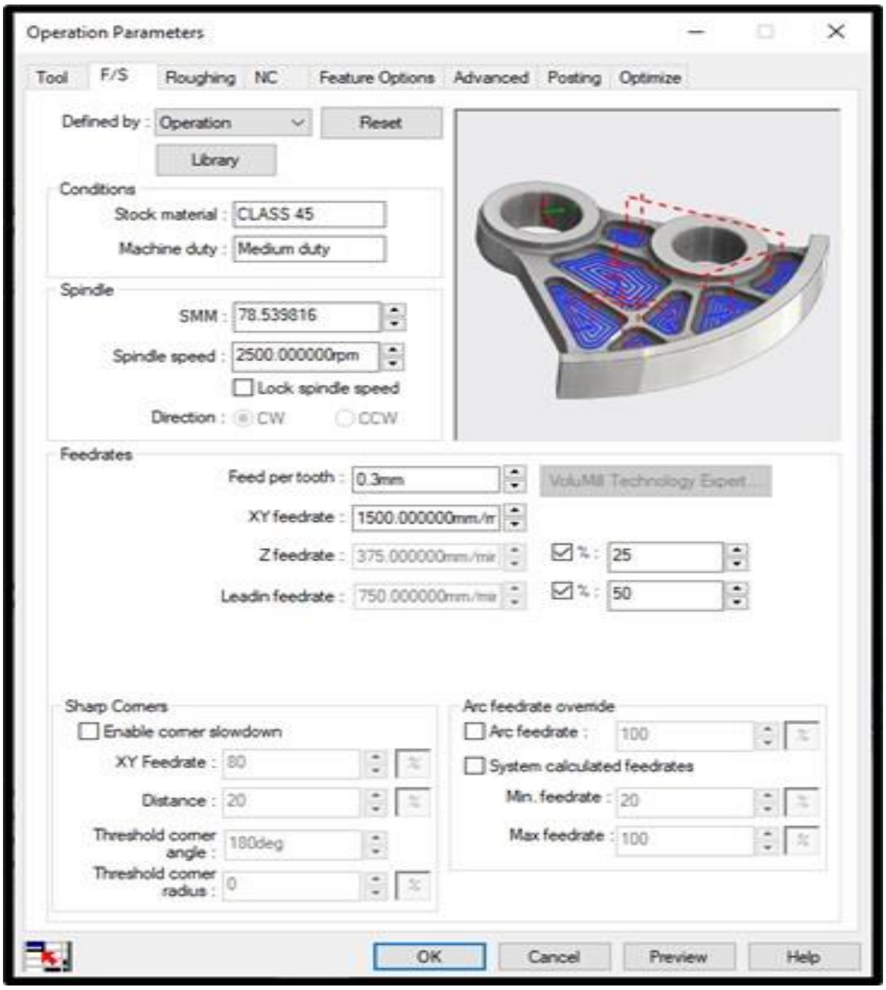
8.2.1 Open pocket operation:

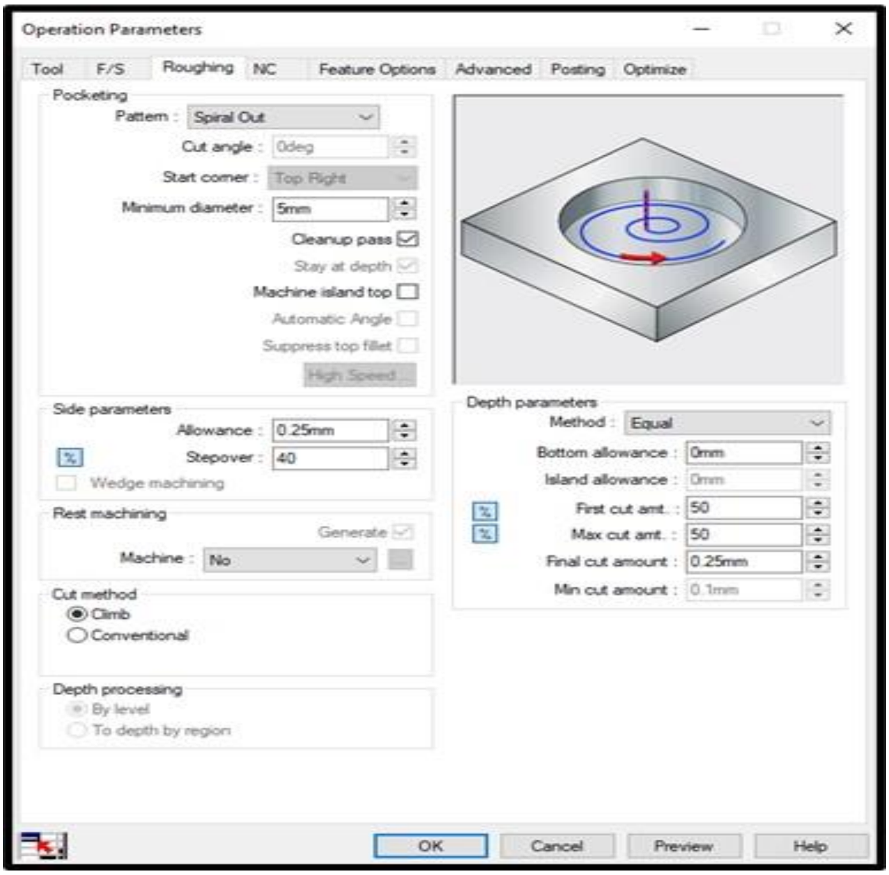


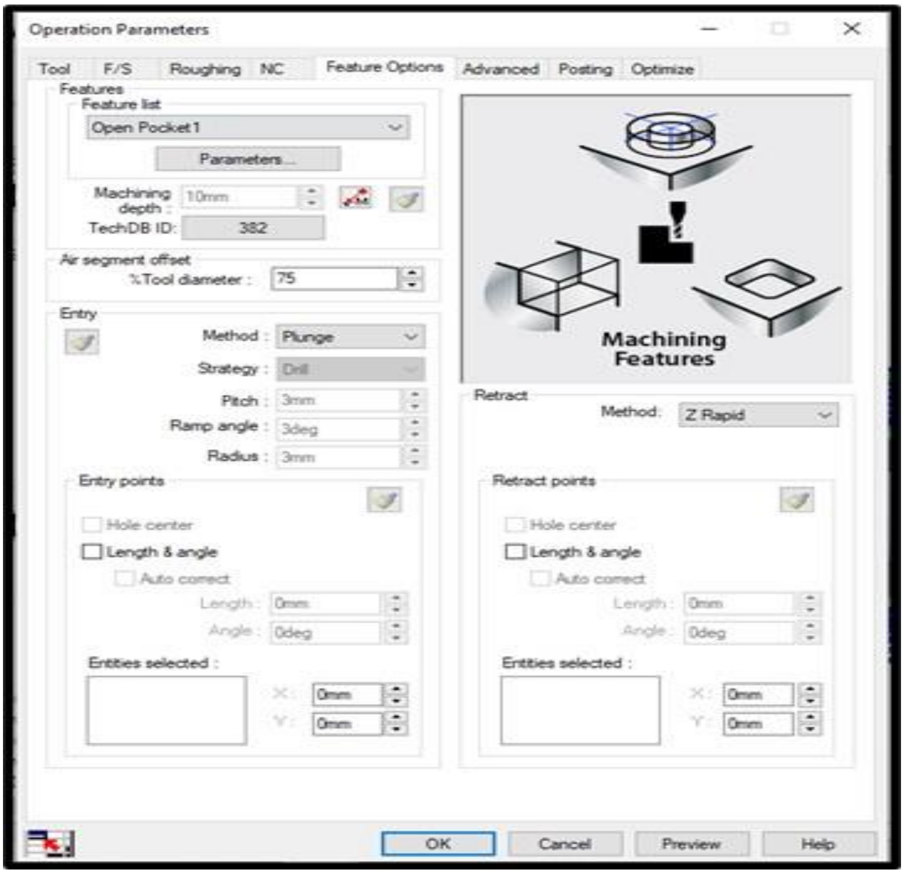


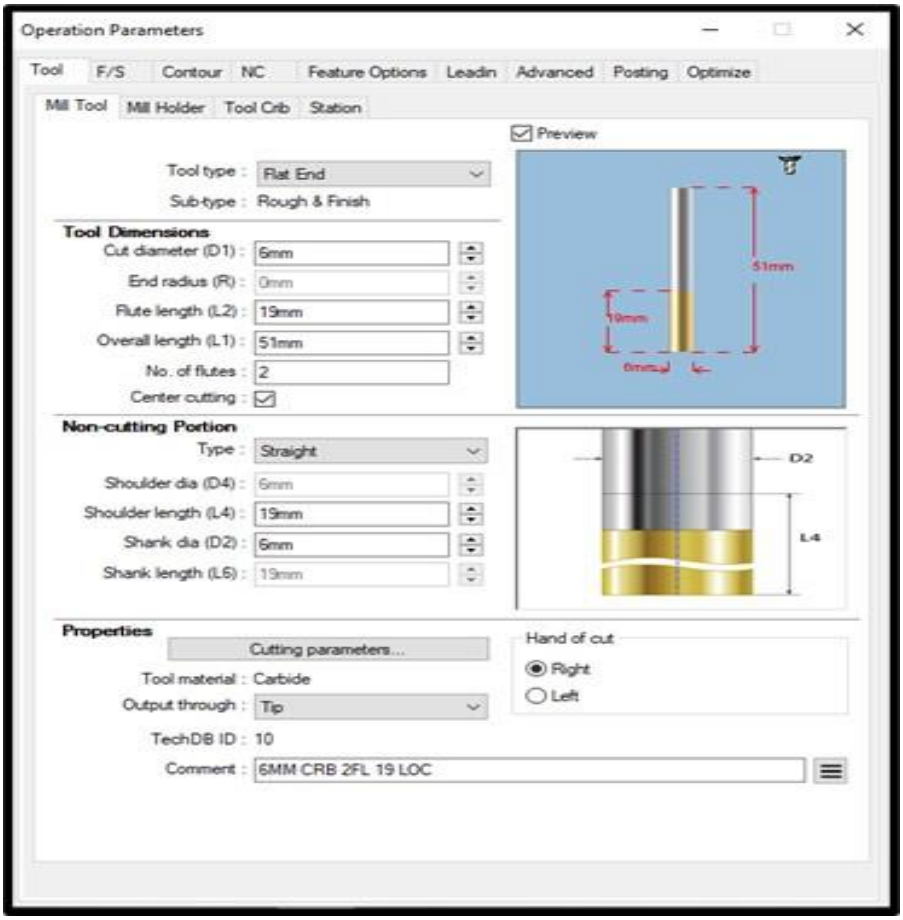
ROUGH AND COUNTOUR MILL:

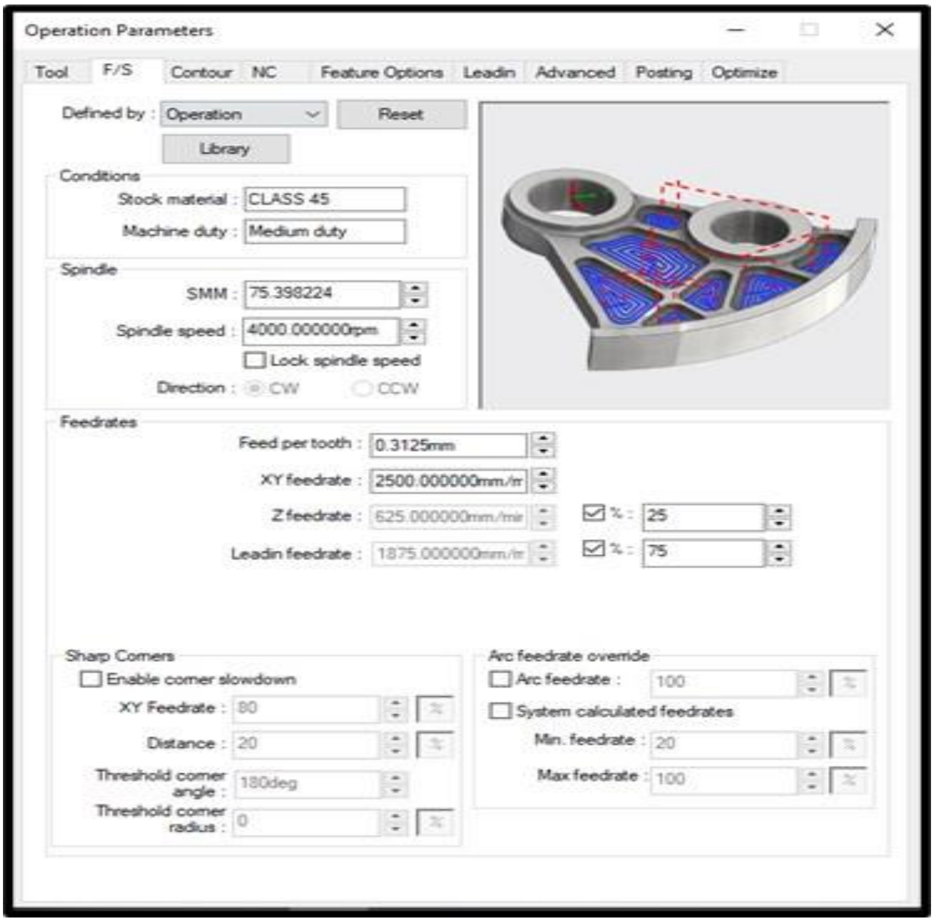


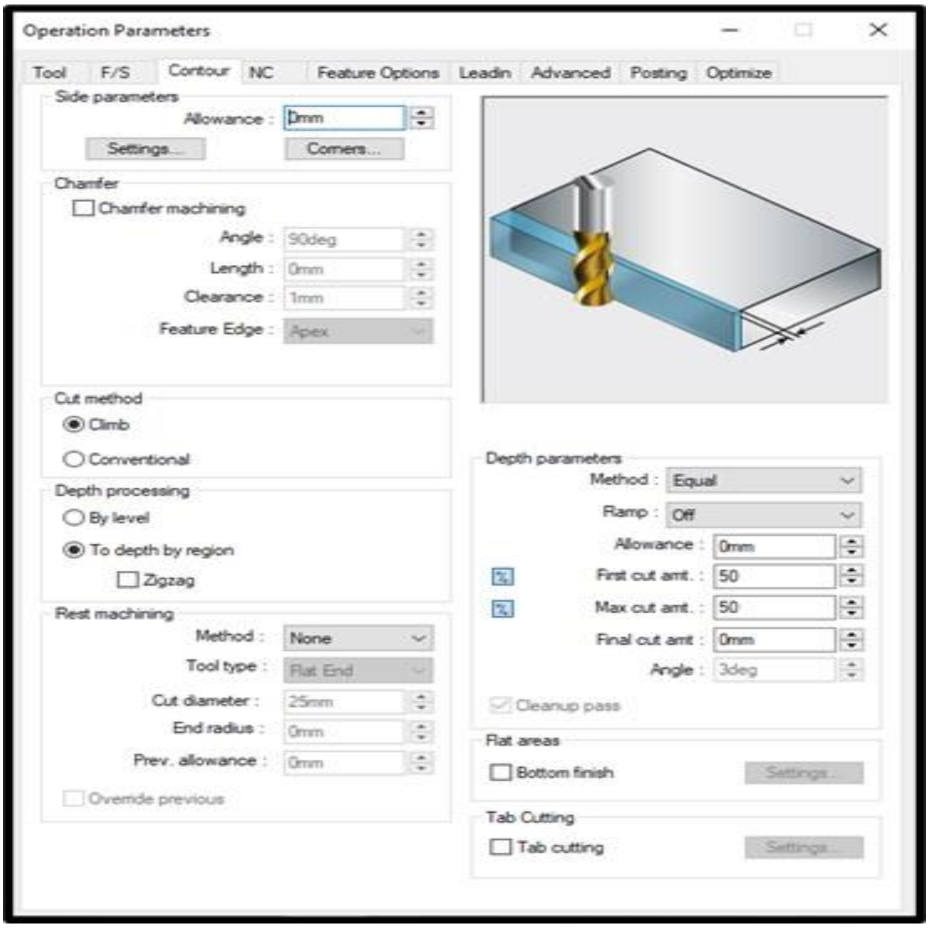

Operation Parameters
Tool
F/S
Roughing
NC
Feature Options
Advanced
Posting
Optimize
Defined by : Operation
Reset
Library
Conditions
Stock material : CLASS 45
Machine duty : Medium duty
Spindle
SMM : 78.539816
Spindle speed : 2500.000000rpm
Lock spindle speed
Direction : CW CCW
Feedrates
Feed per tooth : 0.3mm
VoluMill Technology Expert...
XY feedrate : 1500.000000mm/m
Z feedrate : 375.000000mm/min
% : 25
Leadin feedrate : 750.000000mm/min
% : 50
Sharp Corners
Enable corner slowdown
XY Feedrate : 80
Distance : 20
Threshold corner angle : 180deg
Threshold corner radius : 0
Arc feedrate override
Arc feedrate : 100
System calculated feedrates
Min. feedrate : 20
Max feedrate : 100
OK
Cancel
Preview
Help

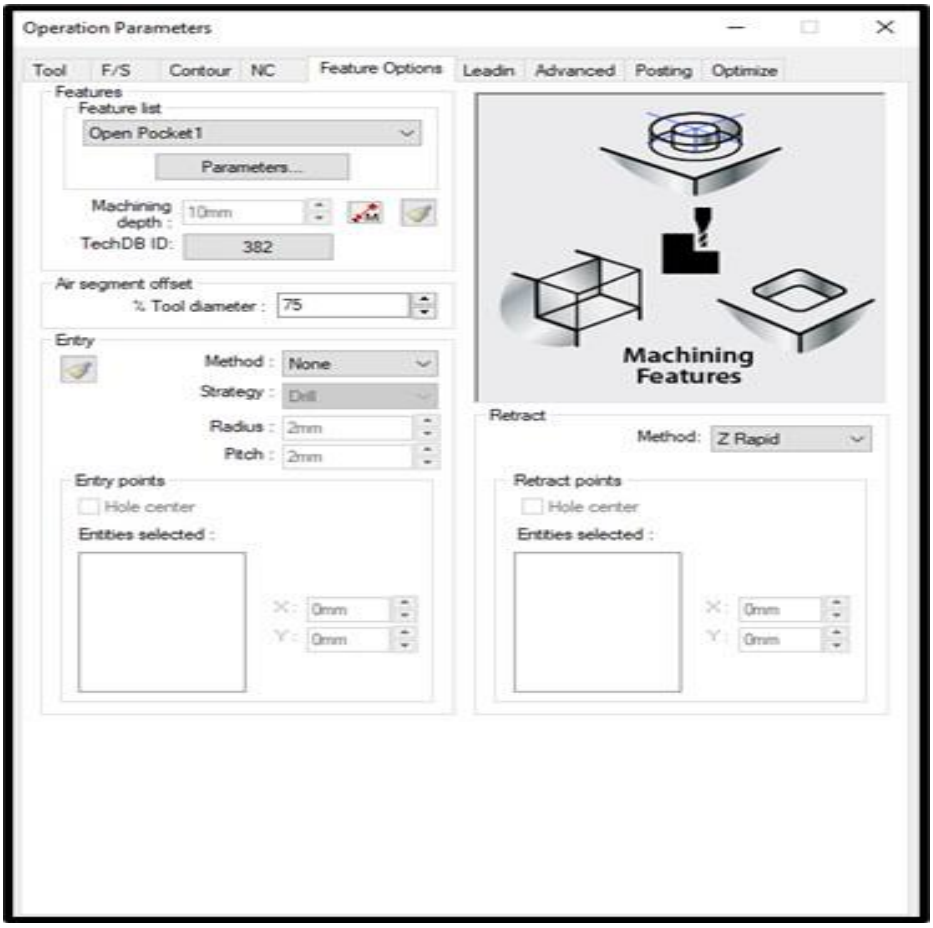

Operation Parameters
Tool
F/S
Roughing
NC
Feature Options
Advanced
Posting
Optimize
Pocketing
Pattern : Spiral Out
Cut angle : 0deg
Start corner : Top Right
Minimum diameter : 5mm
Cleanup pass
Stay at depth
Machine island top
Automatic Angle
Suppress top fillet
High Speed...
Side parameters
Allowance : 0.25mm
Stepover : 40
Wedge machining
Rest machining
Generate
Machine : No
Cut method
Climb
Conventional
Depth processing
By level
To depth by region
Depth parameters
Method : Equal
Bottom allowance : 0mm
Island allowance : 0mm
First cut amt. : 50
Max cut amt. : 50
Final cut amount : 0.25mm
Min cut amount : 0.1mm
OK
Cancel
Preview
Help


Operation Parameters
Tool
F/S
Roughing
NC
Feature Options
Advanced
Posting
Optimize
Features
Feature list
Open Pocket1
Parameters...
Machining depth :
10mm
TechDB ID:
382
Air segment offset
%Tool diameter :
75
Entry
Method :
Plunge
Strategy :
Drill
Pitch :
3mm
Ramp angle :
3deg
Radius :
3mm
Entry points
Hole center
Length & angle
Auto correct
Length :
0mm
Angle :
0deg
Entities selected :
X :
0mm
Y :
0mm
Machining Features
Retract
Method:
Z Rapid
Retract points
Hole center
Length & angle
Auto correct
Length :
0mm
Angle :
0deg
Entities selected :
X :
0mm
Y :
0mm
OK
Cancel
Preview
Help

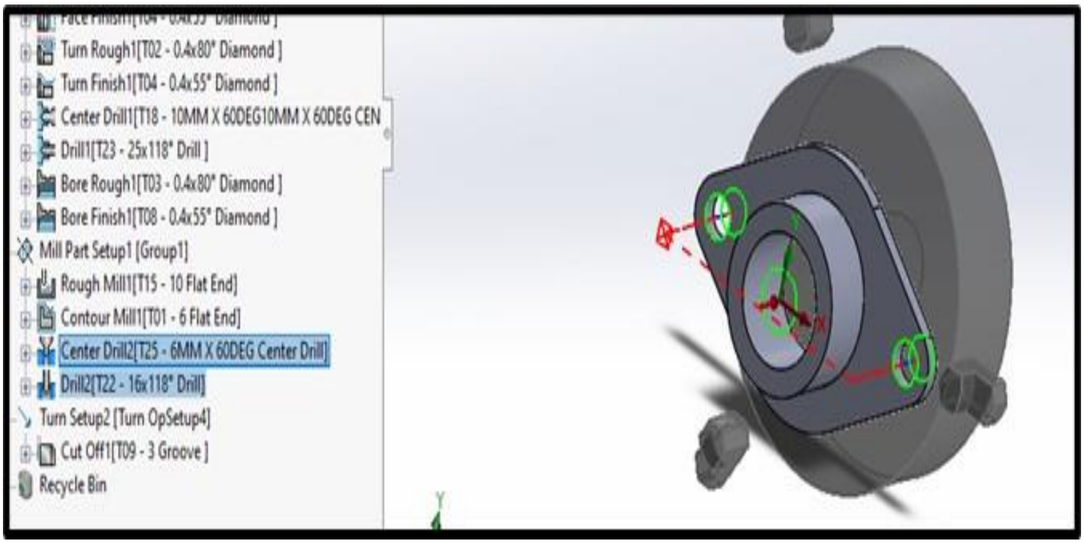

Operation Parameters
Tool
F/S
Contour
NC
Feature Options
Leadin
Advanced
Posting
Optimize
Mill Tool
Mill Holder
Tool Crib
Station
Preview
Tool type :
Flat End
Sub-type :
Rough & Finish
Tool Dimensions
Cut diameter (D1) :
6mm
End radius (R) :
0mm
Flute length (L2) :
19mm
Overall length (L1) :
51mm
No. of flutes :
2
Center cutting :
51mm
19mm
6mm
Non-cutting Portion
Type :
Straight
Shoulder dia (D4) :
6mm
Shoulder length (L4) :
19mm
Shank dia (D2) :
6mm
Shank length (L6) :
19mm
D2
L4
Properties
Cutting parameters...
Tool material :
Carbide
Output through :
Tip
TechDB ID :
10
Comment :
6MM CRB 2FL 19 LOC
Hand of cut
Right
Left

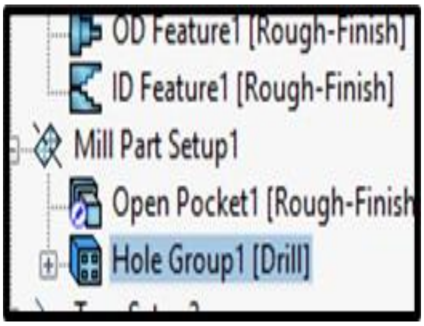

Operation Parameters
Tool
F/S
Contour
NC
Feature Options
Leadin
Advanced
Posting
Optimize
Defined by :
Operation
Reset
Library
Conditions
Stock material :
CLASS 45
Machine duty :
Medium duty
Spindle
SMM :
75.398224
Spindle speed :
4000.000000rpm
Lock spindle speed
Direction :
CW
CCW
Feedrates
Feed per tooth :
0.3125mm
XY feedrate :
2500.000000mm/m
Z feedrate :
625.000000mm/mi
%:
25
Leadin feedrate :
1875.000000mm/m
%:
75
Sharp Corners
Enable corner slowdown
XY Feedrate :
80
Distance :
20
Threshold corner angle :
180deg
Threshold corner radius :
0
Arc feedrate override
Arc feedrate :
100
System calculated feedrates
Min. feedrate :
20
Max feedrate :
100


Operation Parameters
Tool
F/S
Contour
NC
Feature Options
Leadin
Advanced
Posting
Optimize
Side parameters
Allowance :
0mm
Settings...
Corners...
Chamfer
Chamfer machining
Angle :
90deg
Length :
0mm
Clearance :
1mm
Feature Edge :
Apex
Cut method
Climb
Conventional
Depth processing
By level
To depth by region
Zigzag
Rest machining
Method :
None
Tool type :
Flat End
Cut diameter :
25mm
End radius :
0mm
Prev. allowance :
0mm
Override previous
Depth parameters
Method :
Equal
Ramp :
Off
Allowance :
0mm
First cut amt. :
50
Max cut amt. :
50
Final cut amt :
0mm
Angle :
3deg
Cleanup pass
Flat areas
Bottom finish
Settings...
Tab Cutting
Tab cutting
Settings...

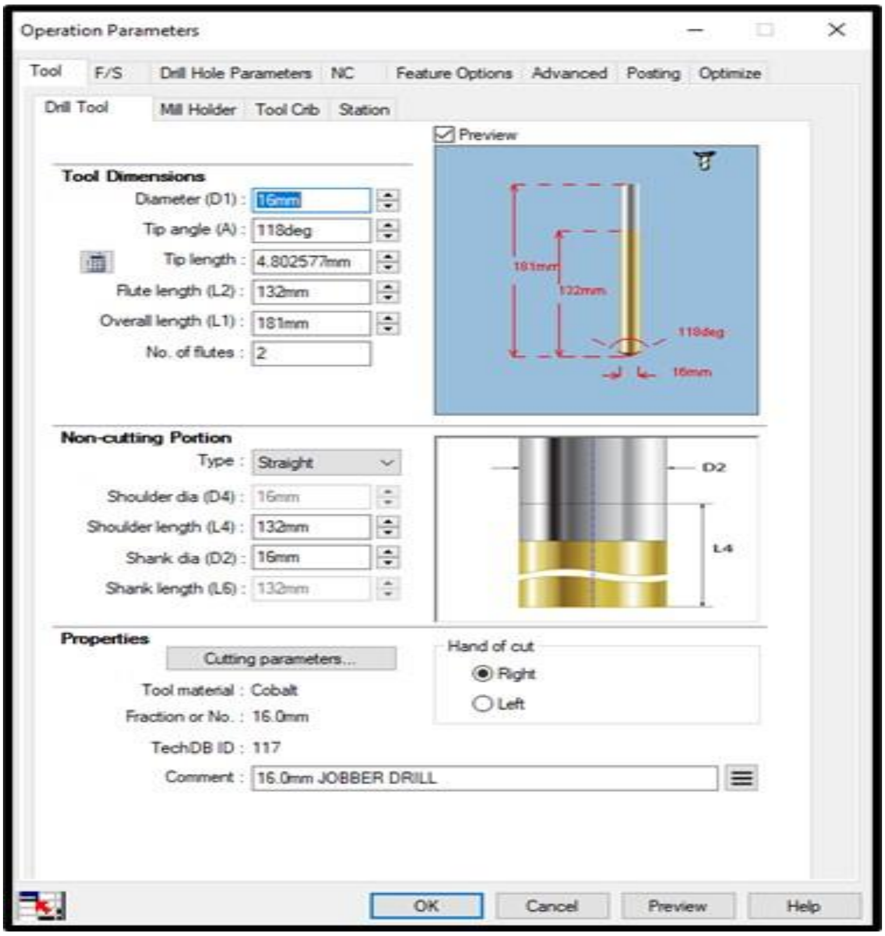


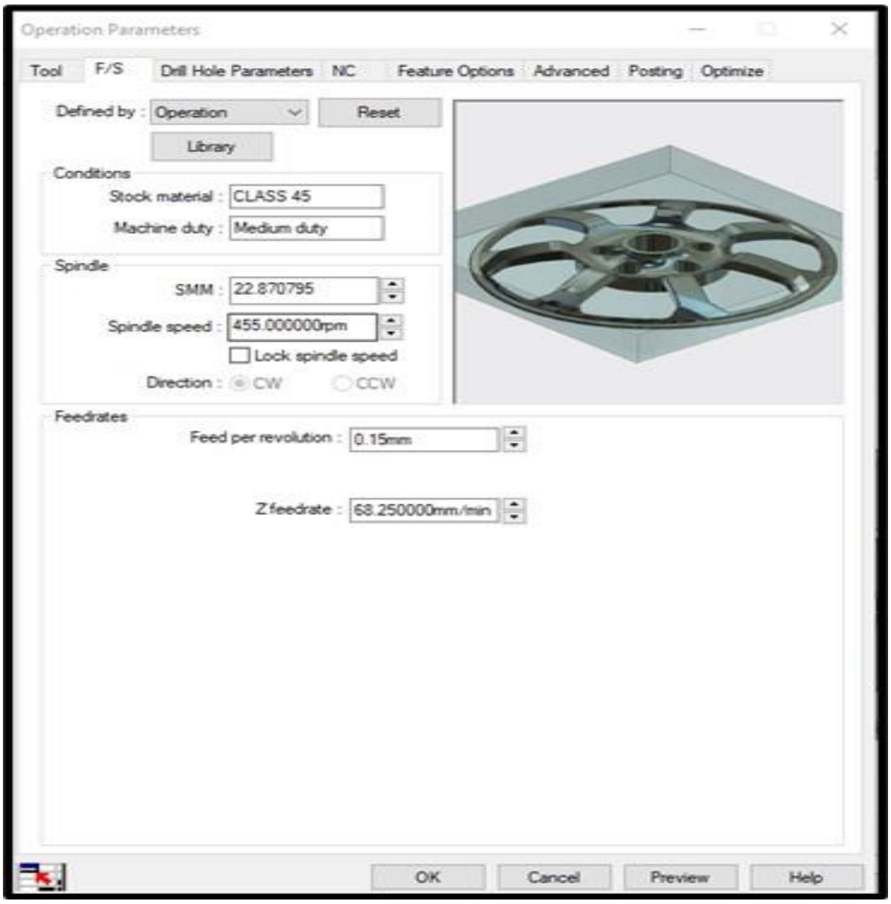
8.2.2 Hole group operation:





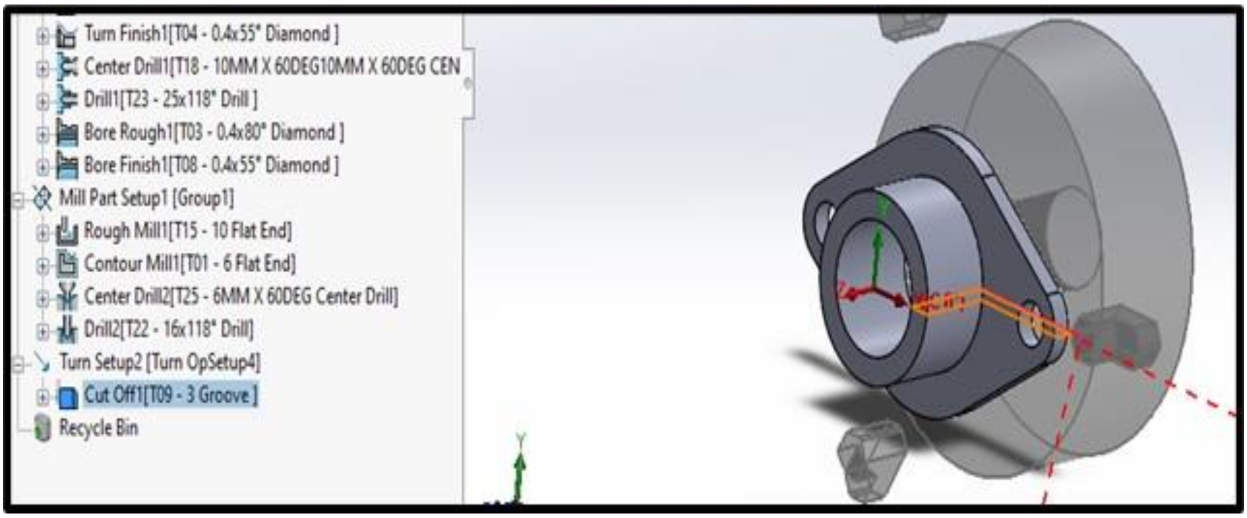
drill operations:

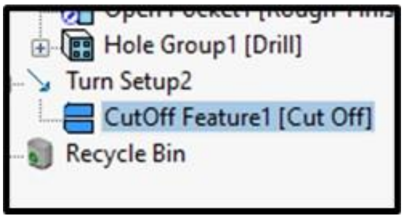




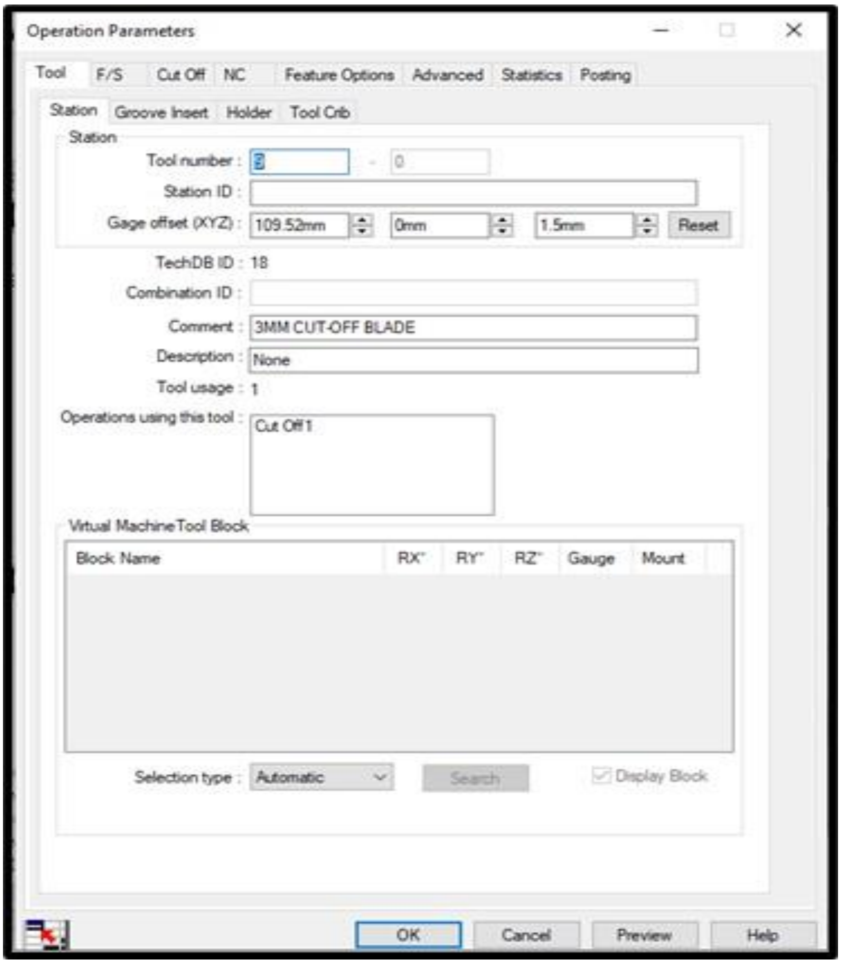
Turn setup:

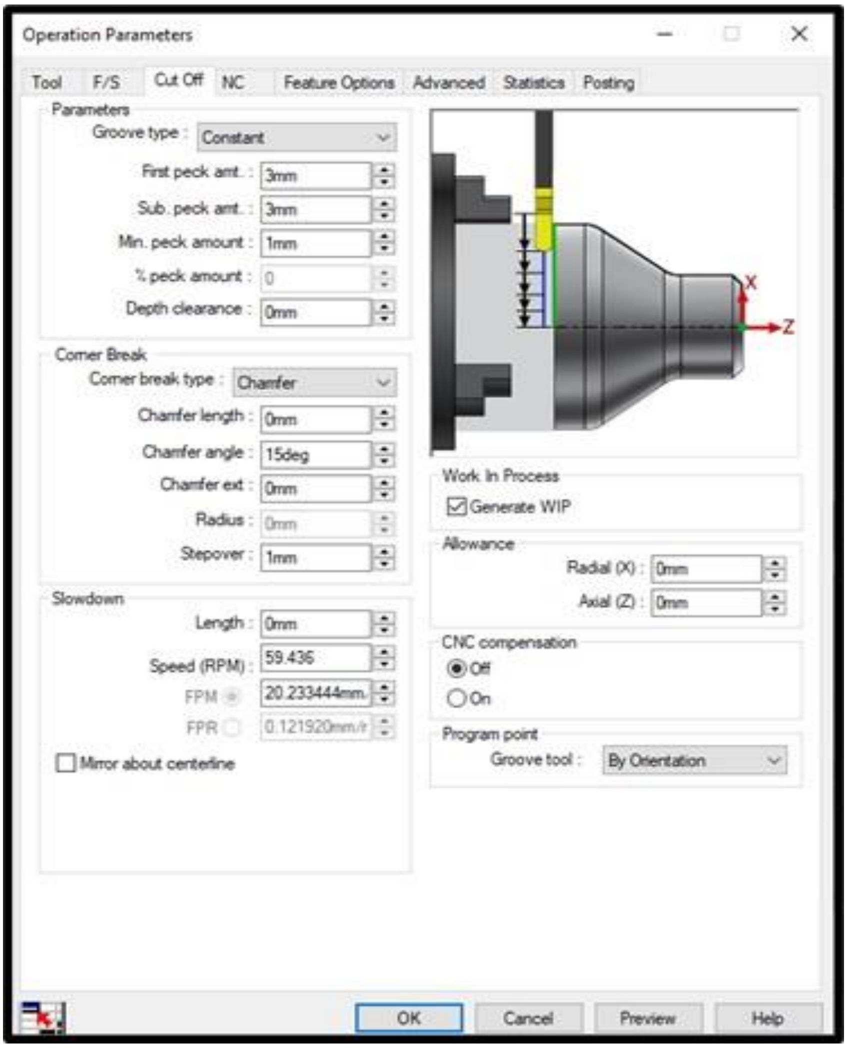
8.3 TURN OPRATIONS:





8.3.1 CUT OFF OPERATIONS:

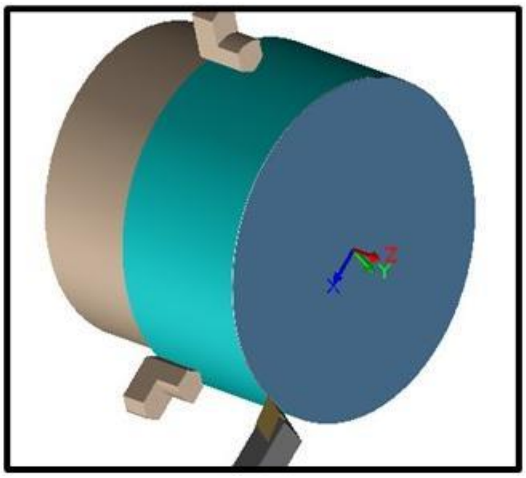


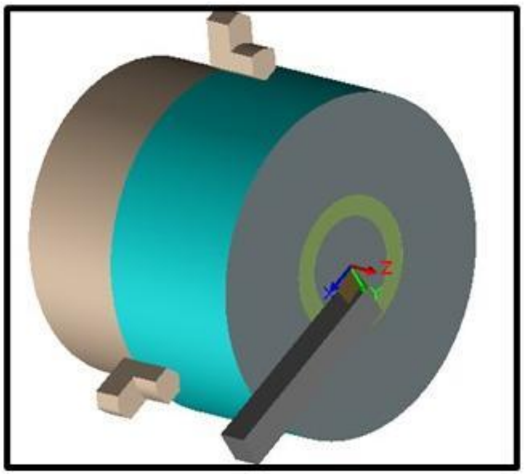

Operation Parameters
Tool
F/S
Cut Off
NC
Feature Options
Advanced
Statistics
Posting
Parameters
Groove type : Constant
First peck amt. : 3mm
Sub. peck amt. : 3mm
Min. peck amount : 1mm
% peck amount : 0
Depth clearance : 0mm
Corner Break
Corner break type : Chamfer
Chamfer length : 0mm
Chamfer angle : 15deg
Chamfer ext : 0mm
Radius : 0mm
Stepover : 1mm
Slowdown
Length : 0mm
Speed (RPM) : 59.436
FPM 20.233444mm
FPR 0.121920mm/r
Mirror about centerline
X
Z
Work In Process
Generate WIP
Allowance
Radial (X) : 0mm
Axial (Z) : 0mm
CNC compensation
Off
On
Program point
Groove tool : By Orientation
OK
Cancel
Preview
Help


Tool
F/S
Cut Off
NC
Feature Options
Advanced
Statistics
Posting
Defined by : Operation
Library
Reset
Conditions
Stock material : CLASS 45
Machine duty : Medium duty
Spindle
Mode : SMM
Surface Speed : 60.000000m/min
Spindle Speed : 165.956723rpm
Direction : CW
CCW
Override spindle direction :
Feed
FPM 25.000000mm/min
FPR 0.150000mm/rev

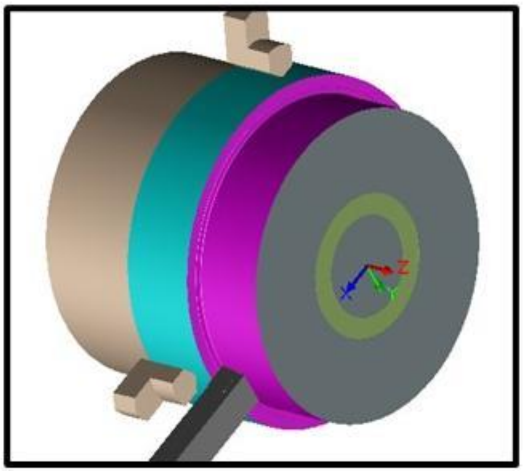
8.3.2 Simulation:

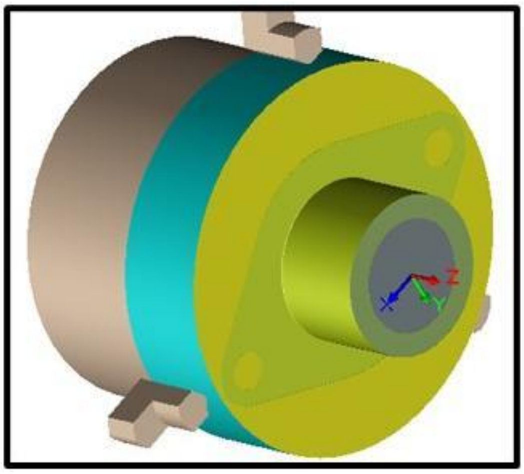
face feature:





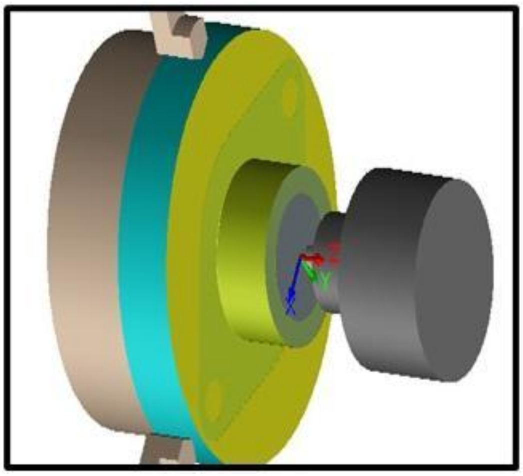
OD feature

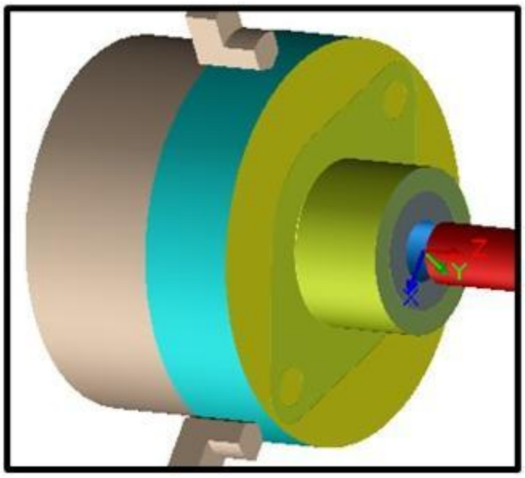




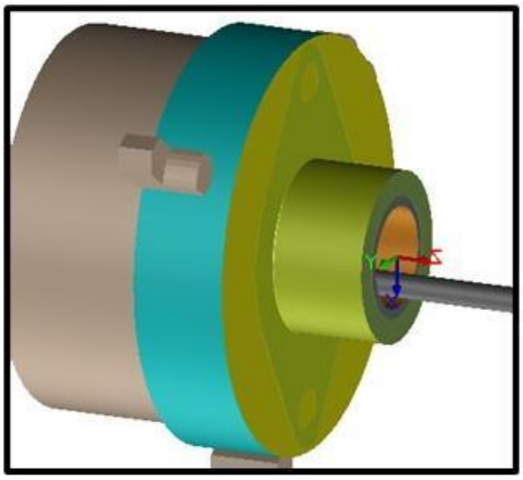
ID feature

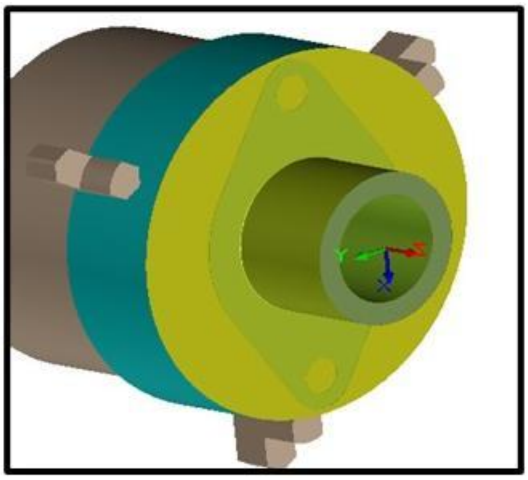
center drill and drill



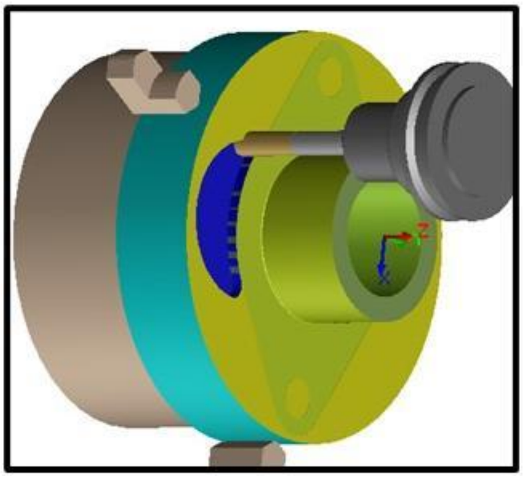


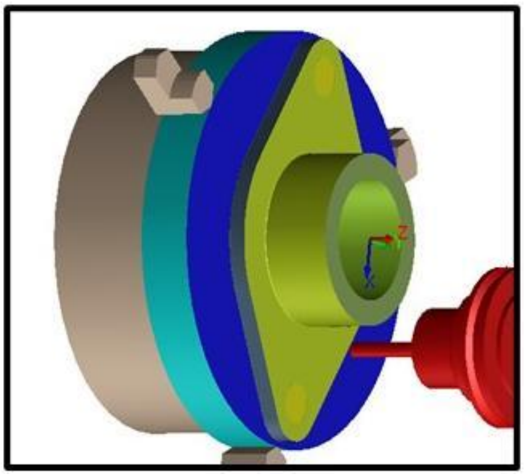
bore operation



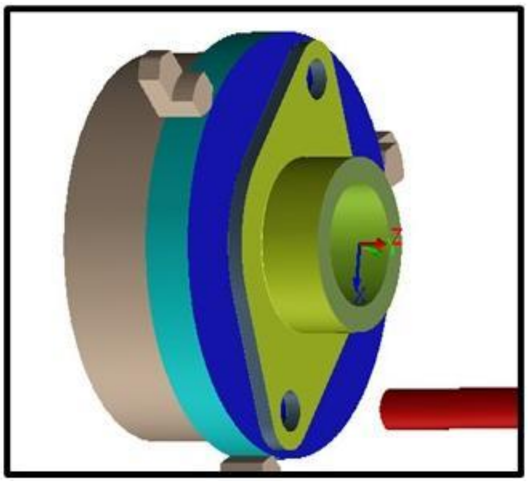


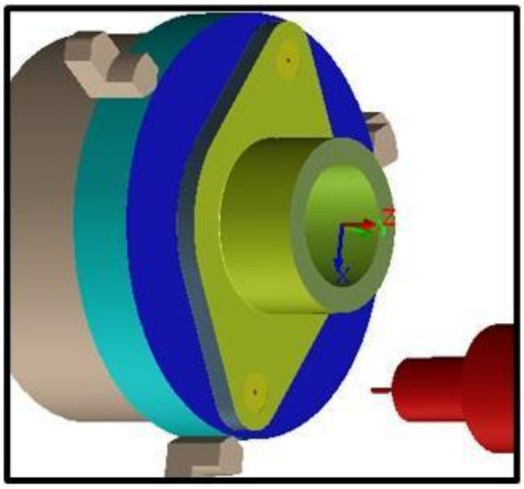
open pocket operation



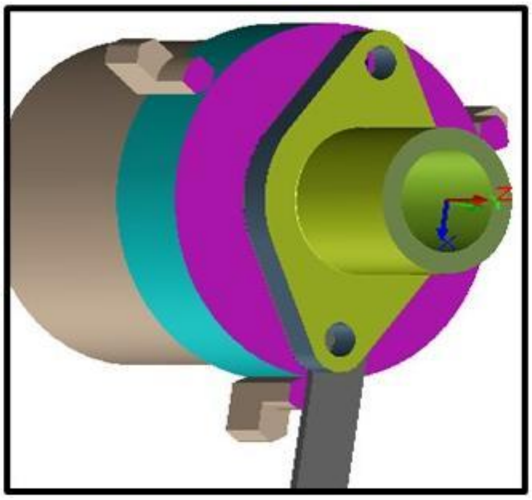


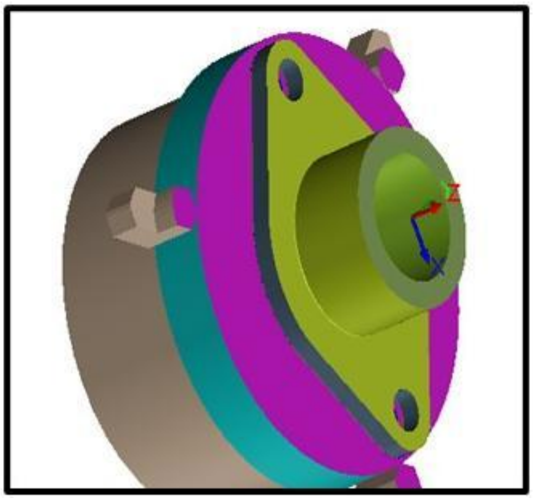
center drill and drill





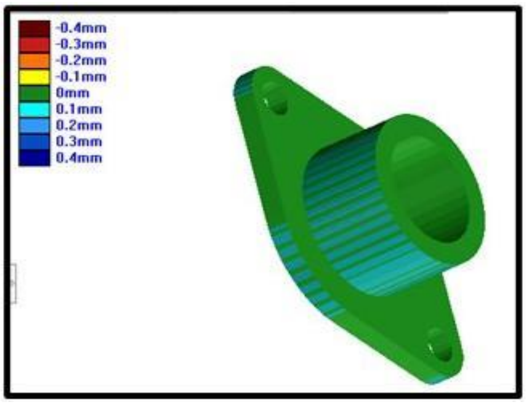
cut off feature:

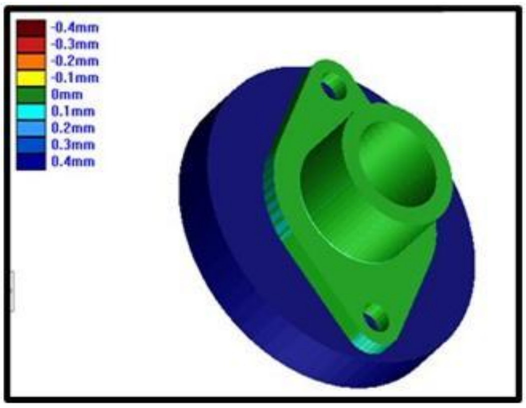




9 Surface quality comparison

between 3 axes milling and 4 axis mill-turn machine

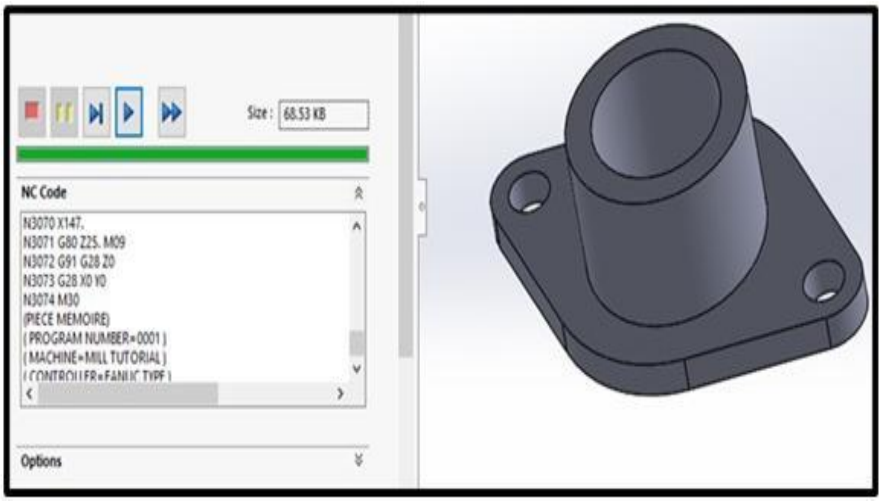


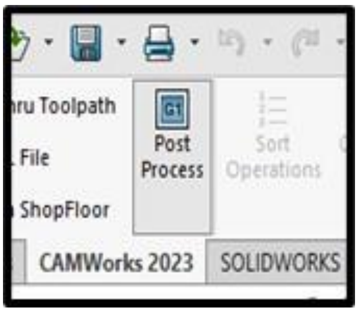


3 axis mill machine 4 axis mill-turn machine

3 axis machines have lower machining efficiency for round part and the tool access is limited for cylindrical shape may show step marks or tool lines on the surface meanwhile the 4axis mill-turn has an excellent tool access and make better surface quality for cylindrical shape

10 G-code extraction:





Conclusion:

This work presents the production of a machining range for a part on a CNC machine, by designing the part using Solidworks software and simulating its manufacturing using CamWorks software. The thesis presents step by step the design and manufacturing simulation process. The development of the different stages demonstrates the complexity of the task and the details of the phases undertaken to successfully complete this work and achieve the set objective.

The internship carried out at Ferrovial, which has CNC machines, is relevant to the proposed subject. Theoretically, the set objective of producing the range using CAD and CAM software is achieved. However, manufacturing the part on the machine requires mastery of the G-Code software package and the acquisition of material piece, which are not included in the agreed work plan. This is possibly left in perspective.

The principal contribution of this work is the application of numerical control to the manufacture of a complex part with a multiform geometry, designed on SolidWorks and machined and manufactured on Camworks. This enabled us to present the simulation of the virtual production of the part.

A presentation of the benefits of simulating the manufacture of a part before deciding whether or not to produce it on the machine. This approach, based on industrial simulation software, enables us to foresee more than one scenario for producing the part, and from there to opt for the scenario that allows us to choose the most cost-effective means, and save time and money

In conclusion, this work shows the interest of manufacturing a part using digital processes, and the use of industrial software, which constitutes a novelty in our training.

Bibliographic References

[1] Groover, M. P. (2020). Automation, Production Systems, and Computer-Integrated Manufacturing (5th ed.). Pearson Education

[2] https://leadrp.net/blog/13-types-of-machining-processes/

[3]https://www.google.dz/books/edition/FUNDAMENTALS_OF_MODERN_MANUFACTURING/U9g2EAAAQBAJ?hl=fr&gbpv=1&printsec=frontcover

[4] https://en.wikipedia.org/wiki/Computer_numerical_control

[5] Prof. Olivier de Weck ©January 6, 2005 Engineering Design and Rapid Prototyping Lecture

[6]https://academy.titansofcnc.com/files/Fundamentals_of_CNC_Machining.pdf

[7]https://www.google.dz/books/edition/CNC_Programming_Handbook/JNnQ8r5merMC?hl=fr&gbpv=1&dq=Fundamentals+of+CNC+Machining&pg=PA4&printsec=frontcover

[8] Farid ASMA 'Introduction à la commande numérique', Notes de cours, 2007/2008.

[9] MorchedDallali © May 2010Concept of CNC Machine / Bachelor thesis /Centers SLOVAK UNIVERSITY OF TECHNOLOGY IN BRATISLAVA Faculty of Mechanical Engineering

[10] Mach motion Specializing in CNC Automation and Motion Control G & M Code Copyright © 2016, Mach Motion All rights reserved

[11] LÁSZLÓ KÁTAI, ÓU COPYRIGHT © 2012-2017 CAD Book,

[12] Fundamentals of CNC Machining Copyright 2014 Autodesk, Inc.

[13]https://my.solidworks.com/solidworks/guide/SOLIDWORKS_Introduction_EN.pdf

[14]https://www.google.dz/books/edition/Virtual_Machining_Using_CAMWorks_2023/CrfLEAAAQBAJ?hl=fr&gbpv=1