

A
DISSERTATION REPORT
on
DEVELOPMENT OF FEATURE RECOGNITION AND FEATURE BASED
MACHINING ALGORITHM FOR THREE AXIS CNC LATHE MILL

Submitted in partial fulfillment of the requirements for the award of degree of

MASTER OF ENGINEERING
IN
CAD/CAM & ROBOTICS

Submitted by:
ATUL KHOSLA
ROLL NO: 801181004

Under the guidance of
Mr. A. S. JAWANDA
Associate Professor, MED



MECHANICAL ENGINEERING DEPARTMENT
THAPAR UNIVERSITY
PATIALA-147004, INDIA
JULY- 2013

CERTIFICATE

I hereby declare that the dissertation entitled "**DEVELOPMENT OF FEATURE RECOGNITION AND FEATURE BASED MACHINING ALGORITHM FOR THREE AXIS CNC LATHE MILL**" is an authentic record of my own work carried out in partial fulfillment of the requirement for the award of degree of **Master of Engineering in CAD/CAM & Robotics** under the guidance of **Mr. A.S JAWANDA**, Associate Professor, MED, Thapar University, Patiala.

Date: 15/07/2013

Atul Khosla
ATUL KHOSLA
801181004

This is to certify that the above statement made by the candidate is correct and true to the best of my knowledge.

A.S. Jawanda
Mr. A.S JAWANDA
Associate Professor

Mechanical Engineering Department
Thapar University, Patiala

Countersigned By:-

S.K. Mohapatra
Dr. S. K. Mohapatra
Senior Professor and Dean of Academic Affairs
Mechanical Engineering Department
Thapar University, Patiala

Ajay Batish
Dr. Ajay Batish
Professor and Head
Mechanical Engineering Department
Thapar University, Patiala

ACKNOWLEDGEMENT

Though only my name appears on the cover of this dissertation, a lot of people have contributed to its completion. I owe my gratitude to all those people who have made this dissertation possible and because of whom my post graduate experience has been one that I will cherish forever.

My deepest gratitude is to my supervisor **Mr. A.S Jawanda**. I have been amazingly fortunate to have an advisor who gave me the freedom to explore on my own and at the same time the guidance to recover when my steps faltered. He taught me how to question thoughts and express ideas. I am grateful to him for holding me to high research standards and enforcing strict validations for each research result and thus teaching me how to do research.

I am also thankful to all the staff members of mechanical engineering department who are directly or indirectly concerned within the completion of my thesis.

Many friends have helped me stay sane through these years. Their support and care helped me overcome setbacks and keep me centered on my study. I greatly worth their friendship and I deeply appreciate their belief in me.

Most significantly, none of this would have been possible without the love and patience of my family. I would like to express my heart felt gratitude to my family particularly my younger brother for his support.

ATUL KHOSLA

ABSTRACT

Over the past few years, manufacturing companies have had to deal with an increasing demand for feature-rich products. There is also a lot of work done on sculptured parts using lathe-mill CNC machines. The present state of the art generates scallops on the part surface. There is a need to finish these scallops using feature recognition and contoured machining of these scallops. A CAD package provides the information about the part geometry saved as stereo lithography (STL) file, which is taken as an input to the tool path generation algorithm. The algorithm calculates the cutter location (CL) points based on various parameters given as input like depth of cut for roughing pass, side step for roughing and finishing operation. Various checks have been applied in the algorithm to find the best suitable point at particular tool position. The tool path for a lathe mill CNC machine follows a helical path to machine the surface, with cutter profile as ball nose end mill with varying radius for roughing and finishing passes, which can be changed by the user. In this work features like cavities, grooves, and protrusions of any shape are recognized based on the roughing and finishing files generated by the computer algorithm. The tool path planning algorithm is developed for skipping the features in the first go and then these features are separately approached for machining them in the last.

The testing of tool path is carried out on a Custom Simulator ‘Tool Sim’ and has been validated by comparing the actual part geometry saved as STL file with the final machined geometry output taken as obj file from ‘Tool Sim’ into a CAD package like CREO.

ABBREVIATIONS

NC:	Numeric Controlled
MCU:	Machine Control Unit
CNC:	Computer Numeric Controlled
CAD:	Computer Aided Design
CAM:	Computer Aided Manufacturing
APT :	Automatic Programmed Tools
ASCII:	American standard code of information interchange
CL:	Cutter Location
CC:	Cutter Contact
DMAC:	Direct Machining and Controlling
STEP:	Standard for the Exchange of Product Model Data
IGES:	Initial Graphics Exchange Specification
STL:	Stereo Lithography
NURBS:	Non-Uniform Rational B-Splines
B-Rep:	Boundary Representation
CSG:	Constructive Solid Geometry

Table of Contents

CHAPTER 1	1
INTRODUCTION	1
1.1 CONVENTIONAL NC/CNC MACHINING SYSTEM	4
1.2 DIRECT MACHINING AND CONTROLLING [DMAC]	7
1.3 PC BASED NC MACHINING SYSTEM	8
1.4 SINGLE CONTROLLED AXIS LATHE MILL [SCALM]	9
1.5 PROPOSED WORK	10
CHAPTER 2	11
LITERATURE REVIEW	11
2.1 TOOL PATH GENERATION METHODS [6] [10]	11
2.2 TOOL PATH PLANNING STRATEGY [7].....	14
2.3 TOOL POSITIONING METHODS	16
2.4 FEATURE RECOGNITION AND TOOL PATH PLANNING FOR THREE AXIS MILLING LATHE	17
CHAPTER 3	18
PROBLEM IDENTIFICATION AND METHODOLOGY FOR SOLUTION	18
3.1 PROBLEM IDENTIFICATION	18
3.2 TOOL PATH PLANNING FOR THREE AXIS CNC MILLING LATHE	19
3.2.1 SPECIFICATION OF 3-AXIS NC MILLING LATHE MACHINE [W4]	19
3.2.2 PARAMETERS REQUIRED FOR NC TOOL PATH PLANNING	20
3.2.3 BALL DROP METHOD.....	24
3.2.4 TOOL PATH PLANNING FOR ROUGHING.....	27
3.2.5 TOOL PATH PLANNING FOR FINISHING	27
3.3 IDENTIFICATION OF GEOMETRY AND FEATURE RECOGNITION BASED TOOL PATH PLANNING	28
3.3.1 IDENTIFICATION OF A CYLINDRICAL GEOMETRY	28
3.3.2 IDENTIFICATION OF A CONICAL GEOMETRY	30
3.3.3 IDENTIFICATION OF FEATURES IN THE CYLINDRICAL AND CONICAL GEOMETRIES	31
3.4 METHODOLOGY FOR SOLUTION.....	32

3.5 FLOW CHART FOR SOLUTION	33
CHAPTER 4	35
RESULTS AND VALIDATION	35
4.1 VALIDATION OF CYLINDER	36
4.2 VALIDATION OF CYLINDER WITH GROOVE AS A FEATURE.....	37
4.3 VALIDATION OF CYLINDER WITH PROTRUSION	38
4.4 VALIDATION OF CYLINDRICAL-CONICAL GEOMETRY WITH VARIOUS FEATURES	39
CHAPTER 5	40
CONCLUSIONS AND FURTHER SCOPE	40
5.1 SCOPE FOR FURTHER STUDY	40
REFERENCES	41
WEB REFERENCES.....	43

List of Figures

Figure 1: ROUGHING CUT [W1].....	2
Figure 2: CONTOUR FINISHING [W2].....	2
Figure 3: PENCIL CUTTING [W3].....	3
Figure 4: CNC TURNING CENTER [W5].....	6
Figure 5: NEW CNC PARADIGM[6].....	8
Figure 6: ISO PARAMETRIC MESH [9]	11
Figure 7: THE OFFSET SURFACE WITH SELF INTERSECTION [15].....	12
Figure 8: PBG2048 CNC LATHE MILL.....	19
Figure 9: STL FILE FORMAT FOR A CYLINDER.....	21
Figure.10: COUNTER CLOCKWISE/CLOCKWISE ORDERS OF STL FACETS [4].....	22
Figure 11: VERTEX TO VERTEX RULE.....	22
Figure 12: A CYLINDRICAL GEOMETRY WITH A POCKET.....	28
Figure 13: IDENTIFICATION OF CYLINDRICAL GEOMETRY.....	29
Figure 14: CONICAL GEOMETRY WITH FEATURES.....	30
Figure 15: IDENTIFICATION OF CONICAL GEOMETRY.....	30
Figure 16: CYLINDRICAL-CONICAL GEOMETRY WITH FEATURES.....	31
Figure 17: BALL NOSE END MILL.....	35
Figure 18: THE .OBJ FILE OF A CYLINDRICAL PART.....	36
Figure 19: STL FILE OF CYLINDER WITH GROOVE.....	37
Figure 20: OBJ FILE FOR VALIDATION.....	37
Figure 21: CYLINDER WITH GROOVE MACHINED IN THE END IN TOOL SIM.....	38
Figure 22: AN OBJ FILE FOR CYLINDER WITH PROTRUSION.....	38
Figure 23: VALIDATION OF ALGORITHM IN TOOL SIM.....	39
Figure 24: OBJ FILE FOR CYLINDRICAL-CONICAL PART WITH FEATURES.....	39

CHAPTER 1

INTRODUCTION

Computer aided design (CAD) and Computer Aided Manufacturing (CAM) technology has been around for decades to assist us with efficient design and manufacturing of complex sculptured surfaces components which is an expensive and time consuming task. With CAD/CAM software, Computer Numeric Controlled (CNC) machines can machine intricate components with highest dimensional accuracy. CAD/CAM software simplifies the programming of CNC machines and helps improve their utilization. The complete CNC machining process is divided into three stages:

- Design phase
- Tool Path planning stage
- Verification and manufacturing stage

In the Design stage, a 3D Computer Aided Design (CAD) model for the design surface is created using the modeling capabilities of a CAD package. In the Tool Path Planning stage, the path for the cutter is planned to produce the component from the CAD model within acceptable dimensional tolerances. In this stage, cutter location data that can be translated into G-Codes that can be understood by CNC machines is generated. In the Verification stage, the generated tool path is verified and checked in a simulator for machining errors or flaws before actually manufacturing the component. In the Manufacturing stage, the actual component is loaded on the machine and the machine tool will read the G codes generated during tool path planning stage to produce final product in its intended application.

For complex surfaces the component is further machined in three sub-stages: Rough machining, finished machining and manual finishing. Conventionally, complex surfaces are machined using ball nose end milling cutter on 3-axis milling machines. CNC milling machines direct the tool via a computerized controller.

Tool path planning is a fundamental task in CNC machining. Tool path planning is the motion planning of machine tool. Tool path gives the trajectory followed by machine tool. Tool path gives information about surface topology because of scallops which are developed during machining. Planning is needed to execute both rough-cuts and finish cuts. In the rough-cut stage, the main goal is to remove material in such a manner that MRR (metal removal rate) should be

maximized. The rough machining is done by machining volumes of material in a slice by slice manner. In finish cutting, producing the desired surface finish and accuracy is the primary driving factor.

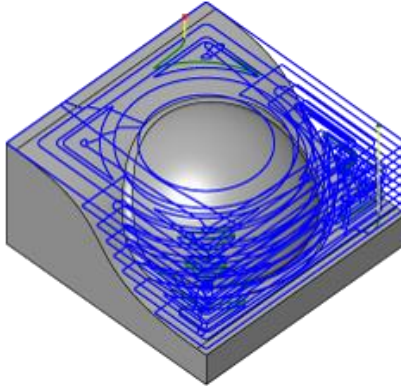


Figure 1: ROUGHING CUT [W1]

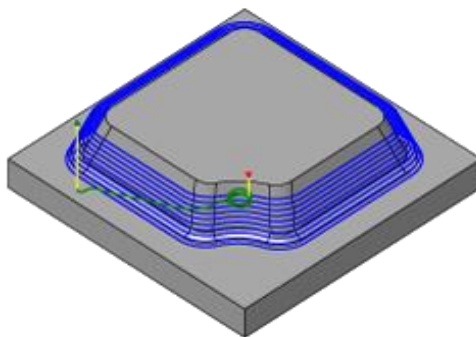


Figure 2: CONTOUR FINISHING [W2]

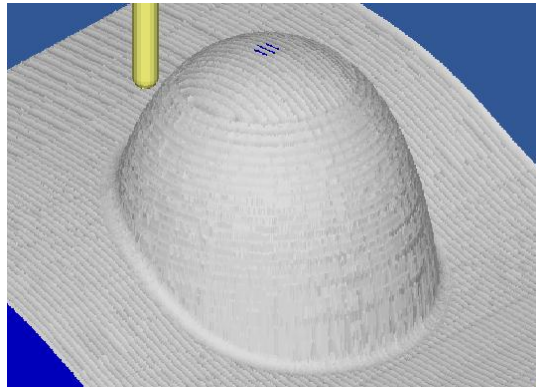


Figure 3: PENCIL CUTTING [W3]

Based on the survey, Gray et al.[1], Bedi et al.[2] and H. T. Yau et.al [3] proposed a model that can be used to produce parts directly from the model. In this approach process planning and CAM module are clubbed in new CNC manufacturing module. Also there is no need of post processor as now G & M codes are not being generated which was a conventional approach for long time and the tool path is generated directly in CAM package. The simulation is kept optional for the time being.

1.1 CONVENTIONAL NC/CNC MACHINING SYSTEM

Numerical control (NC) refers to the automation of machine tools that are operated by abstractly programmed commands encoded on a storage medium, as opposed to manually controlled via hand wheels or levers, or mechanically automated via cams alone. The first NC machines were built in the 1940s and '50s, based on existing tools that were modified with motors that moved the controls to follow points fed into the system on punched tape.

The operation of conventional NC control system consists of following basic components:

- Program of instructions
- Controller unit, also called machine control unit(MCU)
- Machine tool or other controlled process

Due to advancement in technology sooner NC machines were replaced by CNC machines. In modern Computer Numerical Controlled (CNC) machine specifically refers to machine tools that are computer controlled to machine complex parts. The first CNC machine was a continuous path numerically-controlled milling machine developed by the MIT Servomechanisms Laboratory in the early 1950s. CNC machines are blind in the way that they would indisputably follow the input of the part program disregarding whether the program would lead to inappropriate machine movements. CNC machines are highly automated using CAD/CAM programs. The programs produce a computer file that is interpreted to extract the commands needed to operate a particular machine via a postprocessor, and then loaded into the CNC machines for production. The CNC machining paradigm has not changed since its inception. The machine motions are dictated by the part program that contains the tool path. Tool path generation is done prior to the machining process and the tool path is usually generated offline with a commercial CAM package. Different commercial CAM packages like MASTER CAM, EDGE CAM use different tool path generation algorithms depending on the final machining accuracy and required surface finish. The CAD model data and the machining parameters such as the choice of cutting tool, feed rate, and spindle speed are generally the inputs to the tool path generator and are supplied by the operator. The tool path generator follows the machine tool trajectory and computes a list of tool positions for the machine to interpolate between. Most machines offer linear and circular interpolation capability. The tool positions are considered as the instantaneous tool locations with respect to the machine coordinate system or work

coordinate system. Commands that will instruct a specific machine to follow the tool paths that were created during the CAD/CAM phase of design.

The architecture of CNC machines is generally divided into three layers:

- The User Interface
- The Machine Controller
- The Machine Interface

The User Interface

The user interface of the CNC machines is the front-end of the system on the CNC controller console that allows the operators to interact with the machines. The user interface displays the in process machine status and the part program. It also allows the operators to edit and verify the part programs prior to the actual machining process to prevent any errors in the tool path. The part programs are generally generated manually or with Computer Aided Manufacturing (CAM) software packages by using the cutting parameters and the part's Computer Aided Design (CAD) model data. This part program contains the machine movement instructions usually in a standard machine tool language such as ISO G & M codes. The initial machine setup such as entering the tool compensation and the work piece origin are done through the user interface. Simulations of the machining process could also be displayed on the user interface of some modern CNC machines.

The Machine Controller

The machine controller on CNC machines is the kernel of the system. The machine controller is aware of the in process machine status and the part coordinate systems to determine the current tool and part locations. It is also a motion control system that interprets the part program and commands the motions of the hardware devices during production. The machine controller is commonly custom-built for the CNC machine, which accessing the underlying architecture of the controller for modifications is difficult.

The Machine Interface

The machine interface layer is considered as the execution level of the CNC machines. The machine interface consists of the machine motion fixtures and the communication protocol for handling the interactions between the machine and the control system. The operation of the machines is executed by transferring motion control data from the control system via the machine interface. The data transmitted includes the motion (tool positions and velocity) and machine parameters (the motors and drivers parameters).



Figure 4: CNC TURNING CENTER [W5]

DISADVANTAGES OF CONVENTIONAL TOOL PATH PLANNING USING G AND M CODES

- Requirement of specialized labor to operate NC machines.
- Cost of conventional NC machines is very high.
- More human interference.
- Part program is written into G & M codes. The big disadvantage of these codes is that they skip some of the main and important tool points.
- For different machine controllers like FANUC, SIEMENS there are different G & M codes so post processors corresponding to these controllers should be developed and if any manufacturing plant is having different machine tools then many post processors are to be developed which is a tedious task.

1.2 DIRECT MACHINING AND CONTROLLING [DMAC]

DMAC is an acronym for direct machining and controlling. In the conventional CNC machining system the work piece is designed in the CAD package then the tool path is simulated on the CAM package, the tool path is to be converted into G&M codes for machining on the machines. There is a significant loss of data while conversion. Also the curves are not exact while conversion and are interpolated to be exact. That curves may not be suitable for requirement. Thus there should be any alternate for G&M codes, so that this loss can be eliminated and there should be proper interpolation of exact model.

For removing this limitation the concept of direct machining and controlling was introduced [4]. Direct machining and control (DMAC) has provided a unique solution to the shortcomings of typical machine tool controllers which are discussed above. The DMAC controller is software-based and very flexible.

1.3 PC BASED NC MACHINING SYSTEM

Gray et al. [1] proposed a new paradigm, in which the machine operates from simple PC. The work done by Bedi et. al.[2] is an elaboration of such paradigm, and implements the paradigm that can be used to produce parts directly from the model. In order to make a part directly from the model, the information of CAD model is taken as STL (Stereo-lithography) file and tool path is directly generated from STL file, thus eliminating the need of postprocessor, in which is used to generate the G & M codes. Now the tool path is directly made without the conversion of part program into G & M codes. The present thesis work also explores the tremendous capabilities of computer for the development of CAD-CAM integrated system. H. T. Yau et.al [3] proposed a model on Numerical control machining of triangulated sculptured surfaces in a stereo lithography format with a generalized cutter.

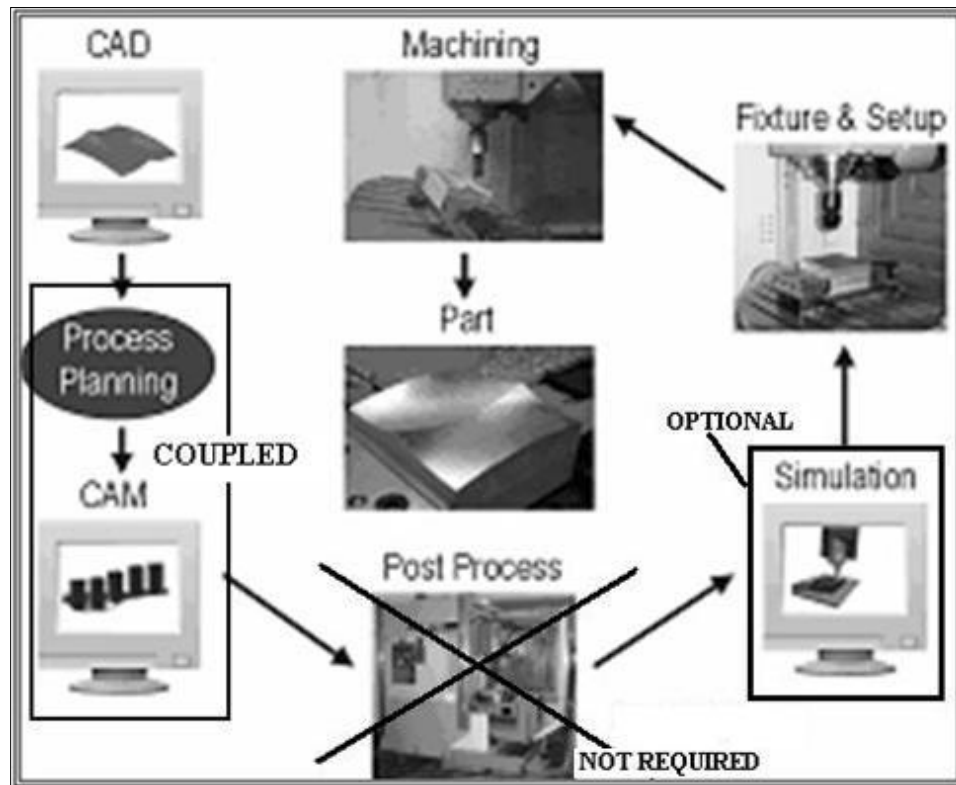


Figure 5: NEW CNC PARADIGM [6]

As shown above, the process planning and CAM module are clubbed in new CNC manufacturing module. Also there is no need of post processor as now G & M codes are not being generated and the tool path is generated directly in CAM package. The simulation is kept optional for the time being.

1.4 SINGLE CONTROLLED AXIS LATHE MILL [SCALM]

Manos et. al. [5] has developed the machine called Single controlled axis lathe mill (SCALM) to present a method for machining complex three-dimensional surfaces using only one axis of controlled motion for positioning a cutting tool on a specially designed numerically controlled (NC) machine. This single controlled axis lathe is configured like a lathe, but is used to produce complex sculptured surfaces out of wood. This is accomplished by mechanically linking two axes of motion to produce a fixed helical footprint of a tool path with constant step-over distance. As the linked axes are rotated, their location is measured by an encoder and passed directly to a personal computer (PC). Software running on the PC determines the depth of the computer controlled axis. The depth information is used to control the depth axis. Several test pieces have been machined out of cedar wood for evaluating the method.

1.5 PROPOSED WORK

The proposed work involves feature based Tool path planning for 3-Axis NC-Milling Lathe for the cylindrical and conical geometries. The main objective will be to develop the tool path for these geometries and then to identify the features in these geometries and then to develop algorithm for tool path planning of the identified features. Features are to be identified based on the tool path files generated by the computer algorithm available for both roughing and finishing machining operation. After features are recognized like pockets, grooves, protrusions of any shape and size, these features will be subsequently machined. So computer algorithm has to be generated for the processing of both roughing files and finished file. The tool path planning algorithm has to be developed for skipping the features in the first go and then these features are separately approached for machining them in the last. The main aim will be to finish the scallops which are generated on the part surface using feature recognition and contoured machining. Afterwards validation of the work has be done both on software.

CHAPTER 2

LITERATURE REVIEW

2.1 TOOL PATH GENERATION METHODS [6] [10]

The conventional tool path generation method used in the industry is to specify the cutter contact point on the part surface and then offset that point to yield the cutter location. The cutter contact point (CC) is the location where the tool touches the part surface. The cutter location (CL) is the location of the Centre of the tool. There exist a number of tool path generation methods that are popular in industry. Some of the common ones are:

1. The Isoparametric Method

The Isoparametric method is one of the simplest tool path generation algorithms in which the cutter contact points are specified along isoparametric lines on the part surface. Isoparametric lines are curves of constant parameter value on parametric surfaces. The isoparametric curves are approximated by linear segments. However, if the linear segments are large, it may results in under-cuts on the sculptured surfaces of the model [8] [14].

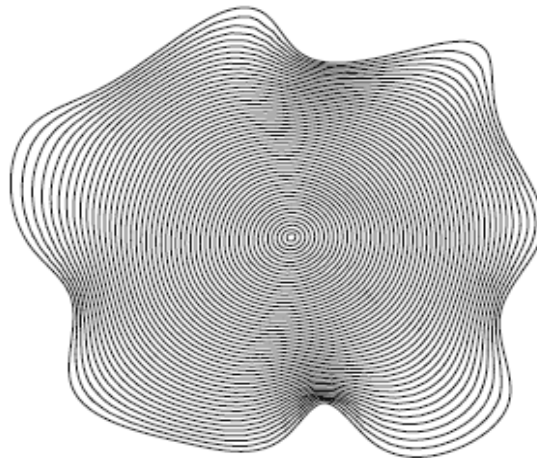


Figure 6: ISOPARAMETRIC MESH [9]

2. The Cartesian Method

In contrast to determining the tool movements based on the part surface's parameters, the **Cartesian** method allow the operators to generate tool paths with respect to the global X-Y-Z Cartesian coordinates. The tool path generator takes the X-Y coordinate of the cutter location as its input and computes the Z-value of the cutter location. The instantaneous tool location is also checked for gouging. This method is more difficult to implement for parametric curved surfaces when compared to isoparametric method. This is due to the complexity of the computational relationship between the cutter contact points in the global X-Y-Z coordinates and the part surface coordinate.

3. The Offset Surface Method

The offset surface method is conceptually similar to the Cartesian method. The offset method generates the tool path by offsetting the part surfaces by the tool radius. The center of the cutter tool travels along the offset surface to machine the part, and the tool path is calculated by identifying tool passes on the offset surface. However, self-intersections tool path that leads to over-cut or cavities of under-cut must be detected and corrected while performing the offset surface algorithm [15]. An advantage of this method is one can find a tool trajectory parallel to X-axis in which the tool moves only along the Y- and Z-axis. Lee [3] has generated helical tool path for offset surface with constant scallop height. Tang [16] has also given the surface offset method.

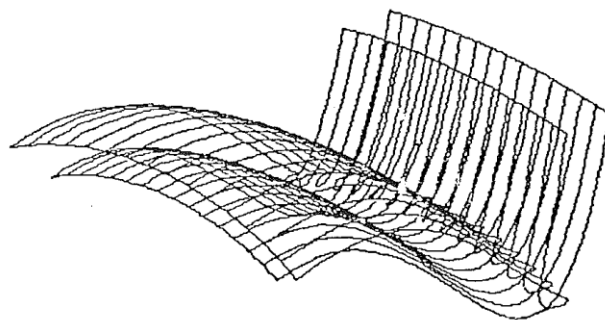


Figure 7: THE OFFSET SURFACE WITH SELF INTERSECTION [15]

4. Feed Forward Method

Feed forward is also important in tool path generation as the machine movements is discretized into finite piecewise motions. Huang et. al. [16] discussed parameterizing the surface to determine the step size while maintain the machining errors within a desired tolerance. The linear feed-forward tool motion between two tool positions is used to determine the deviation between the actual straight line tool paths and the desired surface.

5. Side Step Method

The step-over increments, which are the distance between the adjacent tool paths in the side-step direction, are used to determine the height of cusps that remain after the machining process. Thus, the tool path is basically recognized as a set of tool positions for the tool to traverse and interpolate into a smooth path movement. Hence, the method of determining the tool positions is a factor that can affect the finish quality of the machined parts. There are many significant developments done Yau et.al. [3] Presented the generalized cutter, in which the tool path algorithm is made for generalized cutter. The cutter profile varies with the variation of parameters of generalized cutter. Kishinami [17] has described the inverse offset method for tool path generation.

2.2 TOOL PATH PLANNING STRATEGY [7]

Tool path planning has been divided into two parts:

- FOOT PRINTING
- TOOL POSITIONING

The approach used by Manos et al. [5] requires the specification of the tool axis trajectory, not the cutter contact point (CCP). The tool axis trajectory fixes two of the three coordinates required to specify the cutter location and reduces the problem to solving for one coordinate. In addition, the technique allows for in-built gouge checking. In this method, the tool path is generated by separating tool positioning method with tool path foot print based on the surface to be machined. First, a foot print is generated, and this foot print is discretized into number of points where the gouge-free cutter position is calculated. To calculate gouge-free position, the work piece surface is triangulated, i.e. the surface is divided into 3D triangular elements. The tool is then dropped over the triangulated surface of the work piece along the tool axis whose position is specified. In this work, the method developed by Manos et al. [6] is extended to flat and radiused end milling cutters. Here, the ball-nosed, flat and radiused end milling cutters are modeled as a ball, a coin and a doughnut respectively. The cutter is dropped along the tool axis, the first point of contact between the cutter and the surface is selected as it represents a gouge-free cutter location. To find the first point of contact several cutter locations for different triangles under the tool are calculated and the tool position that represents the highest location of the tool is chosen. To generate the tool path, a series of cutter drops at regularly pre-determined intervals are executed and stored in a file. The file is used to control the depth of cutter as it moves along the specified tool path foot print during machining of a complex surface

For the 3 axis milling machines, the cutter is confined to move in and out from the work piece along the confined axis. The cutter is allowed to move in X, Y and Z axis out of which X and Y axis are decided based on the tool path foot print, and the coordinate of Z axis i.e. depth at which the cutter can penetrate is to be found. Each of the cutter depth is combined and sequenced along the specific foot print to generate gouge-free tool path. A tool path is composed of several tool positions based on the radius of the cutter and step-over distance (distance between consecutive cutter positions). A traditional NC controller can move a tool along any path built up of linear segments. The trajectory of the tool is dictated by a path designed on a plane perpendicular to the

tool axis. This trajectory is called the tool path foot print; there is no limit to the shapes of footprints on commercial NC machines. However, it is extremely important to choose optimum footprint because it will affect the final machining time. The footprint is directly related to the tool path length, thus better the footprint lower would be the tool path length. The path is characterized by its side step i.e. the distance between adjacent straight line paths. To create a tool path, the foot print is discretized into small moves and the Dropping Method" is used to find the gouge-free cutter position to generate tool path. The tool is moved linearly between these gouge-free positions to machine the part. If the foot print is discretized finely the surface can be machined very precisely. However, time required to generate tool path will increase. The advantage of this method of separating foot print and cutter positioning is that it can be applied blindly to create the tool path for machining any surface. Moreover, the footprint is generated once, and the same footprint can be used for machining any number of components only the depth will vary at each position.

If the tool path generated can be developed at any X, Y location on the work piece independent of any specific foot print than it will allow generation of custom foot print depending upon the type of the work piece. Thus, by generating fixed topology with an optimal foot print tool path for a family of parts, the mass production of custom products can be achieved. Wooden sign are by nature custom. A CAD application that creates in finite combination of shapes, backgrounds, borders, logos and names is used to produce 3D models of signs that are to be machined. This work describes an automatic method for generating tool paths to machine these signs.

Once the foot print is developed, the entire foot print is divided into number of points. The discretization of the foot print into several points is done based on the radius of the cutter and the side-step with which work piece must be machined. The cutter is dropped independent of complexity of the surface at each of these points along foot print and gouge-free cutter location is found. The Dropping Method is used to gouge-free cutter location at each of these points along foot print. By separating the foot print with cutter positioning, the surface can be machined with all type of cutters independent of the complexity of surface. Moreover, the foot print can be customized based on the type of surface to be produced to achieve better surface finish. By machining the surface with flat-end and radiused end mills, more volume of material is removed in single pass which improves machining time. The method described provides a powerful

method for positioning a tool on the part surface; however, a part is machined by moving the tool across the surface

2.3 TOOL POSITIONING METHODS

The CNC machine's tool movement is guided by a series of interpolated tool positions, whereas the parts to be machined could be represented by elements with varying slopes and curvatures. Thus, the method of representing the sculptured surfaces is related to the accuracy of the tool positions and the machined part. CAD systems have adopted parametric methods such as NURBS and B-splines as standards for designing sculptured surfaces. Isoparametric lines on parametric surfaces can fan out from one end to the other making them convenient for machining as they result in over-machining in some areas [8] and consume unnecessary machining time. An alternate point-by-point based tool positioning approach is commonly used for generating tool positions by offsetting a desired distance along the surface normal at a given location to determine the tool contact point. This method is similar to the offset surface method for tool path generation other than it is applied to a discretized surface. This method is used for tool positioning with ball nosed end mills. A variation of this method is to project the tool normal onto the curve surfaces and determining the tool contact point along the projected surface normal. This method is used for flat end milling cutter. A combination of the two methods is used for toroidal or ball nose end mill cutter. These offset methods are commonly used in determining gouge-free tool positions while the tool is tangent to a point on the surface. The accuracy of the tool positioning methods relies significantly on the part surface representation. In order to develop a tool path independent of surface representation method, a solution adopted in industry is to use triangulated surface representation. The use of triangulated surface CAD model in tool positioning follows the same steps as any other tool positioning methods. First, the tool path footprint is specified. Next, at each point along the tool path, the triangulated model of the tool is dropped along the tool axis towards the part and the contact point between the tool surface and the triangulated model surface is analytically computed [12]. The direct projection of the tool surface onto the part surface along the tool normal creates an instantaneous tool shadow area where the desired contact point is to be calculated. Under this shadow, the first point of intersection between the tool surface and the part surface is the desired contact point. A series of contact points at regularly pre-determined intervals can produce the tool path. This method is

useful in generating the tool positions for three-dimensional models with irregular shapes, and do not require data converters or geometry engines for remodeling the sculptured surface during tool position generation.

2.4 FEATURE RECOGNITION AND TOOL PATH PLANNING FOR THREE AXIS MILLING LATHE

Bor Tyng Sheen et al. [18] developed an effective method for identifying machining features. While recognizing features, the work piece is sliced at some assigned positions. The sectional curves of the work piece faces and slicing plane constitute the feature profiles. Not only the isolated machining features but also the intersecting machining features can be identified by information from these intersection profiles. Moreover, the recognized machining features can be employed for scheduling the manufacturing sequence. Different kinds of tool paths can be automatically generated for various machining features to improve the cutting efficiency.

Vinodh kumar Somavar Muniappan [19] developed Automatic recognition of features from Boundary representation solid models and tool path generation for precision machining of features with free form surfaces. Using the boundary representation technique, the information required for manufacturing process can be directly extracted from the CAD model. It also consists of a parameterization strategy to extract user-defined parameters from the recognized features. The extracted parameters from the individual features are used to generate the tool path for machining operations regardless of the intersection of one or more features. Various types of tool paths such as one-way, zigzag, contour parallel are generated according to the type of the feature for the roughing operation. The algorithm automatically plans the sequence of machining operation with respect to the feature location, and also selects the type of tool and tool path to be used according to the feature.

CHAPTER 3

PROBLEM IDENTIFICATION AND METHODOLOGY FOR SOLUTION

3.1 PROBLEM IDENTIFICATION

First of all, a generalized tool path generation algorithm has been developed for the three axis CNC milling lathe machine with ball nose end mill cutter. Here tool path was generated which moves in the helical path with uniform side feed and varying radius of cutter for roughing and finishing passes. Taking .STL (Stereo-Lithography) file as input of the algorithm which gives all of the information about the work piece, a gouge free surface was developed. By taking the STL (Stereo-lithography) file as initial input from the CAD modeler, the computer program generates the cutter location (CL) points in a text file. The work piece may be made on the any design software like SOLID-WORKS, PRO-E, and CATIA as all designing software has capability to save the part as .STL format. Tool path generated is capable of generating the all artistic features and complex parts other than making the simple turn parts. Tool cutter radius for both roughing and finishing passes can be given as input by the user as per requirements. Also the depth of cut for both roughing and finishing passes is taken as input from user. After this generalized tool path algorithm, it was identified that there was a need of feature based tool path planning algorithm which would identify the features like cavities, pockets, grooves etc. based on the x values generated in the text files obtained from the previous algorithm. A generalized algorithm for the cylindrical and conical geometries having distinct features like cavities, pockets, grooves etc. has to be developed.

3.2 TOOL PATH PLANNING FOR THREE AXIS CNC MILLING LATHE

Tool path planning will be done according to specification of 3 Axis NC milling lathe machine which are given below:

3.2.1 SPECIFICATION OF 3-AXIS NC MILLING LATHE MACHINE [W4]

The 3 Axis NC Milling Lathe PBG KW 2048[30] machine used is having one rotational and two translational motions, tool penetrate the work piece along X axis, the tool traverse along Z axis and work piece rotate about Z axis. Any pseudo symmetric part (which may or may not have a central axis of symmetry) can be machined by this machine. Since we are using offset technique which already consider tool radius offset for roughing and finishing path, so we assume tool radius as a zero which act like a single point. So whenever a cutter contact (CC) point on an offset surface is found this means cutter location point (CL) is found. The tool path of this machine is helical. The C axis, which is the rotational axis, gives the location of number of points on the surface along the helical curve. Tool axis is consider as a ray in such condition, which is projecting on a surface from a source point which is at the central axis to destination point which is at a far distance along X axis, in a helical path and each contact point is stored as a cutter location point.

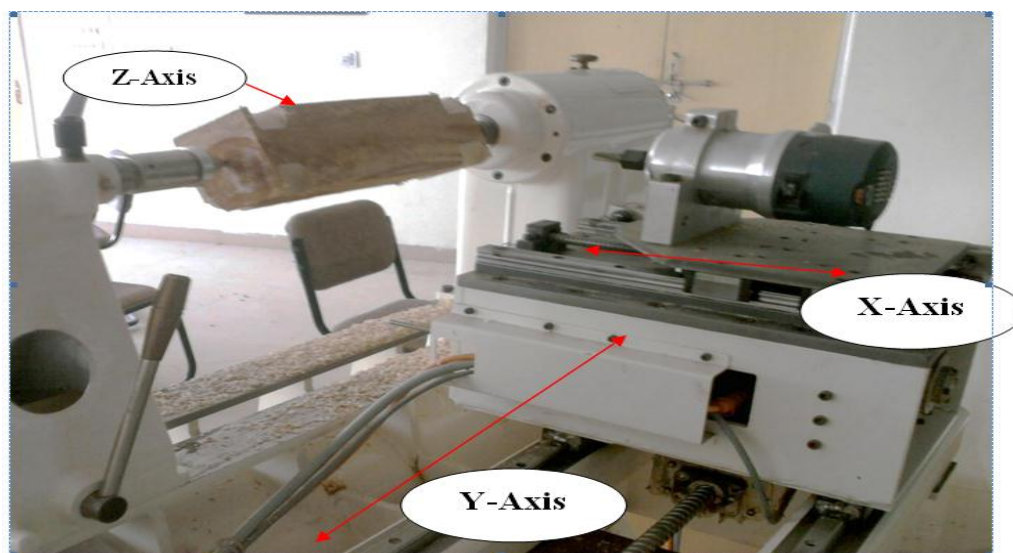


Figure 8: PBG2048 CNC LATHE MILL

3.2.2 PARAMETERS REQUIRED FOR NC TOOL PATH PLANNING

1. STL data for CAD model: The STL file [7] [11] is the first input to the tool path generation algorithm. This STL can be made on any of the CAD model. Most CAD soft wares have capability to store the CAD model as .STL format. The CREO has been used to make the .STL file for the thesis work. This discretizes the CAD model in small number of triangles known as facets. The STL standard includes ASCII data format.

ASCII file is human-readable and can be modified by a text editor if required, thus for this reason we are using the ASCII format, which has structured as:

solid *name*

facet normal *n1 n2 n3*

outer loop

vertex *v11 v12 v13*

vertex *v21 v22 v23*

vertex *v31 v32 v33*

endloop

endfacet

.....

endsolid *name*

Where *n1* to *n3* and *v11* to *v33* are floating point numbers .

White space (spaces, tabs, newlines) may be used anywhere in the file except within numbers or words. The spaces between 'facet' and 'normal' and between 'outer' and 'loop' are required.



```

facet normal 0.000000e+00 1.000000e+00 0.000000e+00
  outer loop
    vertex -4.755336e+01 1.500000e+02 1.544857e+01
    vertex 5.000000e+01 1.500000e+02 0.000000e+00
    vertex -5.000000e+01 1.500000e+02 0.000000e+00
  endloop

```

Figure 9: STL FILE FORMAT FOR A CYLINDER

This ASCII file format of STL is used as input in the tool path planning algorithm. There are two basic rules followed for generation of STL file.

- Facet Orientation
- Vertex to Vertex Rule

Facet orientation: The facets define the surface of a 3-dimensional object. As such, each facet is part of the boundary between the interior and the exterior of the object. The orientation of the facets (which side is \out" and which side is \in") is specified. First, the direction of the normal specified in the format always points towards the outside of the part. Second, the vertices are listed in counter-clockwise order when looking at the object from the outside (right-hand rule). Orientation of a facet is determined by the direction of the unit normal and the order in which the vertices are listed. These rules are illustrated in 3.1.

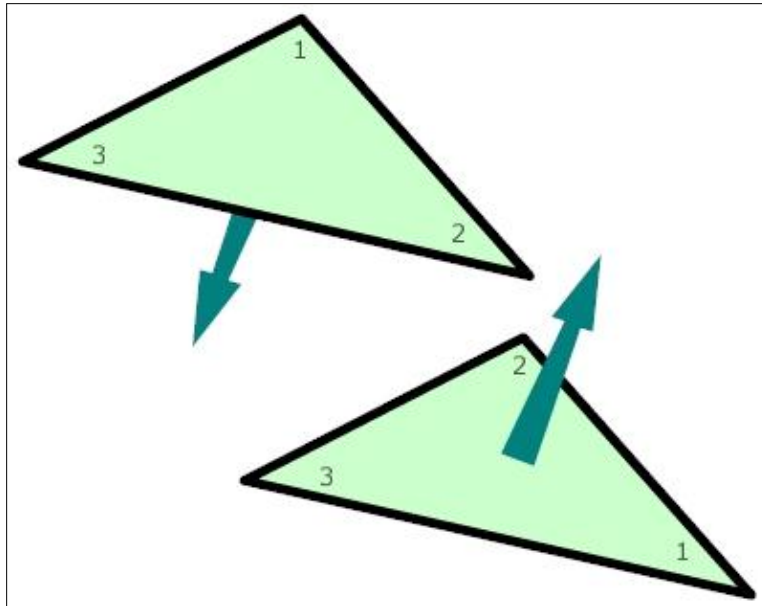


Figure.10: COUNTER CLOCKWISE/CLOCKWISE ORDERS OF STL FACETS[4].

Vertex to Vertex Rule: Each triangle must share two vertices with each of its adjacent triangles. In other words, a vertex of one triangle cannot lie on the side of another. This is illustrated in Figure 3.2.

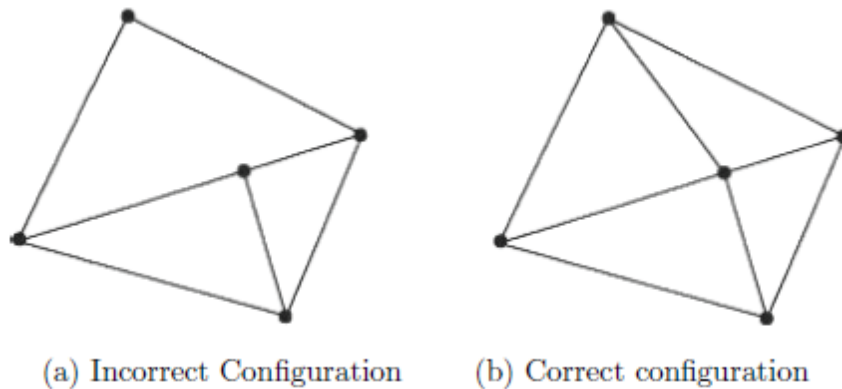


Figure 11: VERTEX TO VERTEX RULE

2. Radius of roughing and finishing tool: The radius of roughing and finishing tool is the next input given by user. This will give user the ability to use the ball end tool for any radius.

3. Depth of cut for roughing: The depth of cut for roughing pass can be given by user as per requirements. The raw material generally is of much bigger size than the part dimensions to be manufactured. In case if the part dimensions are of much smaller in size, then the user has given independence to cut the raw material in multiple passes by giving depth of cut. First, the difference between maximum and minimum dimensions of work piece is displayed on the execution windows. By seeing the difference, the user can put any depth of cut. A computer algorithm automatically calculates the number of passes for the entered value of depth of cut and it displays the number of passes on the execution window. If the depth of cut is more than the difference between maximum and minimum dimensions, then it cuts the material in one go.

4. Finishing allowance: The another value which user inputs in the program is the finishing allowance which the roughing tool should leave for finishing tool. Generally it is kept 2mm but the user may enter another value if the part is not so complex or the tool used for finishing is of higher radius.

5. Side feed: The value of side feed is to be entered by the user. This value is kept large for roughing pass and small for finished passes.

The STL file which is used as the input for the tool path algorithm contains numerals, strings which are not required. Thus there is a need to refine the STL file. A computer algorithm is used for this purpose. It automatically ignores the unnecessary data and strings which are not desired in the coming program. The STL file contains the vector normal and coordinates vertices of all facets. The major dimensions of the work piece are extracted from the STL file as it may help to make the initial stock so that it may help to optimize the stock dimensions.

3.2.3 BALL DROP METHOD

Manos [5] developed Drop the Ball method which could be used to machine a sculptured surface on 3 axis CNC lathes. This method can machine sculptures surface by dropping the ball nose cutter shaped like a sphere. The tool is dropped at each position, independent of the type of surface being machined which makes it efficient for machining complex surfaces. The tool is then moved over the fixed helical foot print to generate gouge- free tool path. The Drop the Ball method was developed and tested for machining complex sculptured surface on CNC lathes. This Drop the Ball method can machine sculptured surfaces efficiently but this method is limited for the use of ball nose end milling cutter with a fixed path along which cutter is moved. The tool path planning method developed by Manos et al. [6] was selected for machining sculptured wooden sign because it can machine sculptures surfaces and complex features efficiently. Moreover, the Drop the Ball Method can position the cutter gouge- free over the work piece surface to machine a surface.

The entire work piece surface which is to be machined is mathematically represented by the triangle known as facets and surface normal called facet normal. In Dropping Method [7], for each cutter position the tool is dropped from a height along the tool axis at any particular position in X-Y plane. The cutter drops steadily along tool axis plane and would intersect one of the triangles representing the work piece surface. There are three possible scenarios for the point of contact between cutter and surface:

- i. Contact on planar surface.
- ii. Contact on edge.
- iii. Contact on vertex.

The point of contact will be dependent of the topography of the surface. The tool path planning algorithm has been designed to check for the projection of the tool in a given position that the generic ball end milling cutter is touching with the part of the triangulated surface. The parameter u is set for the extreme position of the tool, and it keeps on decreasing as goes from Centre to the outer of the work piece. The tool projection may be inside the triangle or may touch triangle on edge or on vertices. The tool positioning strategy must stops the tool forward motion at the first contact of the tool with either of edge, triangle surface or vertex for which following

checks have been applied to extract the information about tool position along X-axis direction of the 3-axis CNC milling lathe.

1. TRIANGLE CHECK

This check is performed to identify if a tool of radius r moving along a tool axis $(1 - u)T_1 + uT_2$ will contact a triangle within the bounds of its vertices P_1 , P_2 and P_3 , where u is the parameter from T_1 to T_2 . As the centre of the tool lies along the tool axis and the tool contacts the plane of the triangle tangentially, we get the following equations:

$$(1 - u)T_1 + uT_2 = P_1 + (P_2 - P_1)t + (P_3 - P_1)s + \hat{N}r, \quad (1)$$

Where

$$\hat{N} = \frac{(P_2 - P_1) \times (P_3 - P_1)}{|(P_2 - P_1) \times (P_3 - P_1)|}.$$

t & s parameters are to define the surface of facet and \hat{N} is the facet normal.

Solving this as:

$$\begin{bmatrix} (T_2 - T_1)_x & (P_1 - P_2)_x & (P_1 - P_3)_x \\ (T_2 - T_1)_y & (P_1 - P_2)_y & (P_1 - P_3)_y \\ (T_2 - T_1)_z & (P_1 - P_2)_z & (P_1 - P_3)_z \end{bmatrix} \begin{bmatrix} u \\ s \\ t \end{bmatrix} = \begin{bmatrix} (P_1)_x + (\hat{N})_x r + (T_1)_x \\ (P_1)_y + (\hat{N})_y r + (T_1)_y \\ (P_1)_z + (\hat{N})_z r + (T_1)_z \end{bmatrix}$$

Where r is the radius of the tool, and s and t are the point of contact between the triangle and the tool. Solving the above equation tells us the contact point P_c in the plane of the triangle and the location of the tool T_c along the tool axis. Then the check is performed to check that the point P_c tool T_c is within the boundaries or not.

•

2. EDGE CHECK

This check is performed to see if the tool moving along the line $(1 - u)T_1 + uT_2$ will touch the edge $P_1 + (P_2 - P_1)t + \hat{N}r$ tangentially within the end vertices P_1 and P_2 (i.e. with $t \in [0, 1]$). This check is performed for all three edges of a single triangle. If the tool touches any edge of the face, it must satisfy the following equation:

$$(1 - u)T_1 + uT_2 = P_1 + (P_2 - P_1)t + \hat{N}r \quad (2)$$

3. VERTEX CHECK

This check is performed to see if a vertex P_i will touch the tool as it moves along the line tool axis $[(1 - u)T_1 + uT_2]$. The check is performed for all three vertices of the triangle. As the tool contacts the vertex P_i , it must satisfy the equation:

$$|P_i - [(1 - u)T_1 + uT_2]| = r . \quad (3)$$

Solving the above equation for u gives the location of the tool axis as the tool touches the vertex P_i . If there are no real-valued solutions, then the tool will not touch P_i , for any value of u . In general, when the tool can touch P_i , there will be two solutions; we choose the solution furthest away from the work piece axis.

After applying all these checks, the value of P_c is checked whether it is in the triangle or not and the appropriate value is stored and then tool is moved to another location. These values of P_c are stored in a file to retrieve for the next step.

3.2.4 TOOL PATH PLANNING FOR ROUGHING

After predicting the tool path, the depth of pass is entered from the user and then the algorithm automatically calculates the number of passes and makes the file for different rough passes. The finishing allowance is also entered from user to make it convenient for him to select these parameters.

3.2.5 TOOL PATH PLANNING FOR FINISHING

In the roughing pass, the maximum cut is kept away as the user input the allowance from the actual dimensions of work piece; this distance is kept for roughing pass. The finishing pass is applied up to the actual dimensions of the work piece, thus making the dimensions of the work piece to its actual dimensions.

3.3 IDENTIFICATION OF GEOMETRY AND FEATURE RECOGNITION BASED TOOL PATH PLANNING

Based on the tool path files generated from the generalized tool path algorithm, it can be identified by comparison of the x values that whether the geometry of work piece is cylindrical or conical. The present thesis work is based on the identification of the geometry mainly cylindrical and conical geometry and then to identify what are the various features in these geometries like cavities, pockets, grooves etc. After identification of various features in a particular geometry, tool path planning strategy has to be adopted in such a way that first whole geometry is to be machined and features are skipped and saved in other files then feature based tool path planning has to done.

3.3.1 IDENTIFICATION OF A CYLINDRICAL GEOMETRY

Cylinder can be identified based on the x values calculated from the roughing passes and finishing pass. If x values remains constant throughout i.e if we calculate the mean of all x values in the roughing passes as well in the finishing pass and it remains same as the integer values of x values then it is identified as cylinder.

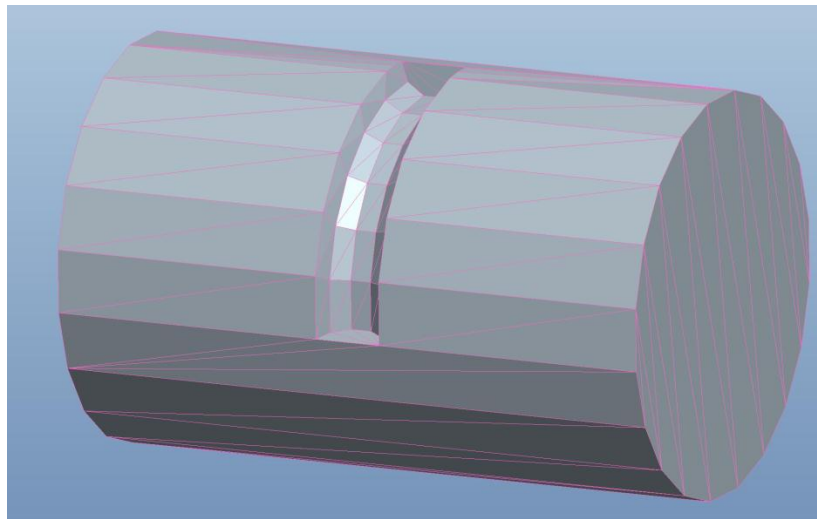


Figure 12: A CYLINDRICAL GEOMETRY WITH A POCKET

0	0	15.6045
0	1	15.5661
0.002205	2	15.5269
0.0033075	3	15.4879
0.00441	4	15.4536
0.0055125	5	15.4238
0.006615	6	15.3984
0.007717	7	15.3776
0.00882	8	15.3612
0.0099225	9	15.3492
0.011025	10	15.3415
0.0121275	11	15.3383
0.01323	12	15.3394
0.0143325	13	15.3449
0.015435	14	15.3548
0.0165375	15	15.3691
0.01764	16	15.3878
0.0187425	17	15.4109
0.019845	18	15.4385
0.0209475	19	15.4707
0.02205	20	15.5074

Figure 13: IDENTIFICATION OF CYLINDRICAL GEOMETRY

3.3.2 IDENTIFICATION OF A CONICAL GEOMETRY

Cone can be identified based on the x values calculated from the roughing passes and finishing pass. If x values follows a definite decremented pattern after a certain interval of time and decrement is constant throughout then it is identified as a cone.

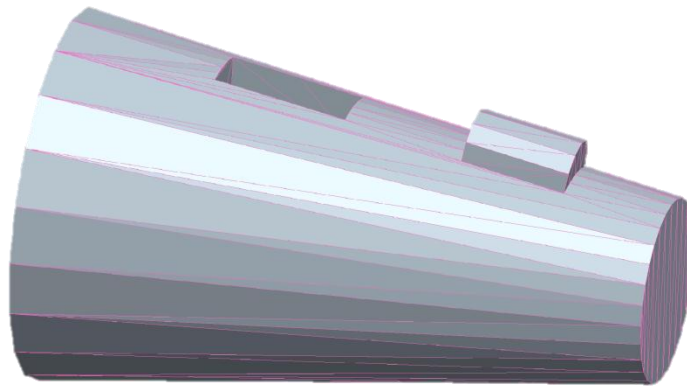


Figure 14: CONICAL GEOMETRY WITH FEATURES

8.23237	7467	14.0024
8.23347	7468	14.0376
8.23457	7469	14.0376
8.23568	7470	14.0376
8.23678	7471	14.0376
8.23788	7472	14.0288
8.23898	7473	13.9941
8.24009	7474	13.9635
8.24119	7475	13.9369
8.24229	7476	13.9145
8.24339	7477	13.896
8.2445	7478	13.8815
8.2456	7479	13.8709
8.2467	7480	13.8643
8.2478	7481	13.8616
8.24891	7482	13.8628
8.25001	7483	13.868
8.25111	7484	13.877
8.25221	7485	13.8901
8.25332	7486	13.9071
8.25442	7487	13.928
8.25552	7488	13.9531
8.25662	7489	13.9821

Figure 15: IDENTIFICATION OF CONICAL GEOMETRY

3.3.3 IDENTIFICATION OF FEATURES IN THE CYLINDRICAL AND CONICAL GEOMETRIES

If we have the combined geometry for both cone and cylinder then the tool path algorithm must verify whether first geometry is a cylinder or a cone then recognize the features in it and skip them and then go to other geometry and again recognize the features in it and skip them also and then in the last it would machine all the features separately in a single go

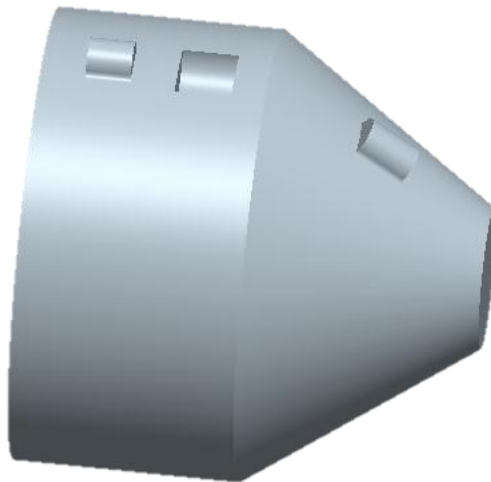


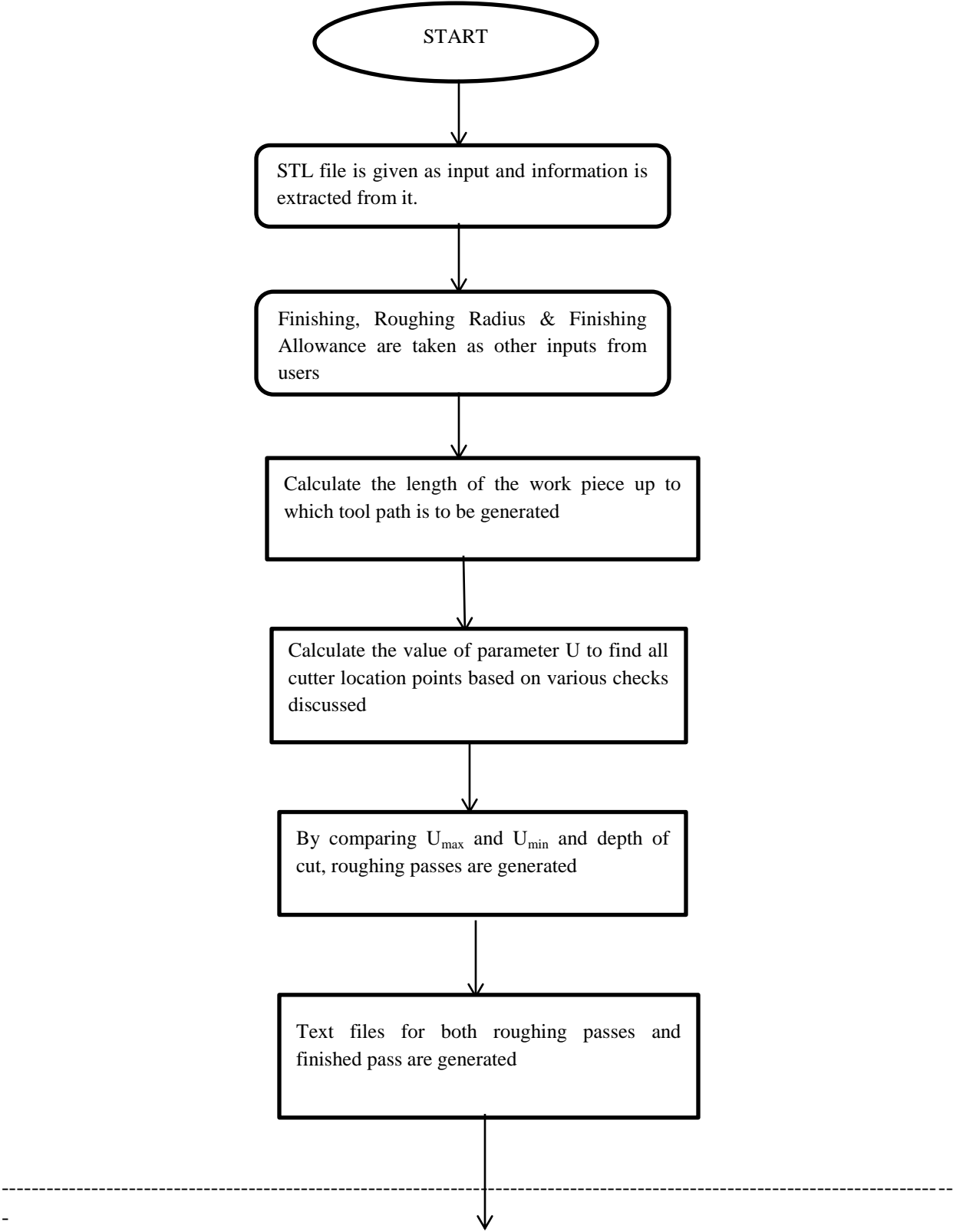
Figure 16: CYLINDRICAL-CONICAL GEOMETRY WITH FEATURES

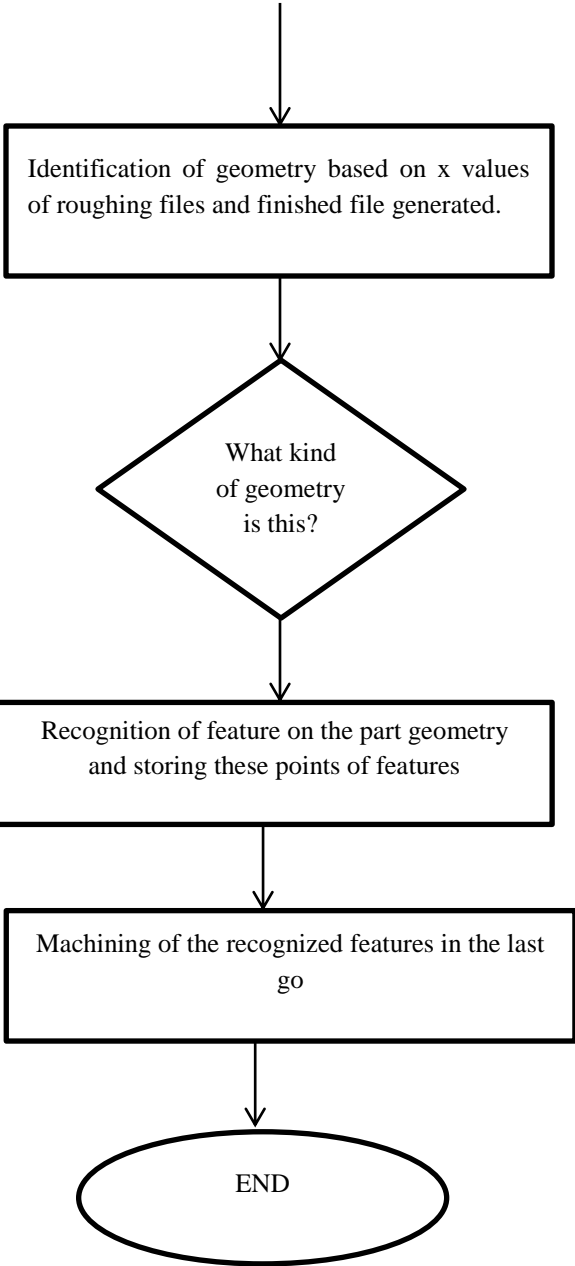
3.4 METHODOLOGY FOR SOLUTION

A computer program has been developed for the mathematical procedure defined. The solution procedure for NC tool path generation for 3 axis NC milling lathe has been discussed stepwise as below:

- 1 The STL file is generated in CREO CAD modeling software.
- 2 The STL file is then send as input to the program and is made to run on the PC.
- 3 The STL data for the three dimensional CAD model of the part extracts the required dimensions of the part for the algorithm. Then facets which are not required are removed and others are saved during the program execution.
- 4 The execution of program starts only after user has provided input like depth of cut for roughing pass, finishing allowance etc.
- 5 The maximum stock size is calculated to get the length of the work piece.
- 6 The program runs for the whole of the work piece. It checks the extreme tool position where the value of u is least at every point and after storing that value, it moves to next position.
- 7 The program then runs for the roughing passes. The calculation of roughing passes is done based on the depth of cut given by the user.
- 8 The program executes now for the last time for finishing pass after roughing passes are calculated. The finishing allowance is predefined by the user in the algorithm.
All Cutter Location (CL) points are saved in auto generated text files with different names for roughing passes and finishing pass.
- 9 After text files are generated for both roughing and finishing passes, based on the geometry of the work piece and various features incorporated in it, another algorithm is developed
- 10 In the next algorithm, roughing files and finished file both are processed based on the features identified in the work piece.
- 11 Here features are identified and then they are skipped in the machining and afterwards these features are separately machined in the last.

3.5 FLOW CHART FOR SOLUTION





CHAPTER 4

RESULTS AND VALIDATION

The testing of tool path is carried out on a Custom Simulator ‘Tool Sim’ and has been validated by comparing the actual part geometry saved as STL file with the final machined geometry output taken as obj file in the CREO software. Tool path have been prepared for different models like cylinder, cone and then various features are incorporated into these geometries like cavities, pockets and protrusions and these features are then machined in the last and tool path is validated by saving the stock as obj file and then validation of tool path algorithm is done by comparing obj file with STL file

TOOLING USED

The tool used in the algorithm is ball nose end mill cutter. The two radiuses of the tool are used, but their radius can be modified and the algorithm works for any tool radius. Then all checks are being applied here and then the tool moves to the next position in helical path. In the algorithm, the tool moves in a helical way. Then based on geometry features are recognised and they are skipped in the first go so as to machine them separately The tool keeps on rotating and finding the suitable points until the length of work piece.

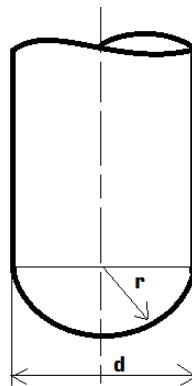


Figure 17: BALL NOSE END MILL

4.1 VALIDATION OF CYLINDER

First of all tool path algorithm is applied to simple cylindrical surface. Tool path is verified on custom simulator Tool Sim .After that stock is saved as obj file in the simulator. Then obj file is directly open in CREO software and compared with actual stl file to validate the algorithm developed.

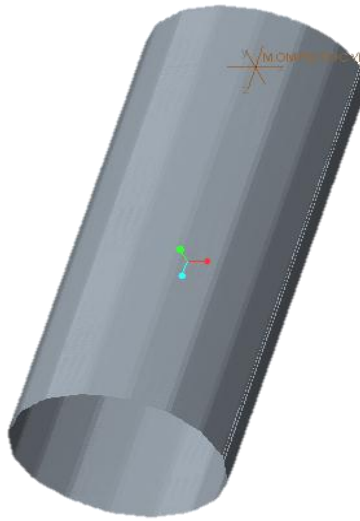


Figure 18: THE .OBJ FILE OF A CYLINDRICAL PART

4.2 VALIDATION OF CYLINDER WITH GROOVE AS A FEATURE

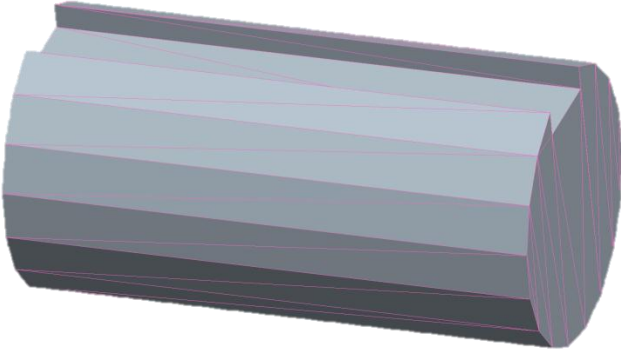


Figure 19: STL FILE OF CYLINDER WITH GROOVE



Figure 20: OBJ FILE FOR VALIDATION

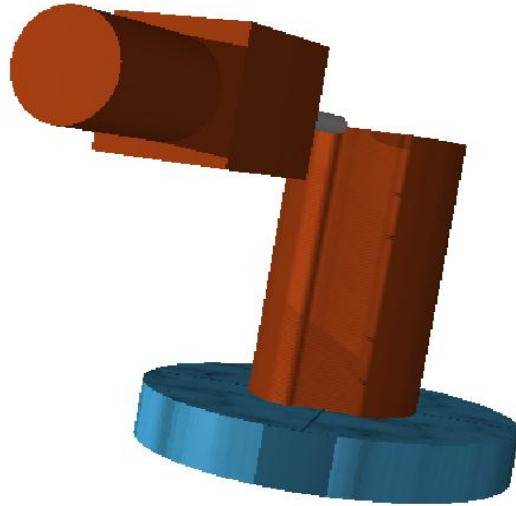


Figure 21: CYLINDER WITH GROOVE MACHINED IN THE END IN TOOL SIM

4.3 VALIDATION OF CYLINDER WITH PROTRUSION

Here cylinder with protrusion is made as a part in CREO software and then algorithm is verified by saving machined file in simulator as obj file and then this file is opened in CREO software to validate the results



Figure 22: AN OBJ FILE FOR CYLINDER WITH PROTRUSION

4.4 VALIDATION OF CYLINDRICAL-CONICAL GEOMETRY WITH VARIOUS FEATURES

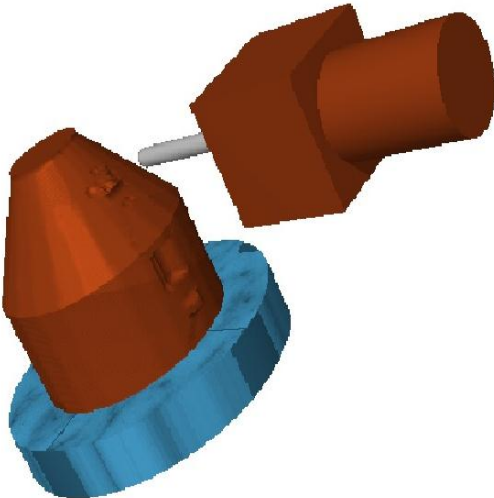


Figure 23: VALIDATION OF ALGORITHM IN TOOL SIM

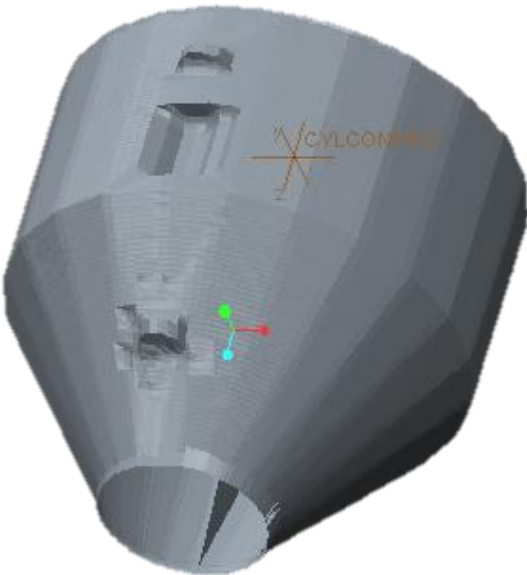


Figure 24: OBJ FILE FOR CYLINDRICAL-CONICAL PART WITH FEATURES

CHAPTER 5

CONCLUSIONS AND FURTHER SCOPE

In the present work, firstly tool path algorithm for the 3-axis lathe mill has been developed. Visual studio is used for programming purpose. Ball nose end mill cutter tool has been used in the algorithm. In the present work, tool of radius 6.35 mm has been used for roughing and 1.5875mm for finishing operations. Various checks have been applied to trace the point of contact of cutter with work piece. Tool and work piece moves in such a way that makes the helical tool path for machining. The accuracy of work piece depends upon the STL triangles, radius of tool for finishing and side feed for finishing pass. After generation of roughing files and finishing file then identification of the geometries have been done by developing geometry based algorithm and then various features like cavities, pocket, and grooves have been separately machined in the last step based on the algorithm developed.

5.1 SCOPE FOR FURTHER STUDY

- The present work can be extended for various other geometries which have not been considered in this algorithm
- The present work can be extended to various tools or a generalized tool from the present ball nose end mill cutter.
- Generalized tool cutter may be incorporated, with the help of which the tool path can be generated for various tools, thus giving user the flexibility of changing the tools according to the geometry of the tool to save the time.
- In this work, the part definition is simplified into a triangulated model before use in Feature Recognition. The choice to recognize features from the B-Rep definition gives the proposed method flexibility to deal with transition feature like fillets and chamfers [19].
- Generation of tool path algorithm for feature recognition for 5-axis machining process.

REFERENCES

- [1] P.J. Gray, S. Bedi, and S. Mann, “A new paradigm for numerically Controlled machines”, Virtual Concept, Biarritz, France, 2003, pp. 128-134
- [2] Nick P.Manos, Sanjeev Bedi, Dan Miller and Stephen Mann, “Single Controlled Axis Lathe Mill”, International Journal of Advanced Manufacturing Technology, vol. 32, pp. 55-65, August 11, 2005
- [3] H. T. Yau, C. M. Chuang, and Y C Lee, “Numerical control machining of triangulated sculptured surfaces in a stereo lithography format with a generalized cutter”, International Journal of Production Research, vol. 42, no. 13, 2004 pp. 2573–98.
- [4] P.J. Gray, G. Poon, G. Israeli, S. Bedi, S. Mann, D. Miller, “Model to Part: A map for the CNC machine of the future,” 7th International Conf. on Adv. Manuf. Systems and Technology, Udine, Italy, 2005.
- [5] Manos N., Bedi S., Miller D., and Mann S. “Single controlled axis lathe mill. International journal of Advanced Manufacturing Technology”, 32:55-65, 2007.
- [6] Paras Mohan Jasra, “Generalized tool path generation algorithm for sculptured pseudo symmetric surface machining”, 2009.
- [7] Kandarp patel, “Web Based Automatic Tool Path Planning Strategy for Complex Sculptured Surfaces”, 2010.
- [8] P. Broomhead, M. Edkins, “Generation NC data at the machine tool for the manufacture of free-form surfaces”, International Journal of Production Research, Vol. 24, No. 1, 1986, pp 1-14.
- [9] Sun Yuwen, Guo Dongming, Jia zhenyuan, Wang Haixia, “Iso-parametric tool path generation from triangular meshes for free-form surface machining.
- [10] Vivek Pal Singh, “Tool path planning for 3-axis NC-milling lathe and 3-axis NC-vertical milling for sculptured surfaces machining using triangular mesh offset.
- [11] P. F. Jacobs, and D. T. Reid, “Rapid Prototyping & Manufacturing: Fundamentals of Stereo-lithography”, McGraw-Hill, New York, 1992

- [12] J. E. Bobrow, “NC machine tool path generation from CSG part representations”, *Computer Aided Design*, Vol. 17, No. 2, 1985, pp. 69-76.
- [13] Bor Tyng Sheen, Chun-Fong You, “Machining feature recognition and tool-path generation for 3-axis CNC milling
- [14] B.H. Kim, B.K. Choi, “Guide surface based tool path generation in 3-axis milling: an extension of the guide plane method.
- [15] Y.J. Chen, B. Ravani, “Offset Surface Generation and Contouring in Computer-Aided Design”, *ASME, Journal of Mechanisms, Transmissions & Automation in Design*, Vol.109, March 1987, pp132-142.
- [16] K. Tang, C. C. Cheng, and Y. Dayan, “Offsetting surface boundaries and 3-axis gouge-free surface machining”, *Computer-Aided Design*, vol. 27, pp. 915–92, 1995.
- [17] T. Kishinami, T. Kondo and K. Saito, “Inverse offset method for cutter path generation”, *Proceedings of the 6th International Conference of Production Engineering*, Osaka, Japan, 1987, pp. 807–812.
- [18] Bor-Tyng Sheen, Chun-Fong You, “Machining feature recognition and tool-path generation for 3-axis CNC milling” *Computer-Aided Design* 38 553–562, 2006.
- [19] Vinodhkumar Somavar Muniappan, “Automatic Feature Recognition and Tool path Generation Integrated with Process Planning” *University of Waterloo Ontario, Canada*, 2012

WEB REFERENCES

W1. <http://www.hsmworks.com/docs/hsmworks/2012/en/#StrategyPocket3D>

W2. <http://www.hsmworks.com/docs/hsmworks/2012/en/#StrategyContour3D>

W3. <http://www.cnccookbook.com/CC3DCAMToolpaths.html>

W4. <http://visor.uwaterloo.ca/~krmoule/webcad>

W5: https://upload.wikimedia.org/wikipedia/commons/6/6c/Small_CNC_Turning_Center.jpg