IJARIIT INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact factor: 4.295
(Volume2, Issue6)
Available online at: www.ijariit.com

# Optimization of Cycle Time through Mastercam Virtual Simulation and Four Axis CNC Milling Machining of Camshaft

**Prasanth Navaneetham***
*M. Tech Graduate*
*Dept. of Mech. Engg. & SRM University*
*prasanth100@gmail.com*

**V. G. Umasekar**
*Assistant Professor*
*Dept of Mech. Engg & SRM University*
*umasekar.g@ktr.srmuniv.ac.in*

**Bibin Baby**
*M. Tech Graduate*
*Dept. Of Mech. Engg & SRM University*
*bibinbaby12345@gmail.com*

***Abstract— in olden days, the complexity of fabricating the mechanical components is higher and time consuming. Manufacturing industries normally use conventional methods like casting and forging process to shape components and use CNC machines to achieve final dimensions within the required tolerance. The difficulty in making components can be minimized by introducing the techniques for reduced setup time, reduced machining time for increase in productivity and increase in accuracy and surface finish as a result of using 5- axis CNC VMC milling machining setup. But prior to start machining, a virtual simulation of the entire operations is to be carried out to check, verify and rectify the occurrence of collision, and the application of suitable parameters to achieve reduced cycle time. And this virtual simulation can be carried out through Mastercam software. This paper is focused on 4-axis finishing process of camshaft using 5-axis CNC VMC milling machine. This work involves geometric modelling of camshaft using Solid Works software. The model is then imported to Mastercam software for applying different methods and parameters to simulate the tool. Then the suitable parameters having collision free and reduced cycle time is taken into account. Then, the NC program is generated using the required post processor and program is entered to the 5- axis CNC milling machine to perform both roughing process and finishing process. Material used is Aluminium Alloy 6061. As per the experimental results, it has been observed that constant overlap spiral tool path for top part machining is selected and high speed tool path is selected for bottom part 3- axis rough machining and rotary 4 axis tool path for final finishing ensuring collision free and reduced cycle time. The cycle time resulted by Mastercam software is less than the cycle time resulted by the 5 axis CNC VMC milling machine. Then, the total machining time of camshaft as per the profile were found to be 5.83 hours and 52 seconds.***

***Keywords — Multi – Axis CNC VMC Milling Machine, Camshaft, Solid Works, Mastercam, Optimization.***

## I. INTRODUCTION

Today, automotive industries require the components having geometrical complexity to be produced ensuring productivity, accuracy, surface finish and reduced human effort. Camshaft manufacturing industries normally use conventional methods like casting and forging process to shape camshaft and use CNC machines to achieve final dimensions within the required tolerance. Cam shafts are complex shaped mechanical components and its production should involve multi-axis CNC milling machines for efficient production. Introducing multi axis CNC machines helps in achieving reduced timing, better surface finish, reduced setup, increased tool life. A new approach was presented to find the 3D shape-generating profiles of different types of cutters for constructing the *G*-buffer models for 5-axis machining

 

[1]. Develop a voxel-based simulator for multi-axis CNC machining. The simulator displays the machining process in which the initial work piece is incrementally converted into the finished part [2]. The cutter path generation method comprises real-time algorithms for cutter-contact path interpolation, cutter offsetting, and coordinate conversion. In addition, a global feedback loop is closed by the CNC interpolator so as to augment the controlled accuracy in practical cutter path generation [3].

### 1.1 *MODELING and SIMULATION*

Solid Works is a solid modelling computer-aided design (CAD) and computer-aided engineering (CAE) computer program that runs on Microsoft Windows. Solid Works is published by Dassault Systems. Mastercam software is the worldwide popularly known computer aided manufacturing software. It was founded in Massachusetts Institute of Technology in the year 1983 and is one of the PC-based computer aided design and computer aided manufacturing software. Mastercam is an effective tool to generate, execute, and simulate tool path. The embedded post processor allows generating tool path according to different manufacturers of CNC machines. In complex machining tasks, Tool changing time, cutting time, traverse time and retract time plays a major role in total machining time. Different types of tool paths are available in Mastercam for both 2D and 3D shapes. Different tool paths are used for roughing and finishing process. Tools are also assigned when defining tool path in simulation, in order to verify whether the assigned machining parameters and tools are correct to the required profile ensuring productivity and surface finish. A camshaft is a shaft used in IC engine for actuating each intake and exhaust valve of an internal combustion engine. The development of CAD/CAM software's plays a vital role in modeling of mechanical components as well as in the generation of CNC part programs for running CNC machines for automated manufacturing of complex shapes and components.  Selection of appropriate parameters among different available parameters and simulation verification results in efficient machining and productivity. Five –axis CNC machines have the ability to perform movements in five different axes simultaneously. Most CNC manufacturers define the machine movements in three different axes namely X-axis, Y-axis, and Z-axis. The five-axis machines can move their axes as like three axis machines and the other two axes namely A-axis and C-axis which can rotate about X-axis and Z-axis. The different configurations of five-axis CNC milling machine, as shown in Fig. 1. The four-axis configuration that is used to fabricate cam shaft, is as shown in Fig. 2.

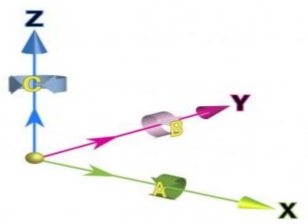


Fig. 1 Machine configurations

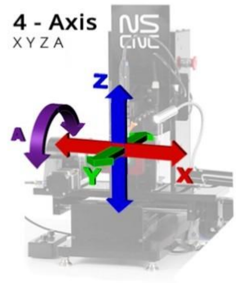


Fig. 2 Four axis configuration

## II. **METHODOLOGY**

### 2.1 *GEOMETRIC MODELING of CAMSHAFT*

The cam terminology involves base circle, prime circle, nose, pitch curve, lobe separation angle. The dimensions used in cam profile are as follows and the steps involved in geometric modeling of camshaft are, as shown in Fig. 3. The model of the camshaft is, as shown in Fig. 4. The drawing of the camshaft is, as shown in the Fig. 5.

1) Base circle  = 42mm
2) Prime circle = 48mm
3) Distance between base circle and prime circle = 1.5mm
4) Lobe separation angle = $30^o$
5) Nose radius = 7.5mm

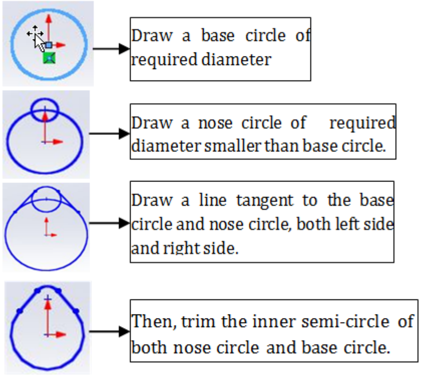


Fig. 3 Geometric modeling of camshaft

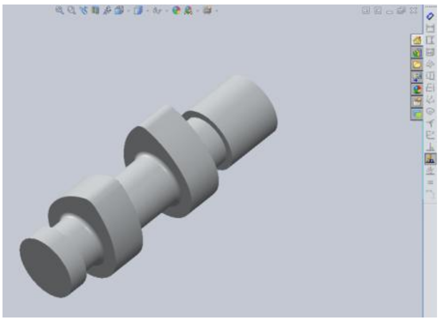

Fig. 4 Model of the camshaft

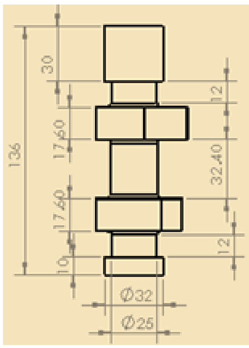


Fig. 5 drawing of the camshaft

## 2.2 *MASTERCAM EXPERIMENTAL SETUP*

For, the cam shaft to be machined, there are three processes involved. They are 3-axis rough machining, 4-axis semi-finishing and 4-axis finishing. Prior to the course of action, stock definition need to be defined, that is, the raw material dimensions need to be defined, as shown in Fig. 6. And that is applied to execute machining. In Stock definition, the form of the stock is chosen to be cylindrical rod positioned relating to x-axis. The measurement lengthwise of stock is 185mm and the diameter of stock is 64mm. The part oriented with respect to x-axis in the graphics screen is shown in the Fig. 7.

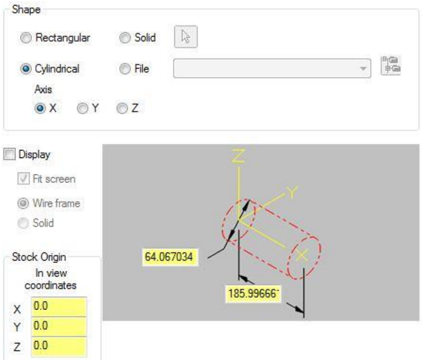


Fig. 6 Stock definition

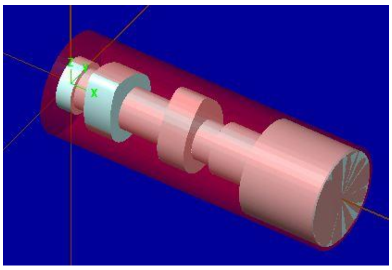

Fig. 7 Part oriented with respect to x-axis in the graphics screen

The method for this part to perform 3 axis roughing involves, dividing the part into two divisions and identified as top segment, as shown in Fig. 8 and bottom segment, as shown in Fig. 9.

 

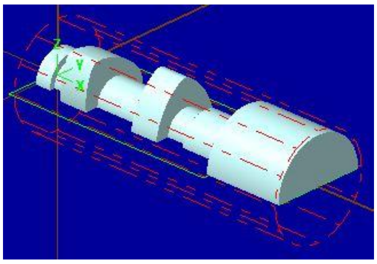

Fig. 8 Mastercam top segment in graphics screen

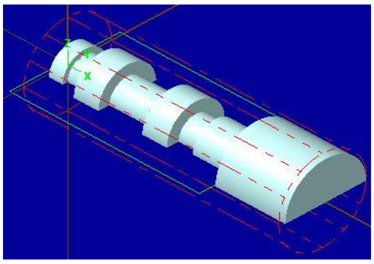

Fig. 9 Mastercam bottom segment in graphics screen

For top and bottom segment of cam shaft, a green colored rectangular boundary was drawn to define the boundary with in which the tool should move. The red color lines on the top of part show the stock definition in wire frame. The level manager means of Mastercam were used to systematize the surfaces, model and wireframe. The set up for 3-axis rough machining of cam shaft was that, at first, the top segment was machined as shown in Fig. 8, and then the work holding device with the part was rotated to 180º and the 3-axis machining for the bottom segment of the cam shaft was carried out. Then, the semi finishing and finishing of the part were carried out. The level manager dialog box is shown in Fig. 10.

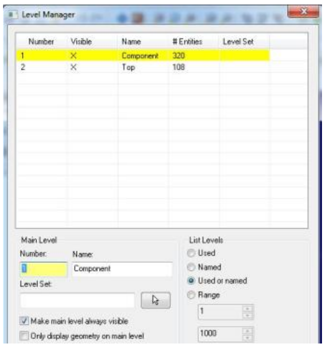


Fig. 10 Level manager dialog box

In level manager, it is shown that the full part is assigned to level 1 and the top segment is assigned to level 2. It means that, when the level 1 is set to main level, the full cam shaft is able to seen on the graphics screen and when level 2 is set to main level, only the top segment is visible in the graphics screen as shown in Fig. 8.

## 2.3 *TOOLPATH PLANNING and GENERATION*

There are number of tool paths available in Mastercam for roughing and finishing operations regarding 3-axis and multi axis machining. So here, for both sides, that is, top and bottom segment, surface rough pocket and surface rough rest mill operations were selected as per the profile to perform 3-axis roughing and rotary 4-axis tool path was selected to perform 4-axis semi finishing and finishing process. The selection of surface rough pocket is, as shown in Fig. 11 and selection of rotary 4 – axis is as shown in Fig. 12.

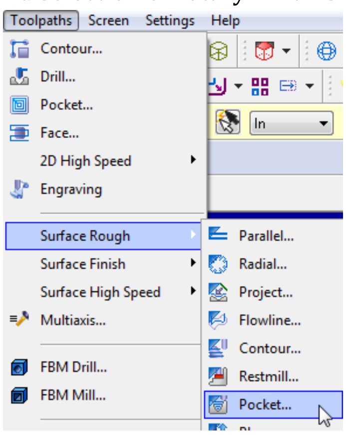


Fig. 11 Selection of surface rough pocket

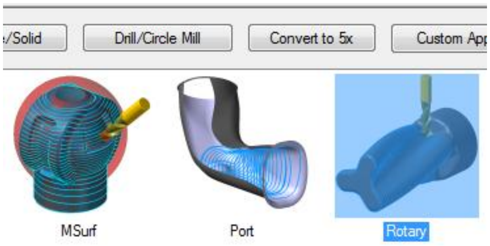


Fig. 12 Selection of rotary 4 - axis

 

For surface rough pocket operation, bull end mill of 25mm diameter and 0.8mm corner radius were used in order to reduce the machining time, since a bigger diameter tool can remove more material per rotation of the tool. But when 25mm diameter were used, the cutter did not remove material on some parts of the work piece, due to bigger diameter cutter cannot enter the sharp corners of the profile. So, in order to remove the remaining stock, surface rough rest mill operation were used with 10mm diameter flat end mill, since the tool can access the sharp corners to remove the material and the formation of radius between the side of the cam and the shaft can be minimized. The parameters applied for surface rough pocket and rest mill for both top and bottom sides are shown in Fig. 13.

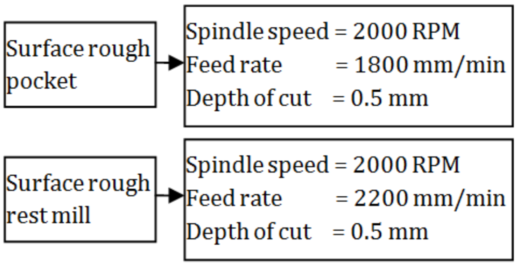


Fig. 13 Parameters applied for pocket and rest mill

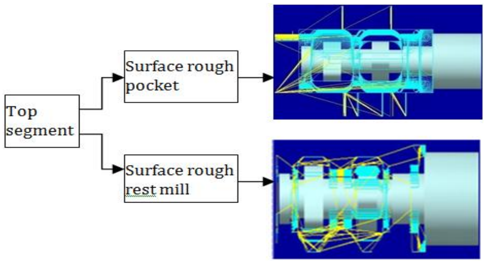


Fig. 14 Tool paths for surface rough pocket and surface rough rest mill

Theoretically, in roughing process, spindle speed should be less than the feed rate. But, in pocketing operation, the amount of material to be removed is high. So, spindle speed applied is greater than feed rate. These parameters were selected considering the material chosen for machining and its cutting speed, diameter of the cutting tool. Except the parameters mentioned in Fig. 13, pocket cutting method also need to be considered. Among the different cutting method, constant overlap spiral method for top segment was selected and high speed method for bottom segment was selected because these method results in reduced cycle time and collision free. The tool paths generated for both surface rough pocket and surface rough rest mill process for top segment is shown in Fig. 14 and bottom segment is as shown in Fig. 15.

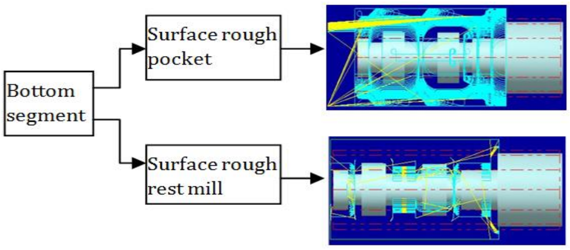


Fig. 15 Tool paths for surface rough pocket and surface rough rest mill

In the tool paths shown, it can be seen that there are difference between pocket and rest mill operations, pocketing process can access most of the region of the profile to remove the material, and rest mill process focuses only on the regions, which cannot be accessed by pocket. If smaller diameter tool was used in pocketing process, it can remove the materials even in sharp corners without rest mill process but the cycle time will be high. So, pocketing along with rest mill can reduce the cycle time as much as possible. The parameters applied for semi finishing and finishing operation are as shown in Fig. 16 and the tool paths generated for semi finishing and finishing operation using rotary 4 axis tool path is as shown in Fig. 17.

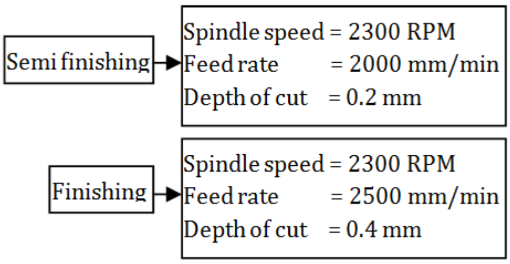


Fig. 16 Parameters applied for semi finishing and finishing operation

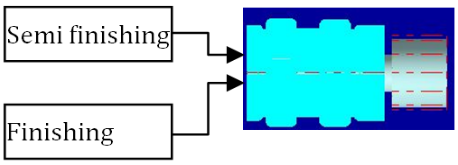


Fig. 17 Tool paths generated for semi finishing and finishing process

It can be seen in the Fig. 17, that the tool path for semi finishing and finishing process is same, and only the parameters applied were different.

 

2.4 *SIMULATION VERIFICATION*

The simulation is verified to ensure there is no collision between the tool and the stock and the machine parts. The verification is done to check the path of the tool around the job. The verification is also used to calculate the cycle time taken by each operation. The graphics after simulation of pocket and rest mill of top segment is as shown in Fig. 18. The graphics after simulation of pocket and rest mill of bottom segment is as shown in Fig. 19.

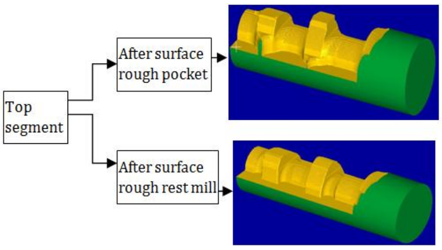


Fig. 18 After simulation of pocket and rest mill

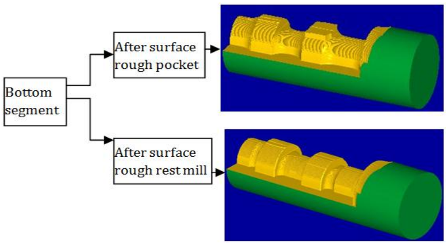


Fig. 19 After simulation of pocket and rest mill

The graphics after semi finishing and finishing simulation process is as shown in Fig. 20.

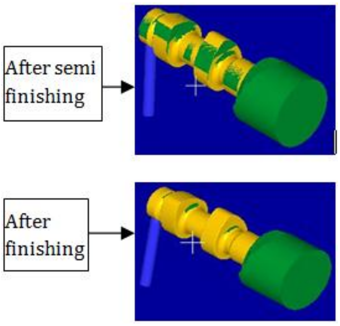


Fig. 20 After semi finish and finish simulation

2.5 3 – *AXIS MACHINING*

The 5 - axis CNC milling machine has totally 5 – axis and they are X - axis, Y – axis, Z – axis, A – axis and C – axis. X and Y axis are the table movement in X and Y direction, Z axis is the tool movement up and down and A – axis is the rotation of work holding device as shown in Fig. 2 and C – axis is kept constant at -90$^o$. The machining after executing pocket operation on top segment is as shown in Fig. 21.

Fig. 21 After machining pocket on top segment

It can be seen on the above figure, that most of the material as per the profile were removed by the bigger cutter and there are left out material to be removed, as the bigger cutter cannot access the sharp corners between the shaft and the face of the cam and at the width of the cam. The machining after executing rest mill operation on top side and rotating the job to 180$^o$ is as shown in Fig. 22. The machining after executing pocket on bottom segment is as shown in Fig. 23.

 

Fig. 22 After machining rest mill rotating the job to 180$^{o}$

Fig. 23 Machining after executing pocket on bottom segment

It can be seen that there are some left out material that is to be machined using rest mill operation. The completed rough machining is as shown in Fig. 24.

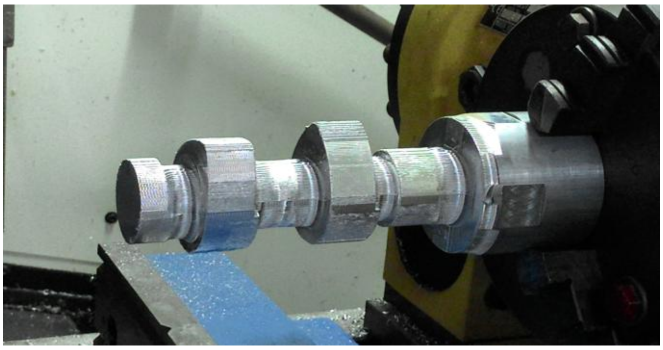

Fig. 24 Completed rough machining

There is a numerical control program to rotate the job to 180$^{o}$ and that is, G0G53A180C-90. This means, the A – axis, that is the chuck fitted with the job rotates from 0$^{o}$ to 180$^{o}$ and the C-90 indicates that, the chuck is to be positioned to horizontal direction, as per the setup shown in Fig. 22.

2.5 4 – *AXIS MACHINING*

Now, the semi finishing and finishing process is to be carried out by applying rotary 4-axis numerical control program using varied machining parameters as indicated in Fig. 16. The job after semi finishing process is as shown in Fig. 25. The finished camshaft is as shown in Fig. 26.

Fig. 25 Semi finishing completed

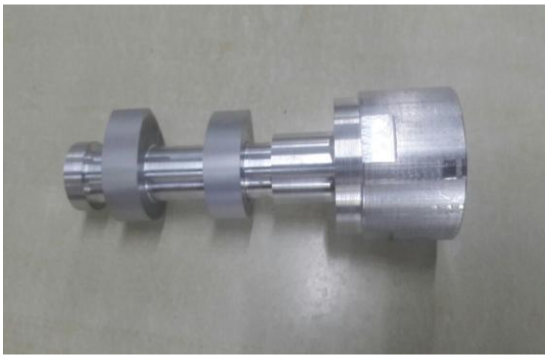

Fig. 26 Completed camshaft

## III. **RESULTS AND DISCUSSION**

During surface rough pocket operation on top and bottom side, different types of pocketing methods were applied simulated and verified using Mastercam software. Using Mastercam software, the cycle time taken by each operation and whether the machining is collision free can be known prior to real machining and this process can be used to avoid collision in real machining and the best tool path and the best parameters, which results in enhanced productivity can be known earlier to real machining. So, using the Mastercam software, damage to the machine parts, danger to operator and the parameters for better machining can be identified easily by applying different parameters, and analyzing them. For a better machining, the pocket cutting parameters applied should result in collision free between the tool, job and the machine parts, and should result in enhanced productivity. Different pocket cutting methods and the cycle time taken for top and bottom side, resulted from Mastercam software is illustrated in Table 1.

TABLE I
Pocket Cutting Methods, Cycle Time taken for Top and Bottom side

| Top side | | Bottom side |
|---|---|---|
| Pocket cutting methods | Cycle time taken | Cycle time taken |
| Zig zag | 31mins:28.17secs | 36mins:46.69secs |
| Constant overlap spiral | 33mins:37.27secs | 36mins:55.61secs |
| Parallel spiral | 35mins:23.76secs | 42mins:34.17secs |
| Parallel spiral and clean corners | 37mins:37.26secs | 48mins:31.91secs |
| High speed | 33mins:12.69secs | 35mins:45.08secs |
| True spiral | 1hrs:1mins:50.33secs | 1hrs:23mins:28.63secs |
| One way | 46mins:11.72secs | 53mins:8.34secs |
| Morph spiral | 1hrs:5mins:49.76secs | 1hrs:40mins:24.51secs |

Table 2 illustrates the occurrence of collision during rest mill operation when applying each pocketing method, that is, during rough machining; pocket machining was carried out along with rest mill operation. So, when rest mill operation was applied during some pocket methods, collision occurred in rest mill operation alone and not during the specified pocket method. So, if a pocket cutting method is applied and if it results in reduced cycle time, the collision occurrence during rest mill operation need to be considered for selecting the pocket parameter.

TABLE II
Occurrence of Collision during Rest Mill Operation

| Pocket cutting method | Rest mill | |
|---|---|---|
| | Top side | Bottom side |
| Zig zag | Collision found | Collision found |
| Constant overlap spiral | No collision | No collision |
| Parallel spiral | Collision found | No collision |
| Parallel spiral and clean corners | Collision found | No collision |
| High speed | No collision | No collision |
| True spiral | No collision | No collision |
| One way | Collision found | Collision found |
| Morph spiral | No collision | No collision |

It is already known that, not only cycle time is considered for better machining, but also there should be no collision during pocket as well as rest mill operation. After analyzing the Table I and Table II, it can be noted that, for top side, methods like zig zag, parallel spiral, parallel spiral and clean corners, and one way results in rest mill collision and high speed method results in pocket collision. So, the constant overlap spiral method for top side seems to be efficient method of pocketing operation since it results in reduced cycle time and collision free during both pocket and rest mill. For bottom side, one way and zig zag method results in rest mill collision, and the other methods results in increased cycle time except high speed method. So, high speed method is preferred for bottom side, as it results in reduced cycle time and collision free. So, the best parameters are selected among other different parameters using the software, without wasting time to find the parameters directly using real life machining. The cycle time resulted by Mastercam during rotary 4 axis semi finishing and finishing operation is illustrated in Table 3.

 

TABLE III
Cycle Time resulted by Mastercam Software

| Rotary 4 axis | Cycle time |
|---|---|
| Semi finishing | 15mins:8.70secs |
| Finishing | 23mins:20.02secs |

Same as like, results of cycle time by Mastercam software, 5 axis CNC VMC also resulted the cycle time taken for each operation. The cycle time resulted by 5 axis CNC VMC after applying the efficient pocketing parameters during roughing on top and bottom side and 4 axis rotary semi finishing and finishing operation is illustrated in Table 4.

TABLE IV
Cycle Time resulted by 5 axis CNC VMC

| 5 – axis machining cycle time | |
|---|---|
| Top side | |
| Operations | Cycle time |
| Surface rough pocket (constant overlap spiral) | 36mins:52secs |
| Surface rough rest mill | 17mins |
| Bottom side | |
| Surface rough pocket (High speed) | 42mins |
| Surface rough rest mill | 11mins |
| Semi finish and finish | |
| Rotary 4 axis (Semi finish) | 1hr:27mins |
| Rotary 4 axis (Finish) | 2hrs:37mins |

For top side pocket, it takes 36mins and 52secs, for top side rest mill, it takes 17mins. For bottom side pocket, it takes 42mins and bottom side rest mill, it takes 11mins. For rotary 4 axis semi finishing, it takes 1hr and 27mins. For rotary 4 axis finishing, it takes 2hrs and 37mins. After comparing, it was found out that, the cycle time taken by 5 axis CNC VMC machine is higher than the cycle time taken by Mastercam software and it may be due to tool travelling time, idle passes, time taken to control the tools and the other machine parts by the machine controller. It was noted that, the total time taken for real life machining is 5.83 hours and 52 seconds.

## CONCLUSIONS

The advantages of using simulation software to identify the best parameter that avoids collision and improve productivity by reducing the cycle time were studied in this paper. The use of computer aided manufacturing software really helped in understanding the tool direction, identified collision between the machine parts and the type of tool that can be used in an earlier stage, before machining. Finding collision and rectifying them and reducing cycle time by changing the parameters during virtual simulation helped in avoiding danger to machine operator and the machine parts and the reduction in time consumption for finding out best parameters. If there was no simulation software's, then it would be difficult for engineers to find out the tool directions, occurrence of collision, and appropriate parameters that results in collision free and productivity. And the another advantage of using 5 axis CNC VMC has resulted in better surface finish, reduced setup time and reduced machining time. The main advantage of 5 axis CNC VMC mill is machining capability of complex shapes under single setup in reduced cycle time. Different parameters of surface rough pocket methods were applied, verified in simulation and constant overlap spiral method was found to be efficient for top side and high speed method was found to be efficient for bottom side. The total cycle time taken for machining of camshaft was 5.83 hours and 52 seconds.

 

## REFERENCES

[1] C. J. Chiou and Y. S. Lee, "A Shape Generating Approach for Multi axis Machining G-buffer Models", Computer Aided Design, Vol. 31, Issue 12, pp. 761-776, 1999.

[2] Jang. D, Kim. K, Jung. J, "Voxel Based Virtual Multi axis Machining" , The International Journal of Advanced Manufacturing Technology, Vol. 16, Issue 10, pp. 709-713, 2000.
C. J. Chiou and Y. S. Lee, "Swept Surface Determination for Five-axis Numerical Control Machining" , International Journal of Machine Tools and Manufacture, Vol. 42, Issue 14, pp. 1497-1507, 2002.

[3] Chih-Ching Lo, "Real-time Generation and Control of Cutter Path for 5 – axis CNC Machining", International Journal of Machine Tools and Manufacture, Vol. 39, Issue 3, pp. 471-488, 1999.

[4] S. Nakkeeran, "Development of Part Program, Simulation and Analysis on Motor Coupler using Mastercam", International Journal of Computer and Organization Trends, Vol. 2, Issue 5, 2012.

[5] www.nscnc.com

[6] blog.hurco.com