OPTIMIZATION OF TOOL PATH THROUGH VERICUT

Kabade D.K.[1] Wawre S.S.[2]
1,2 Assistant Professor , SRESCOE, Kopargaon
Email:[1]kabadedk@rediffmail.com

Abstract--- Computer Aided Manufacturing finds application in highly competitive and constrained industry needs. A program generated by cam module of high end software for given application not directly suited to machine tools. The program to be filtered for keen machining parameters like cycle time, tool life and quality. In present work we are optimizing these machining parameters by using VERICUT software.

Keywords--- NC Program Verification, Inspection & Analysis, CAD Export, Post processor, Simulation environment, Optimization of tool path, Verification of optimized tool path

Introduction:

Nowadays, in metal cutting applications, the "*tape tryout*" (TTO), the first attempt part program execution upon CNC machine tool, is a very difficult and expensive phase in terms of: process time, raw material and human resources**.** Most CAM software tools carry out a cutter location file (CLF) validation, CLF can be considered the tool path before the post processing phase. This simulation offers a preliminary idea of the physical application because, it is totally disconnected from the machine tool and its numerical control. Any tool motion has to be kinematically checked during the first execution in particular the operator has to perform a step by step machining, turning off the rapid motions tool path parts. Rapid motions for modern machine tools can easily reach speed values up to 50 m/min, for this reason they could be the cause, if they are not well designed, of high speed collisions with possible serious damages to work piece and/or machine tools.

Virtual manufacturing is introduced into several CAM tools as additional utilities able to provide a kinematic inspection of machine tool motions. In particular, VERICUT® which is the application used in the present work, it is able to verify the tool motion generated from the post processor.

These computer aided techniques, applied to machining simulation, ensure a good tool path verification only in terms of proper path. Post processing phase, in this virtual environment, detects machined model features and possible collisions (rapid motions that cause contacts between tool and physical part, collision among machine tool components, etc.) No information is available about dynamic interactions (forces, residual stress, and strain) among work piece/tool/machine tool. Machined model creation involves an exact knowledge of the contact conditions existing between the tool model and the raw material model; this information allows the implementation of the optimization module able to find cutting conditions more useful in terms of cycle time reduction, tool wear and surfaces quality.

Methodology:

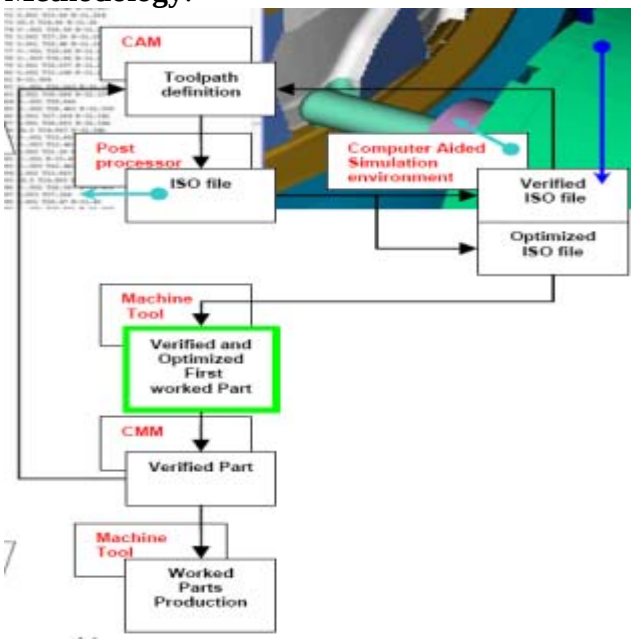

Figure 1 Proposed Methodology

In order to validate the proposed procedure, several steps have been defined as it is reported below.

- Creation of 3D model in UG NX 6
- Tool path creation in manufacturing module in UG NX 6
- Post processor carrying out.
- Simulation environment generation using VERICUT
- Process simulation in VERICUT & verification
- Correction if any
- Optimization of toolpath using OptiPath module in VERICUT
- Verification of optimized toolpath
- Carry actual machining
- Validate results.

In this work tool path is optimized for a part which is machined on three axis milling machine (DMU 50 DECKEL MAHO) with Heidenhain control.

Validation

Creation of 3D model in UG NX 6

It is the first step in this where part is modeled in UG NX 6. Standard modeling procedure followed is boundary representation systems.

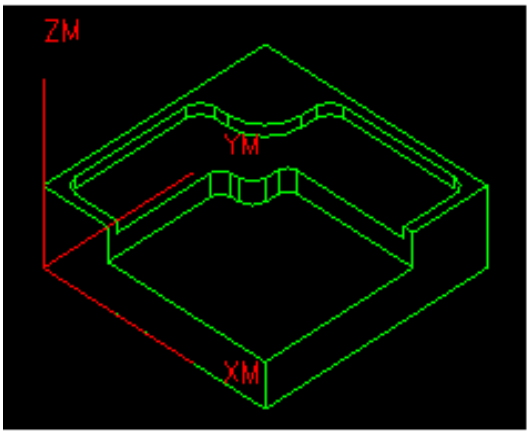

Figure 2 Model in UG NX 6

After creating model it is imported to machining module of UG NX 6

Tool path creation in manufacturing module in UG NX 6 It is the process in which tool path is created in UG NX 6 for given job. For this tool parameters like dimension, feed rate, depth of cut is specified.

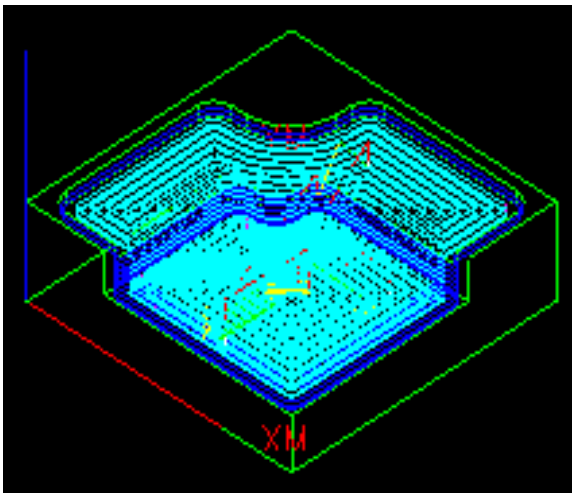

Figure 3 Tool path creation UG NX 6

Post processing in UG NX 6

The main task of post processing is to convert machine tool path source file generated by ug cam into the nc code files that nc program can be understood by cnc system and machine tool. Post processing is the preparation of machine code used to drive a specific machine tool.

Fig.4 Shows tool path source file APT or Automatically Programmed Tool which is a high-level computer programming language used to generate instructions for numerically controlled machine tools.

```
TOOL PATH/FIXED_CONTOUR,TOOL,
TLDATA/MILL,15.0000,7.5000,80
MSYS/0.0000,0.0000,0.0000,1.0
PAINT/PATH
PAINT/SPEED,10
PAINT/COLOR,4
RAPID
GOTO/0.0000,45.0000,38.0000,0
PAINT/COLOR,1
RAPID
GOTO/0.0000,45.0000,3.0000
PAINT/COLOR,6
FEDRAT/MMPM,250.0000
```

Internal Tool Path

Figure 4 tool path source file

Fig.5 Shows NC code file which consists of G-codes and M – Codes. G- Codes are preparatory codes which defines mainly tool motion and M-Codes which are Miscellaneous code used for ON – OFF Functions.

```
N70 G01 Z0.0 F250. M08
N80 Y100.
N90 X100.
N100 Y0.0
N110 X0.0
N120 Y45.
N130 X7.5
N140 Y92.5
N150 X92.5
N160 Y7.5
N170 X7.5
```

Figure 5 NC code file

Optimization of tool path through VERICUT software

Next step is to optimize given CAM generated NC program for optimization of machine safety, machining cycle time and machine finishing and tool life. This is accomplished by using VERICUT software. The procedure for optimization is discussed below

Addition of OptiPath

In order to optimize tool path various optimization parameters are to be set for each cutter.

Feed /Speed**: -** In this window axial depth, radial width, chip thickness, spindle speed & air cut feed-rate is to be entered.

Combined chip & volume method :- Feed per minute is adjusted to maintain either 1) constant chip thickness, or 2) constant volume removal rate, whichever produces a lower feed per minute.

Settings: - Enter minimum feed rate change, cleanup feed rate & leave other entries as is.

Limits: - Set minimum & maximum cut feed rate.

Plunge: - Enter plunge feed rate & clearance distance.

Entry/Exit: - Enter entry feed rate, clearance distance & cut distance.

Angle: - Adjust feed rate for entry angle & click on add

After processing NC program through VERICUT software NC program is optimized for cycle time. This process involves block to block verification of feed rate for given machining condition. If machining is there controlled feed rate is applied whereas for non- maching zones higher feed rate is applied. Fig. 6 shows original nc program generated through CAM software and Fig. 7 Shows Optimized NC program optimizing cycle time by defining feed rate as highlighted.

Fig. 8 Shows a comparison of cycle time for NC Program generated by CAM software to that of optimized tool path generated by VERICUT software. It obviously shows decrease in cycle time increasing productivity.

Original NC Program

\\172.18.50.150\current\5111-tr1\DKK\vericut\101-VERI\101BH-B.h

```
1 BEGIN PGM 1 MM
2 BLK FORM 0.1 Z X0.0 Y0.0 Z-20.000
3 BLK FORM 0.2 X+100.000 Y+100.000 Z0.0
4 TOOL CALL 1 Z S4500; TDia: 9.98, CRad: 1.00
5 CYCL DEF 32.0 TOLERANCE
6 CYCL DEF 32.1 T0.03
7 L Z2.00 F MAX
8 L X-52.691 Y-41.862 F MAX M3
9 L Z+2.000 F MAX
10 L Z-3.433 F MAX
11 L X-52.691 Y-41.862 Z-5.433 F1260
12 L X-49.173 Y-41.862
13 L X-48.964 Y-41.851
14 L X-48.757 Y-41.819
15 L X-48.555 Y-41.765
16 L X-48.359 Y-41.689
17 L X-48.173 Y-41.594
18 L X-47.997 Y-41.480
19 L X-47.834 Y-41.349
20 L X-47.686 Y-41.201
```

Figure 6 Original NC Program (UG NX 6)

Optimized NC Program

\\172.18.50.150\current\5111-tr1\DKK\vericut\101-VERI\101BH-B.opti

```
1 BEGIN PGM 1 MM
2 BLK FORM 0.1 Z X0.0 Y0.0 Z-20.000
3 BLK FORM 0.2 X+100.000 Y+100.000 Z0.0
4 TOOL CALL 1 Z S4500; TDia: 9.98, CRad: 1.00
5 CYCL DEF 32.0 TOLERANCE
6 CYCL DEF 32.1 T0.03
7 L Z2.00 F MAX
8 L X-52.691 Y-41.862 F MAX M3
9 L Z+2.000 F MAX
10 L Z-3.433 F MAX
11TOOL CALL S4510
11LX-52.691Y-41.862Z-4.233F6000
11LZ-5.433F600
12LX-51.909Y-41.862
12LX-49.173F6000
13LX-48.964Y-41.851
14LX-48.757Y-41.819
15LX-48.555Y-41.765
16LX-48.359Y-41.689
17LX-48.173Y-41.594
18LX-47.997Y-41.48
19LX-47.834Y-41.349
20LX-47.686Y-41.201
```

Figure 7 Optimized NC Program by Opti Path Module

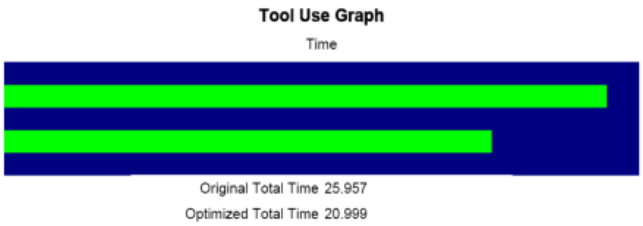

Figure 8 Reduction in cycle time by optimized program

Conclusion

In this paper we have discussed optimization of NC program by Optipath software. Optipath software optimizes tool path by optimizing feed rate for block to block operations so that cycle time is reduced. Again due to constant load on cutting tool its life and surface finish of work piece is also improved.

References

[1] A.Del Pretel, D. Mazzottal and A. Anglani, “Control and optimization of toolpath in metal cutting applications through the usage of computer aided instruments”, 8th AITeM Congress Montecatini Terme 10-12 settembre 2007

[2] A.Soepardi, M. Chaeron, F.L. Aini, “Optimization Problems related to triangular pocket machining”, pp.562-565 IIEM, 2010.

[3] S. Arya, W.Cheng , D.M. Mount, “Approximation Algorithm for Multiple-Tool Milling”, International Journal of Computational Geometry and Applications, vol.11, no.3, pp. 339-372, 2001.

[4] R. Kramer, “Pocket milling with tool engagement detection”, Journal of Manufacturing System, vol.11, no.2, pp. 114-123, 1992.

[5] Y.M. Kyoung, K.K. Cho, C.S. Jun, “ Optimal tool selection for machining in process planning”, Computers Industrial Engineering, vol.33, pp.505-508, 1997.

[6] Unigraphics Solutions User manual

[7] VERICUT, User Manual.

[8] Escar Tools Manual.