CUTTING SIMULATION SOFTWARE FOR MOLD INDURSTRY

Su-Jin Kim
School of Mechanical and Aerospace Engineering
Gyeongsang National University
Gajwa-dong 900, Jinju, Gyeongnam, 660-701, South Korea
E-mail: sujinkim@gnu.ac.kr, http://ma.gnu.ac.kr
Tel: +82-55-751-6075, Fax: +82-55-762-0227

KEYWORDS
Mold Production, NC Milling, Machining Simulation, Tool Path Optimization

ABSTRACT

The mold machining simulation software that will be introduced in this paper is developed to make NC machining more effective. This software includes many digital technologies related to plastic injection mold or press mold machining. It changes the cutting conditions automatically for every different working status. Many mold and press die companies are using the simulation software to get about 27% productivity increasement. Not only machining time but also easier management of machining process is the effect of the simulation software.

1. INTRODUCTION

Mold industry market has been demanding higher quality and shortening manufacturing time. Those demands will be intensifying in the future. In these market circumstances, NC milling processes should be improved to innovational ways for satisfactory of the mold industry market and improvement of competitiveness. NC cutting of the mold manufacturing takes the main portion of time and money.
But there are many difficulties about the NC machining. Machining time and finishing quality come out differently according to CAM data quality and machine operator's skill. Operating NC machine can not progress fluently by lacking of the cutting information. Therefore, it is hard to expect to reducing error rate, and it can't get to maximize productivity through ATC (Automatic Tool Change) unmanned operation. A cutting simulation software was developed by the help of NC machining engineers in mold & die companies to solve those difficulties. During simulation, it regenerates CAM data into optimized NC data in practical way. It makes a NC machine to perform higher productivity, and workshop floor can be change to be better environment by stress less machine operation.

2. CUTTING SIMULATION

The software read tool path generated by conventional CAM software and simulate NC machining. It controls and modifies cutting conditions of tool paths automatically to increase productivity during simulation. In this chapter the implemented algorithms of it is introduced.

2.1 Control Feed Rate Automatically

Cutting load is continuously changed along the work shape. Original tool path use normal feed rate would be F1000 mm/min. The simulation software controls the feed rates, from F800 to F4000 automatically. Faster and safer machining is possible.

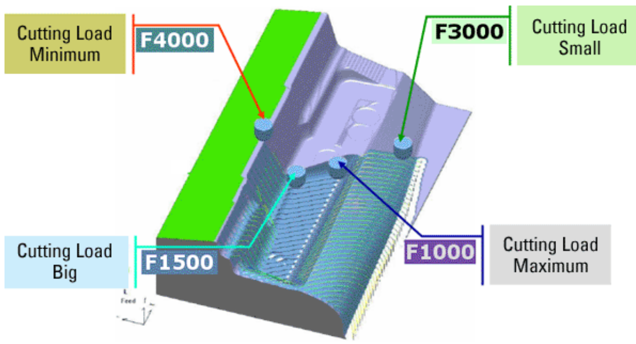

Fig. 1: Control Feed Rate Automatically

2.2 Control Spindle Speed

Spindle speed should be low when cuts massive volume with outer blade and high when cuts slight volume with inner blade. But practically, the real cutting spindle should be set to the left side case. It changes spindle speed for the high efficient manufacturing with corresponding feed rates value.

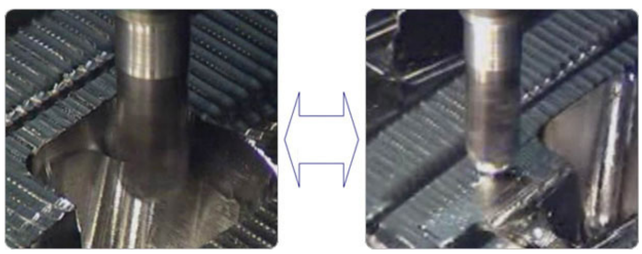

Fig. 2: Control Spindle Speed

2.3 Arc Fitting for Old Controller

There are huge amount of small line segments at corner of parts and these are problems for old-type controller. As all NC engineers know, there are many problems of cutting quality because of sudden feed decrease and hesitant milling motion at corners. It translates massive line segments into simple arc paths for smooth ac/deceleration and fine finishing. The processing time reduced 30% and the quality increased in case of old-type controller.

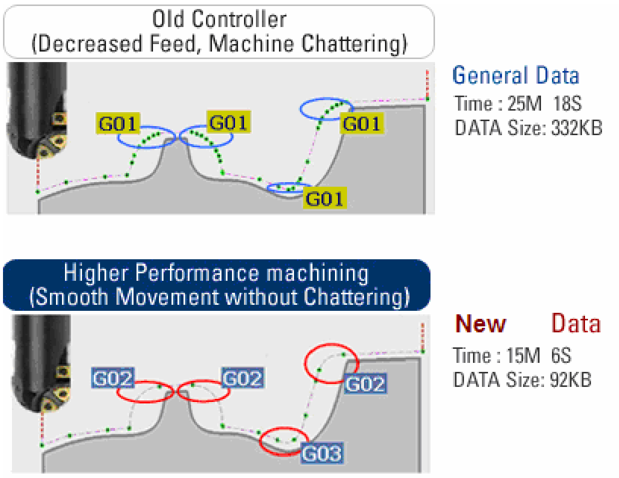

Fig. 3: Arc Fitting for Old Controller

2.4 Add Tool Path at Over Loaded Regions

If the machine makes sounds loud in cutting, feed rate should be reduced or stop the machine to change tool path pattern at CAM software. Some cases can not be solved with feed rate control and spindle speed control. Develop simulation system solve that with self-creating tool path in extremely loaded area. As it shows on figure, it adds tool paths at overloaded area. NC milling can be safe enough to run at night without an operator.

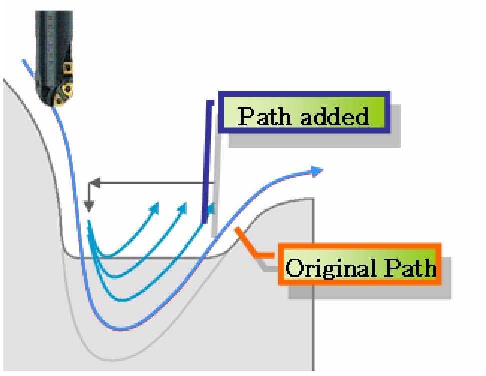

Fig. 4: Add Tool Path at Over Loaded Regions

2.5 Delete Air Cut Tool Path

During the core machining, air cut paths reach 5%, up to 30%. In the figure, the air cut tool paths should be deleted. Only necessary tool path should be remained. Developed simulation system deletes unnecessary air cut paths.
There are many other algorithms, such as output tool and holder setting length information, modifying tool paths for preventing the bottom collision of the cutter.

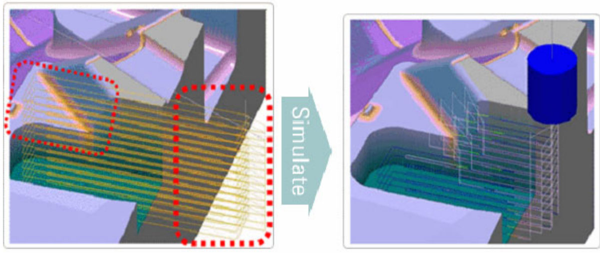

Fig. 5: Delete Air Cut Tool Path

3. APLICATION FOR A MOLD

The difference between general way and proposed way of the NC machining is explained in this chapter. A technical role is also explained.

3.1 A Tested Mold

A mold in figure is difficult to cut because of various size slots and bottom corners between slop faces and curved surfaces. An engineer using CAM software should deeply consider of tool patterns and cutting load when creating tool paths.

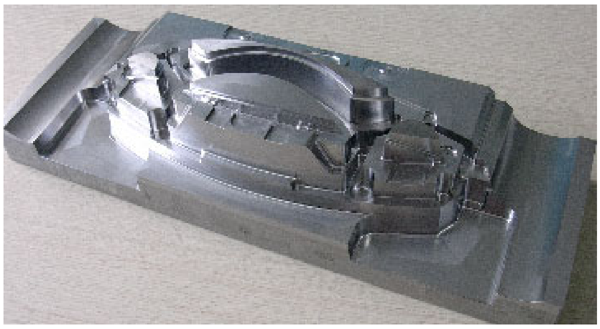

Fig. 6: A Tested Mold

3.2 Shorten Tool Path Length

General mold cores have been cut off by pre-roughing process before NC milling, and it is good to make short machining time and prevents deformation of the stock model. In those cases, tool path of CAM software includes air cut paths a lot. Developed system simulates these tool paths and deletes itself to reduce milling time. Also programming task of CAM could be less stressful.

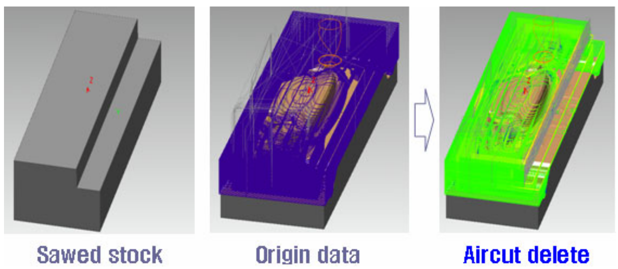

Fig. 7: Delete Air Cut Tool Path

3.3 Protect Tool from Large Cutting Load

Tool breaking when the path met large volume happens frequently because previous step of NC data did not cut off. CAM software couldn't know that kind of big volume area. NC operator should be careful all the time to prevent a tool breakage. This tool breaking problem will never happen with it, because it recognizes remained stock volume and creates additional tool paths.

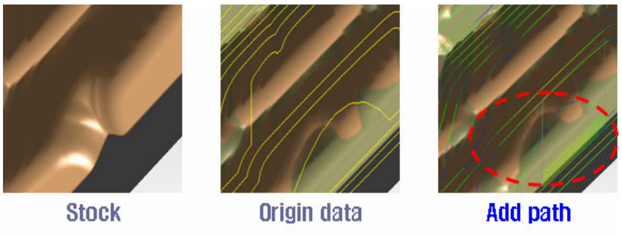

Fig. 8: Add Tool Path at Over Loaded Regions

3.4 Protect Flat Tool from Bottom Collision

Pencil process or re-machining process are usually used to cut off the corner volume repeatedly after the roughing or semi-finishing. But this tool pattern is pencil with Ø16 0.4 tool tip-radius cutter. It is only possible to use for the practical cutting since developed system check up any possibilities to have problems and modifies every tool paths that it may happen any interruption. As all engineers know, flat cutter has no bottom blades. It is very dangerous to use the cutter at –Z direction tool paths. Generally the cutter is not using for the pencil tool pattern. At the picture, tool paths after the simulation have lamp lead-in paths which the system creates by recognizing of –Z direction paths. It prevents tool damages or over-cut. The pencil process is the fastest pattern for removing corner volume and developed system makes it possible, even practically safer.

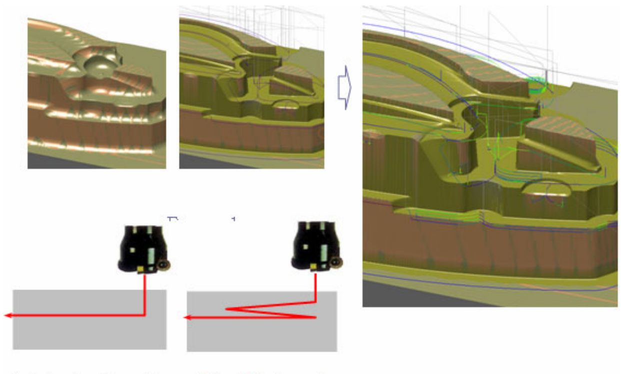

Fig. 9: Add Slop Flange Tool Path

Fallowing case is an opposite function of creating ramp path. At a very narrow shape, the cutter can not cut off stock material at the center area of the cutter. That remained stock will collide with bottom of cutter body. Tool path is deleted when the cutter can't avoid bottom collision.

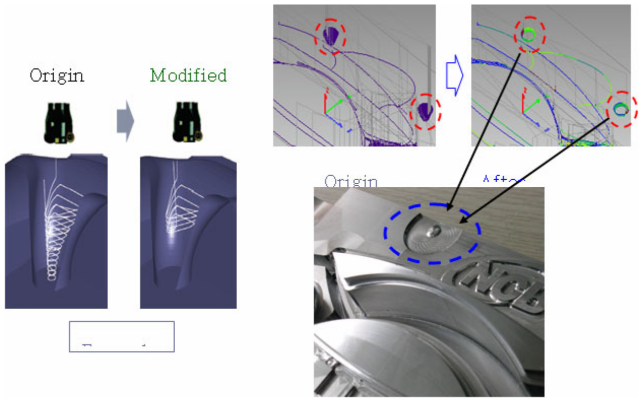

Fig. 10: Delete Bottom Collision Tool Path

3.5 Adjust Rapid Height

Adjusting rapid height makes lower rapid path height to the shape. It makes G0 rapid motion time minimize for higher efficiency. Every rapid path will always have 5mm gap from the surface. Developed system adjusts the G0 rapid motion height and also it rectifies the G1 approach length for the safe machining.

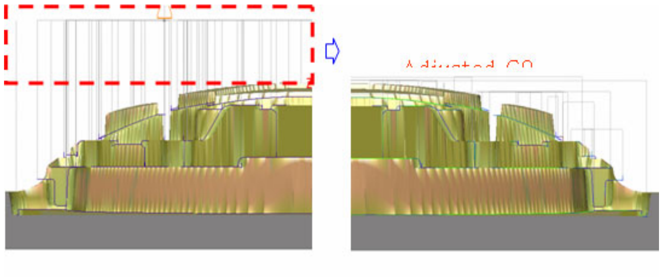

Fig. 11: Adjust Rapid Height

3.6 Control Feed Rate

Cutting load is continuously changed along the work shape. The developed system commands the optimum feed rates value at every NC data blocks for the safe and the fast machining. The tool path color shows feed rate differences. Operator doesn't need to control the feed override dial.

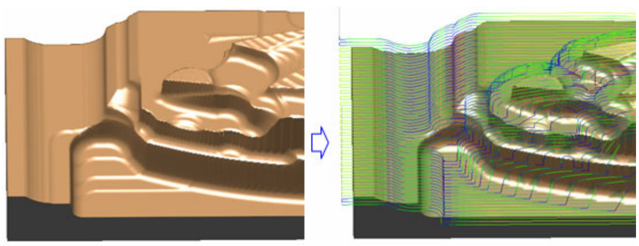

Fig. 12: Control Feed Rate

3.7 Compare to CAD Model

It detects differences between the CAD model and the simulated model. It outputs comparison result by color. The blue shows the under cut, and the red is the over cut area.

The blue area needs the additional electro discharge machining process.

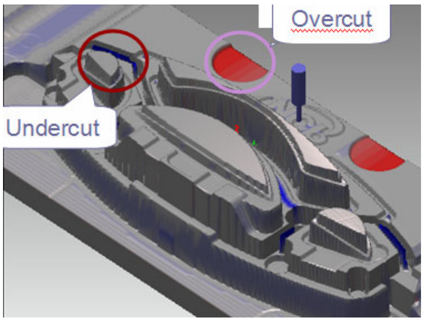

Fig. 13: Compare to CAD Model

3.8 Comparative Results

Comprehensive view about this case, CAM programming time and task stress could be cut short and tool consumption and milling time could be also saved. All functions, such as adding tool paths at overloaded regions, air cut path delete, feed control, are working at once during that developed software simulates every NC data with a stock model. NC data from it could be operated safe and fast. It will be the best way to achieve the productivity with the higher efficiency.

Table 1: Comparative Results

	Conventional style	**Developed system**
CAM	Lots of NC data files are needed Very complex tool path style with tool selection Stressful tasks of CAM	Simple tool path style Less than a half time for CAM tasks
Tool	Use more than 10 kinds of tool used	Less than 6 kinds of tool used
Milling time	About 20 hours Frequent tool break ATC operation disable	About 10 hours (50%) Full ATC operation possible

4. CONCLUSIONS

Developed system upgrades NC machining efficiency practically. Almost all of technologies and techniques about NC milling operation for the mold and press die making are converted into the digital database of it. And it will change the cutting conditions for every different working status automatically. NC machining process can be finished faster and safer by it. It will be helpful to NC operating engineers as a secretary.

Machining time, tool consumption, easier management of manufacturing process, and uncountable benefits including reducing the task stress will make mold makers get positive effect. The result simulation software will give mold & die manufacturers the alternative competitiveness. It will be helpful for mold & die industry.

REFERENCES

K.J. Lee, T.M. Lee, S.J. Kim and M.Y. Yang, 2004, “Hybrid approach to high performance machining”, *Proceedings of the International Conference on High Speed Machining*, (Nanging, China, March 31-April 2), 215-219.

S.J. Kim, 2000, “A study on the feed optimization and NURBS tool path for CNC high speed machining”, *KAIST*, M.S. Thesis

S.J. Kim, T.S. Jung and M.Y. Yang, 2004, “Feedrate optimization using CL surface”, *Journal of Korea Society of Precision Engineering*, Vol. 21, No. 4, 39-47.

S.J. Kim, 2007, “Short and safe tool setting by safe space in NC machining”, *International Journal of Advanced Manufacturing Technology*, Vol. 33, 1017-1023.

BIOGRAPHY

SU-JIN KIM is an asistant professor of mechanical engineering at Gyeongsang National University in South Korea. He obtained a BS degree at Soul National University and MS and PhD degrees at KAIST in 1998, 2000 and 2005 respectively. He was a postdoctoral candidate at MIT in 2005. His research interests include Computer-Aided Manufacturing and NC Machining Simulation.

ACKNOLIGEMENT

The software tool introduced in this paper is developed through the industrial research proects between the author and NCB Co. Ltd (http://www.ncbrain.com). Thanks to president Du-Jin Kim who supported the research and allowed the figues and texts of the company to be used for ISC’2008.