PREPARING OF NC PROBING PROGRAM BY CAM SOFTWARE

Ph.D., Eng. Arkadiusz Kolka
Silesian Technical University
Machine Technology Department
ul. Konarskiego 18a, Gliwice
Poland

ABSTRACT
One of the important an useful option for CNC machine tools are touch probe. It is used mainly for automatic program zero settings. Producers of CNC controllers gives some measuring cycles for some inspection measurement. Practically programs for that purposes are created manually, because requirements of the special measuring cycles. At the last time some options for probing have been included into some CAM software. Example solution for preparing of probing programs for CNC machine tools by use of CATIA Manufacturing software are presented in the paper. Main part of solution is specialized advanced postprocessor. Presented solution enables to prepare advanced inspection programs, which are impossible to be programmed by standard cycles or must be prepared manually. Since now, it is possible to include measuring cycles as a part of cutting NC programs, directly at the process preparing step, by use one environment of CAM system.
Keywords: Touch probe, CNC, Machine Tool

1. USING OF TOUCH PROBE ON MACHINE TOOL

A lot of machine tools are equipped into touch probe for workpiece measuring. The one can be used by CNC machine tool operator during setup, by use specialized CNC controller function. Measurement is possible both in manual and auto modes. Automatic measuring cycles can be used in to CNC programs to improve machining setup of workpiece or for inspection. Practically programs for that purposes are created manually, because requirements of the special measuring cycles. In the practice using of cycles requires to implement manually created probing programs into NC programs created by CAM software. At the last time some options for probing have been included into some CAM software. But use of this options are not so simple like machining programming. To be relay useful, some additional information must be added to planed process. In most cases CAM options are only for simple measuring cycles for elements like holes, slots, points etc..

2. CAM PROBING FUNCIONALITY

One of the most powerful CAM software is CATIA, produced by Dassault. Manufacturing modules of

Figure 1. CATIA probing operations toolbar

CATIA are improved by additional probing options. In this case are possible to use a hole ore pin probing cycles, slot or rib probing cycles, corner probing and multipoint probing. This options gives possibility to define basic measurements cycles. To do it is necessary to define geometrical information, it is done by clicking on the 3D model. For example to measure hole it is necessary to set center point of holes (by point or circle selection), measured body, and top surface. Geometry of measuring tool can be taken from catalog or defined by user if necessary. Most important is to

define probing strategy. At this point measuring depth of holes, safety distance, security distance and number of points are set. Very important is, that user parameters can be set at this point. Knowledge about CNC measuring cycles options is very important at this stage. For example selection of number of points depends not only on cycle parameters but in many cases on the CNC controller parameters and in this case can't be changed by programming.
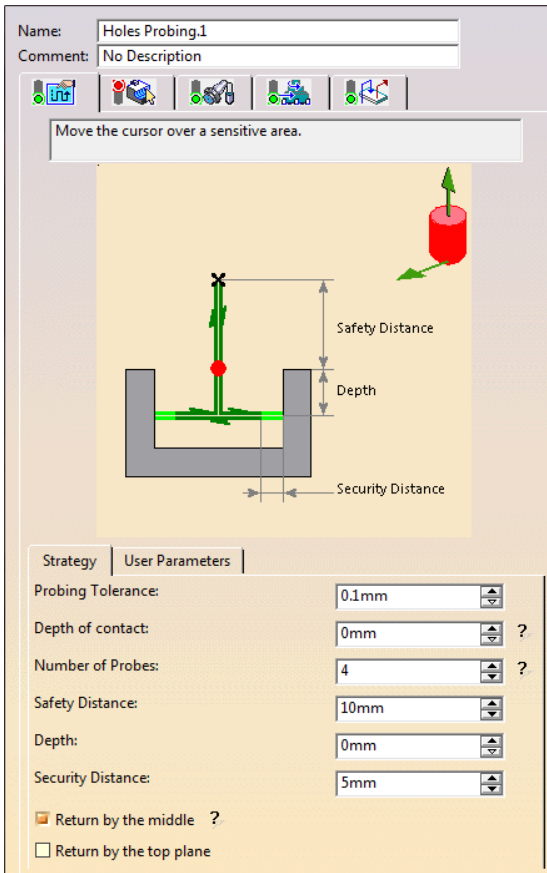

Figure 2. Hole probing strategy definition window

Figure 3. Postprocessing algorithm basing on simple probing operations

3. POSTPROCESSING

Understanding of CNC measuring cycles requirements, and CAM cycles definition is most important thing to prepare good postprocessor. There some assumption must be done. Two kinds of measurement can be done: first – probing for WCS (Work Coordinate System) setup, second – inspection of workpiece dimension. For first situation we can assume, that probing is done for simple geometry ex. WCS can be relative to center of hole or in the corner. In another case multipoint probing must be applied. If post is prepared only for WCS setting, always after o measuring WCS offset values must be setup into controller. Values of offset must be calculated by CNC controller, because real position of workpiece in the machining space isn't known during programming.
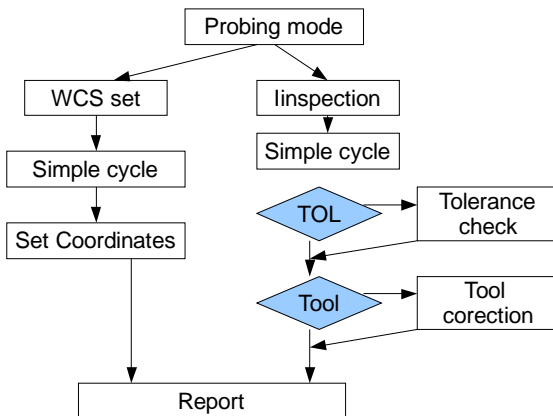
Relative offsets, number of WCS are taken directly from process setup. In the second example, when probing are for inspection only, setting of WCS can't be done. It means first problem for post processing. It is necessary to recognize when We are planning to setup WCS and when We are doing inspection. In the CATIA environment there is no switch for this purpose, fortunately it is possible to include user POST words into process. In this case we include Our new POST word MODE. This word can be applied with two switches MODE/ WCS – for WCS setting mode, and MODE/ INSPECT – for inspection mode. This word must be applied before probing cycles. Another problem is to applying probing results. In the practice probing

results can be used for few purposes: for simple measuring, for tolerance checking, for tool offset correction. Finally We must decide where results of measurement will stored. That makes postprocessing significantly difficult.

To fulfill all above considerations it Was necessary to permit for CAM user to setup all of information which are possible to use for any CNC control. We decided to enlarge number of parameters for MODE/ INSPECT. They can be applied for simple measuring like hole diameter inspection. In this case is only important to setup Probing tolerance, if this value is greater than zero it means that tolerance is checked (if diameter is outside tolerance – program can stop), if is equal to zero then measurement is done.

For tool offset correction it is necessary to inform POSTPROCESSOR that correction of tool based on inspection results is required. For this example it is done by MODE/ TINSPECT command.

At least in the practice most of the probing programs parameterized, and are based on the simple conditional movement command (ex. G31), in this case results of measuring are only the points coordinates. It is possible to store this results as variables, which are used for some calculations by NC programmer. In this case on the CAM side it must be possible to setup names of variables for result storing. We can image that some universal macro procedures can be prepared on this side, and posted to NC program.

5. CONCLUSIONS

An some example and idea of preparing an NC probing programs are presented in the paper. As You Can see problem of touch probe cycles programming by use of CAM system isn't truly solved and a lot of work on the side of POSTPROCESSOR must be done. Author presents some work results which are preparing CATIA Manufacturing for more complex solution of the problem, and prepare enough data for postprocessor. Software for some controllers (Heidenhain, OKUMA) are tested. Future works are taken consideration that preparing of probing programs gives more functionality for programming of complex jobs based not only on basic cycles, but also on conditional moves in any configuration.

6. REFERENCES

[1] CATIA Documentation, Version 5 Relase 18, Dessault Systemes 2007
[2] CATIA V5 AS MULTITASKING MACHINE TOOLS PROGRAMMING ENVIRONMENT, Arkadiusz Kolka, TMT 2008
[3] Dessault System on-line: http://www.3ds.com