

Full Length Research Paper

The design and development of G-code checker and cutting simulator for CNC turning operation

Bagus Arthaya^{1*}, Ari Setiawan² and Santoso Sunardi¹

¹Industrial Engineering Department, Parahyangan Catholic University, Bandung, Indonesia.

²Industrial Engineering Department, Institute Teknologi Harapan Bangsa, Bandung, Indonesia.

Accepted 2 May, 2010

CNC machines have been becoming popular in manufacturing sector for the last four decades. It still costs a lot of money to afford some aspects related to the operations of these machines, such as the hardware, the software and the brainware. One important aspect is the programming stage where this activity requires a skillful programmer and an appropriate programming media. To hire a good programmer is not a cheap and the best solution while having a commercial programming media is also not affordable for Small Medium Enterprises (SME) in Indonesia. This research was focusing on the design and development of a kind of program checker to see whether the program for a CNC turning machine is correct and ready to be fed to the machine. At once, this program was also capable of displaying cutting simulation of how the product to be cut in the machine. Therefore this program will help a small manufacturing company to avoid hiring an expensive expert or buying a special programming media. This program was specifically developed for a CNC Okuma Howa ACT 3 turning machine but it can be extended to other machine types. Some results were showed that the program can easily point out error location if there is a logical or syntax error in the NC program. Moreover, the simulation can be executed only if all errors have been corrected. This feature ensured that the simulation can run properly. To run the simulation, one has to set up some variables such as workpiece dimension, tool position with respect to the machine, cutter dimension and so on.

Key words: CNC machine, turning operation, G-Code checker, cutting simulation.

INTRODUCTION TO CNC MACHINING

Computer technology has been increasingly and intensively used in modern manufacturing processes. CNC machines have been becoming a very popular tool to produce machined parts. These machines are operated by means of computer control and the movements of the cutting tool are directed by an NC program. Quality of the machined part depends on the selection of cutting parameters and the way the part is being cut. To facilitate these conditions, an appropriate NC program has to be prepared. This program contains a list of codes that can be understood by the CNC machine controller.

NC programming is a monotonous and time consuming task but it is in fact an unavoidable job. Incorrect NC program may lead to wasting of raw material, shortening of tool life time, wasting of production time and so forth.

Concerning these kinds of problem, an idea to develop a program that can check the syntax of NC program, path of the cutting tool during the cutting operation, the changes of the part shape, is very useful. It helps an NC programmer to deal with these monotonous and time consuming tasks.

The features of this program are ability to view and check the NC program line by line, to point out where the error occurs, to identify what error source might exist. These can therefore prevent any waste caused by incorrect programming in the very early stage. Visual simulation will help a programmer to see where the mistake might happen and to watch whether the written program runs as required. An NC program is ready to be fed to the CNC machine, when all logical and syntax errors in the program have been corrected and product final shape is somehow correct.

The MTS CNC Simulators are employed in production as well as in vocational training and further education. Every effort has been made to meet the complex

*Corresponding author. E-mail: bagusart@home.unpar.ac.id.
Tel: +62-85793338681. Fax: +62-022-2032700.

requirements in both these fields of application by including appropriate new functions in the current software, which makes the MTS Simulators one of the most efficient systems on the market today. This high performance is accompanied by increased clarity and convenience of operation, due to the new user interface and function keys (Metal Cutting Processes, 2010).

MTS Simulators can simulate all aspects of a CNC machine tool, as well as they can also test NC programs for actual production; this includes checking for possible collisions as well as establishing work piece geometries based on the actual state of machining. No complicated changes of programming or control systems are necessary. The NC Editor supports direct entry or editing of NC blocks via its programming interface, especially suited for NC programs. Each NC block entered is subject to an automatic syntax check ("syntax" in this context being the internal structure of the NC blocks). There is also a check to ensure that the values entered fall within the permitted ranges (Metal Cutting Processes, 2010).

CNC 2000 is windows compatible software for simulating CNC program of turn/mill/top/WEDM. It allows user to enter NC program in G/M codes and see their movement. This package also has various video files to see actual working of CNC. This package is useful to provide training in CNC Technology. CNC 2000 has turning simulator which allows user to enter NC program using G/M codes and to checks its syntax as well as to simulate them on PC screen. This covers all the cycles available with CNC. It gives user a better environment for making efficient NC program. It has built in time estimation facility as well (CNC, 2000).

Chen et al (1998) presents a model that characterizes interactions among the subsystems of a computer numerically controlled (CNC) lathe. The model is combined with a cutting force model to obtain a comprehensive turning simulator that simulates the cutting forces and part dimensions. The interactions among the subunits of a CNC lathe and the cutting process are found to be potentially important. It is necessary to understand the interactions among machine tool components and the cutting process in order to improve productivity and quality in machining.

All of those mentioned machining simulators still cost a lot of money especially small and medium size industries in Indonesia that have been suffering from the national crisis since the last decade. They can not afford to work with modern manufacturing facilities such as CNC machines. The one who wants to work using CNC machines, they mostly can not afford to recruit/hire a skillful CNC programmer. By using this kind of simulation program, they can prepare an NC program by ordinary machine operators without having to physically have a CNC machine. At first, they have to prepare and check the NC program and simulate the cutting operation carefully. Afterwards, error free program can be sent to the CNC machine. Eventually, a small industry can involve their operations with CNC machine without having to hire

expensive operator and can prepare the NC Codes by themselves.

THE RESEARCH OBJECTIVES

This research is conducted to facilitate small and medium size industries in Indonesia in manufacturing parts/product using modern manufacturing processes. It is also meant to provide a simulating media for some higher educational institutions (vocational schools) in Indonesia that still have not had any CNC machine. It is very helpful when they can be more educated towards this relatively modern technology and can be prepared to get ready utilizing CNC machines. In this work, turning process and NC programming related matters become the focus of discussion. In turning, cylindrical or symmetrical rotating parts are produced. Based on the requirements briefly explained, two problems are stated such as: how to build an algorithm that is capable of checking and verifying the NC Code (program) of a turning process and how to design a simulation facility that visually simulates the cutting in turning operation.

In this research there are some constraints/limitations applied to this work such as:

1. The turning machine used is CNC Okuma Howa ACT 3 with Fanuc 6T controller.
2. G canned cycle and macro have not been implemented yet.
3. The cutting operations are valid only for turning, finishing, threading, drilling, boring, and parting off or necking.
4. The machine vibration and cutting forces are neglected.

The software developed in this work is called "G Lathe Simulation Pro 2006 v1.0". This software is developed within a C++ programming language environment (Zukhri, 2005) and based on basic machining knowledge described in (DeGarmo et al., 1999; Rochim, 1990).

TURNING PROCESS AND NC PROGRAMMING

By its nature, turning is a process of cutting external or internal cylindrical, conical or any symmetrical rotating surfaces on a lathe or turning machine. Turning is one of the several cutting processes that are performed in a lathe whether it is manually operated or aided by computer control. The work piece is held by a rotating chuck and then cut by horizontally positioned single point cutting tool that can move on a horizontal plane (ZX plane) as depicted in Figure 1a. In contrast to turning, milling is a machining process generally performed in a milling machine where the work piece is fixed/clamped on the machine table and is fed into a rotating cutting tool as shown in Figure 1b.

The machined parts produced by turning operation are

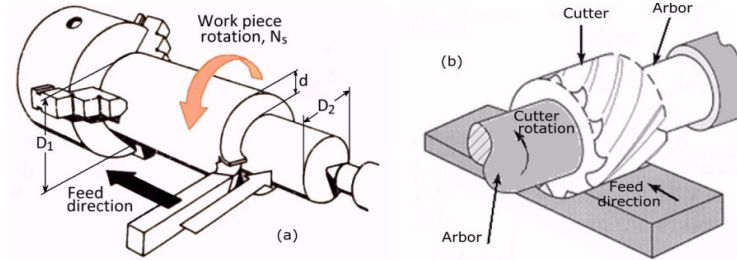


Figure 1. Basic cutting operation performed in turning and milling.

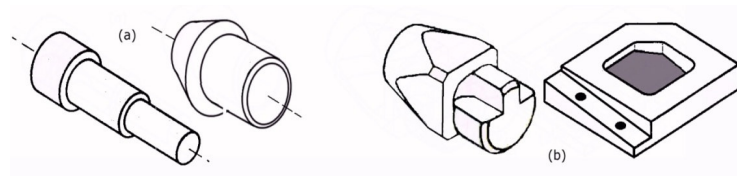


Figure 2. Typical parts produced by (a) turning and (b) milling processes.

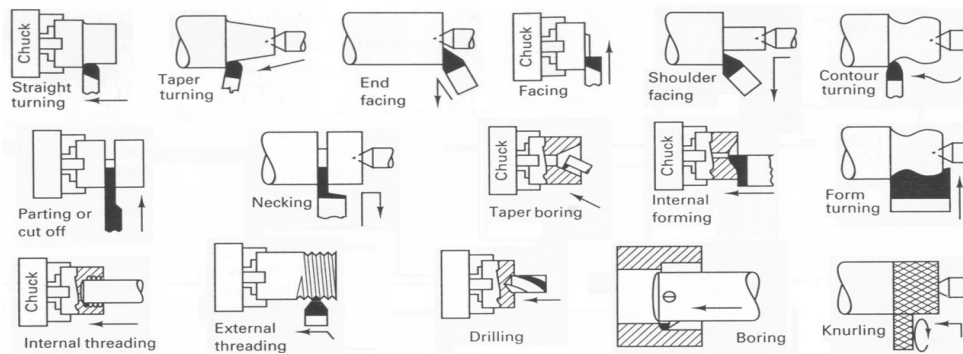


Figure 3. Several machining operations typically performed on a lathe (DeGarmo et al., 1999)

symmetrical rotational parts, such as shafts, cones, cylinders, irregular surface of limited length and so on (Figure 2a) as explained in DeGarmo et al. (1999). On the other hand, work pieces produced in milling process are mostly rectangular like parts, as clearly shown in Figure 2b, although rotational parts or sculptured surfaces are often the case. In fact, turning is not the only machining operation can be done in a lathe. Other machining operations that can be carried out in a lathe are drilling, boring, facing, parting (cut off), threading, profile and contour turning and necking as shown in Figure 3.

CNC lathe is a turning machine equipped with a computerized controller that replaces the human role in a conventional machine to synchronously control all movements of cutting tool when performing the cutting operation. At the same time, cutting parameters such as cutting speed, feed-rate and depth of cut can be adjusted easily. CNC machine has higher performance than a conventional machine in the sense of accuracy, precision, and productivity, as well as the complexity of

work pieces that can be processed.

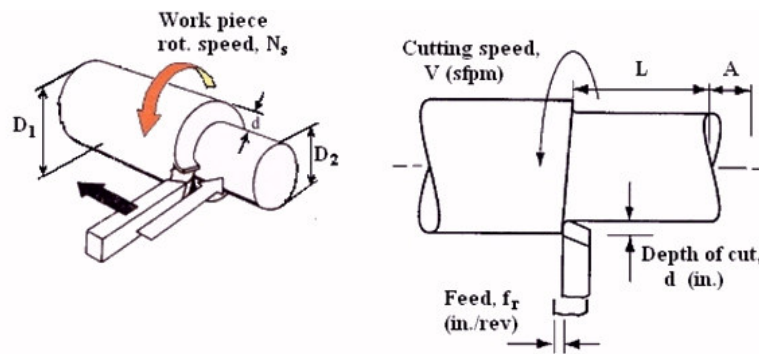
To perform these actions, the machine needs a program that translates all the tasks into a list of machine codes. These machine codes associate with cutting tool motions relative to the work piece. The codes are standardized and have been accepted worldwide by the machine manufacturers as well as the industries. A list of machine codes is known as NC program and it has some functions such as:

1. To define the sequence of operations in the form of coded instructions that is understood by CNC machine controller and is required to fabricate the part.
2. To give the X and Z coordinate positions of the cutting tool.
3. To specify the tool traverse that determines the depth of cut, feed-rate and spindle speed, etc. so that the machine can perform the cutting operations automatically.

A general NC program consists of the following part (De Garmo et al., 1999; Stenerson et al., 1997; Falck, 2008).

Table 1. Definition of common NC words (G-codes) as in DeGarmo et al. (1999).

NC word	Use
N	Sequence number: identifies the block of information
G	Preparatory function: requests different control function, including preprogrammed machining routines
X,Y,Z,I,J,K	Dimension coordinate data: linear and angular motion commands for axis of the machine
F	Feed function: sets feed rate for the operation
S	Speed function: sets cutting speed
T	Tool function: tells the machine the location of the tool in the tool holder or tool turret
M	Miscellaneous function: turns coolant on or off, open spindle, reverse spindle, tool change, etc.
EOB	End of block: indicates to the CPU that a full block of information has been terminated and the block can be executed

**Figure 4.** Definition of machining parameters related to turning operation.

1. Blocks: are basic unit of the NC program containing some NC words as tabulated in Table 1 which consist of sequence numbers, preparatory function, dimension coordinate data, feed function, speed function, tool function, and miscellaneous function.
2. Sequence numbers: are numbers pointing to the sequence of block that is being executed.
3. Preparatory functions: are some reserved/standardized words used to determine a specific motion followed by a reserved integer.
4. Dimension coordinate data: are numbers following the axis of motion, theoretically valid for real number between 0 and 9999.
5. Feed functions: is a word determining the feed of cutting tool when executing the cutting process.
6. Speed functions: is a word determining the speed of cutting the work piece.
7. Tool functions: is a word determining the number/ position of cutting tool to be used.
8. Miscellaneous functions: is a word to control some specific machine functions that is turning on or off the coolant, reversing the spindle speed, changing the cutting tool and so on.

Cutting process in turning

The most well known metal cutting processes are turning

and milling. Turning is a process of removing material from a rotational part both on the external as well as in the internal surfaces. Using a lathe or turning machine one can make a cylindrical bar, a stepping shaft, hollow rotational part, thread and so forth. As mentioned above, other machining processes that can be performed in a lathe are drilling, boring, facing, threading and so on as shown in Figure 3. These types of cutting are possible by feeding the cutting tool at an angle to the axis of rotation of the part or by adjusting the movement of the tool with respect to the rotational axis. As an example, a flat surface is produced by moving the tool at an angle of 90° to the work piece axis of rotation.

The speed of rotation (N_s) defined in rpm has to be defined based on the material to be cut and dept of cut chosen. If it is defined too big, it can generate a high cutting force and then vibrate the tool mounting structure. It is normally defined according to Referring to Figure 1. Equation for calculating N_s is derived as follows:

$$N_s = \frac{12V}{\pi D_1} \quad (3.1)$$

Where 12 is a conversion factor from feet to inch as depicted in Figure 4.

From this, cutting time (CT) defined in minutes can be calculated as follows:

Table 2. Definition of standard G-Code used for Fanuc 6T controller (Fanuc Ltd., 1983)

Standard G-Code	Group	Function	Syntax checking by GLSPro2006	Simulation by GLSPro2006
G00		Positioning (Rapid traverse)	Yes	Yes
G01	01	Linear interpolation (Feed)	Yes	Yes
G02		Circular interpolation CW	Yes	Yes
G03		Circular interpolation CCW	Yes	Yes
G04		Dwell	Yes	Yes
G07	00	Feed rate sine curve control	Warning	---
G10		Offset value setting	Yes	Yes
---	---	---	---	---
G90	03	Absolute input	Yes	Yes
G91		Incremental input	Yes	Yes
---	---	---	---	---
G96	02	Constant surface speed control	Yes	---
G97		Constant surface speed control cancel	Yes	---
G98	05	Per minute feed	Yes	---
G99		Per revolution feed	Yes	---

$$CT = \frac{L + A}{f_r N_s} \quad (3.2)$$

Where L is the cutting length and A is the overrun allowance (both in inches) and d is depth of cut, while f_r is the feed (in feet) per revolution. In this case, A is assumed to be small enough (neglected) compare to L .

The value of CT in Equation (3.2) will be considered as the contact time between tool and part in the simulation stage which in turn becomes the cutting time although when in A the cutter is not in contact with the part. In turning operation, the relative motion between the tool and the part (in the form of tool path) must be generated to get a desired external and/or internal forms of the workpiece. The movement of this tool can be a straight line, circular or contour line as well as a combination of them all. For a CNC turning machine, all of these motions should be prepared before executing them directly into the machine. Generating an NC program is not an easy job, whilst it should be done carefully and be based on the knowledge about machining processes and related topics

NC part programming

Part programming is a critical step in operating a CNC machine. The program is a kind of communication language between the operator and the machine. Moreover, in this communication the language has a certain grammar that has to be obeyed. The language consists of a list of standardized codes that briefly mentioned before.

The preparation of NC program is obviously an important part of CNC programming. It involves compiling

some numerical codes (words) that are generally already standardized and some of them in NC programming listed in Table 1. These words become the basic ingredients to write an NC program and some of them are obligatory while the rests are optional. The NC words are put together in a specified order to define a block of information needed to execute an operation.

This order has a special meaning and the algorithm developed in this work is based on this regulation. Machine operator (programmer) first must ensure that the words order and all necessary words within a block are already placed correctly, before the machine can execute the block. In this work, a dedicated algorithm for checking the syntax is developed. Moreover, checking and simulation widows are prepared to allow the programmer to identify any possible mistakes.

To define a correct block, order and completeness of the words are very important. If one of the obligatory words is missing, the machine will not move to the right position. An example of missing the value of S word in a program block would be:

“N0100 G01 X40.5 Y20.0 S F100”

This block is constructed using six words that is N0100, G01, X40.5, Y20.0, S and F100. It tells the tool to move towards a point $(x, y) = (40.5, 20.0)$ at a feed rate of 100 mm/s, but no cutting speed is defined. This block can certainly not be executed in the machine because the controller will not let the machine work without rotating the spindle. If correction has already been done, the NC checker will allow the user to proceed to the next step, that is, the simulation step.

Table 2 provides some standard G-Codes (Preparatory functions) used by controller Fanuc Ltd., (1983). This

Table 3. Definition of standard M-Code used for Fanuc 6T controller (Fanuc Ltd., 1983).

Code	Use	Syntax checking by GLSPro2006	Simulation by GLSPro2006
M00	Program stop	Yes	No
M01	Optional program stop	Yes	No
M02	Program end	Yes	Yes
M03	Spindle ON clockwise	Yes	No
M04	Spindle ON counterclockwise	Yes	No
M05	Spindle OFF	Yes	No
M06	Tool change	Yes	Yes
M07	Coolant 1 ON	Yes	No
M08	Coolant 2 ON	Yes	No
M09	Coolant OFF	Yes	No
M30	End of program, reset to start	Yes	Yes

Table 4. Definition of standard T-Code used for Fanuc 6T controller (Fanuc Ltd., 1983)

Code	Use	Syntax checking	Simulation
T01	Turning / Facing left	Yes	Yes
T02	Turning / Facing right	Yes	Yes
T03	Finishing turning / Facing left	Yes	Yes
T04	Finishing turning / Facing right	Yes	Yes
T05	Grooving / Parting	Yes	Yes
T06	Threading	Yes	Yes
T07	Internal threading	Yes	Yes
T08	Drilling	Yes	Yes
T09	Boring left	Yes	Yes
T10	Boring right	Yes	Yes

table also explains that the software “G Lathe Simulation Pro 2006 v1.0 (GLSPro2006)” can already check the syntax for the codes marked by sign “Yes”. If the sign “warning” appears in the column, it means that the software has been able to check the G-code, but it has not been able to check the parameter code that follows the G-code yet. When the sign “Yes” appears in simulation column then this software can perform the G-code simulation. Those sign let the user to have a general idea about what the software already can do. The functions that can already be simulated are the most basic movement example: G00, G01, G02 and G03 etc. They can give enough visual representation when a cutting tool moves and cuts the work piece according to the prepared NC program.

In addition to all of these words, an extended version of M-code used in Fanuc oi-MC controller is listed in Table 3. This table provides more detailed information to be used in controlling miscellaneous functions as mentioned in Table 1.

The words enable the users to control many of the I/O functions of a machine, for instance to start spindle, to turn on/off the coolant. The words can also signal the end

of a program or a stop within a program. Table 4 shows the use of T-code for tool changing purposes from which one can perform other processes in turning machine. It can be clearly seen that not all of the M-codes can be simulated in the software tool. In contrary to the T-codes, all of the functions have been able to be simulated in this software.

There are many types of cutting tool normally used in turning operations. Based on what operation will be performed, the suitable and correct geometry of cutting tool can be chosen. A lot of parameters have to be considered in order to choose the most suitable cutting tools for a certain operation. Types of cutting tools usually used for turning are depicted in Figure 5.

Right and left turnings use different type of tools. Tools in group (a) generally perform operations from external/radial direction, that is: facing left or right, grooving or parting, and threading. Parameter of cutting tool such as tool offset in X and Z directions are very important in positioning the tools correctly. Meanwhile, group (b) contains tools for working from the tailstock direction. They are normally used for internal operations, such as internal threading, drilling and boring.

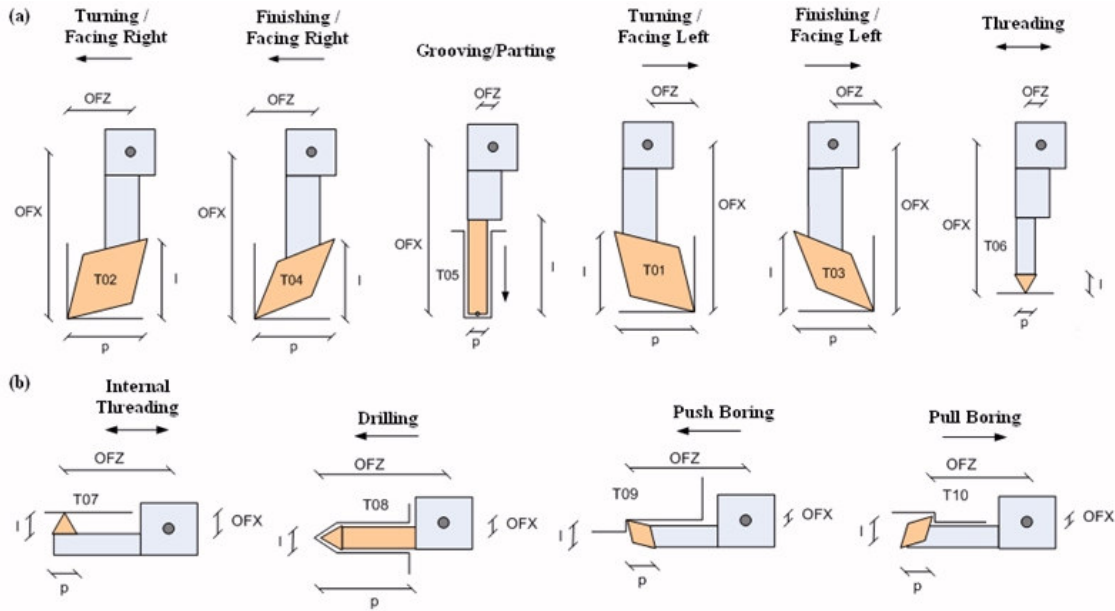


Figure 5. Several cutting tools used in CNC turning for: (a) external and (b) internal cutting.

Table 5. Machine specification used in GLSPro2006.

Specification	Range
Controller	Fanuc 6T
Feed rate (F)	1 – 12112,752 mm/min
Spindle speed (S)	35 – 35000 RPM
Max. traveling distance in X-axis	156,46 mm
Max. traveling distance in Z-axis	605,79 mm
Max. work piece length	585,47 mm
Max. work piece radius	106,61 mm
Number of tools in the magazine	10

Machine specification

Fanuc 6T Controller is commonly used to control CNC lathes, and one of them is Okuma Howa ACT 3. This machine is chosen as a reference for developing GLSPro2006 software in demonstrating the simulation capability. Table 5 lists the machine specification and Figure 6 shows the effective working area in the machine.

The work piece is clamped using 3 or 4 jaws chuck securely and the maximum length and radius of work piece to be cut is 585,47 and 106,61 mm. Outside this area there is a free area for the cutting tools to move freely after machining steps. The maximum space available in this machine is 605,79 x 156,46 mm. A tool home start point is defined at the corner of this space and is used in the simulation as starting point for cutting tools to move.

Development of the algorithms

The development process of G-Code checker and

simulation program is started by making some algorithms used for preparing the NC program. These algorithms apply when one executes a program listing into this software tool. The general algorithm is divided into two main programs, that is. syntax checking algorithm and cutting simulation algorithm as they are depicted in Figures 7 and 8.

The syntax checking algorithm consists of the following, that is:

- Main Syntax Checking Algorithm, used when performing syntax check of an NC program.
- Comment Checking Algorithm, applied when a comment in an NC program is found. Any character stood between the sign “(“and the sign “)” will not be checked or will be neglected.
- NC Program Opening Checking Algorithm, used for checking whether there exist a symbol of “%” as the opening of NC program and there exists a symbol of “O” or “:” as the program number.

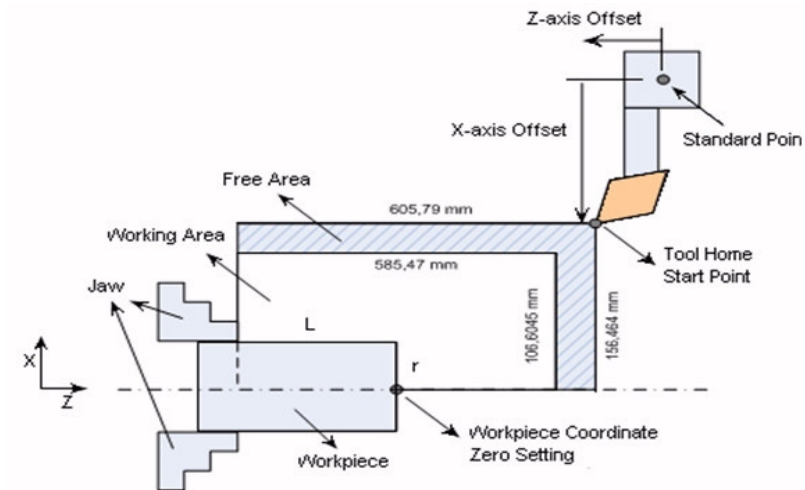


Figure 6. Effective working area inside Okuma Howa ACT 3 lathe

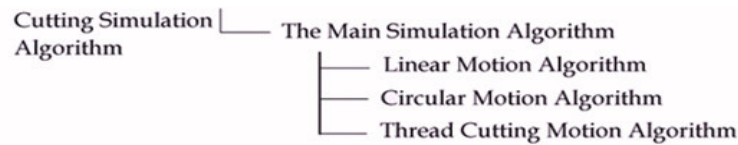


Figure 7. The motion algorithms contained in cutting simulation.

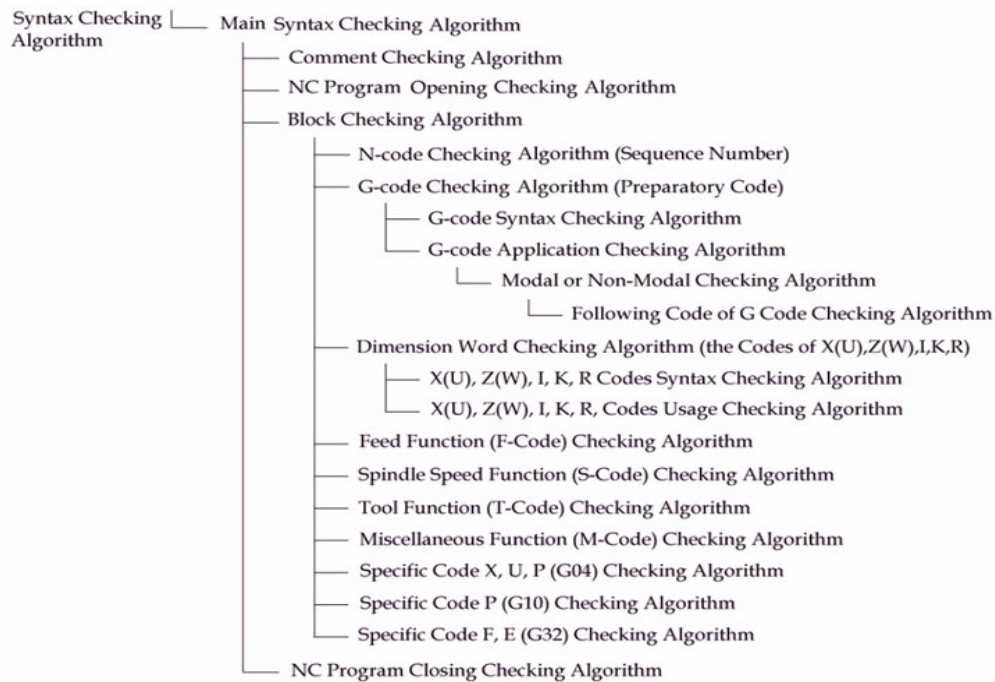


Figure 8. The hierarchy of syntax checking algorithm.

d. Block Checking Algorithm, calls other corresponding algorithm when a specific code is detected.

e. N-code (Sequence Number) Checking Algorithm, applies when N code is detected. It will check the digit

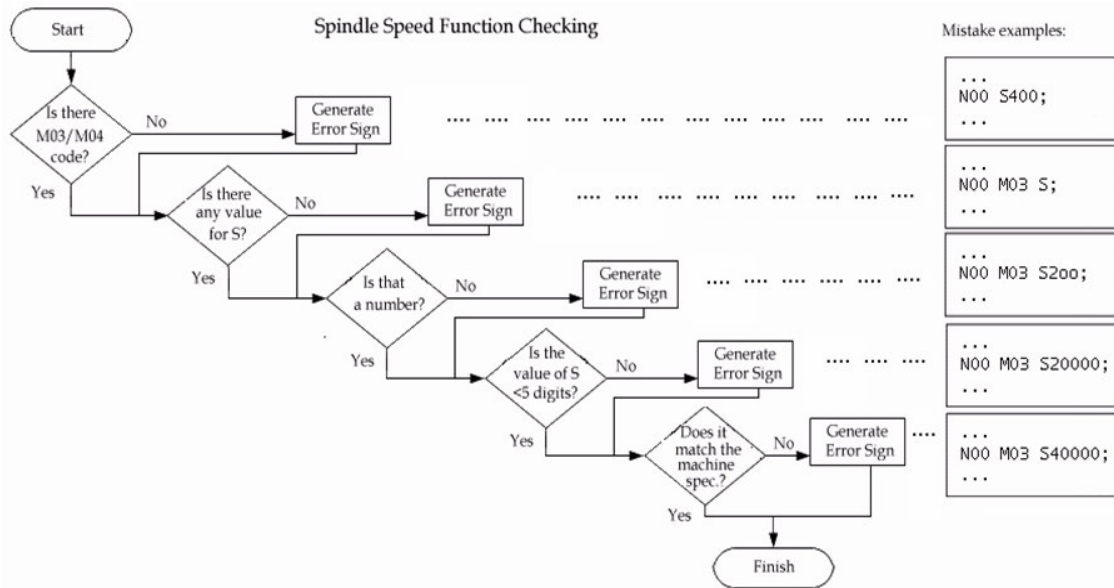


Figure 9. Flow chart of spindle speed function checking algorithm.

and value of the corresponding N code.

f. G-code (Preparatory Function) Checking Algorithm, applies when G code is detected. This is divided into two main flows: G code syntax checking and G code application checking algorithms.

g. G-Code Syntax Checking Algorithm, used to check the values of the code whether it has two digits or consists of number or other characters and so on.

h. G-Code Application Checking Algorithm, used to check whether usage of G code has complied the implementation prerequisites such as G modal or non modal, and other specific following codes.

i. Modal or Non-Modal Checking Algorithm, used for checking whether the G exists in the database and also for checking whether the G includes in modal or non modal. It also checks the correlation with the G codes appeared in previous blocks.

j. Following Code of G-Code Checking Algorithm, used for identifying whether a G code has been followed by appropriate codes.

k. Dimension Word Checking Algorithm (the Codes of X(U), Z(W), I, K, R) applies when the dimension word is detected such as X(U), Z(W), I, K, R. It is divided into two parts: syntax checking algorithm and usage checking algorithm, X(U), Z(W), I, K, R Codes Syntax Checking Algorithm is used to check the code value such as number of digit, type of character and so on.

l. X(U), Z(W), I, K, R, Codes Usage Checking Algorithm is used to check whether the usage has obeyed the rules. The rules are for instance the availability of following code, and prohibition for code I and K to be appeared together with code R.

m. Feed Function (F-Code) Checking Algorithm, used when code F is detected then check is done for number

of digit, accordance to machine specification.

n. Spindle Speed Function (S-Code) Checking Algorithm, used when code S is detected. Similarly some checks are done for instance number of digit, accordance to machine specification, the availability of code M03 or M04.

o. Tool Function (T-Code) Checking Algorithm, used when code T is detected. Check involves examining number of digit and the database.

p. Miscellaneous Function (M-Code) Checking Algorithm, used when code M is detected. Check involves examining number of digits and the database.

q. Specific Code X, U, P(G04) Checking Algorithm, used when code X or U or P is detected where previously code G04 (Dwell) is defined.

r. Specific Code P (G10) Checking Algorithm, used when code X or U or P is detected where previously code G10 is defined.

s. Specific Code F, E (G32) Checking Algorithm, used when code E is detected where previously code G32 or G34 is defined.

t. NC Program Closing Checking Algorithm, used for checking the closing of an NC program, such as the existence of M02 or M30 code as the end of NC program and symbol % as final closing.

An example of syntax checking algorithm is Spindle Speed Function Checking (S-code). It starts with checking the existence of speed codes in the corresponding block: the M03 or M04. Afterwards checking the existence of speed value is done followed by examining whether that value is a number or other character, how many digits are used as the speed value and finally checking if this value is within the machine specification. The action is presented in a flow chart as shown in Figure 9.

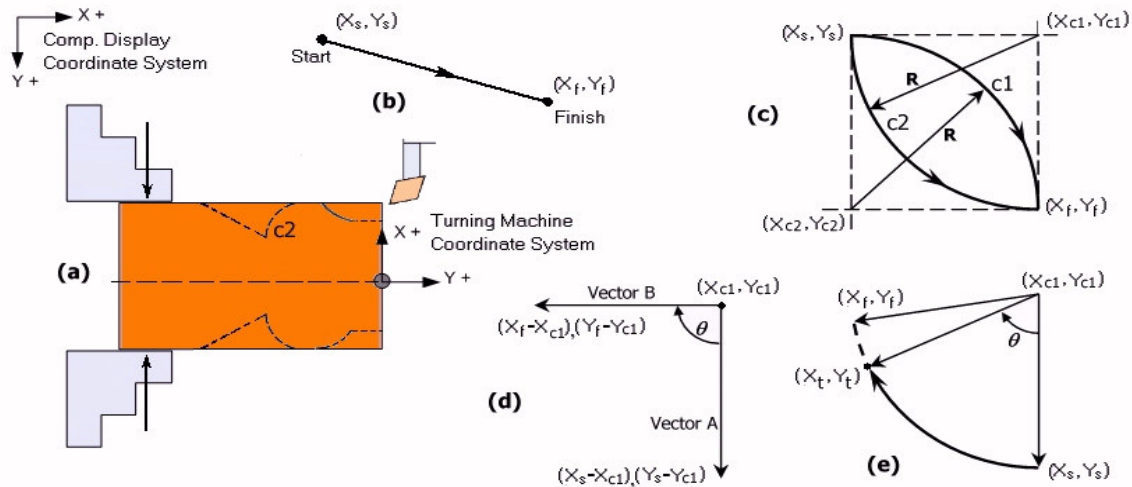


Figure 10. Three types of basic motion of cutting tool used in the simulation.

When the error free program is ready, the next step is simulating the codes to check whether tools movements are correctly defined and match to the requirement. In realizing the cutting tool motion, some general formulas have been developed based on G-code detected that represents certain movement. Linear and circular motions are treated in different ways. The cutting simulation algorithm consists of several parts, that is:

1. Main Simulation Algorithm, used to run particular command when some codes are recognized within a block such as: G00, G01, G02, G03, G04, G10, G20, G21, G28, G29, G32, G50, F, S, T, X, Z, U, and W.
2. Linear Motion Algorithm, used when related codes to linear motion are detected such as G00, G01, and so on. This function will automatically collect all necessary data to move the tool tip to the destination point along a linear line. The linear line is generated based on the idea as follows:

- a. The value related to destination point is extracted from the words: X/U 123.4 Z/W 678.99
- b. If G90 (absolute motion) is detected, then coordinate of destination point is defined by:

$$X_f = X_o + \text{value_of_word}(Z) \quad (3.3)$$

$$Y_f = Y_o - \text{value_of_word}(X) \quad (3.4)$$

- c. If G91 (incremental motion) is detected, then coordinate of destination point is defined by:

$$X_f = X_s + \text{value_of_word}(W) \quad (3.5)$$

$$Y_f = Y_s - \text{value_of_word}(U) \quad (3.6)$$

d. Example of this motion is depicted in Figure 10b. The minus signs Equations (3.4) and (3.6) are the consequence of different orientation of two coordinate systems, that is computer screen and machine coordinate systems. X_o and Y_o are coordinates of the absolute zero point which were assigned by G50 in the beginning of NC program.

3. Circular Motion Algorithm, used when G02 or G03 are detected and the simulator determines which arc should be taken and drawn on the screen. The circular line is generated based on the idea as follows:

a. The values related to destination point is extracted from the words, for instance: X20.33 Y40.55 R 12.34 or I 56.77 K 88.99.

b. If G02 (cw circular motion) or G03 (ccw circular motion) and R word are detected, then center of rotation should be first determined, where from the solution of the following equations, two possible answer are obtained:

$$(X_f - X_c)^2 + (Z_f - Z_c)^2 = R^2 \quad (3.7)$$

$$(X_s - X_c)^2 + (Z_s - Z_c)^2 = R^2 \quad (3.8)$$

c. The two possible answers for the center of rotation are (X_{c1}, Z_{c1}) and (X_{c2}, Z_{c2}) as depicted in Figure 10c.

d. To correctly choose the center point, one can just perform the cross product between two vectors generated from center point to start and finish points. When the cross product results in positive sign, it means that the center point is the correct center point for G02 or G03, otherwise, the other point is the correct center point for G02 or G03 as seen in Figure 10d.

e. When the circular motion is only a part of a circle, the words I and K are used in simulation.

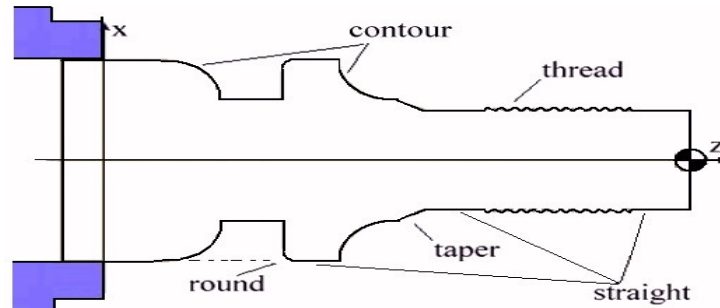


Figure 11. The cutting features contained in the simulated work piece.

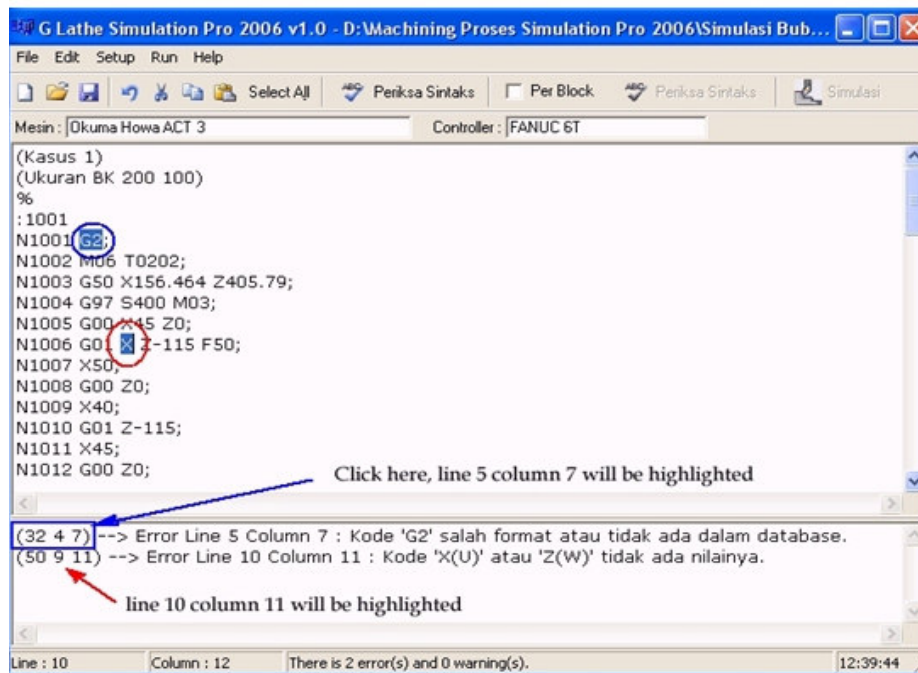


Figure 12. Error detection and possible solution for a certain NC program.

4. Thread Cutting Motion Algorithm, used when G32 (thread cutting) is detected. The cutter motion will be based on Equations 3.3 to 3.6 will be used, and the word F is also incorporated. Thread cutting simulation is carried out by performing small linear motions along the thread.

CASE STUDIES

To see the capability of this software tool, an interesting cylindrical work piece which contains some cutting/machining features is chosen as the research object. A blank as the original form of the work piece is a massive cylinder of diameter 200 × 100 mm length. This work piece needs three straight turnings and two contour turning cuts, tapering, round cutting, necking and also thread cutting as depicted in Figure 11. These cutting operations will provide a good insight how the machining are going to carry out and also how all of the developed functions work to perform cutting

motion on the screen display.

The blank to be machined should be securely mounted in the machine spindle and it is clamped by the chuck before machining is started. For this application, we assumed that about 50 mm length of the part is clamped. This amount of clamping will give quite space for the spindle chuck to grasp the work piece firmly. The blank normally has dimensions as close as possible to the final form, as to reduce the machining cycle time. Anyhow, when a deep cut should be done this concept of course would not work.

Syntax checking facility

After preparing the G-codes and making sure they are error free, the syntax checking facility plays its role in carefully checking the blocks line by line. If there is a mistake, the message box will highlight where the error occurs. It is done by clicking the left mouse button on the error message for instance: (32 4 7), then characters at line 5 and column 7 will be highlighted. This is shown in Figure 12 where two writing errors appear in the NC program.

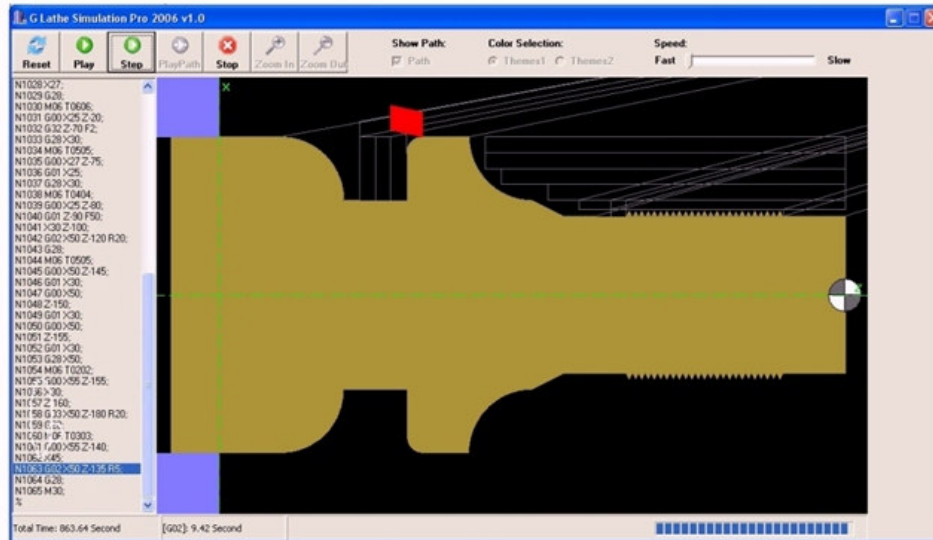


Figure 13. Progress of turning operation in the simulation window.

The first error was mistyping of G2 instead of G21 in line 5 column 7. This code (G2) is not recognized and is not contained in the database. The second error was mistyping X (without any numerical value) instead of X105 for instance. Code X should always be accompanied by a numerical value and they must not appear alone at all. In this case, X must have a real number or otherwise not appear at all. These blocks command the tool to perform straight cutting until positions $X = 105$ mm, $Z = -115$ mm are reached at a feed of 50 fpr.

Other things related to work piece also have to be prepared. Setup menu directs the operator to check the work piece dimension whether it is still within the machine range, to check type of machine to be used, and to choose the type of cutting tools that are available in the tool database. When this program is completely free from errors, "Simulasi" icon (right-up side) flips into red color.

Stage of simulation

Having clicked the simulation icon, one will arrive at the following window (Figure 13). The NC commands can be executed block by block, and the appearance of the tool path is optional. Simulating the cutting operation in this way gives a good and clear insight on how the operation will run in the reality. Unfortunately, this software still cannot visualize isometric view of the work piece being cut.

By sliding the speed bar, the speed of cutting simulation can be adjusted. In the top button, one can choose step by step NC block execution. This action shows the correspondence between the executed NC command and the cutting or motion performs by the tool. In other case, one can also display the tool path by selecting "PlayPath" button. Attention should be paid to the position of the work piece, as if they are not properly defined in the "Setup" stage, none won't be shown on the display screen.

This software tool will also automatically calculate the length of cut. This is done by executing a simple algorithm that calculates the total cutting time as defined in Equation 3.2. Cutting time is obtained by cumulatively adding all the time when the cutting tool is in contact with the work piece. The period during contacts with the work piece is assumed to be the stage when the cutting is performed to cut the material. This is closely related with one of the G01, G02 or G03 codes.

RESULTS AND DISCUSSIONS

Preparing a CNC program is an interesting activity but is not an easy task. It becomes easier because this programming media is equipped with checking and simulating facilities. A programmer is easily guided to work in this environment to make the program in a correct way. Right in the beginning, they are asked to write the blocks according to standard NC programming method and some special function related to the machine controller. In the simulation stage, the programmer is able to check the motion of the tool block by block, and supervising the path of cutting tool as well.

Providing an NC programming facility to small-medium industries is a great step to prepare them towards the new coming machining technology. People in industrial field who do not have enough production budgets can at least prepare their NC program of their works. And even further, they can also simulate the machining process. This software tool is very easy to be operated step by step as it is equipped with a help menu.

Beside for the industries, many higher educational institutions (vocational schools) in Indonesia still do not have any kind of NC machine and even the students cannot image how to program a CNC machine. This programming environment gives the basic idea and the guidance to them on how to program an NC machine. This software has not been completely ready yet, and therefore some volunteers may contribute in supporting the uncompleted functions.

Conclusions

From the text and discussion mentioned above, some conclusions may be drawn as the following:

1. Software tool developed in this work has been able to check the syntax of NC program based on Fanuc 6T Controller on Okuma Howa ACT-3 lathe.
2. The syntax checker points out the errors position and gives the solutions to the problem right away, such as mistyping dimension coordinate data or ambiguity between numbers and characters.
3. Simulation stage provides a clear insight on how the cutting process will be carried out and almost all aspects of the machining can be shown, such as position of tool related to command block, path of the cutting tool during the cutting operation, the changes of the workpiece while cutting is performed, and total cutting time and so on.
4. This programming media helps NC programmers to deal with the monotonous and time consuming programming tasks.
5. Software tool containing checking and simulating abilities for an NC program no doubly is useful for NC programmers, small-medium industries and vocational schools in Indonesia.

REFERENCES

- Arthaya B, Setiawan A, Setiawan A (2007). The Design and Development of G-Code Checker and Simulation of ISO G-Code for CNC Milling, Proc. of the 1st Asia-Pacific Conference on Manufacturing System.
- Chen SG, Ulsoy AG, Koren Y (1998). Machining Error Source Diagnostics Using A Turning Process Simulator, <http://cat.inist.fr/?aModele=afficheN&cpsidt=2294497>.
- CNC (2000). [2 Jan. 2010] <http://www.antechmicro.com/cnctrainsoft.htm>.
- De Garmo EP, Black JT, Kohser RA (1999). Materials and Processes in Manufacturing, John Wiley Sons, Inc.,.
- Falck D (2008). *RS274NGC G-Code Programming Basics*, <http://www.linuxcnc.org/handbook/gcode/g-code.html>.
- Fanuc (1983). Fanuc System 6T Operator's Manual, Fanuc Ltd.
- Metal Cutting Processes (2010). 2-Milling, [12 Jan.] <http://mmu.ic.polyu.edu.hk/handout/0103/0103.htm#1>.
- MTS (2010). CNC Simulator for Turning and Milling - System Description, [2 Jan.] <http://www.mts-cnc.com/espanol/sd5e.htm#1.1>.
- Turning Simulator (2010). <http://mobitechworld.com/search/turning-simulator.html>. 2 Jan.
- Rochim T (1990). Pemrograman NC (In Indonesian). Institut Teknologi Bandung, Bandung, Indonesia.
- Stenerson J, Curran K (1997). Computer Numerical Control. Prentice Hall Inc.
- Zukhri Z (2005). Visual Programming using C++ Builder, In Indonesian, Graha Ilmu, Yogyakarta, Indonesia.