Turkish J. Eng. Env. Sci.
29 (2005) , 75 – 88.

Time Minimization of CNC Part Programs in a Vertical Machining Center in Terms of Tool Path and Cutting Parameter Criteria

Ahmet Murat PINAR, Abdulkadir GÜLLÜ
Gazi University, Technical Education Faculty, Ankara-TURKEY
e-mail: ahmetp@gazi.edu.tr

Received 17.02.2004

Abstract

An algorithm was prepared for minimizing the machining time of CNC part programs used in a vertical machining center. The CNC program is transferred from the machine to the CNC code editor with a serial cable (RS-232). The time calculator examines all the codes in the code editor and calculates the time of all the moves in minutes. The other processor, which does the code formation, analyzes each individual movement of machining. As a result, it orders all the operations formed with the same cutting tool on the x and y axes. It also generates the G codes by minimizing the rapid movements of the tool in terms of the shortest tool path. With the cutting parameters module, machine table feed and spindle speed are modified using the machine power and tool life as limiting factors for each machining operation. In this way, the CNC program is modified. By reanalyzing the new CNC program with the time calculation processor, the time saved can be seen.

Key words: Cutting parameters, Tool path planning, Machining time, NC program.

Introduction

Machining centers are used widely in precise machining operations with high speeds (Yan *et al.*, 1999). In particular, a cutting tool's rapid movements can reach 5000-15,000 mm/min. The shared aim of tool manufacturers and user companies is to produce more. To achieve this, the role of the CNC programs used with the machine is very significant. In order to use these programs effectively, machining time should be minimized (Deshmukh and Wang, 1993). This process can be achieved through:

1. Minimization of the tool path,
2. Optimization of cutting parameters.

Studies about the optimization of tool path and cutting parameters include the operations of machining of $2^1/_2$D simple features and 3 and 5 axis sculptured features in die/mould and aerospace areas. The first studies in this field mainly focused on rapid movements (global motions) and machining movements (local motions) of $2^1/_2$D drilling, pocketing, and face milling operations.

Jung and Ahluwalia (1994) suggested an integer programming approach in order to minimize the tool travel distance by the selection of optimal sequences of the features to be machined. However, this study is only used for simple $2^1/_2$D features such as holes, rectangular pockets and slots. Khan *et al.* (1999) modeled global tool path planning as a traveling salesman problem and they solved the problem by simulated annealing techniques. Patil and Pande (2002) generated tool path and parametric NC code using the feature-based modeling technique. They minimized the global tool paths by sequencing the features. Wah *et al.* (2002) proposed a new strategy using the combination of the genetic algorithm-based asymmetric traveling salesman problem and integer programming in order to solve the problem of the minimization of tool rapid (global) motions between features. Jamil (1997) determined the sub-optimum tool path for both convex and concave pockets using zig-zag pocketing. Lakkaraju *et al.* (1992) investigated the effect of cutting direction angle on tool path length for face milling operations. Sun and Tsai (1994) improved the algorithm by adding an extra tool path to Lakkaraju's approach. In this way, tool paths were shortened by 17% . Kamarthi *et al.* (1997) calculated boundary points of each pass with polygon segments. Minimization of the tool paths is achieved by rotating those polygons from 0° to 360° in both directions. Vosniakos and Papapanagiotou (2000) developed an algorithm that enables us to machine convex pockets without islands with 3 different tools. The first 2 of these tools have the volume necessary for the third tool by offsetting the pocket boundaries. The last tool completes the pocket with a zig-zag operation. In this study, tool path minimization was realized by choosing the optimum combinations of tool diameters and by parametizing zig-zag machining. Kim *et al.* (2003) suggested a method to generate global and local tool paths from the feature data defined automatically from a part solid model for 2½D machining. The start and end points of features were examined to minimize each path segment in the global tool path. Pateloup *et al.* (2004) suggested modifying the corner radius in order to increase the feed rate of pocketing. With this algorithm, a shorter tool path and higher feed rate were obtained. In addition, in the tool path calculation, when compared with straight line and circle arcs, the use of a B spline provided a significant improvement in machining time reduction. Taiber (1996) developed software that helps the process planner in subjects such as the optimization of cutting parameters and the determination of tool paths and process sequences. The study achieved drilling and milling optimization in the process of planning in the light of data obtained from a feature-based CAD model. Tekiner and Güllü (2001) realized the optimization of cutting parameters based on maximum production and minimum cost criteria in single-pass milling operations. After the model of the workpiece to be processed was recognized by the feature recognition module, the tool was chosen depending on these factors. By means of these data, the optimization of cutting parameters was realized by crystal search, which is one of the most common optimization techniques. Tandon *et al.* (2002) used the new evolutionary calculation technique and particle swarm optimization (PSO) and achieved the precise and efficient optimization of multi-machining parameters in pocketing. The prediction of cutting forces used in the algorithm developed with PSO for critical machining parameters was achieved by an artificial neural networks prediction model. In the study, a reduction of 35% in the machining time was obtained.

Innovations in the control systems of machines and CAD/CAM softwares have contributed greatly to the manufacturing of die/mold and aerospace products. In the manufacturing process of these kinds of product, it takes a long time to rough 3 and 5 axis sculptured surfaces. Typical die/mold fabrication takes 1200 to 3800 h and from 55% to 65% of this time is spent roughing (Lee and Koc, 1998). A reduction in the roughing time will dramatically reduce the total manufacturing time, which is provided by efficient tool paths and cutting parameters.

Veeramani and Gau (1998) examined the tool path planning optimization problem for the machining of parts having multi-surface patches patch by patch for 3 different cases. For each case, the transformation of the tool path planning optimization problem into conventional optimization problems such as the shortest path or traveling salesman problem by graph-theoretic models was described. Reddy and Babu (1996) proposed a tool path planning procedure for the rough machining of free formed cavities modeled by the AutoCAD software. In this study, several cutting planes were used to generate a CNC tool path. In this way, a contour map approach was established that divides the feature into different 2D cutting regions. Lee and Koc (1996) developed a method for 5 axis rough and finish machining of only free formed ruled surface pockets. First, the core region of the pocket was machined by the developed inclined zig-zag method. Then the remaining part in the residual material region was semi-roughed by the new ellipse-offset method. Lastly, the operation was finalized by finishing the pocket surface using the tool traveling along the ruled surface. Park and Choi (2000) proposed a tool path planning algorithm for parallel direction surface machining. This algorithm was generated with 3 modules:

1. Finding the optimal inclination by reflecting the form of the machining region.
2. Calculation and storage of the tool path elements using plane sweeping paradigm and monotone chain concepts.
3. Linking the tool path. This problem is modeled as a tool path element net traversing problem.

In the study, one-way and zig-zag tool path linking algorithms were proposed for 2 direction parallel surface machining. Warkentin *et al.* (2003) compared multi-point machining (MPM) with the method of inclined tool and principal axis. In addition, 3 axis ball nose machining was compared with them. The results were shown with computer simulations and experimental tests. Consequently, it was seen that scallop heights obtained with MPM were shorter than the others. Feng and Li (2002) suggested an approach based on constant scallop height for the machining of sculptured surfaces in the 3 axis machining center with a ball end mill. In their study, the stabilization of scallop heights in a machined surface in order to minimize unnecessary tool paths was aimed. Scallop surface and tool center surface were defined from scallop height necessity and tool radius based on design surface, respectively. Masood *et al.* (2002) presented the minimum distance algorithm to machine the sculptured surfaces on CNC machines. The algorithm was developed in order to show the simulation of the tool path of specified surfaces and to produce NC programs. In addition, the study was prepared in accordance with reverse engineering applications. Lee and Yang (2002) proposed a direct calculation method using a new path interval algorithm based on real geometric features of the parametric surface without using surface curvatures. Chan *et al.* (2003) developed a CAM module for a 3 axis CNC machine in order to develop more effective CNC programs in the machining of molds/dies. The system called high efficiency roughing included a unidirectional straight line tool path, convex hull tool path and climb milling tool path.

As seen above, tool paths that consist of global and local tool movements have been examined in a wide range. The important common features of the studies are given in Table.

Table shows that tool path studies mainly focus on local tool movements called machining motions. Rapid tool movements (global motions) are examined less than the other and the optimization problem in this area is solved by the traveling salesman problem, integer programming approach, and sequencing the features and examining their start and end points. In the study proposed, rapid tool movements are shortened by sequencing the machining operations, taking the start and finish points into consideration. The required data for the study are obtained with a CNC part program different from traditional approaches.

Developed Algorithm

As shown in Figure 1, the algorithm consists of 5 modules. The CNC code module is where the CNC part program transferred from the machine is taken to be analyzed. In this module, the time of all the movements and total manufacturing time except for tool change operations are calculated by the machining time calculator. With the cutting geometry determination processor, cutting width and cutting depth values of each operation are determined. With the tool path minimization module, the new CNC codes where the rapid movements are minimized are generated and they are transferred to the CNC code module again. With the cutting parameters module, the new feed and spindle speeds are reorganized by getting maximum cutting values for each operation. Finally, turret and workpiece data modules are the sub-modules where the necessary data for calculations are taken.

Turret data module

In this module, the cutting data of the turret are determined. The information about the ordering code of cutting tools, the diameter, the tooth or insert number and the tool life are saved as a computer file with a txt extension.

Workpiece data module

In this module, the information about the material, length, width, height and reference points of the workpiece is given by the user and it is saved as a file with a txt extension to be called in the CNC code editor. In the first movements in this program and in the movements after the G28 command, the information about the geometric data and user reference points is taken from the user in order to calculate total movement distance.

CNC code module

This module includes a cutting geometry determination processor and machining time calculator. In order to achieve these processes, the part program is transferred to the code editor in the CNC code module. The CNC part program in the code editor is shown in Figure 2.

Table. The main common features of tool path studies.
● = Present ☐ = Not present ↑ = Unknown

Article Title	Tool Path Analysis		Machining Type			Feature Type	Data Required From
	Global	Local	$2^1\!/_2$ d	3 axis	5 axis axis		
Feature-Based Non-Cutting Tool-Path Selection (Jung and Ahluwalia, 1994)	●	☐	●	☐	☐	General	↑
Determination of Optimal Path under Approach and Exit Constraints (Khan *et al.*, 1999)	●	↑	●	☐	☐	↑	↑
An Intelligent Feature-Based Process Planning System for Prismatic Parts (Patil and Pande, 2002)	●	●	●	☐	☐	General	CAD model
Tool Path Optimization in Layered Manufacturing (Wah *et al.*, 2002)	●	☐	●	☐	☐	General	↑
A Computerized Algorithm for Milling Non-Convex Pockets with Numerically Controlled Machines (Jamil, 1997)	☐	●	●	☐	☐	Pocketing	↑
An Analytical Model for Optimization of NC Tool Cutting Path (Lakkaraju *et al.*, 1992)	☐	●	●	☐	☐	Face milling	↑
A Modified Analytical Model for Optimization of NC-Tool Cutting Path (Sun and Tsai, 1994)	☐	●	●	☐	☐	Face milling	↑
Foundations for Analytical Models of Staircase Traversal of Convex Polygonal Surfaces (Kamarthi *et al.*, 1997)	☐	●	●	☐	☐	Face milling	↑
Multiple Tool Path Planning for NC Machining of Convex Pockets without Islands (Vosniakos and Papapanagiotou, 2000)	☐	●	●	☐	☐	Pocketing	CAD model
Optimization of Process Sequences Considering Prismatic Workpieces (Taiber, 1996)	●	●	●	☐	☐	General	CAD model
Integrated Machining Tool Path Planning Using Feature Free Spaces (Kim *et al.*, 2003)	●	●	●	☐	☐	General	CAD model
Corner Optimization for Pocket Machining (Pateloup *et al.*, 2004)	☐	●	●	☐	☐	Pocketing	↑
Models for Tool Path Plan Optimization in Patch-by-Patch Machining (Veeramani and Gau, 1998)	●	☐	☐	●	☐	General	CAD model
Tool Path Planning Procedure for Rough Machining of Free Form Cavities (Reddy and Babu, 1996)	☐	●	☐	●	☐	Cavity machining	CAD model

Table. Contunied.
• = Present □ = Not present ↑ = Unknown

Article Title	Tool Path Analysis		Machining Type			Feature Type	Data Required From
	Global	Local	$2^1/_2$ d	3 axis	5 axis axis		
Ellipse-Offset Approach and Inclined Zig-Zag Method for Multi-Axis Roughing of Ruled Surface Pockets (Lee and Koc, 1998)	□	•	□	□	•	Ruled surface pocketing	CAD model
Tool Path Planning for Direction-Parallel Area Milling (Park and Choi, 1996)	□	•	□	•	□	Parallel surface machining	↑
Comparison between Multi-Point and Other 5-Axis Tool Positioning Strategies (Warkentin *et al.*, 2000)	□	•	□	•	•	General sculptured surface	↑
Constant Scallop Height Tool Path Generation for 3-Axis Sculptured Surface Machining (Feng and Li, 2002)	□	•	□	•	□	General sculptured surface	CAD model
A Computerized Minimum Distance Algorithm for Machining of Sculptured Surfaces (Masood *et al.*, 2002)	□	•	□	•	□	General sculptured surface	CAD model
Efficient Tool Path Planning for 5-Axis Surface Machining with a Drum-Taper Cutter (Cai *et al.*, 2003)	□	•	□	□	•	General sculptured surface	CAD model
Rough Pocketing of Multi-Sculptured Surface Cavities (Vafaeesefat and El-Maraghy, 2001)	□	•	□	•	□	General sculptured surface	CAD model, Reverse engineering
CNC Tool Path Planning for High-Speed High-Resolution Machining Using a New Tool-Path Calculation Algorithm (Lee and Yang, 2002)	□	•	□	•	•	General sculptured surface	CAD model
A High-Efficiency Rough Milling Strategy for Mold Core Machining (Chan *et al.*, 2003)	□	•	□	•	□	Core machining	CAD model
Optimization of Rotations of a 5-Axis Milling Machine near Stationary Points (Munlin *et al.*, 2004)	□	•	□	□	•	General sculptured surface	↑
Iso-Planar Piecewise Linear NC Tool Path Generation from Discrete Measured Data Points (Feng and Teng, 2005)	□	•	□	□	•	Ruled surface machining	Reverse engineering

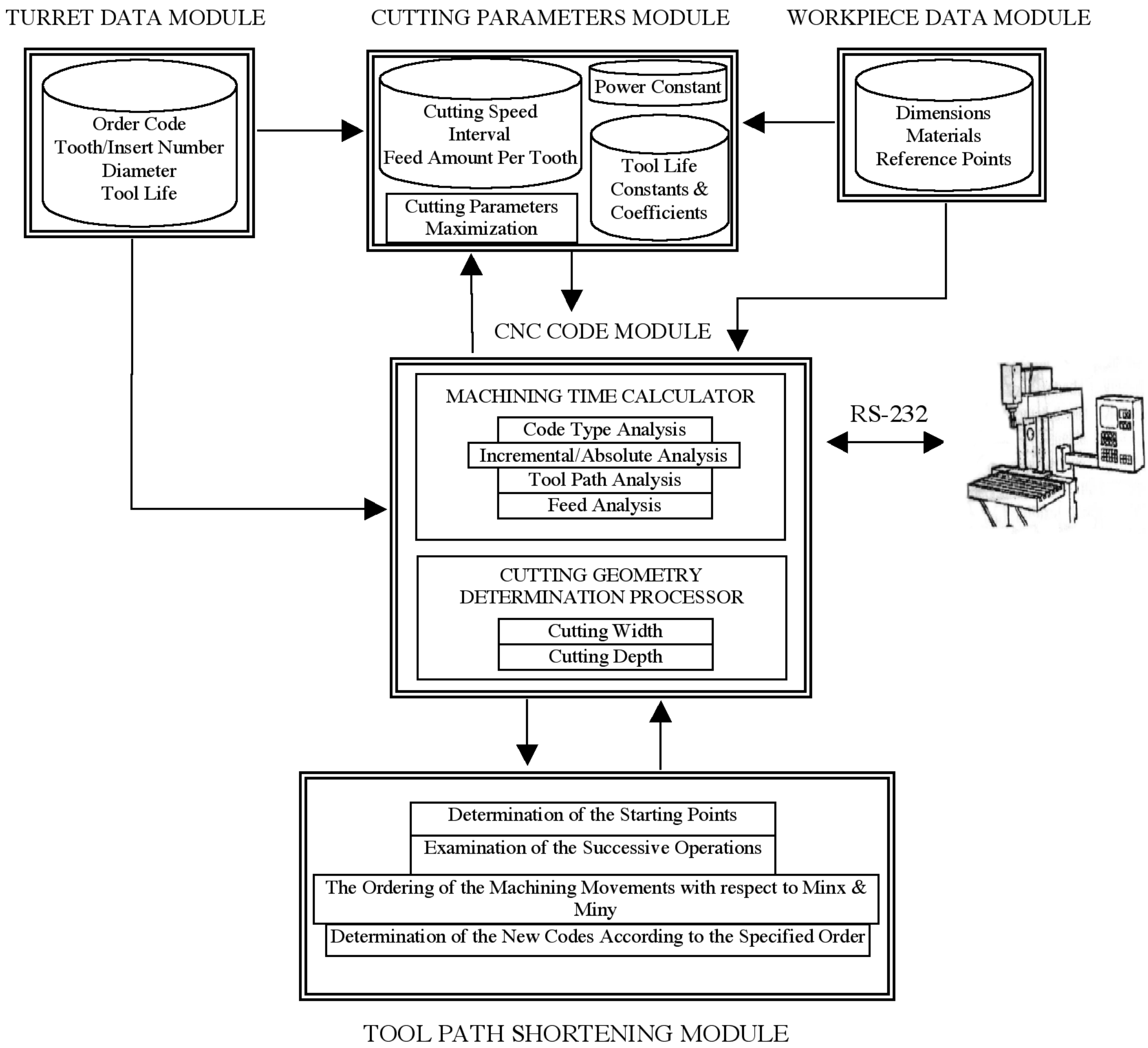
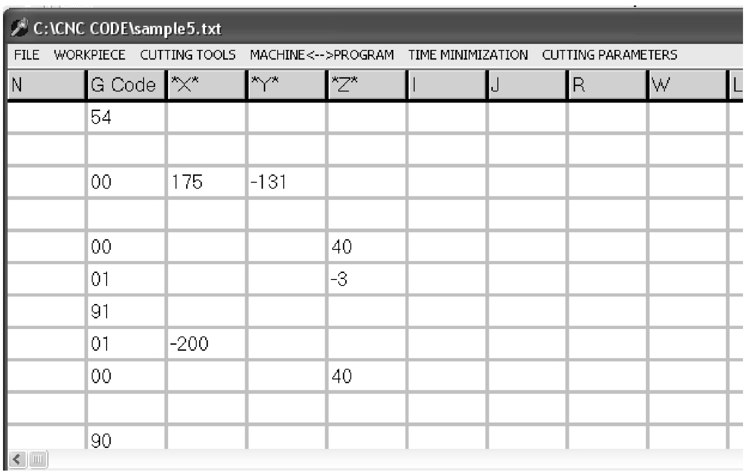


Figure 1. General structure of the generated program.



N	G Code	*X*	*Y*	*Z*	I	J	R	W	L
	54								
	00	175	-131						
	00			40					
	01			-3					
	91								
	01	-200							
	00			40					
	90								

Figure 2. CNC part program in the code editor.

Machining time calculator

After completing the data input into the code editor, the time calculation is performed by this processor. The steps are as follows:

1. The analysis of code type
2. The analysis in terms of the absolute and incremental coordinate system
3. The analysis of tool path for machining movements and the analysis of movement distance for rapid movements
4. The analysis of feed.

The analysis of code type

This module includes the determination of the code types of operations in the program. In addition to G01, G02 and G03 movements, various drill and pocket cycles can be determined. Below the steps of a G01 machining movement are shown:

For k:=0 to Gcodenumber do
Begin
If memo1.lines[k]='01' then
Begin
G01procedure;
End;

Absolute/incremental analysis

The algorithm gives each movement in the program a sequence number. By comparing these numbers with their situations in terms of G90 and G91, absolute/incremental code distinction of the movement is performed.

G54;
T01;
G90;
G00 X20 Y20 Z60;
G00 Z5;
G91;
G01 Z-12 F200;

If we examine G01 movement with the program lines above, the number for G01 movement is 7, for G90 is 3 and for G91 is 6. These 3 numbers are examined and the result is that this command is an incremental one. If the number is absolute for the variable, "true" is used. If the number is incremental, then "false" is used.

Choice[7]:=False;

Tool path analysis

In this processor, the distance for each movement of the tool is found. While finding these distances, first of all it is examined whether the tool is absolute or incremental. If it is incremental, in the CNC code editor, the X, Y and Z values in that sequence number give the movement distance directly. If the code is absolute, necessary calculations are performed by examining the prior movements. In the software, the distance of each axis is expressed with an array variable in the type of real number that shows the type of movement and the sequence number

G00distx[k] = x distance of the k. G00 movement
G01disty[k] = y distance of the k. G01 movement
G02distx[k] = x distance of the k. G02 movement

The displacement of the specified code is determined by taking the resultant of the distances in all axes.

G00distr[k]:=sqrt((sqr(G00distx[k])+
sqr(G00disty[k])+sqr(G00distz[k]))

If the movement is a circular interpolation, the movement distance is found by determining the length of the arc with the arc radius and given coordinate values.

Feed analysis

As is known, there are 2 types of movement in CNC machines: machining movements, which shape the workpiece (G01, G02, and all other machining cycles), and rapid movements, where the tool has no contact with the workpiece. In the software, in the G00 movements on the X, Y and Z axes, 9000, 6000 and 6000 mm/min feed values are used respectively. If rapid movement is generated in more than one axis simultaneously, the resultant of each feed value on each axis is taken. In machining movements, the feed value is determined by the F parameter in the line where the code is written. If the F parameter is not entered, the program uses the F parameter of the prior machining movements. The determined F value is appointed to the integer typed array variable expressed by Fvalue[k]. K is the sequence number of the related code.

With the 4 steps above, times for all movements in the program are determined. In the program line below, calculation of the G00 movement number 205 is shown.

Th[205]:=G00distr[205] / Fvalue[205]

The time for the movements is expressed by the array variable in the type of real number shown as th[k].

Cutting geometry determination processor

This is the processor where cutting width and depth are calculated for each operation to be used in force calculations of cutting parameters.

Tool path shortening module

In this module the CNC codes are generated by minimizing rapid movements. The main steps are as follows:

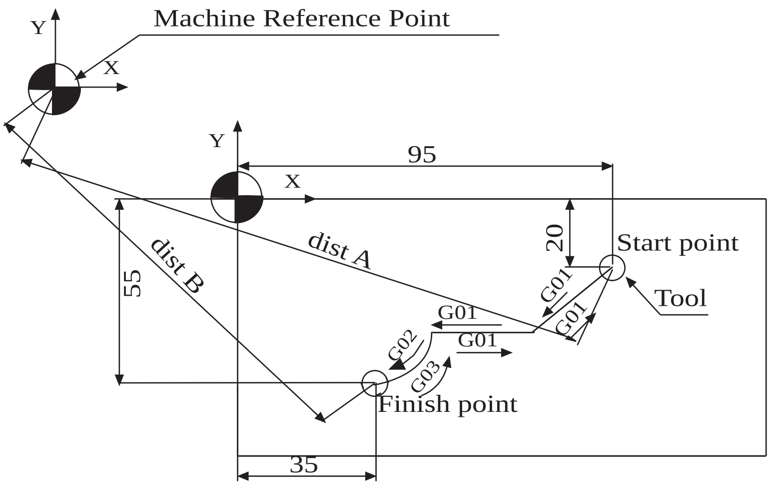
1. Determination of the start points.
2. Examinations of the successive operations.
3. Ordering of the machining movements respecting minimum X and Y.
4. Determination of the new codes according to the specified sequence.

Determination of the starting points

In this part of the program, absolute starting points according to the machine reference points are determined for each machining movement to be used in the ordering processes of the operations. The aim of this module is to guide the determination of the sequence of the machining operation.

Examination of the successive operations

In this step of the program, machining movements called successive operations, which do not have G00 rapid movement between each other, are examined. The coordinates where successive operations having many machining movements start and finish are compared considering the machine reference point. If the finish point is closer than the start point, the operation is started from the finish coordinates and CNC codes are modified. In Figure 3, a simple contour application is shown. In this figure, the start point of the operation is modified as the point where the arc finishes.




```
G54
T01
S1200 M03
G90
G00 X95 Y-20 Z10
G01 Z-1 F60
G91
G01 X-20 Y-20 F200
G01 X-25 F250
G02 X-15 Y-15 R15 F180
IF DIST B < DIST A THEN
G54
T01
S1200 M03
G90
G00 X35 Y-55 Z10
G01 Z-1 F60
G91
G03 X15 Y15 R15 F180
G01 X25 F250
G01 X20 Y20 F200
```

Figure 3. The examination of the successive operations [3].

The ordering of the machining movements with respect to minimum x and y

In this step, cutting operations generated by the same cutting tool are put in order for the x and y axes considering their distances to the machine reference point. Prior to this process, for each operation, a corresponding line number in the code editor is appointed integer typed array variables named Minx and Miny. However, in successive operations, these numbers are appointed only for the start movement. Following this process, the start values of each operation are compared depending on their distances to the machine reference point and they are put in order from smallest to biggest. Based on this sequence, new Minx and Miny values are appointed. If there are any operations with the same start point, the ordering is done considering the other axis. In Figure 4, ordering is done for a simple application that has 7 drilling operations.

In Figure 4, the values in parentheses are the x and y coordinates according to the workpiece reference point. The abbreviations like op1, op2, ...op7 are the sequence numbers of the operations. Below, the start coordinates in the x and y axes and operation sequence numbers are given.

Operation no.	X start	Y start
1	90	-81
2	15	-12
3	70	-16
4	34	-50
5	50	-16
6	87	-51
7	27	-30

The program determines the distance of each operation on the x and y axes in terms of the machine zero point for these specified starting points. These distances are explained by array variables in the real number type according to the sequence number of each operation.

Comparedistx[1]:=340;

In this part, "Comparedistx" is the name of the variable, "1" is the operation sequence number and "340" is the distance from the related operation to the machine zero point on the x axis. The distance on the y axis is also expressed with the "Comparedisty" variable. Below, the new Minx and Miny values found for the operations are given.

Operation no.	Min x sequence	Min y sequence
1	7	7
2	1	1
3	5	3
4	3	5
5	4	2
6	6	6
7	2	4

Determination of the new codes according to the specified sequence

In this module, the total movement amount of the new code sequences according to the Minx and Miny sequences is examined and new codes are created according to the sequence that is less in terms of distance

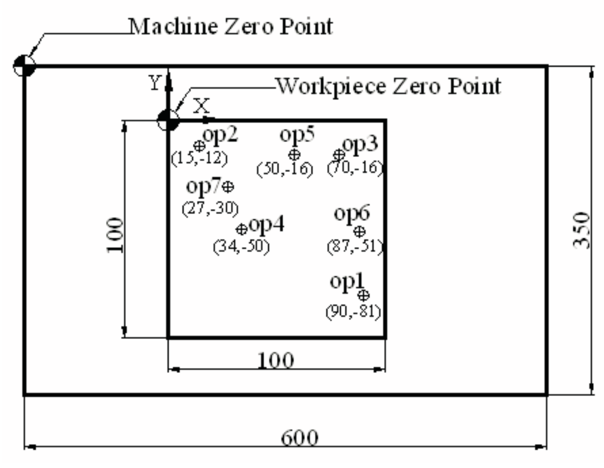


Figure 4. Sample application for the ordering process.

If (totaldistancex<totaldistancey) then
Begin
codecreatex;
End;
If (totaldistancey<totaldistancex) then
begin
codecreatey;
End;

In the program lines above "totaldistancex" and "totaldistancey" show the total cutting movement amount for the Minx sequence and the Miny sequence, respectively. The total movement amount for Minx and y of the sample in Figure 4 is shown in Figure 5.

Cutting parameters module

In this module the cutting speed is maximized considering machine power and tool life as restrictive factors. In this way, the spindle speed and feed rate of each operation in the NC program are reorganized based on the maximum cutting speed. This module consists of the steps below:

• A cutting speed interval and a feed per tooth for each operation are found from the database prepared by the Sandvik Coromat Catalogue in the light of the combination of the data of the cutting tool order code and the workpiece material taken from turret and work piece modules. Starting from the lower values of this interval, spindle speed, table feed and tool life values are found in Eqs. (1)-(3).

$$n = (1000 \times V)/(\pi \times D) \qquad (1)$$

where

n = Spindle speed (rpm).
V = Cutting speed (m/min).
D = Tool diameter (mm).

$$Vf = Zn \times n \times fz \qquad (2)$$

where

Vf = Table feed (mm/min).
Zn = Tooth number or insert number.
fz = Feed per tooth (mm).

$$T = \frac{((Cv^{\frac{1}{m}})\times(D^{\frac{q}{m}})\times(Ku^{\frac{1}{m}})\times(Kn^{\frac{1}{m}})\times(Km^{\frac{1}{m}})\times(K\phi^{\frac{1}{m}}))}{((V^{\frac{1}{m}})\times(ap^{\frac{x}{m}})\times(fz^{\frac{y}{m}})\times(Zn^{\frac{n}{m}})\times(ae^{\frac{\theta}{m}}))} \qquad (3)$$

where

T = Tool life (min).
Cv = Constant for a specific material.
Ku = A factor for tool material.
Kn = Skin factor.
Km = A factor for workpiece material.
$K\phi$ = A factor for corner angle.
q, m, n, x, y, θ = Exponents for a specific material.

The specified values are appointed to the integer typed array variables expressed as Feedrate[k] and spindlespeed[k] in the algorithm. K is the sequence number of the operation.

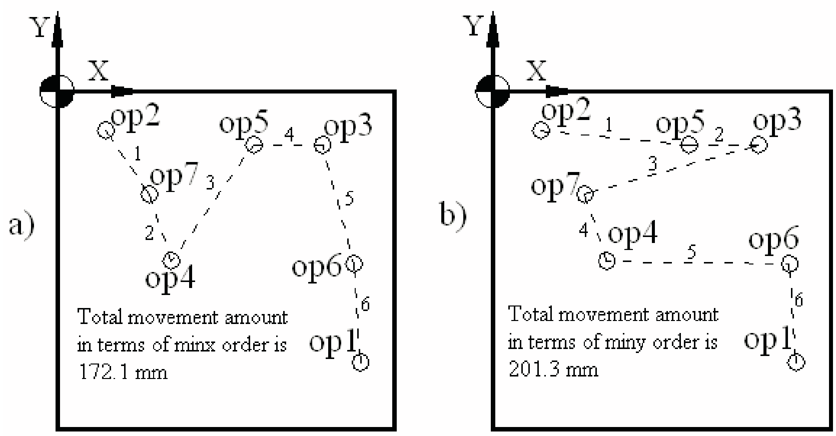


Figure 5. a) The calculation of the totaldistancex value schematically. b) The calculation of the totaldistancey value schematically.

• By means of the cutting geometry processor in the CNC code module, the data about ae and ap are found. By using Eq. (2) force formulation in the Sandvik Catalogue is applied in Eq. (4).

$$Pc = (ap \times ae \times Vf \times K)/100000 \tag{4}$$

where

Pc = Power consumption (kW).
ap = Cutting depth (mm).
ae = Cutting width (mm).
K = Experimental constant that can change according to feed per tooth, cutting width, and tool diameter.

• Tool life and power calculated starting from the first value of the cutting speed interval are compared with the tool life value entered for the corresponding tool in the turret data module and machine power in Eq. (5), respectively. As long as the difference between Pm and Pc is more than 0.01, the computation continues by increasing the cutting speed by 0.1 until the absolute value of difference between Tuser and T is less than 0.1. The NC program is rearranged according to the table feed and spindle speed calculated depending on the obtained maximum cutting speed.

$$Pm = P \times \eta \tag{5}$$

Pm = Power in the supposed efficiency.
P = Power value of the machine catalogue.
η = Supposed efficiency.

Below, the maximization of the cutting parameters of the face milling operation of the sample program is shown:

Data:

Cutting Tool: R245-050Q40-12M   Insert: R245-12T3M-PM GC4030

Workpiece Material: Stainless Steel 15.12 cm   ae = 40, ap = 2.45, Zn = 4, D = 50

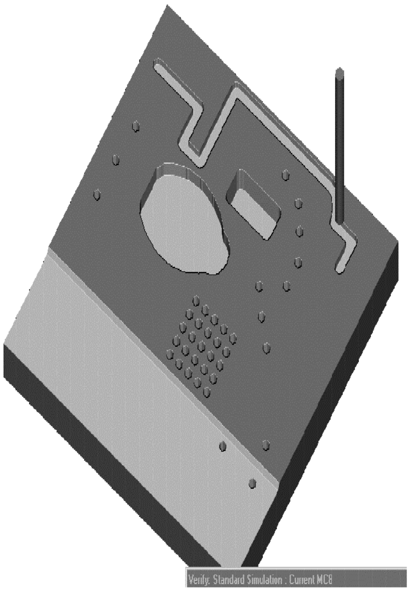
Pm = 5 x 0.8 = 4 kW.   Tuser = 28 min

V = 115-140 (m/min).

K = 5.4

Determination of the maximum values by the cutting parameters module

Cutting Speed:115	Tool Life: 29.020496870009	Spindle Speed:732.112738222719	Table Feed: 702.82822869381	Power:3.71936698624764
Cutting Speed:115.1	Tool Life: 28.9700921954217	Spindle Speed:732.749357995086	Table Feed: 703.439383675283	Power:3.7226012184096
Cutting Speed:115.2	Tool Life: 28.9198187260282	Spindle Speed:733.385977767454	Table Feed: 704.050538656755	Power:3.72583545057155
Cutting Speed:115.3	Tool Life: 28.8696760068474	Spindle Speed:734.022597539821	Table Feed: 704.661693638228	Power:3.7290696827335
Cutting Speed:115.4	Tool Life: 28.8196635848686	Spindle Speed:734.659217312189	Table Feed: 705.272848619701	Power:3.73230391489546
Cutting Speed:115.5	Tool Life: 28.7697810090418	Spindle Speed:735.295837084556	Table Feed: 705.884003601174	Power:3.73553814705741
Cutting Speed:115.6	Tool Life: 28.7200278302665	Spindle Speed:735.932456856924	Table Feed: 706.495158582647	Power:3.73877237921937
Cutting Speed:115.7	Tool Life: 28.6704036013824	Spindle Speed:736.569076629291	Table Feed: 707.10631356412	Power:3.74200661138132
Cutting Speed:115.8	Tool Life: 28.6209078771592	Spindle Speed:737.205696401659	Table Feed: 707.717468545593	Power:3.74524084354328
Cutting Speed:115.9	Tool Life: 28.5715402142865	Spindle Speed:737.842316174026	Table Feed: 708.328623527065	Power:3.74847507570523
Cutting Speed:116	Tool Life: 28.5223001713637	Spindle Speed:738.478935946394	Table Feed: 708.939778508538	Power:3.75170930786718
Cutting Speed:116.1	Tool Life: 28.4731873088905	Spindle Speed:739.115555718762	Table Feed: 709.550933490011	Power:3.75494354002914
Cutting Speed:116.2	Tool Life: 28.4242011892569	Spindle Speed:739.752175491129	Table Feed: 710.162088471484	Power:3.75817777219109
Cutting Speed:116.3	Tool Life: 28.3753413767334	Spindle Speed:740.388795263497	Table Feed: 710.773243452957	Power:3.76141200435305
Cutting Speed:116.4	Tool Life: 28.3266074374616	Spindle Speed:741.025415035864	Table Feed: 711.38439843443	Power:3.764646236515
Cutting Speed:116.5	Tool Life: 28.2779989394441	Spindle Speed:741.662034808232	Table Feed: 711.995553415902	Power:3.76788046867696
Cutting Speed:116.6	Tool Life: 28.2295154525352	Spindle Speed:742.298654580599	Table Feed: 712.606708397375	Power:3.77111470083891
Cutting Speed:116.7	Tool Life: 28.1811565484316	Spindle Speed:742.935274352967	Table Feed: 713.217863378848	Power:3.77434893300086
Cutting Speed:116.8	Tool Life: 28.1329218006625	Spindle Speed:743.571894125334	Table Feed: 713.829018360321	Power:3.77758316516282
Cutting Speed:116.9	Tool Life: 28.0848107845804	Spindle Speed:744.208513897702	Table Feed: 714.440173341794	Power:3.78081739732477



```
%
O0000
N100G21
N102G0G17G40G49G80G90
(UNDEFINED TOOL - 1 DIA. OFF. - 1 LEN. - 1 DIA. - 50.)
N104T1M6
N106G0G90G54X0.Y15.A0.S500M3
N108G43H1Z25.
N110Z10.
N112G1Z-2.45F200.
N114X150.F700.
N116G0Z25.
N118M5
N120G91G28Z0.
N122G28X0.Y0.A0.
N124M01
(4. DRILL TOOL - 2 DIA. OFF. - 2 LEN. - 2 DIA. - 4.)
N126T2M6
N128G0G90G54X87.348Y135.228A0.S50M5
N130G43H2Z25.
N132G98G81Z-14.R10.F300.
N134X116.58Y36.444
N136X9.984Y71.724
N138X10.74Y87.852
N140X115.824Y77.772
N142X107.004Y85.584
N144X99.444Y93.9
```

Figure 6. Sample application (Mastercam version 8, CNC Software).

```
1102 G00 CODE:0.0118585412256351
1103 G01 CODE:0.100000000000023
1104 G00 CODE:0.00208333333333499
1105 G00 CODE:0.00989949493660447
1106 G01 CODE:0.100000000000023
1107 G00 CODE:0.00208333333333499
1108 G00 CODE:0.00636832439150936
1109 G01 CODE:0.100000000000023
1110 G00 CODE:0.00208333333333499
<<TOTAL TIME >>(SAMPLE1.NC)
24.14 MIN
```

```
1208 G00 CODE:0.00276134025429542
1209 G00 CODE:0.0010416666666675
1211 G01 CODE:0.1599899447793661167
1212 G00 CODE:0.0312500000000071
1214 G00 CODE:0.00219215773965686
1215 G00 CODE:0.0010416666666675
1217 G01 CODE:0.102312559
1218 G00 CODE:0.00312500000000071
<<TOTAL TIME>>(MODIFIED SAMPLE1.NC)
22.32 MIN
```

Figure 7. The analysis results of a sample program.

The change made to the NC program

M03 S750 → *(M03 S744)*

G00 Z10

G01 Z-3 F400

*G*91

G01 X-200 F500 → *(F714)*

Sample Application

In Figure 6, the machined shape of a workpiece made with 4 tools is shown. With the results of the analysis done with the algorithm, a reduction of 1 min and 42 s is achieved by shortening the tool's rapid movements and maximization of the cutting parameters at the time of production (Figure 7).

Discussion and Conclusion

The CNC part program transferred from the machine by means of the algorithm generated using Delphi 3.0 programming language is examined according to the tool path and cutting parameter criteria. With the tool path shortening module, rapid movements are minimized. With the cutting parameters module, machining movements are modified based on maximum cutting speed considering machine power and tool life as restrictive factors. In the sample application, about 7% time shortening is obtained, which is a very significant advantage for mass production. The simplicity of the program and being able to obtain the time for all operations in detail increase its validity. In addition, with the machining time calculator, different machining types used for the same features can be examined in terms of cutting time. In this algorithm, the rapid movements of the operations generated by the same tool are minimized. Therefore, since the tool change number cannot change, the time of this process is not taken into account.

Below, the factors limiting the usage of the algorithm are given:

- The software can only be used for $2^1/_2$D cutting operations including pocket, contour, drill and face milling processes. It is not appropriate for examining 3D sculptured geometry where different cutting planes are used.
- In the maximization of the cutting parameters, a surface roughness criterion is not taken into account.

References

Cai, Y., Xi, G. and Wang, S., "Efficient Tool Path Planning for Five-Axis Surface Machining with a Drum-Taper Cutter", Int. J. Prod. Res., 41, 3631–3644, 2003.

Chan, K.W., Chiu, W.K., Tan, S.T. and Wong, T.N., "A High-Efficiency Rough Milling Strategy for Mould Core Machining", Proc. Instn. Mech. Engrs., 217, Part B: J. Engineering Manufacture, 335-348, 2003.

Deshmukh, A. and Wang, H.P., "Tool Path Planning for NC Milling of Convex Polygonal Faces: Minimization of Non-Cutting Area", The International Journal of Advanced Manufacturing Technology, 8, 17-24, 1993.

Feng, H.Y. and Teng, Z., "Iso-Planar Piecewise Linear Nc Tool Path Generation from Discrete Measured Data Points", Computer-Aided Design, 37, 55–64, 2005.

Feng, H.Y. and Li, H., "Constant Scallop Height Tool Path Generation for 3 Axis Surface Machining", Computer Aided Design, 34, 674-654, 2002.

Jamil, A.T.M., "A Computerized Algorithm for Milling Non-Convex Pockets with Numerically Controlled Machines", Int. J. Prod. Res., 35, 1843-1855, 1997.

Jung, J.Y. and Ahluwalia, R.S., "Feature-Based Noncutting Tool-Path Selection", Journal of Manufacturing Systems, 13, 165-176, 1994.

Kamarthi, S.V., Pittner, S. and Bukkaratnam, S.T.S., "Foundations for Analytical Models of Staircase Traversal of Convex Polygonal Surfaces", Int. J. Prod. Res., 35, 2009-2024, 1997.

Khan, W.A., Hayhurst, D.R. and Cannings, C., "Determination of Optimal Path under Approach and Exit Constraints", European Journal of Operational Research, 117, 310–325, 1999.

Kim, Y.S., Wang, E., Hwang, I.K. and Rho. H. M., "Integrated Machining Tool Path Planning Using Feature Free Spaces", Int. J. Prod. Res., 41, 3237–3255, 2003.

Lakkaraju, R.K., Raman, S. and Irani, S.A., "An Analytical Model for Optimization of NC Tool Cutting Path", Int. J. Prod. Res., 30, 109-127, 1992.

Lee, S.G. and Yang, S.H., "CNC Tool Path Planning for High-Speed High-Resolution Machining Using a New Tool-Path Calculation Algorithm", Int. J. Adv. Manuf. Technol., 20, 326–333, 2002.

Lee, Y.S. and Koc, B., "Ellipse-Offset Approach and Inclined Zig-Zag Method For Multi-Axis Roughing of Ruled Surface Pockets", Computer-Aided Design, 30, 957–971, 1998.

Masood, S.H., Bagam, V.K. and Chantanabubpha P., "A Computerized Minimum Distance Algorithm for Machining of Sculptured Surfaces", Computers and Industrial Engineering, 42, 291-297, 2002.

Munlin, M., Makhanova, S.S. and Bohez, E.L.J., "Optimization of Rotations of a Five-Axis Milling Machine near Stationary Points", Computer-Aided Design, 36, 1117–1128, 2004.

Park, S.C. and Choi, B.K., "Tool-Path Planning for Direction-Parallel Area Milling", Proceedings of the Japan/USA Symposium on Flexible Automation, 1, 715-722, 1996.

Pateloup, V., Duc, E. and Ray, P., "Corner Optimization for Pocket Machining", International Journal of Machine Tools & Manufacture, 44, 1343–1353, 2004.

Patil, L. and Pande, S.S., "An Intelligent Feature Based Process Planning System for Prismatic Parts", International Journal of Production Research, 40, 4431–4447, 2002.

Pinar, A.M., "Time Analysis Based on Alternative Starting Points and Operation Phases in CNC Milling", MSc Thesis, Gazi University Institute of Science and Technology, 2000.

Reddy, V.S. and Babu, N.R., "Tool Path Planning Procedure for Rough Machining of Free Form Cavities", Proceedings of the Japan/USA Symposium on Flexible Automation, 1, 715-722, 1996.

Sandvik Coromat Rotating Tools Catalogue, Sweden, 2001.

Sun, R.H. and Tsai, Y.C., "A Modified Analytical Model for Optimization of NC-Tool Cutting Path", Int. J. Prod. Res., 32, 2335-2344, 1994.

Taiber, J.G., "Optimization of Process Sequences Considering Prismatic Workpieces", Advances in Engineering Software, 25, 41-50, 1996.

Tandon, V., El-Mounayri, H. and Kishawy, H., "NC End Milling Optimization Using Evolutionary Computation", International Journal of Machine Tools & Manufacture, 42, 595–605, 2002.

Tekiner, Z. and Güllü, A., "Optimization of the Cutting Parameters in Milling Operations", Machine Design and Manufacture, 4, 3-10, 2001

Vafaeesefat, A. and ElMaraghy, H. A., "Rough Pocketing of Multi-Sculptured Surface Cavities", Proc. Instn. Mech. Engrs., 215, Part B, 745-753, 2001.

Veeramani, D. and Gau, Y.S., "Models for Tool Path Plan Optimization in Patch by Patch Machining", Int. J. Prod. Res., 36, 1633-1651, 1998.

Vosniakos, G. and Papapanagiotou, O., "Multiple Tool Path Planning for NC Machining of Convex Pockets without Islands", Robotics and Computer Integrated Manufacture", 16, 425-435, 2000.

Wah, P.K., Murty, K.G., Joneja, A. and Chiu, L.C., "Tool Path Optimization in Layered Manufacturing", IIE Transactions, 34, 335-347, 2002.

Warkentin, A., Ismail, F. and Bedi, S., "Comparison between Multi-Point and Other 5-Axis Tool Positioning Strategies", International Journal of Machine Tools & Manufacture, 40, 185–208, 2000.

Yan, X., Shirase, K., Hirao, M. and Yasui, T., "NC Program Evaluator for Higher Machining Productivity", International Journal of Machine Tools & Manufacture, 39, 1563-1573, 1999.