Automatic Generation of CNC Part Program for Cylindrical Components from DXF File

Jatin Dave and V.R.Iyer

***Abstract*—his paper describes the methodology for the automatic extraction of the co ordinates from dxf file and generation of CNC part program for the cylindrical components. The paper uses the method discussed in the survey for the extraction of the co ordinate from dxf file and recognition of some basic feature like horizontal line, vertical line and inclined line. After collecting the co ordinates the algorithm is developed to arrange these coordinate to write a CNC part program. While arranging the co ordinate, sequences of the features are also considered so that it will be useful in creating tool motion. For writing a CNC part program, the practical aspect like depth of cut is also considered and sequence of features are also taken care so that proper tool path is generated. his paper describes the methodology for the automatic extraction of the co ordinates from dxf file and generation of CNC part program for the cylindrical components. The paper uses the method discussed in the survey for the extraction of the co ordinate from dxf file and recognition of some basic feature like horizontal line, vertical line and inclined line. After collecting the co ordinates the algorithm is developed to arrange these coordinate to write a CNC part program. While arranging the co ordinate, sequences of the features are also considered so that it will be useful in creating tool motion. For writing a CNC part program, the practical aspect like depth of cut is also considered and sequence of features are also taken care so that proper tool path is generated. T**

***Index Terms*—CNC Program, Tool motion,feature extraction, recognition**

I. Introduction

FEATURING plays an important role in the decision-making of design and manufacturing related activities. The rationale of feature based design is to provide explicitly design/manufacturing features at the modeling level and to relieve the designer's load of manipulating low-level graphic primitives to convey the design intent. The importance of the features lies in the fact that features are very promising in establishing a good link between the CAD and CAM, which is essential for improving product quality and reducing time to market. Feature technology allows use of export system to automatically generate manufacturing instructions from design input. Feature based design is a process in which parts are specified in terms of their constituent parameterized form features. Feature design has allowed users essentially to capture design intention. Functional relationships among part features can be defined. Manufacturing information can be captured and associated with a feature. CAD/CAM system uses mainly design by feature or feature recognition technique to capture information of co ordinates values and feature. The recognition process may be automatic by rule based system or semi automatic [1],[2].

Jatin Dave and V.R. Iyer are with Department of Mechanical Engineering, Institute of Technology, Nirma University, SG Highway, Ahmedabad 382481

The automatic system involves algorithms that are used to read information of the object from its geometric file, extract values of various features and recognize various features. Semi automatic system involves user intervention for input of various parameters like manufacturing parameters of a particular process. Researchers for manufacturing processes, to store information for design features and for generation of process plan, develop many CAD/CAM and CAPP systems. Also there are systems available that mainly emphasizes generation of NC part programs. Continuous efforts are going on to develop algorithms and software to directly generate NC part program just by having a solid model or drawing as an input without user involvement. This work is concentrated on developing a computer program that uses a data exchange file of a drawing of a model to generate NC part program. This program runs in three stages. The first stage consists of retrieving co - ordinates values from drawing file, second stage arranges the co - ordinate and features as per the sequence of machining operation and third stage generates the NC part program

II. CAD/CAM Data Exchange

Product design and manufacturing procedure uses variety of software to prepare solid models, assembly, analysis files and manufacturing models. It becomes increasingly important to find effective procedures for exchanging these databases. Fundamental incompatibilities among entity representations greatly complicate exchanging modeling data among CAD/CAM systems. Even simple geometric entities such as circular arcs are represented by incompatible forms in many systems.

The database exchange problem is complicated further by the complexity of CAD/CAM systems, the varying requirements of organizations using them, the restrictions on access to proprietary database information, and the rapid pace of technological change Transferring data between dissimilar CAD/CAM systems must embrace the complete product description stored in its database. Four types of modeling data make up this description.

1) Shape data. This data consists of both geometrical and topological information as well as part or form features.
2) Non shape data. It includes graphics data such as shaded images, and global data as measuring units of the database and the resolution of storing the database numerical values.

3) This data has to do with the information that designers generate from geometric models for analysis purposes.
4) Manufacturing data. It consists of information as tooling, NC tool paths,tolerancing, process planning, tool design, and bill of material.

There are many neutral file formats that can be used to transfer data and fulfill above mentioned requirements. Some of them are Initial Graphics Exchange specification (IGES), Drawing Interchange format(DXF), standard for The Exchange of Product model Data(STEP), Product Design Exchange Specification(PDES) etc. IGES and DXF file formats are used to transfer geometric data and STEP and PDES formats are used for transferring geometric as well as manufacturing data. This paper uses DXF file format as geometric data i.e. co ordinates values are required to prepare NC part program.

III. DXF File Format

Drawing Interchange Format was developed to transfer AutoCAD Drawing into other file format so that it can be used by any CAD/CAM/CAE system.
A DXF File is an ASCII Text File that consists of five sections:
Header - describes AutoCAD environment that exist when file is created
Table - contains information about line types, layers, text style, views etc.
Block - contains list of graphic entities that are defined as a group.
Entity - immediately follows the block section, and serves as a main part of the DXF File, with all entities described in it.
Terminate - Describe end of the file The following is a sample DXF file for a line: 0
SECTION
2 ENTITIES
0
LINE
8
0
10
50
20
50
30
0
11
100
21
50
31
0
0
ENDSEC
0
EOF
The command for a particular entity is described after "ENTITIES". In this sample DXF File, "LINE" is a command used to create a line between (x1, y1) and (x2, y2) co ordinates. The value of x1, y1, z1 and x2, y2, z2 is given after 10, 20, 30 and 11, 21, 31 i.e. 50, 50, 0 and 100, 50, 0. Similarly information of other entities can be obtained.

IV. Data Extraction Algorithm

To capture the value of co ordinates, it is required to compare 10, 20, 30 and 11, 21, 31. when match is found the respective co ordinate can be stored in an array of x, y and z co ordinate. The flow chart for this procedure is shown in figure 1.

V. Arranging Data as per Actual Situation of Machine

The drawing may be drawn using actual co ordinates, but to consider program zero, absolute dimensioning system on CNC Turning center is required to set co ordinates from location of program zero on work piece
The location of program zero also depends on position of chuck i.e. left or right. If we assume chuck position left side that program zero would be on the right side.
It is also very important to set value of co ordinates as per motion of various axes. The turning center consists of X - axis and Z - axis. TheAutoCAD drawing is prepared in XY - plane. Hence our x co ordinate value would become Z value in the part program and y co ordinate value would become X value in part program. The motion along Z axis toward chuck side will be negative and along X axis towards saddle side will be negative.
Let's consider chuck position on the left side and hence program zero will be on the right side of geometry. To arrange co ordinates as per axis of Turning center, maximum x co ordinate is subtracted from all x co ordinate value and mean value of y co ordinate is subtracted from all y co ordinate value. The flow chart for this procedure is shown in figure 2.

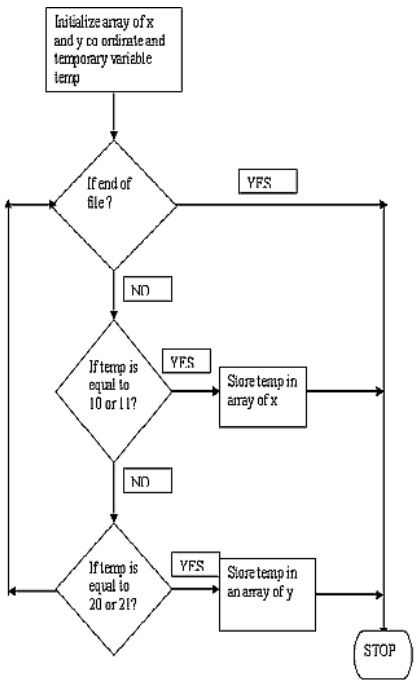

Fig. 1. Flow chart for data extraction

VI. Feature Recognition

This paper describes the procedure for recognizing feature for straight turning, step turning, taper turning and facing operation. It is necessary that all condition to recognize a line should be implemented in the program. This program gives output as per absolute

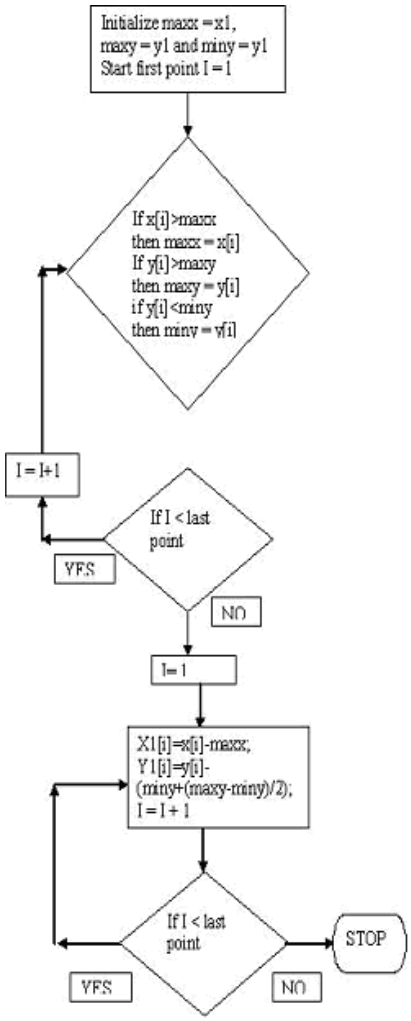

Fig. 2. Arranging data as per chuck position

dimensioning system; hence y co ordinate value is multiplied by 2. The following rules are used to determine various types of line and further type of operation required.

IF((X1[i-1]!=X1[i])&(Y1[i-1]==Y1[i]))THEN line is vertical and step turning Z[n1-1]=X1[i] AND X[n1-1]=2*Y1[i];

IF ((X1[i-1]==X1[i])&(Y1[i-1]!=Y1[i])) THEN line is Horizontal and straight turning Z[n1-1]=X1[i] AND X[n1-1]=2*Y1[i];

IF((X1[i-1]!=X1[i])&(Y1[i-1]!=Y1[i])) THEN line is Inclined and taper turning Z[n1-1]=X1[i] AND X[n1-1]=2*Y1[i];

IF(X1[i-1]==0&Y1[i-1]==0) THEN end of file and reading of file is stopped

VII. Sample run of the Program

The program is written in JAVA Language. A user interface is developed to enter a particular DXf file to be read. The output contains generation of drawing, display of co ordinate value and co ordinates after arranging as per the actual arrangement of machine. Finally an out put file "job.nc" is generated which contains NC Part program. This part program is now ready to transfer on CNC machine and can be used for further manufacturing of a component

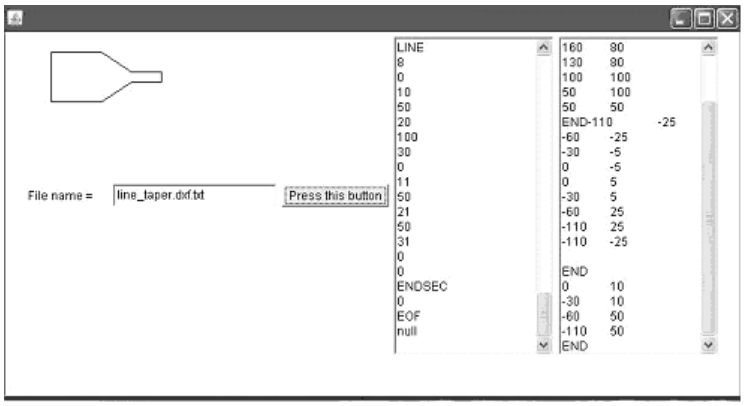

Fig. 3. User Interface of the program

VIII. The NC Part Program

N005 G71 G90 G54 G95 lf
N010 G00 X0 Z5 lf
N015 G01 X0 Z0 F0.5 lf
N020 G01 X52.0 lf
N025 G01 Z-110F0.5 lf
N030 G01 X54.0F0.5 lf
N035 G01 Z0 F0.5 lf
N040 G01 X52.0 F0.5 lf N045 G01 X50.0 lf N050 G01 Z-30F0.5 lf
N055 G01 X50 Z-60 F0.5 lf
N060 G01 X52 F0.5 lf
N065 G01 Z0 F0.5 lf
N070 G01 X50.0 F0.5 lf
N075 G01 X48.0 lf
N080 G01 Z-30F0.5 lf
N085 G01 X50 Z-60 F0.5 lf
N090 G01 X52 F0.5 lf
N095 G01 Z0 F0.5 lf
N0100 G01 X48.0 F0.5 lf
N0105 G01 X46.0 lf
N0110 G01 Z-30F0.5 lf
N0115 G01 X50 Z-60 F0.5 lf
N0245 G01 Z0 F0.5 lf
N0250 G01 X38.0 F0.5 lf
N0255 G01 X36.0 lf
N0260 G01 Z-30F0.5 lf
N0265 G01 X50 Z-60 F0.5 lf
N0270 G01 X52 F0.5 lf
N0275 G01 Z0 F0.5 lf
N0280 G01 X36.0 F0.5 lf
N0285 G01 X34.0 lf
N0290 G01 Z-30F0.5 lf
N0295 G01 X50 Z-60 F0.5 lf

N0300 G01 X52 F0.5 lf
N0305 G01 Z0 F0.5 lf
N0310 G01 X34.0 F0.5 lf
N0315 G01 X32.0 lf
N0320 G01 Z-30F0.5 lf N0325 G01 X50 Z-60 F0.5 lf
N0330 G01 X52 F0.5 lf
N0335 G01 Z0 F0.5 lf
N0340 G01 X32.0 F0.5 lf
N0345 G01 X30.0 lf
N0350 G01 Z-30F0.5 lf
N0355 G01 X50 Z-60 F0.5 lf
N0360 G01 X52 F0.5 lf
N0365 G01 Z0 F0.5 lf
N0370 G01 X30.0 F0.5 lf
N0375 G01 X28.0 lf
N0380 G01 Z-30F0.5 lf
N0385 G01 X50 Z-60 F0.5 lf
N0390 G01 X52 F0.5 lf
N0395 G01 Z0 F0.5 lf
N0400 G01 X28.0 F0.5 lf
N0405 G01 X26.0 lf
N0410 G01 Z-30F0.5 lf
N0415 G01 X50 Z-60 F0.5 lf
N0420 G01 X52 F0.5 lf
N0425 G01 Z0 F0.5 lf
N0430 G01 X26.0 F0.5 lf
N0435 G01 X24.0 lf
N0440 G01 Z-30F0.5 lf
N0445 G01 X50 Z-60 F0.5 lf
N0450 G01 X52 F0.5 lf
N0455 G01 Z0 F0.5 lf
N0460 G01 X24.0 F0.5 lf
N0465 G01 X22.0 lf
N0470G01 Z-30F0.5 lf
N0475 G01 X50 Z-60 F0.5 lf
N0480 G01 X52 F0.5 lf
N0485 G01 Z0 F0.5 lf
N049 0 G01 X22.0 F0.5 lf
N0495 G01 X20.0 lf
N0500 G01 Z-30F0.5 lf
N0505 G01 X50 Z-60 F0.5 lf
N0510 G01 X52 F0.5 lf
N0515 G01 Z0 F0.5 lf
N0520 G01 X20.0 F0.5 lf
N0525 G01 X18.0 lf
N0530 G01 Z-30F0.5 lf
N0535 G01 X50 Z-60 F0.5 lf
N0540 G01 X52 F0.5 lf
N0120 G01 X52 F0.5 lf
N0125 G01 Z0 F0.5 lf
N0130 G01 X46.0 F0.5 lf
N0135 F0.5 lf
N0150 G01 X52 F0.5 lf
N0155 G01 Z0 F0.5 lf
N0160 G01 X44.0 F0.5 lf
N0165 G01 X42.0 lf
N0170 G01 Z-30F0.5 lf
N0175 G01 X50 Z-60 F0.5 lf
N0180 G0 X52 F0.5 lf
N0185 G01 Z0 F0.5 lf
N0190 G01 X42.0 F0.5 lf
N0195 G01 X40.0 lf
N0200 G01 Z-30F0.5 lf
N0205 G01 X50 Z-60 F0.5 lf
N0210 G01 X52 F0.5 lf
N0215 G01 Z0 F0.5 lf
N022 G01 X40.0 F0.5 lf
N0225 G01 X38.0 lf
N0230 G01 Z-30F0.5 lf N0235 G01 X50 Z-60 F0.5 lf
N0240 G01 X52 F0.5 lf
N0545 G01 Z0 F0.5 lf
N0550 G01 X18.0 F0.5 lf
N0555 G01 X16.0 lf
N0560 G01 Z-30F0.5 lf
N0565 G01 X50 Z-60 F0.5 lf
N0570 G01 X52 F0.5 lf
N0575 G01 Z0 F0.5 lf
N0580 G01 X16.0 F0.5 lf
N0585 G01 X14.0 lf
N0590 G01 Z-30F0.5 lf
N0595 G01 X50 Z-60 F0.5 lf
N0600 G01 X52 F0.5 lf
N0605 G01 Z0 F0.5 lf
N0610 G01 X14.0 F0.5 lf
N0615 G01 X12.0 lf
N0620 G01 Z-30F0.5 lf
N0625 G01 X50 Z-60 F0.5 lf
N0630 G01 X52 F0.5 lf
N0635 G01 Z0 F0.5 lf
N0640 G01 X12.0 F0.5 lf
N0645 G01 X10 Z0 F0.5 lf
N0650 G01 X10 Z-30 F0.5 lf
N0655 G01 X50 Z-60 F0.5 lf
N0660 G01 X50 Z-110 F0.5 lf
N0665 G01 X55 F0.5 lf
N0665 G00 X300 Z300 lf
N0670 M02 lf

IX. Conclusion

This program can be customized to meet a specific family of components. Hence this software may be cost effective for small scale industries which can not afford high cost software. Further it can be expanded to consider features like arc, groove, internal holes, thread etc. as per same programming technique.

References

[1] Ersan ASLAN, Ulvi SEKER, Nedim ALPDEM_IR "Data Extraction From CAD Model For Rotational Parts to be Machined at Turning Centres" , J. of Engineering and Environmental Science, 23 (1999) , 339 - 347

[2] Yakup YILDIZ*, Ihsan KORKUT, Ulvi SEKER, "Development of a Feature Based CAM System for Rotational Parts", G.U. Journal of Science 19(1): 35-40 (2006).

[3] Autodesk , Autocad help center.

Jatin Dave is Asst. Professor in the Department of Mechanical Engineering, Institute of Technology, Nirma University, he has obtained B.E. From Gujarat University in 1999 and M.E. in 2002 from the same University, . He is currently pursuing Ph.D. at Nirma University. His current reserch interst is in CAD, Kinemetics and Robotics. His life member of Indian Society for Technical Eduaction.

V. R.Iyer is Professor in the Department of Mechanical Engineering, Institute of Technology, Nirma University, he has obtained B.E. From RegionalEngineering college, Trichy in 1972 and M.E. from Indian Institute of Technology, Bomabay in 1985. He is currently pursuing Ph.D. at Nirma University. His current reserch interst is in CAD, Kinemetics and Robotics. His life member of Institution of Engineers and Indian Society for Technical Eduaction.