JOURNAL OF ENGINEERING AND TECHNOLOGY, Vol. 9 No. 2 December 2011 ISSN 1684-4114
http://www.iutoic-dhaka.edu/jet

Development of a User-friendly Integrated CNC System Part Program Generation Software Focusing on Various Turning Operations

Ahmed Sayem* and Md. Anisul Islam*

Received 19 October 2011; Accepted after revision 30 December 2011

ABSTRACT

The paper discusses the development of a user-friendly AutoCAD based Part Program generation software, named Design CAM Turning, suitable for various turning operations. Under Visual Basic®6.0 environment, this feature based CNC system software allows automatic CNC codes generation by a unified CAD and process planning and CAM integration. Graphic feature identification and geometric parameter extraction from CAD-part-drawing saved as DXF format are executed for process planning and to generate part program automatically. ActiveX Automation of AutoCAD®14.01 was used to sort out the nodal information and Visual Basic as the programming language as it helps to build a user-friendly and suitable graphical interface. However, software has been facilitated with a 2D simulation of VR Turning software to verify the Part Programs generated prior to using these programs as input to Denford CNC lathe. It has also been provided with the manual data input facility so that it can serve the purpose even to an unskilled user.

Keywords: CNC, Part Program, Auto CAD, Visual Basic, and Turing operation.

* *Department of Industrial and Production Engineering, ShahJalal University of Science and Technology, Sylhet-3114, Bangladesh. Email: sayem-ipe@sust.edu*

1 INTRODUCTION

Time and cost reduction, low volume-large variety production and the high productivity are the prime objectives of developing an automated machining process while modern manufacturing faces several challenges. During last two decades, CAD/CAM technology has been extensively developed to automate and integrate various stages in the design and manufacturing cycle [1]. Andrew and Huw examined the implementation of a sophisticated computer aided design (CAD) package in a small company intending to reduced design cycle times through the development of parts libraries; improved efficiencies in planning; and reduced manufacturing cycle times [2]. The basic role of CAD is to precisely define the geometry of a design. Similarly CAM is the technology concerned with the use of computer systems to plan, manage and control manufacturing operations through either direct or indirect computer interface with the plant's production resources so that a design can be materialized. The Computer Numerical Control (CNC) concerns a modality of the automation and integration of the machine tool in CAD/CAM technology and is used in almost all industrial fields. Yaling Jin et al. proposed if the numerical control system can directly read CAD/CAM generated design documents, it can significantly reduce the processing time, and conducive to the realization of CAD/CAM/CNC integration of three systems [3]. Paul summarized the results of field research on the skill requirements associated with the development and implementation of integrated computer-aided design (CAD) and computer-aided manufacturing (CAM) technology [4].

In recent years, the automatic CNC programming technology is developed rapidly. Faizul Huq explored the capabilities and benefits of CAD are discussed in terms of how CAD can assist companies in becoming more product and cost competitive [5]. So, the mistake caused by programmer is decreased obviously. The quality and the efficiency of NC programming have been enhanced. Omirou and Barouni proposed a series of machine codes selected for integrating advanced programming capabilities into the control of a modern CNC milling system [6]. One study by Li Gang showed the automatic NC programming technology based on the graphic feature principle to realize the integration of CAD/ CAM system [7]. Liu et al. researched the application of an intelligent NC program processor for CNC system of machine tool [8]. Kovacic and Brenzocnik proposed a concept for programming of CNC machines based on genetic algorithms on the basis of CAD models of manufacturing environment [9]. But there are still some shortcomings (complex, lengthiness and inefficiency programming process) as oppose to simple, petite and open-structured programming process. For example upmarket CAM system for complex surface modeling could generate NC codes quickly and efficiently by its huge post-processing module, but it does exist non-ignorable disadvantages [10]. It's usually very expensive and more difficult to learn and use, so it's too costly for small-and-medium-sized enterprises to do on-site processing. These works are efficient for process, but they are usually dependent on CAD system and can't satisfy the demand of the users to input the graphics at anytime.

In an integrated environment, Computer-aided process planning (CAPP) is a critical interface between the automation of product design (i.e., CAD) as well as manufacturing processes (i.e., CAM). There are two types of interfaces, the interface for neutral (mostly geometric) data exchange between CAD and CAM or two different CAD or CAM systems, and the interface for communications between a CAM system and a CNC machine tool [11]. CAPP optimizes and computerizes process planning by using software programs and optimization techniques. Due to the disappearance of experienced process planners in industry, shorter product life cycles and the importance of CAD/CAM integration, research in areas related to CAPP is receiving widespread attention and growing more than ever before [12], though computer aided process planning (CAPP) systems have had limited success in integrating business functions and product manufacturing due to the inaccessibility and incompatibility of information residing in proprietary software [13].

In this paper, a novel solution of many drawbacks of CAD/CAM/CNC automation and integration has been introduced through Auto-CAD based part program generating software called DesignCAM turning. Authors had been influenced by several previous research works in these field, for examples, David and William developed a software framework that demonstrates how computer aided design (CAD)/computer aided process planning (CAPP)/computer aided manufacturing (CAM) technology can be combined, utilizing Visual basic®6 interface, with costing and business tools and made available to small and medium sized firms [13]. Nassehi et al. proposed a new framework that would allow bi-directional and knowledge driven information transfer throughout the manufacturing network, enabling adaptable CNC manufacturer to overcome incompatibility barriers in achieving interoperability in the CAD/CAM/CNC chain [14]. Julio and Luis proposed a feature-linked mechanism to integrate manufacturing traceability data and traceability processes in CAD/CAM/CNC chains [15]. Adem and Mahmut developed a new part recognition approach that was applied to computer aided system, where solid modeler of AutoCAD was used as the geometric modeler and assembly algorithms was completely written in Visual BASIC [16]. Deloyer and Kabir developed an feature based computer aided process planning method (consists of 3 method) to generate G-code by utilizing AutoCAD and Visual Studio(Visual C++) [17]. Sankha Deb, et al. developed an integrated and intelligent CAPP methodology for machined rotationally symmetrical parts, where an interface between design and process planning had been created for automatic feature recognition from commercial CAD software [18]. Bahram et al. wrote feature recognition in data translation in CAD system can be done via implementing the algorithm within independent platforms, such as C++, Visual Basic etc [19]. Yaling Jin et al. described NURBS curve extraction method through ActiveX technology to make CAD/CAPP/CAM system integration Using AutoCAD software [3]. Pengfei Li et al. designed an open architecture CAD graph-driven technology based CNC system software including extraction and auto-optimizing tool path function modules is developed under visual C++ 6.0 environment [20]. Yusri Yusof et al. developed a new STEP-NC code generator(GEN-MILL) which focused on milling process that is able to generate STEP-NC codes and necessary interfaces [21]. They highlighted most of the previous researchers used Visual C++ and Java as their development tools but GEN-MILL uses Visual Basic as the development tools. Cao and Miyamoto used AutoCAD as a basis of direct slicing approach to meet the requirement of faster and precise slicing in Rapid Prototyping and as VBA is a subset of VB, they used VB to develop the slicing program as AutoCAD supports the ActiveX Automation interface [22].

2 THE DESIGNCAM TURNING SOFTWARE DEVELOPMENT METHODOLOGY

The whole methodology involves a series of operational steps and procedures that are employed to develop the software see in figure 1.

Step 1: Tool Bar Creation

Two types of tools were created such as “Draw Tool” for drawing the parts in AutoCAD environment and “DesignCAM Tool” tool to switch from AutoCAD environment to main software windows.

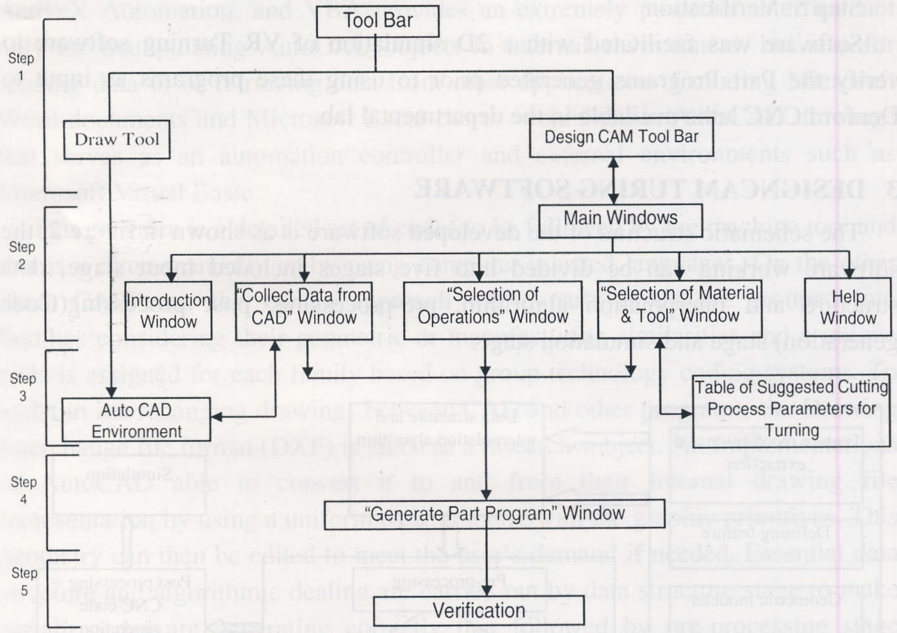


Figure 1: Flow Diagram Showing Various Steps for Software Development

Step 2: Software Windows Creation

"Main Window" was created at this step. This window included the following windows as "Introduction" Window; "Collect Data from CAD" Window; "Selection of Material & Tool" Window; "Selection of Operation" Window; "Help" Window.

Step 3: Linkage Creation

At this stage the following linkages were made through ActiveX Automation and VB, such as "Draw Tool" bar with Auto CAD environment; "Design CAM" tool bar with "Main Windows"; "Collect data from CAD" window with Auto CAD environment; and "Selection of material & tool" window with the table of "Suggested Cutting Process Parameters for Turning"

Step 4: Part Program Generation

Part Program was generated automatically collecting the required data from all the links in format-suitable for VR turning software. Users need to click on the "Generate Part Program" button to generate the Part Program at this stage.

Step 5: Verification

Software was facilitated with a 2D simulation of VR Turning software to verify the Part Programs generated prior to using these programs as input to Denford CNC lathe available in the departmental lab.

3 DESIGNCAM TURING SOFTWARE

The schematic structure of the developed software is as shown in figure 2, the software working can be divided into five stages included input stage, data structure and interpolation algorithm, pre-processing, post processing(Code generation) stage and simulation stage.

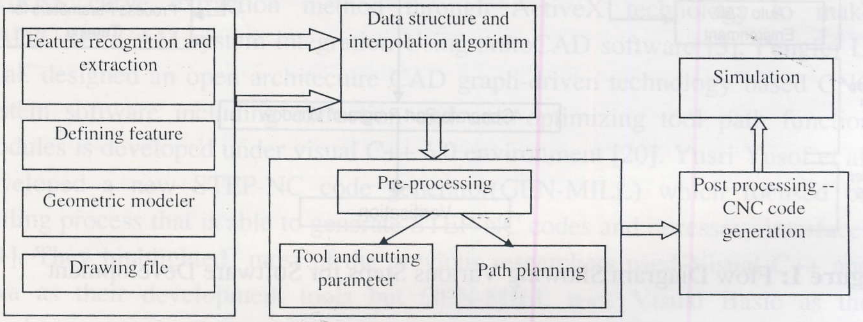


Figure 2: Schematic structure of the developed software.

The DesignCAM turning software developed under Visual Basic®6 environment to ensure user-friendly AutoCAD based part program generation and CAD/CAM integration technically called computer aided process planning(CAPP) on window platform, Autodesk began to support Visual Basic Application (VBA) in the AutoCAD®14.01 (Autodesk, Inc., 1999). Microsoft VBA is an object-oriented programming environment designed to provide rich development capabilities similar to those of Visual Basic (VB). VBA sends messages to AutoCAD by the AutoCAD ActiveX Automation interface(Microsoft's component technology standards, are in fact embedded objects and links (OLE) the new version, ActiveX Controls ActiveX norms are followed to prepare the executable code). AutoCAD VBA permits the VBA environment to run simultaneously with AutoCAD and provides programmable control of AutoCAD through the ActiveX Automation interface. With ActiveX Automation, the features of many applications can be combined in a single application for communication with other programs. This coupling of AutoCAD, ActiveX Automation, and VBA provides an extremely powerful interface not only for manipulating AutoCAD objects in internal environment, but also for sending data to or retrieving data from other applications- including Microsoft Word documents and Microsoft Excel charts - contained in AutoCAD drawings that serves as an automation controller and external environments such as Microsoft Visual Basic.

Part program is a detailed set of codes to be followed by the machine tool and this is written manually or by using computer assisted languages. On the other hand, CAPP use the variant approach where parts firstly are grouped into families considering their geometric or manufacturing similarities and a unique code is assigned for each family based on group technology coding systems. To assist in interchanging drawings between CAD and other programs, the Drawing Interchange file format (DXF) is taken as a research subject. All implementations of AutoCAD able to convert it to and from their internal drawing file representation by using a uniform data structure with the graphic primitives. This geometry can then be edited to meet the user's demand if needed. Essential data structure and algorithmic dealing are carried out by data structure stage to make certain codes are generating correctly that followed by pre-processing stage which ensures proper tool selection and cutting parameters and tool planning so on. After then, CNC programming stage in post-processing is used to generate automatic CNC codes.

3.1 Input stage

A DXF file consists of up to seven sections: a header, tables, blocks, classes, objects, entities, and an end-of-file (EOF) marker. The BLOCKS section contains predefined drawing elements that might be present in the drawing and The ENTITIES section contains the actual object data of the drawing. This can include raw data such as line, circle, ellipse, spline and arc entities as well as insert commands that place a predefined block definition at a certain position in the drawing.

The recognition and pulling out of part machining feature from CAD drawing is the most important work for DesignCAM turning software. The geometric data of CAD drawing are saved in the ENTITIES section after being examined of DXF file structure. The ENTITIES section is divided into many group units. Each group unit is composed by two elements, which are called group code and group value. Group code is equal to the data type code, whose meaning is defined by AutoCAD system. Group value is the value of data type. The combination of two elements can express a data's meaning and its value. By reading the contents of ENTITIES section in the programming procedure, the machining feature parameters of parts are identified and extracted from CAD drawing. Firstly, the contents of DXF drawing are read and followed by the line until ENTITIES section. Secondly, after the group values of ENTITIES section being read, the different graph features are identified according to the group value.

3.2 Data structure and interpolation algorithm stage

The codes to collect data from CAD are written in the 'Form' named 'DesignCAM turn'. After data input into the system using DXF format a set of processes is done automatically according to a specific hierarchy. Firstly, the features in the drawing are found and defined to the system. After these processes, all coordinates of the features are sequenced according to their start points and transferred to the origin. Finally the shape of the work piece is drawn with the program with its sub symmetry.

Another main part is to select material and tool for some specific data that are industrially recommended for some parameters like feed, cutting speed, incremental depth of cut and hardness. Here, the essential actions are being taken for data input with the purpose of the development of G code correctly and efficiently. The whole process is done automatically according to specific algorithm. The process contains sequencing of graphic elements and interpolation matching for input elements. An adaptive interpolation algorithm combined with the shape information for non-arc curve is adopted. However for ease of interpretation all variables are declared in one module named 'Variable Definition'.

3.3 Pre-processing stage

This part provides a list of parameters for machining, including depth of cut, cutting speed, finishing allowances, step depth, step-over, and spindle speed et al. by virtue of human-machine interface. Then the header and tail of the code file can be generated with this information.

3.4 NC code generation module

CNC codes are generated from the part drawings or DXF files after being amended by process parameters and geometric information organized by means of sequencing and interpolation algorithm. Firstly, the header of code file is

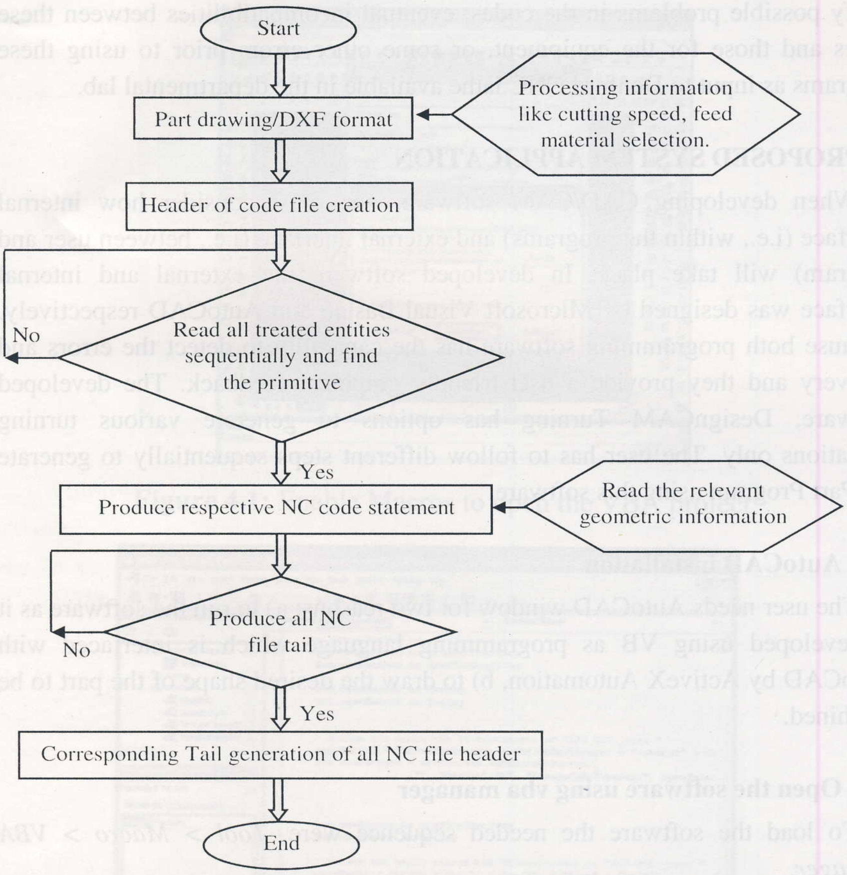


Figure 3: The Flow sequence of CNC code generation

created from the information like processing parameters and tool selection and error controlling. Secondly, read the all treated entities and find the primitive and then read the geometric information and place it into CNC code statement. Thirdly, correspond CNC file tail is developed. The flow sequence of CNC code generation is shown in figure 3.

3.5 Simulation stage

Simulation aims to make the best use of resources (machine, material, and men: called entities in simulation jargon) by helping the user to decide what is needed, and where the best position to locate it. In this regard, the developed software has been facilitated with a 2D simulation of VR Turning software to verify possible problems in the codes, eventual incompatibilities between these codes and those for the equipment, or some other errors prior to using these programs as input to Denford CNC lathe available in the departmental lab.

4 PROPOSED SYSTEM APPLICATION

When developing CAD/CAM software, one must consider how internal interface (i.e., within the programs) and external interface (i.e., between user and program) will take place. In developed software the external and internal interface was designed by Microsoft Visual Basic6 and AutoCAD respectively. Because both programming software has the capability to detect the errors and recovery and they provide a user-friendly graphical feedback. The developed software, DesignCAM Turning has options to generate various turning operations only. The user has to follow different steps sequentially to generate the Part Program using this software.

4.1 AutoCAD installation

The user needs AutoCAD window for two reasons: a) to run the software as it is developed using VB as programming language which is interfaced with AutoCAD by ActiveX Automation, b) to draw the desired shape of the part to be machined.

4.2 Open the software using vba manager

To load the software the needed sequence were, *Tool > Macro > VBA Manager*.

4.3 Creation of tool bar

Performed tasks needed to create the tool bar were:

- Open the VBA manager window by following the command given below:

Click on load button first to load the program browsing from D:\DesignCAM Turning\ADesignCAM\ACAD Proj.\ Project1.dvb (if DesignCAM Turning is installed in Drive D). And then Click two extra buttons named **Enable Macros** and **Visual Basic Editor**, as shown in the figure 4.1 given below, to open the file fully.

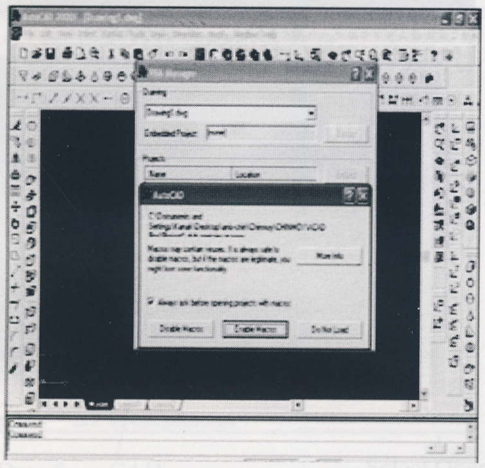


Figure 4.1: Enable Macros to open the VBA project

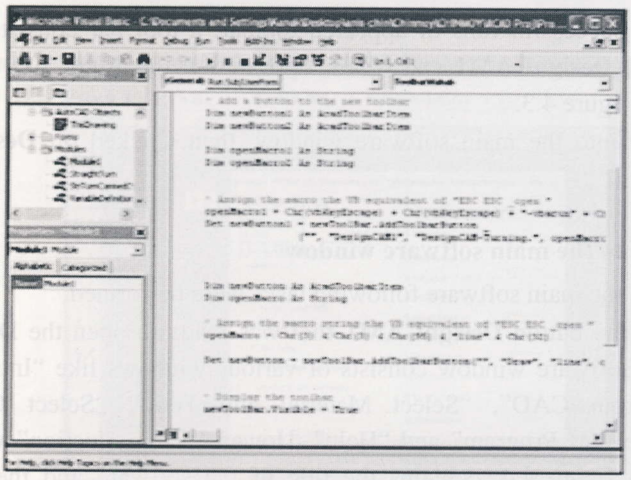


Figure 4.2: Microsoft Visual Basic window to run the project.

At the end of using all the above buttons the visual basic program appeared at the screen as shown in figure 4.2.

- The program was run by using the symbol button of **Run sub/user form** or by using the key **F5** of the keyboard.

The users have to be careful about the modules that are in sequence at the top-left corner. If the Module1 is not shown as selected, the user has to select and open Module1 first and then use **run** button.

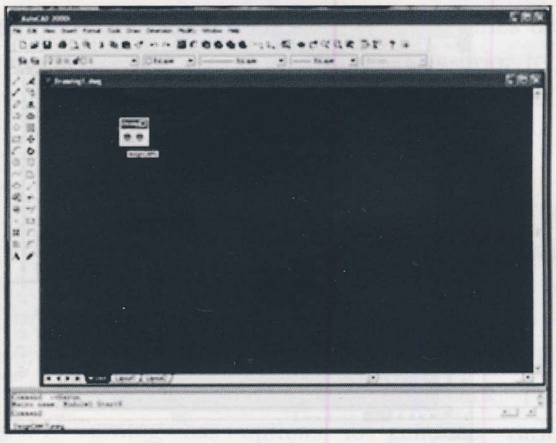


Figure 4.3: Lathe CAM design toolbar at AutoCAD window

After the successful runs of appeared toolbar, a tool bar consisting of two tools namely, **DesignCAM1** and **Draw**, appeared at the AutoCAD window as shown in the figure 4.3.

- To enter into the main software window, then Clicked on **DesignCAM1** button.

4.4 Launch to the main software window

To lunch the main software following tasks were performed:

- Click on the button **DesignCAM1 tool** was made to open the DesignCAM Turning Software window consists of various windows like "Introduction", "Data From CAD", "Select Material and Tool", "Select Operation", "Generate Part Program" and "Help". However, "Introduction" window, as shown in figure 4.4, contains the title of the software and the copyright regulations only.
- The chosen operation was performed after opening the window by Click on 'Data From CAD'. It was ensured that the operation chosen has well matched with the image shown at the top-left corner of this window shown in figure 4.5.
- Part of the desired shape was drawn by Clicking on 'Go to CAD' button given at the bottom of the window to go back to AutoCAD environment.

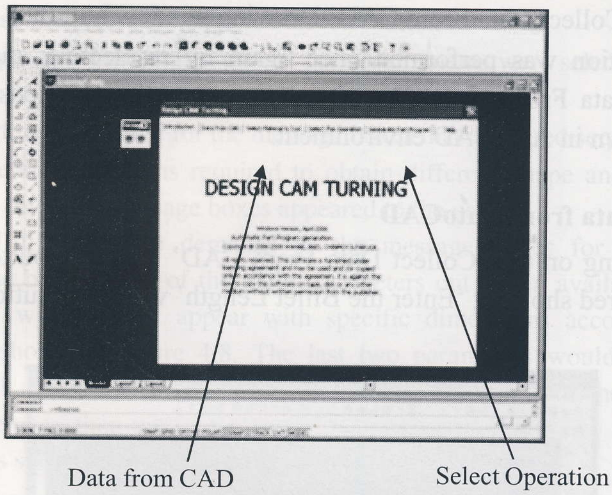


Figure 4.4: The introduction window

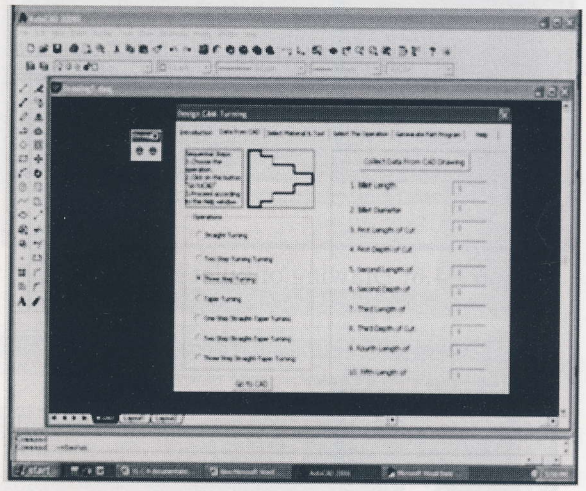


Figure 4.5: Collect Data from CAD window.

4.5 Drawing at the autocad environment

- As shown in figure 4.6, the part of the desired shape was drawn by clicking on Draw Tool button. At this stage, the users have to draw the desired shape according to the image shown in 'Data from CAD' window.
- To go back to the main software windows a Click was done on 'DesignCAM1. Normally, the Introduction window was appeared to the users.

- Click on 'Collect Data From CAD' Drawing window button.
- The operation was performed once again by reselecting and clicking on 'Collect Data From CAD' Drawing button to collect dimensions from the figure drawn in AutoCAD environment.

4.6 Collect data from AutoCAD

After clicking on the 'Collect Data From CAD' Drawing button, a massage box was appeared showing 'Enter the Billet Length' with 'ok' button as shown in figure 4.7.

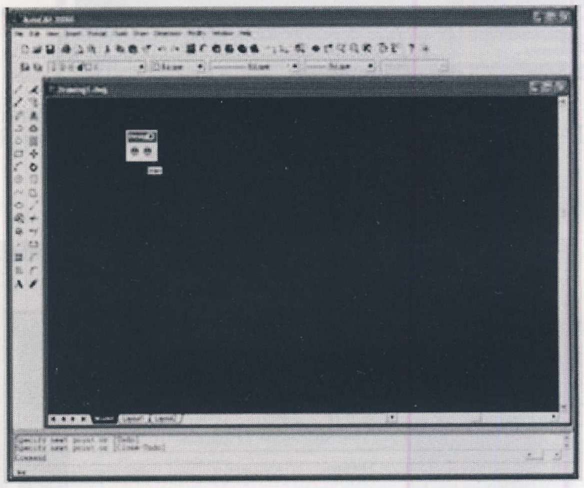


Figure 4.6: Drawing of billet by draw tool.

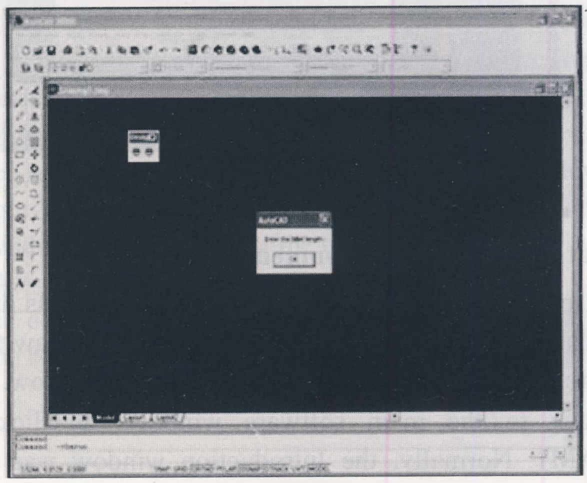


Figure 4.7: Collection of Billet Length Data from CAD

- To release the message box Click on 'ok' button
- Certain points of the drawn part as per requirements were selected as shown on the massage box.
- The work was repeated for the message boxes that appeared sequentially
- For different operations required to obtain different shape and size of the product, different message boxes appeared to the user.

After fulfilling all the demands of the message boxes for the selected operation, the blank space of the eight parameters out of ten available in 'Data From CAD' window will appear with specific dimensions according to the drawing as shown in figure 4.8. The last two parameters would contain the dimensions 1 as a default, i.e. during collection of data if four message boxes appear to the user, then the four parameters of the second window will be filled up and others six will show unity by default.

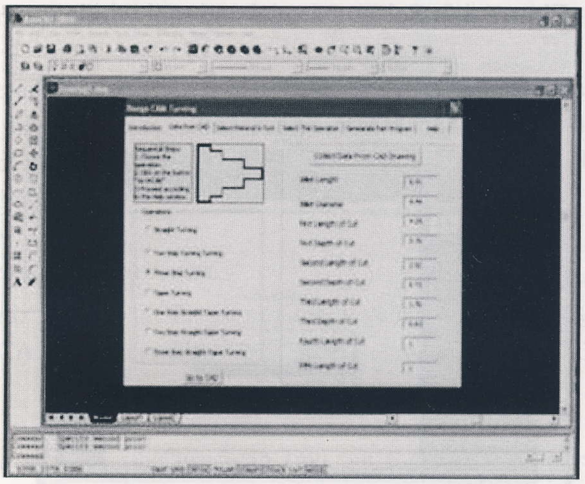


Figure 4.8: Collection of Data from AutoCAD.

4.7 SELECTION OF MATERIAL AND TOOL

Tasks to be performed in this step are

- Click on the Select material and tool window button to open the window as shown in figure 4.9
- Select the required working material and cutting tool from the Select Material and Select Tool lists respectively.

Once the required working material and cutting tool are selected, the corresponding values of material hardness, feed, incremental depth of cut (only

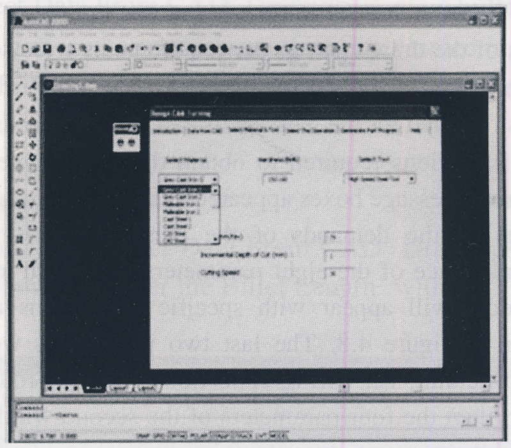


Figure 4.9: Select material and tool window.

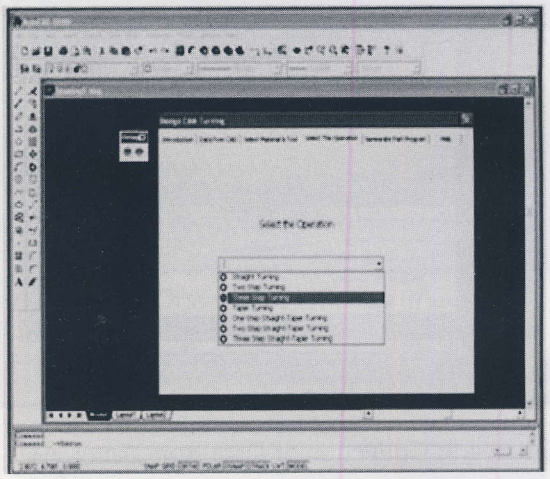


Figure 4.10: Select the operation window.

for step turning otherwise, by default, it would show unity) and cutting speed will be shown in the respective text boxes.

4.8 Selection of operation

Tasks to be done in this stage

- Click on "Select the operation" window button to open this window as shown in the figure. 4.10. This window consists of the list of operations such as Straight Turning Operation, Two Step Turning, Three Step Turning, Taper Turning, One Step Straight Taper Turning, Two Step Straight Taper Turning and Three Step Straight Taper Turning.

- Select the operation that was chosen in "Data from CAD" window.

4.9 Part program generation

Tasks to be performed at this stage

- Click on "Generate Part Program" window button to open this window. However this window contains two buttons named "Generate Part Program" and "Exit" as shown in figure 4.11.
- Click on "Generate Part Program" button to generate a part program for the selected operation to be performed. However with clicking on this button a message box will appear with a message "The Part Program for the selected operation has already generated as CA.fnl in drive C:\." as shown in figure 4.12.

If the operation from the list is not selected, an error message will appear to the user and the user has to select the operation, going back to the forth window named 'Select the Operation'. Then by clicking the button exit, the user can exit from the software.

4.10 Generated part program

For visualizing the Part Program already generated, the users have to follow the path: C:/CA.fnl and open it.

4.11 Help

From this window, the user can find out the sequence of step for getting output from the software.

5 Experimentation and verification

At this stage the verification of the generated Part Program for the selected operation has been done with the support of 2D simulation which is built in VR Turning software suitable for Denford CNC Lathe available in the departmental lab (shown in figure 5.1) which is used for academic, testing and consultancy purposes.

To verify the program a 2D simulation was run prior to manufacture. User can simulate the contents of the CNC file, if there is an unwanted move in the

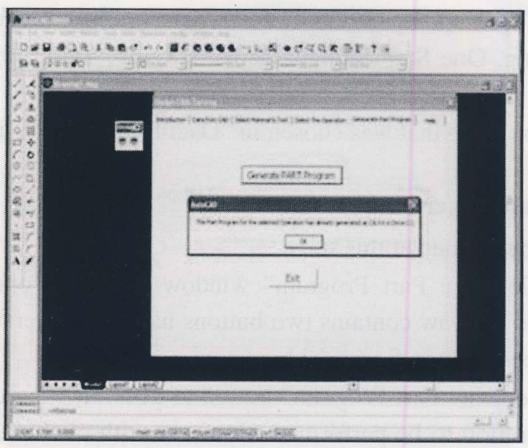


Figure 4.11: Generate Part Program Window.

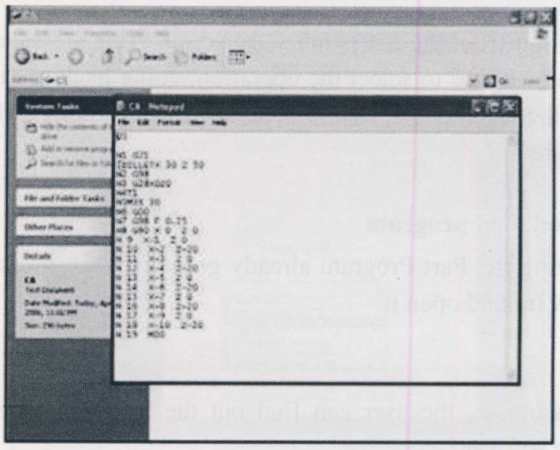


Figure 4.12: Generated Part Program.

program, this will show in the simulation. The simulation uses the information in the tooling window to generate the graphics. Click on the 2D simulation button of VR turning software, the 2D Simulation window provides a side view of the billet as shown in the figure 5.2.

6 CONCLUSIONS

A novel part program generating software "DesignCAM turning" is suggested and developed with most exploited, cheap and user-friendly AutoCAD®14.01 and VB®6 software for different types of turning operations only as Straight Turning, Two Step Straight Turning, Three Step Turning,

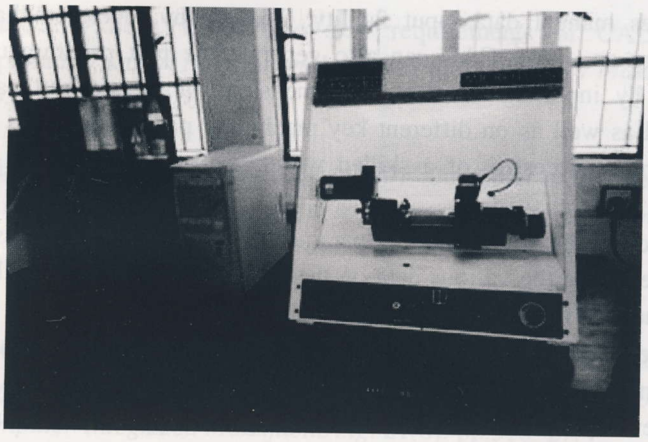


Figure 5.1: Denford CNC Lathe used for the experimentation.

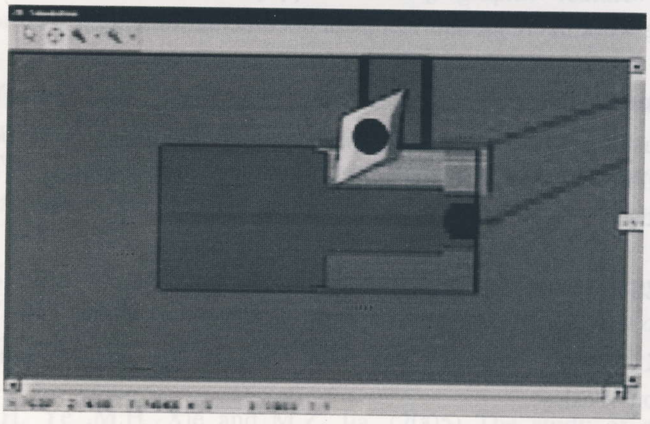
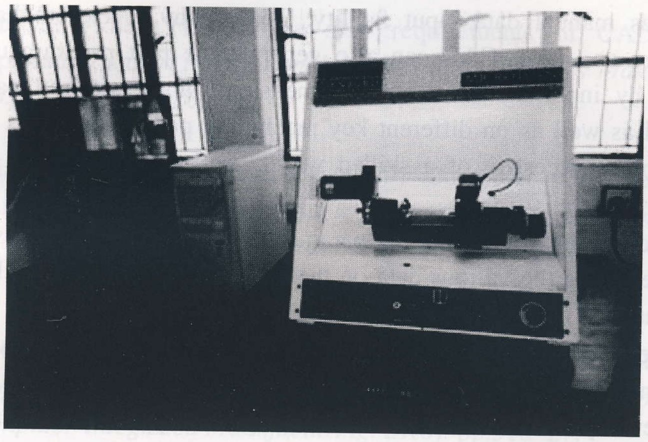


Figure 5.2: Verification of the generated program using VR Turning software.

Taper-Straight-Taper Turning, Two Step Straight-Taper Turning, and Three Step Straight-Taper Turning. This software was tested and validated on Denford CNC lathe available in the departmental lab which is used for academic, testing and consultancy purposes. Apart from this, the developed software named Design CAM Turning has the following features:

- It is able to take data from Auto CAD environment, and generate Part Program automatically.
- It provides Computer Aided Process Planning (CAPP) facilities, like material, tool and operation selection.



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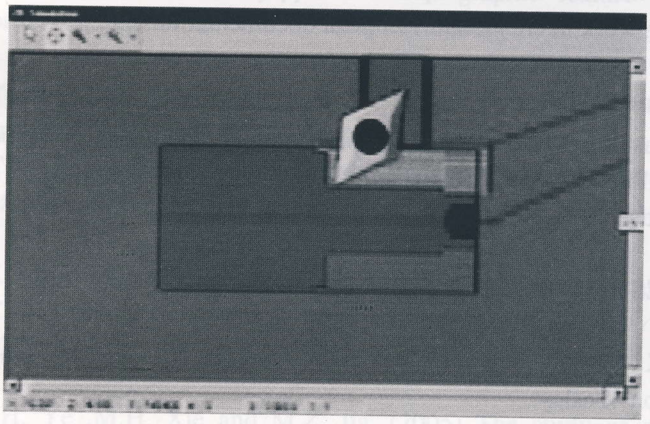

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- It is able to take data from Auto CAD environment, and generate Part Program automatically.
- It provides Computer Aided Process Planning (CAPP) facilities, like material, tool and operation selection.

- It includes manual data input facility. That's why; operators having no knowledge on AutoCAD can also generate the part program using the software by inputting the data on dimensions of the desired part to be machined as well as on different key machining parameters. This program could replace necessity of a skilled worker or part programmer in some extents.
- The values of the key machining parameters are built-in, so the users need not to memorize the various data on these parameters. However, these data can be changed manually as per requirements.
- The time required to create properties and a technical drawing was significantly reduced as well as the number of errors.

Future developments would include linking the application to the other machining operations. Additionally, extensive testing is under way in a real work environment. This will further improve the application to meet the designer's needs regarding work in creation of technical documentation.

Acknowledgement

This work was supported by the Department of Industrial & Production Engineering (IPE), ShahJalal University of Science & Technology (SUST), Sylhet, Bangladesh.

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