MODERN CAD/CAM APPLICATIONS- INTUITIVE AND EFFICIENT

Adrian BUT
"Politehnica" University of Timisoara
E-mail: adi.but@gmail.com

Eniko BUT
Tibiscus University of Timisoara
E-mail eni.but@gmail.com

Abstract: *In this paper, based on an example, it shows how to create a NC program assisted by computer. It highlights the role of graphical programming, ease of interacting with the computer, easy to send commands to echelon operations yet leaving to the human programmer the freedom to intervene and impose their views of creatively. Using this freedom of action, the programmer can optimize the processing operation. It can get different processing times, applying different processing strategies. You can also change the cutting tools parameters. Comparisons can be made before the process through advanced 2D and 3D simulations. Are given opportunities to import the CAD file or drawing graphics effectively use the facilities offered by the CAD capabilities of the software. Follow the raw part defining, establishing and implementing the clamping systems. Using the database of the program can select the cutting tools or we can create others. Using specific post processors for different machine tools will be developed the NC program. Another advantage of the program is how to spread from computer to program NC machine tool, or from computer simulations which verify NC program designed. After more than five year experience and practice with KELLER CAD/CAM software are present the advantages and disadvantages. Methodology adopted to prove the efficiently of the CAD/CAM software presented was the cases analyzed with their results and one real application with the impact efficiently in industrial area. The ease with which the programmer is able to transmit information, to obtain a correct NC program has led to the successful implementation of this software in our teaching process.*

Keywords: *CNC, manufacturing, training center*

I. INTRODUCTION

The part programming assisted by computer what will be machined on CNC machine tools requires multidisciplinary knowledge's. The complexity of NC program elaboration in the classical method, who consist in translation the technological information's in alphanumeric codes, makes the young people or those who come first time in contact with this activity, to have difficulties   to learn to program the CNC machines tools.

The paper presents an alternative at the classical programming mode, called: graphical programming. By this method, who was implement in Keller CAD / CAM software, are use graphical icons to guide the user in the most direct and simple way.

In addition, the programming method includes several alternative input modes - from NC editor through to graphical geometry editor, interactive programming and teach-in operation right up to dialog programming.

The operator can switch between these functions as often as he wishes while making entries.

The software offers a lot of standard basic functions (in CAD as well as in CAM):

- CAD data exchange using interfaces (for read in for example DXF, IGS)

- automatic generation of work plans and set-up sheets,
-work plan editor with graphical display of the individual machining steps, a lot of the functions 'move', 'delete', 'insert', 'activate',
- editing of NC tool paths by subsequent re-entry in the machining dialog,
-styles generated for repeated technology on different contours and bodies,
- tool path associatively to contours,
-bidirectional DNC program transmission between programming system and CNC control.

II. METHODOLOGY

To emphasize efficiency and the freedom to generate the NC program in the "graphical mode", will be presented on an example, the programming steps and CAM software facilities.

In the first contact with the software, will be guide to five different operating mode: F1-the CAD module, F2 –the CAM applications, F3 – SIMULATOR –the part where will be able to check the work, F4- The instrumentation to SETUP the tools and F5- the solutions to TRANSFER the program from computer to the NC equipment or vice versa. (Fig.1.)

Figure 1. The five operating mode

In the 'Geometry' operating mode (Fig.2a), the design of the work piece geometry is created in the graphic dialog using pictograms or available CAD data. DXF and IGES files can also be used. (Fig.2b)

Programming systems normally import the work piece geometry from CAD data and initially generate NC programs in a neutral, generic format, which is subsequently converted by postprocessors, into the programming key* of the target control system.
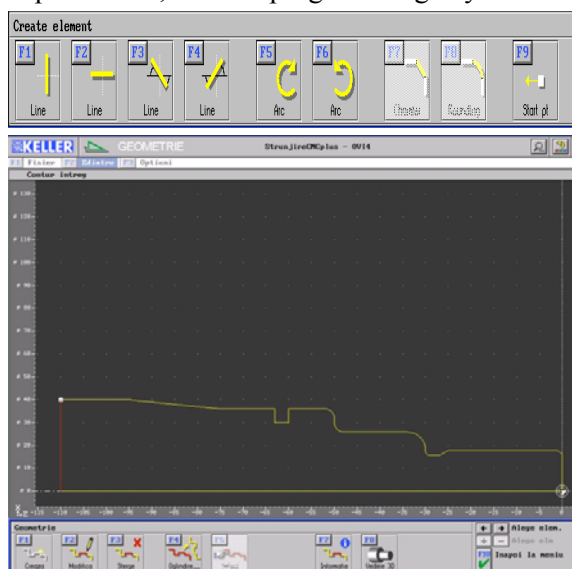

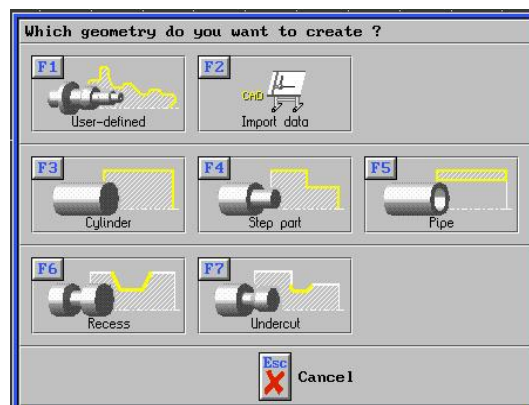**Figure 2a.** Example in 'Geometry' operating mode   **Figure 2b.** CAD Import data

In the 'Work plan' operating mode (Fig.3), you can progress quickly and easily with the graphic programming from the drawing to the NC program without ever having to type a G or M function.

The material who are in the "Material Menu" are used for technology assignment of the individual tools. If you then set a certain material when creating a work plan, the systems automatically offers you the corresponding technologies for the tools in the individual machining steps.
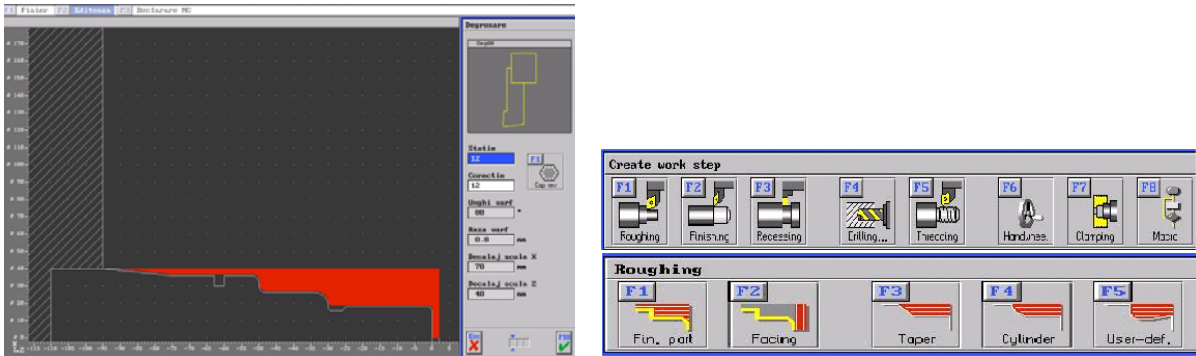
Figure 3. Example in 'Workplan' operating mode

In the 'Simulator' operating mode (Fig.4), you can write and simulate NC programs for different control formats in a "guided" editor with an integrated, control-specific help system. In addition, a keyboard help system is available for each set of controls, which explains the functions of the individual control keys.
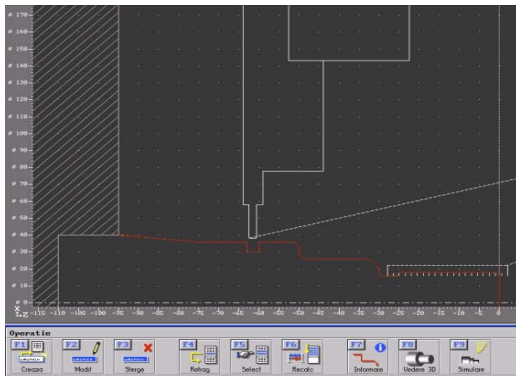

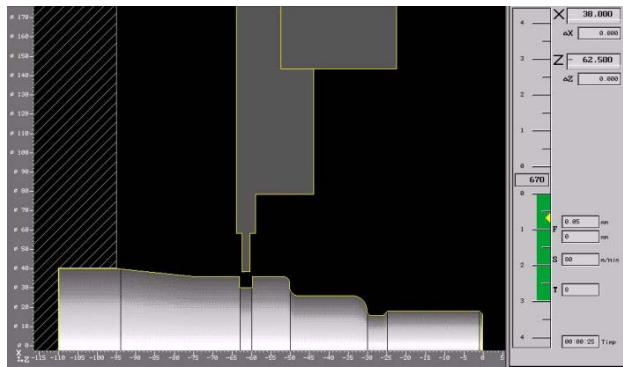**Figure 4a.** 2D Simulation of Roughing operation

Figure 4b. 3D Simulation of Roughing operation

In the 'Setup' operating mode(Fig.5), you can map your tools geometrically, provide cutting data and implement turrets or magazines. The folders for the different file types of the plus Systems are also managed here, and settings can be made in the system configuration including all machining cycles, in particular radial and axial recess cycles with selectable recessing or roughing machining, macros for work part grooves form according to DIN/ISO, radii and chamfers, milling with driven
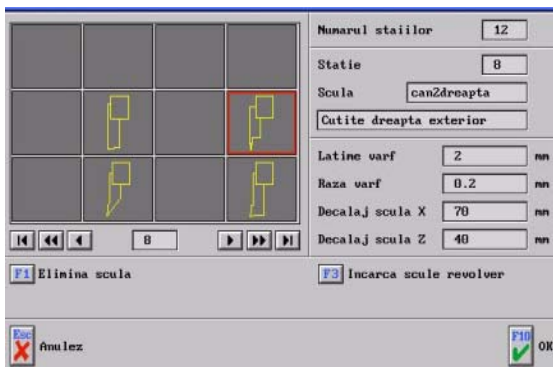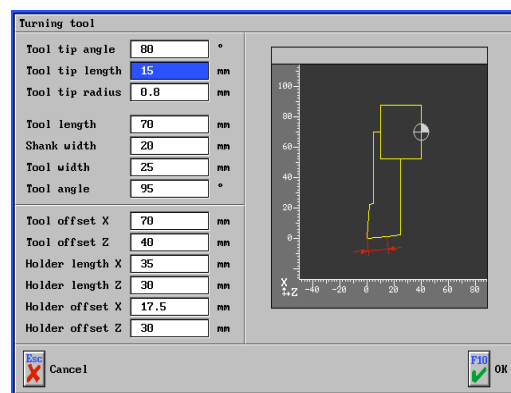

Figure 5a. Data base of cutting tool.

Figure 5b. The possibility to change the tool parameters’

tools on the planes: front face, chords and slanted chord surfaces as well as circumferential surfaces with using the machining cycles: pocket-milling, drilling, reaming, boring, thread-cutting and hole patterns with user macros for sequences of operations with different tools**.** In the same time you have the possibility to change the geometrical parameter for different tool.

In the Transfer' operating mode (Fig.6), you can transfer NC programs between PC and controls. You can also view or adapt NC programs and tooling sheets in a simple text editor, although no simulation is possible with this.
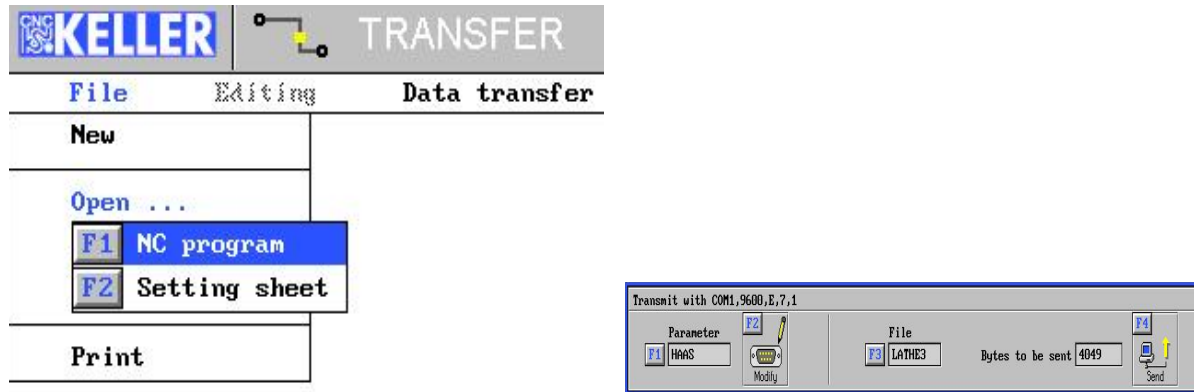

Figure 6. Commands in "Transfer" operating mode

Perfect data transmission indispensably requires the correct setting of the transmission parameters. As an example, a sample file is supplied for the transmission parameters. However, the parameters have to be individually set to the corresponding control system and saved as a new parameter set

3D programming system for milling using all common machining methods such as drilling, reaming, boring, threading cycles, circular and rectangular pockets and pins, as well as, in particular, contour parallel or meandering milling of pockets with any type of edges and islands, holes and finishing
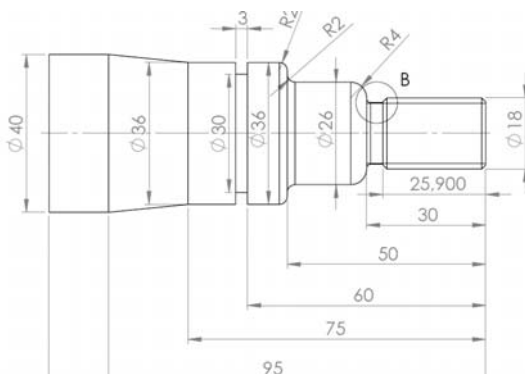

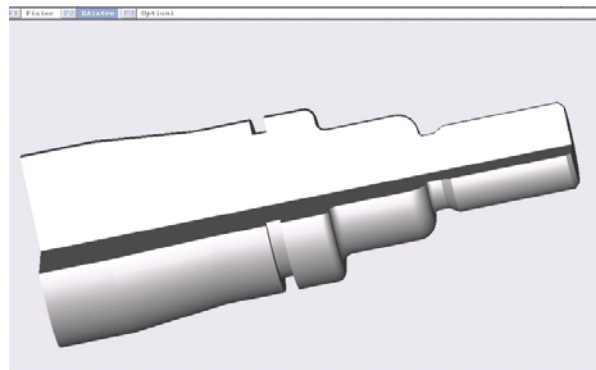**Figure 7a.** 2Drawing of the work piece   **Figure 7b.** 3D Simulation

also with creation of radial and linear hole patterns, approach and withdrawal macros for contour milling and engraving of groups of entities with optimization of the tool path.

The postprocessor translates NC programs created with the KELLER programming system into the programming key* of the target CNC control. These can then be simulated, tested for collisions and further processed, optimized, in direct connection with the LATHE and MILL, shop-floor programming workstations which have been set to this programming key.

It is this unbroken line of processes, connecting the programming system with an easy-to-use dialog programming method in the programming key of any desired CNC control, which his the outstanding feature of CAD/CAM solution.

This combination of the programming key and the postprocessor (Fig. 8) can then be used as a universal NC converter.
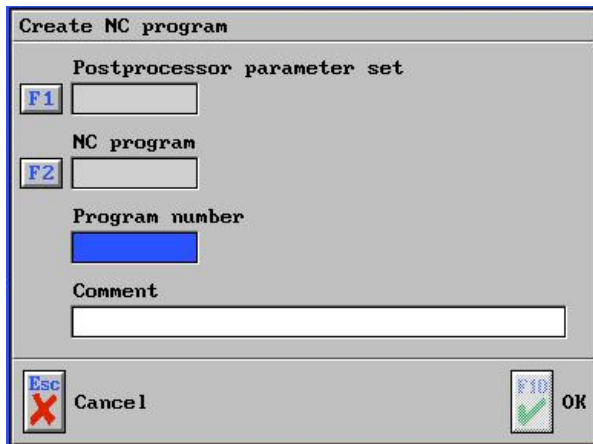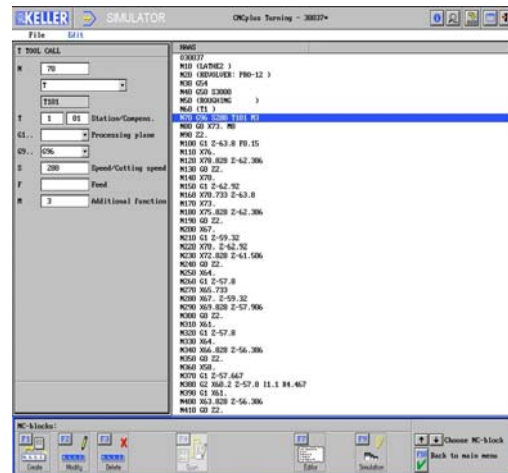
481

Figure 8. Set the postprocessor

Figure 9. NC program

2.1. Programming system

In everyday operations, we encounter both so-called computer-assisted programming, using programming systems as employed in work planning departments, as well, such as is carried out directly on the CNC control of a machine tool.

Programming systems normally import the work piece geometry from CAD data and initially generate NC programs(Fig.9) in a neutral, generic format, which is subsequently converted by postprocessors, into the programming key* of the target control system.

One main feature of the Keller programming system, which is embedded in a fully-functional CAD front-end, is the graphical user interface linking it directly to the CAD data. CAD system offer solution from parametric 2D up to nurbs surface modeler.

Interface is able to support the job with 2D geometries (in work planes included). In this system, the programs are generated with the aid of a simple interactive dialog, the work plan being automatically compiled as the program is created.

2.2. Postprocessors

Path is generated in graphical CAD interface from position of tool pick point in contact with body, surface or contour. Predefined set of technology data together with machined geometry are on demand immediately transferred via above subscribed postprocessor into the language of control system.

Dialog programming can be used to create and edit NC programs in the respective programming key without having to resort to explicit editing of NC blocks. Dialog steps are selected individually, starting from the overview of the machining functions.

III. CONCLUSION

The most efficient method of creating a program is to combine dialog programming with the geometry editor and interactive programming, whereby the NC editor can be used to display the resulting NC program at any time. Another solution to generate NC program is automatic mode.

In automatic mode, NC programs can be simulated in continuous sequence or block by block in real time with override function, or in an accelerated test run. Dynamic simulation is displayed graphically in two dimensional or optionally three dimensional views. The manufacturing, feed and tool usage times as well as the volume of work piece material removed by machining are calculated concurrently.

References

[1] S. KELLER-Manual of Plus System-2005
[2] But Adrian-(2003) Influența construcției strungurilor cu comandă numerică asupra preciziei de prelucrare- Editura POLITEHNICA-Timisoara2003, ISBN-973-625-014-8