BULETINUL Universităţii Petrol – Gaze din Ploieşti	Vol. LIX No. 4/2007	45 - 52	Seria Tehnică

CAD and CAM Integration

Răzvan Duţă*, Marius Gabriel Petrescu**

* SonaHess GCR Project, GEA, Algeria,
e-mail: razvan.duta@hess.com
** Petroleum-Gas University of Ploieşti, Bd. Bucureşti 39, Ploieşti, România
e-mail: pmarius@upg-ploiesti.ro

Abstract

In this paper it is presented the methodology to write a program for a CNC machine, including all the necessary steps and commands. The outcome of the machining phase should be a flange with the dimensions and tolerances as per attached draft.

Key words: CNC machine, Computer Assisted Drafting/Drawing, Computer Aided Machining /Manufacturing

General Considerations

The purpose of this exercise is to show that the Computer Assisted Drafting/Drawing software (CAD) can be integrated with Computer Aided Machining/Manufacturing (CAM), and that it is a proficient manner to approach nowadays manufacturing.

The outcome of the machining phase should be a flange with the dimensions and tolerances as per attached draft (see fig. 1).

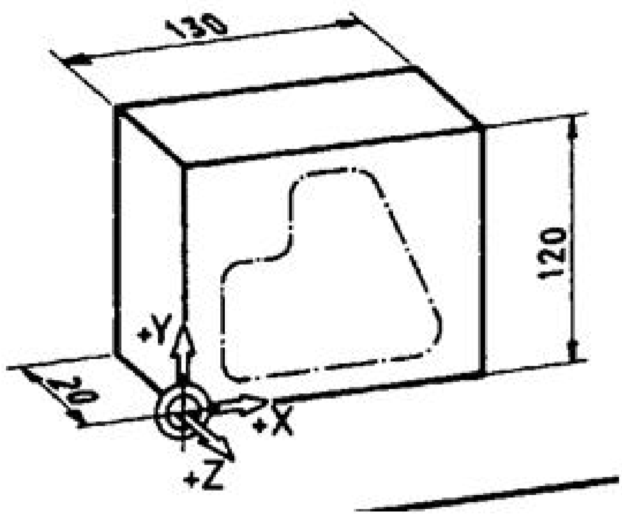

Fig. 1. Flange draft: dimensions and tolerances

From the today multitude of drafting/design software, AutoCAD from AutoDesk has been chosen, as one of the most reliable and packed with features software. The version used was AutoCAD 2007.

Design Stages for a Manufactured Part

The design stages are the following: drawing the part accurately in AutoCAD; export the part drawing to a CNC software; writing the program in the chosen CNC software.

a. **Drawing the part accurately in AutoCAD**
Proper drawing techniques must be applied since upon importing the final drawing into MasterCAM X, all dimensions will be inherited and become manufacturing dimensions. Therefore, accuracy during drawing phase shall be paramount. Basic drafting rules shall be applied, as it follows:

- All lines shall be drawn only by entering START, respectively END POINT coordinates;
- Use of the ORTHO MODE feature, under Drafting Settings is very useful while drafting perpendicular lines;
- The TRACKING feature come at hand when drafting a line from a START point which coordinates are deducted in conjunction to those of a known element (already drawn) coordinates;
- CHAMFER and FILLET modifying commands shall be used not only according to the standards stipulating safe radius of manufactured parts, but also giving consideration to the tools (usually end mills) to be used to create these finished surfaces;
- Overall part dimensions shall be taken into account as well since these are going to have a direct impact on the machine chosen for manufacturing the part, at a later stage.

An example of drawing made in AutoCAD 2007, can be seen below:

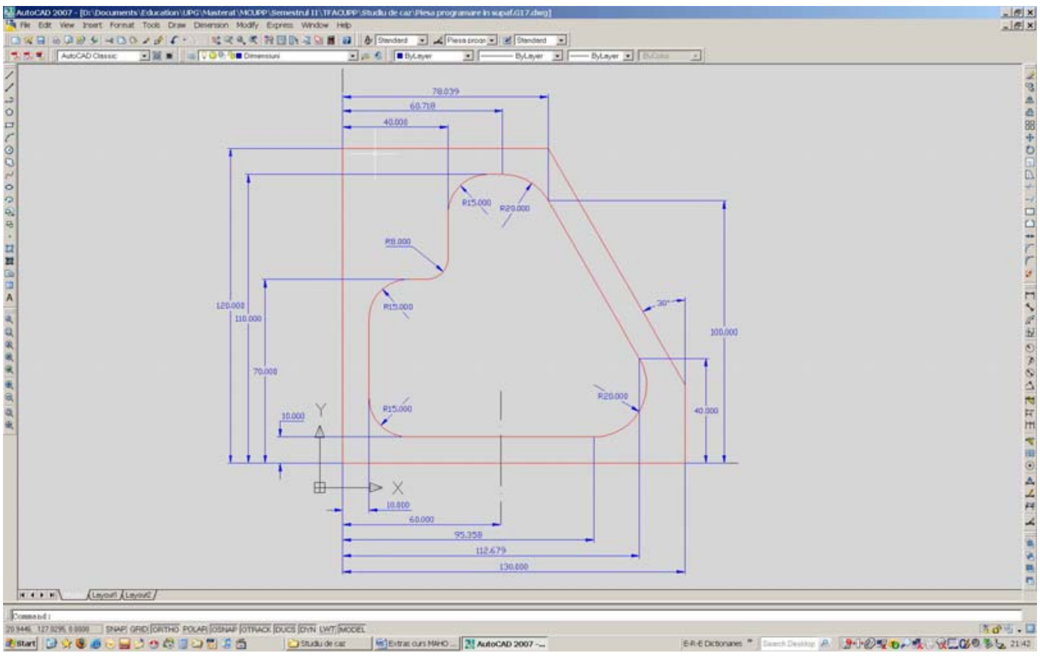

Fig. 2. The part drawn in AutoCAD 2007

b. **Exporting the part drawing to a CNC software**

Drawings made with AutoCAD series software have .dwg extension. They are fully recognized during an import session into MasterCAM X, and only now the well spent time during drawing the part in AutoCAD is being paid off.

A 2D view, respectively a 3D view of the imported part drawing, are shown in figure 3. The 3D view is useful during part manufacturing design, as the three-dimensional view is always easy to mentally place on the machine table, see the tools paths around the contour and inside the part, design and use specific holding down devices, etc.

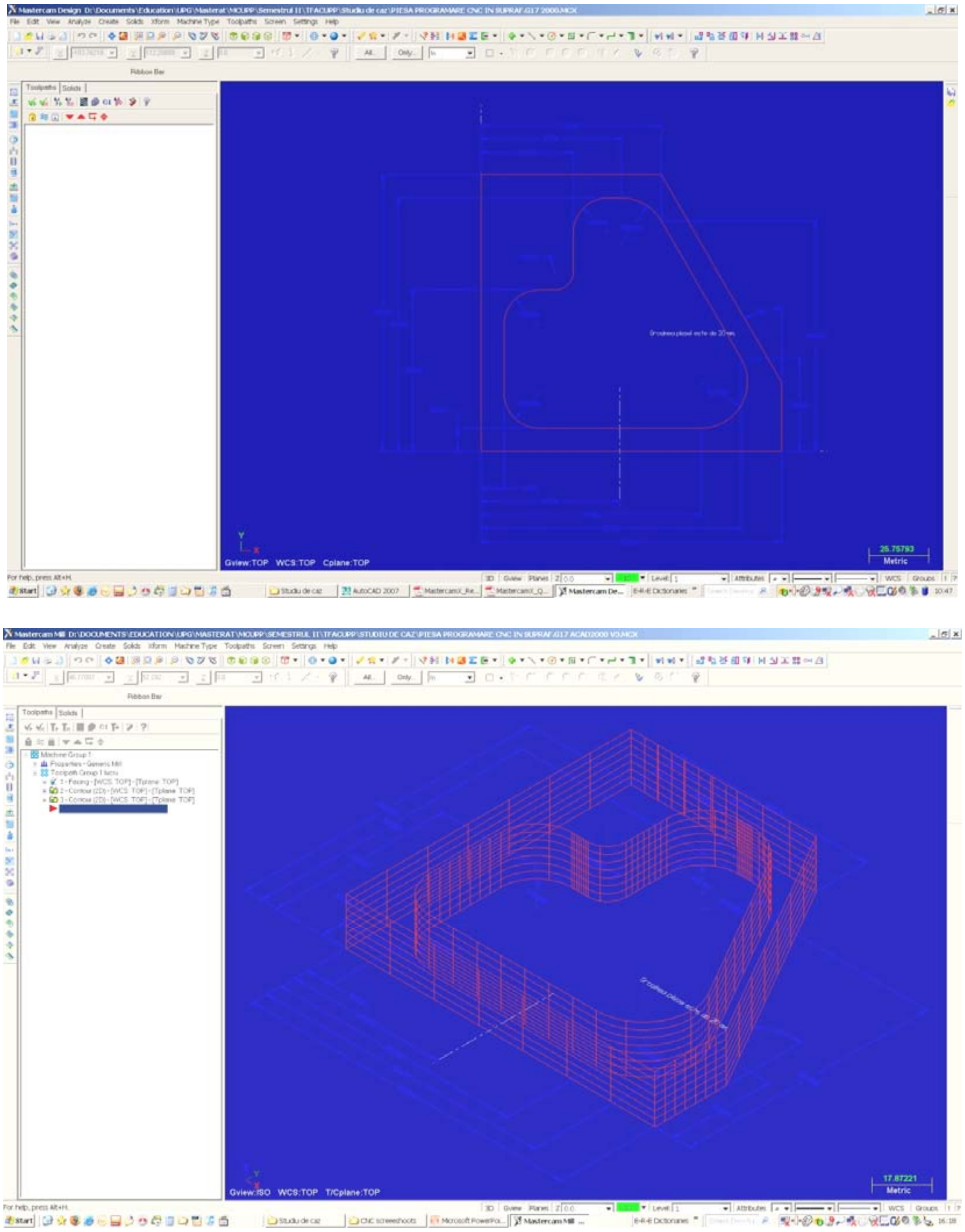

Fig. 3. A 2D view, respectively a 3D view of the part imported in MasterCAM X

c. **Writing the CNC file**

Great consideration during writing the CNC file shall be given to machines and tools to be used, as a general consideration.

It can be overwhelming choosing the right tool for instance, but good knowledge of the traditional manufacturing is a very good point to start.

Although the CNC machines are using specialized tools, especially designed not only for that particular type of machine, but also for special operations or parts, they all are based on a classic design. The cutting portion of the tool is performing basically the same function which has been designed for, in both cases. The only difference resides in the shape of the tool tail, as well as on the solution chosen for holding, transporting, and engaging the tool during machining process.

The first action to be taken is choosing the most appropriate machine for part manufacturing. Usually the CNC software is versatile and come along with a large library of machines to choose from. Again, knowledge of traditional machining is very useful and it should be used by no means.

In the case studied here, a MILL is the right machine to be used for flat surfaces, with an irregular contour. Since the contour is made of shapes easy to describe geometrically, the DEFAULT MACHINE is the best choice.

When it comes down to choosing the tool, the CNC software is also suggesting the default tool, as it we will see later on.

Setting the machine from a drop down list is shown in the figure 4.

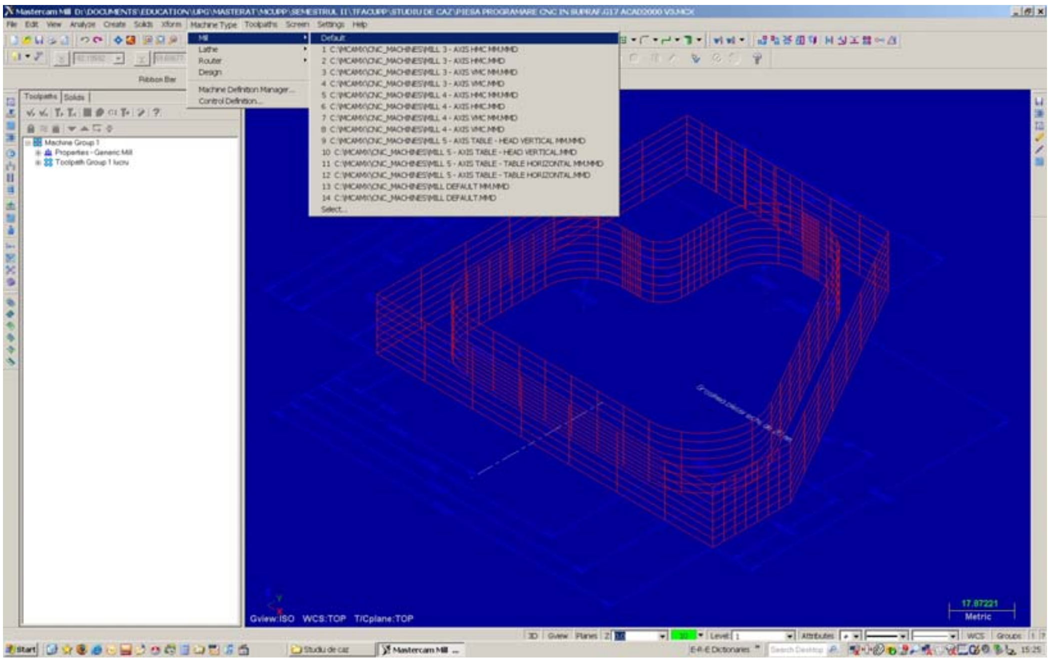

Fig. 4. Setting the milling machine from a drop down list

The steps taken until now are quite similar to those followed when designing a classical technological machining process, in which the machines and tools would have been chosen from paper based data bases such as tooling catalogs, machining equipment books, datasheets, etc. In our case, the software come very handy since it carries electronic libraries of tooling and machines one can choose from.

For such a simple part as the flange used as an example, the necessary machining operations needed to perform are minimal, as it follows:
- o milling: facing;
- o milling: inside contour;
- o milling: outside contour.

This can be done by accessing the menu TOOLPATHS, and then the contour type (OUTSIDE, INSIDE, etc). By doing this, one can create the chain of operations the machine logic will follow.

An example of how a tool-path is chosen is shown in the figure 5.

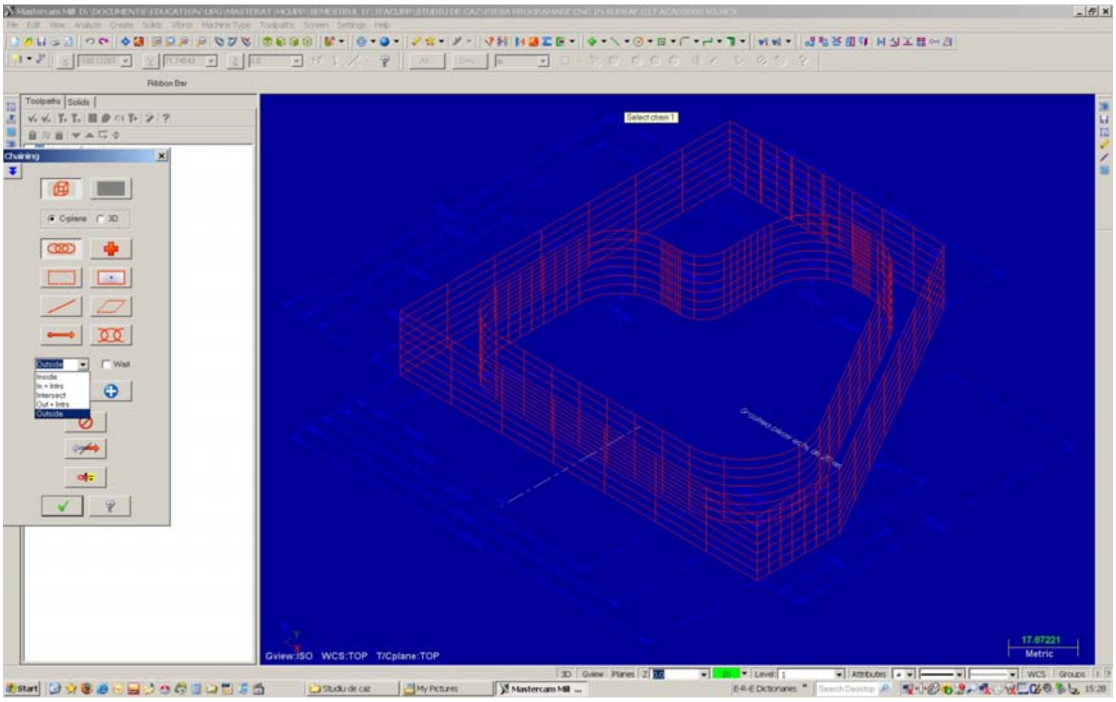

Fig. 5. An example of how a toolpath is chosen

Special built-in subroutines will calculate the tools proximity to the part or machine body, therefore inhibiting (not validating) any possible interfering paths or collisions.

A warning window will usually pop-up letting the user know that a miscalculation occurred. That might be possible due to different reasons, such as:
- o wrong or inappropriate tool dimensions;
- o wrong or inappropriate toolpaths;
- o wrong input provided by the user.

Some information can be usually inferred from the warning message, thus leading the user back to that point where he had provided the inappropriate input or had chosen the setting causing the trouble.

Although the machine is has been set, each machining operation needs to be attributed a tool. They are available, in a similar way to that where the machine are set, from a drop down list. However, a multitude of tools are available and the DEFINE TOOL window allows the user not only to choose the tool shape, but also to adjust different parameters for that particular tool such as:
- o tool angles and diameters;
- o taper and radius values;

o holder type and diameter;
o tool capability (rough, finish or both operations).

An example how these parameters are set is given below, in the figure. 6.

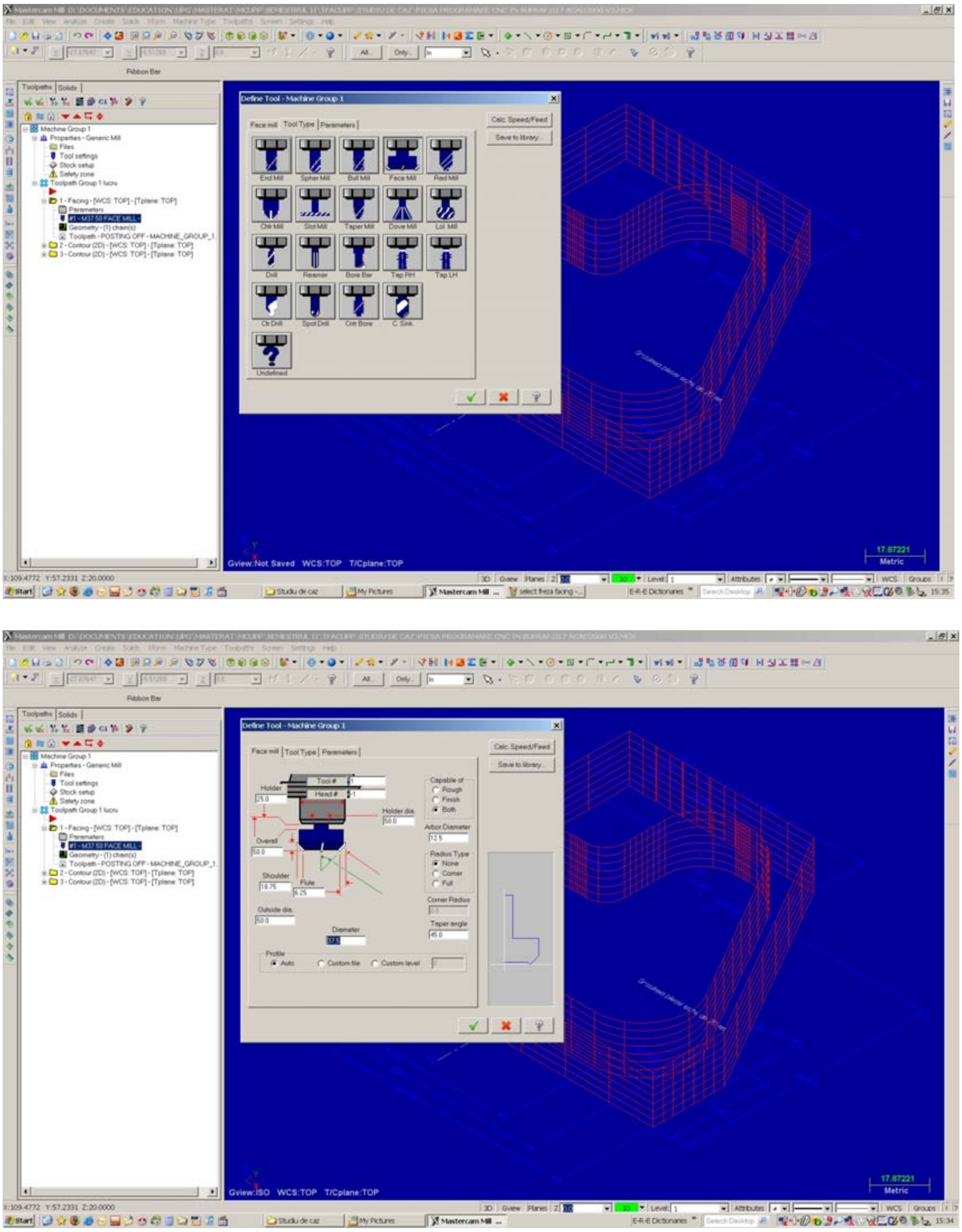

Fig. 6. An example of how the tool parameters are set

The software has built-in simulation capabilities which cab be used to:
o show the part lying on the machine table
o the path followed by the tool cutting edge (showed in motion)
o the entering (tool descending) and exiting (tool ascending) points
o the thickness of the material being machined, etc.,

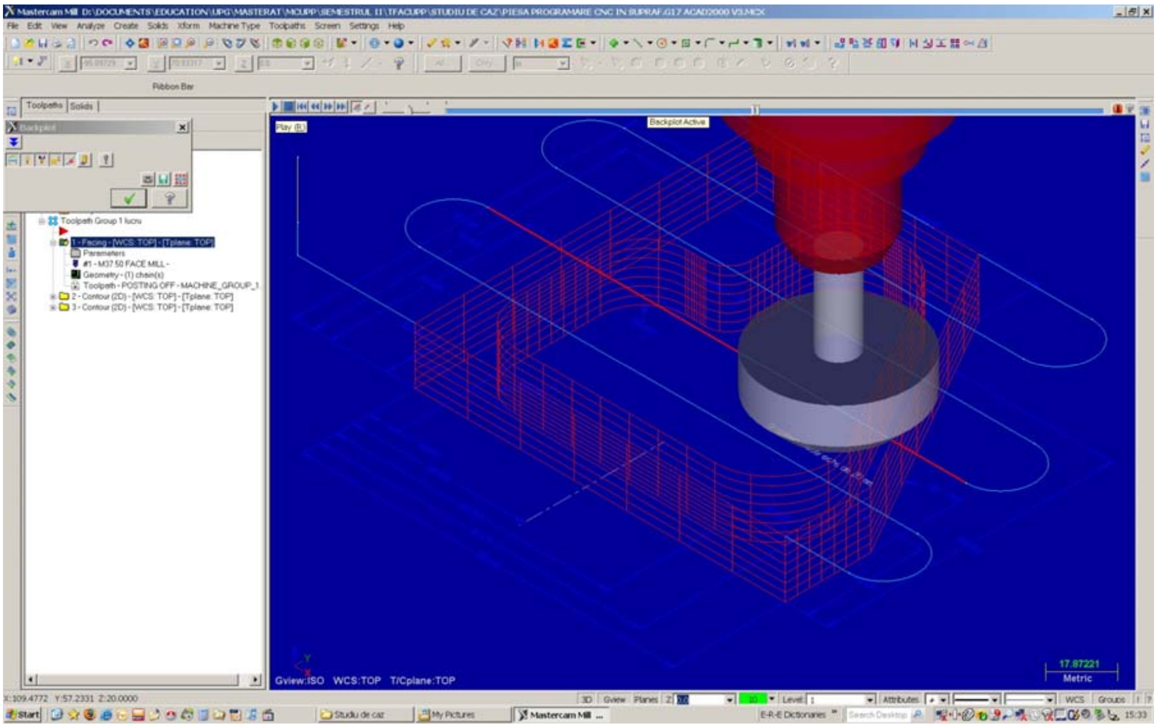

Fig. 7. Manufacturing simulation

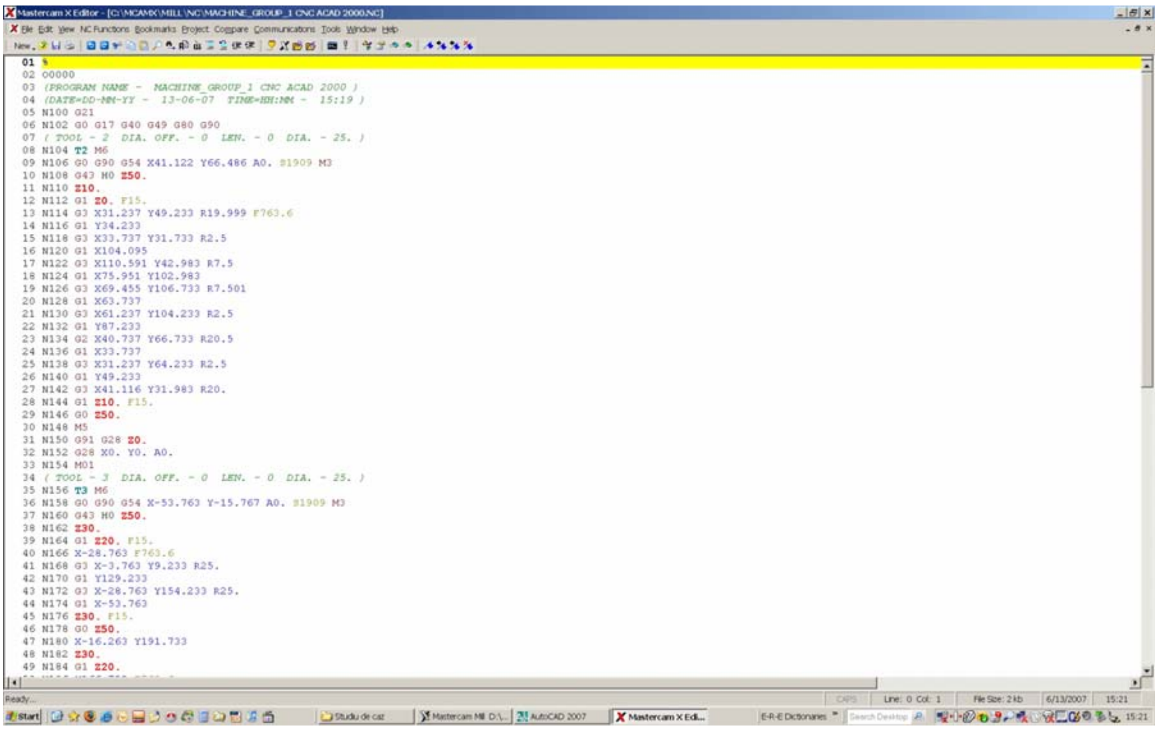

Fig. 8. Screenshot of the CNC file

Similarly, the remainder of the operations is sorted following the same procedure as above. Then, all the operations can be concluded and bonded together in a unique file, which will incorporate the information the CNC machine is going to use during machining process.

Under OPERATIONS MANAGER, by pressing the POST radio button will create the CNC file. A screenshot of this file is given in figure 8, as an example.

Conclusions

The basic rules of drafting still apply, however, and CAD is a very powerful tool that provides accurate drawing. It also allows the user to export the drawing into complementary software and integrated in an ample project.

Thorough understanding and command of classical machining is still very important. Without such knowledge, it is almost impossible to design and accomplish a machining project, even a fairly small one.

]
Through such a simple succession of examples and several images we think that we had proven that CAD and CAM can be easily integrated, and provide powerful and reliable tools for part or assembly design.

References

1. Nae I., Petrescu M.G., Tehnologii în fabricația asistată de calculator, Editura Universității din Ploiești, Ploiești, 2003
2. Norton R.L., Cam Design and Manufacturing Handbook, Industrial Press, 2001
3. * * * MastercamX, Quick Reference Card
4. * * * Curs de programare CNC 432 – MAHO
5. * * * Clasificarea şi simbolizarea parametrilor de aşchiere (ghid) – Norme europene, Editura BREN, Bucureşti, 2002

Proiectare integrată CAD - CAM

Rezumat

În această lucrare este prezentată metodologia de scriere a unui program pentru o maşină-unealtă cu comandă numerică (centru de prelucrare), incluzând toate etapele şi comenzile aferente. Datele inițiale pentru proiectarea în limbaj maşină le-au constituit prescripțiile de calitate corespunzătoare schitei piesei din figura 1 si desenul de execuție executat in AutoCAD 2007 (figura 2).