Solving problems with contact in machining process simulation

L. Masset[*] & J.-F. Debongnie
University of Liège, Manufacturing Laboratory, Chemin des chevreuils 1, 4000 Liège, Belgium
Website : www.ltas19.meca.ulg.ac.be
e-mail : luc.masset@ulg.ac.be; jf.debongnie@ulg.ac.be

F. Berger
Auto Chassis International, 15 Avenue Pierre Piffault, 72086 Le Mans, France
Website : www.auto-chassis.com
e-mail : frederic.berger@auto-chassis.com

ABSTRACT : At the 2001 Esaform conference, we have presented a simulation tool designed to predict form errors of part surfaces obtained by face milling and turning processes. The most relevant aspect of the developed method is its ability to solve industrial cases with huge 3D finite element meshes in a very small time. Until recently, we were only considering pure linear problems. However, the simulation of several machining operations requires the introduction of contact. The present paper describes the numerical method developed to solve such non linear problems.

Key words : finite elements, machining simulation, milling, turning, form error, contact

1 INTRODUCTION

In manufacturing industries, there is a strong demand for machining simulation tools. The available commercial codes do not yet cover the wide panel of industrial needs. Among them is the ability to predict the form error of machined parts. In process planning phase, engineers are faced with technical choices (fixture, tool, cutting conditions, …) but they rarely have the possibility to foresee their influence, so that expensive actual tests are unavoidable. A few research works deal with the form error prediction. Let us cite Schulz and Bimschas [2], Gu *et al.* [3] and more recently Liao [4]. The tool presented in this paper uses a similar framework: the form error is obtained by computing several part deformations thanks to the finite element method. The most interesting capability of our method is to deal with large industrial applications with a small computation time thanks to the superelement technique (see § 3.1). Face milling and turning operations are considered. For most processes, standard boundary conditions (constraints or linear stiffness) can be adopted. However, there are several cases where contact has to be introduced to successfully model the part behavior. This is notably the case when the part geometry prevents from using clamps such as for transverse turning operations on brake disks or when the contacting surfaces are not perfect (rough parts for example). The introduction of contact in the simulation shifts the problem in the non-linear domain that is well-known to be time-consuming. The aim of the present paper is to describe a finite element method that is able to solve efficiently this type of problem.

2 METHOD DESCRIPTION

2.1 Hypotheses

Generally, several phenomena may affect the surface obtained by a given machining process. Some of them are usually unexpected and need to be fixed (chatter, vibrations). Other are peculiar to the process itself such as the bending of thin end-mills or the thermal workpiece expansion in cylinder boring operations. In this research, we assume that the main source of the form error is the part flexibility. The tool and the machine-tool are supposed to be perfectly rigid. This is a reasonable assumption for face milling and turning operations [3]. The workpiece response under the load applied by the tool presents both a static and a dynamic part, especially in milling. As stated by Liao [4], the dynamic part is small when the teeth entering frequency is far enough from the system natural frequencies. Therefore, only the static part of the workpiece response is considered.

2.2 Principle

The defect of a machined surface point depends on its displacement while the tool is cutting through it. The error is simply its normal displacement changed of sign. The figure below illustrates this principle for the straight turning of a cylindrical bar. In this particular case, the part deformation leads to a certain height of uncut material on the surface, the result being a barrel form. It has to be mentioned that in some situations the tool may also cut too much material, especially in milling.

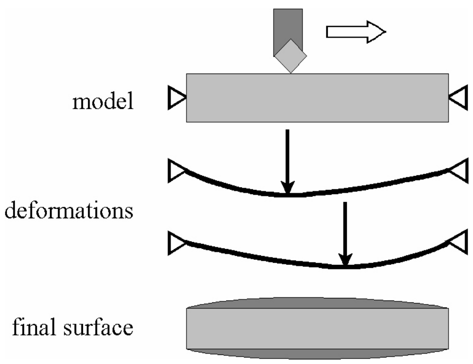

Figure 1: Principle

2.3 Finite element approach

For complex shaped parts, the finite element method is necessary to compute the deformations. Although the part geometry changes as the tool removes material, a single finite element model is used to perform the calculation. Usually we chose a model corresponding to the part geometry after the material removal to obtain the greatest flexibility.
The strategy adopted to obtain the form error has a great influence on both the efficiency and the accuracy of the method. Schulz and Bimschas [2] compute defects at a limited number of surface points to limit the computational cost. They use a complex interpolation scheme to find back the error of the whole surface. The correctness of the solution is doubtful if the number of computation points is small. At contrary, Gu *et. al* [3] use a time description of the machining process which leads to a great number of calculation points. According to us, a time description is useless in the frame of a static computation.
The method that we have adopted is to compute the defect of all the nodes of the machined surface. Doing so, the number of calculation points is naturally set by the mesh and no interpolation scheme is needed. The only interpolation made is the intrinsic finite element interpolation. The defect of one node is obtained as follow:

- first the tool position where the node is cut is determined;
- then the cutting forces are computed and applied on the mesh;
- the part deformation is computed thanks to the finite element method;
- finally the node displacement is picked among the whole displacement fields.

The final surface shape is obtained by applying the above scheme to the n nodes of the machined surface (figure 2).

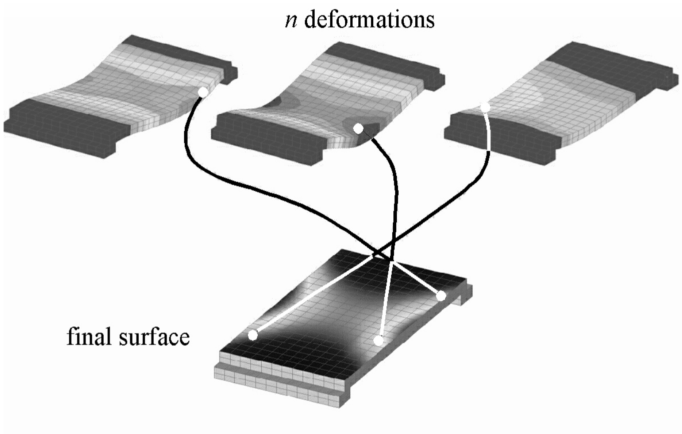

Figure 2: Finite element scheme

More explanations on the cutting force computation and the force application on the mesh can be found in previous paper [1]. Let us just mention that extra features are also implemented such as the back cutting effect in milling and the form error computation (flatness and cylindricity). As illustrated on figure 2, a relevant point of our method is that n deformed structures need to be computed in order to obtain the machined surface. That means that n load cases are applied to the finite element model, n being a large number in the case of industrial applications (up to a few thousands). Solving problems with such a high number of load cases requires special algorithms to achieve a reasonable computation time.

3 FINITE ELEMENTS ANALYSIS

3.1 Superelement method

The system to solve is

$$K\,q = g^{(l)} \;\; l = 1 \cdots n$$

where K is the stiffness matrix, q is a vector containing the degrees of freedom (dof) and $g^{(l)}$ are the n load vectors applied to the structure. Applying directly the n load cases to the whole structure proved to be highly time consuming and even unfeasible for huge models [1]. So a special method called the *superelement method* is adopted to reduce the size of the system to solve. In fact, only a small number of nodes are of interest, namely the machined surface nodes and the nodes where boundary conditions are applied. The major part of the system dof are 'useless' in the frame of form error simulation.

If the n_R retained dof are denoted q_R and the n_C condensed ones q_C, the system can be written in the following way

$$\begin{bmatrix} K_{RR} & K_{RC} \\ K_{CR} & K_{CC} \end{bmatrix} \begin{bmatrix} q_R \\ q_C \end{bmatrix} = \begin{bmatrix} g_R \\ g_C \end{bmatrix}$$

As the loads g_C are all equal to zero, this leads to the reduced system

$$\left[K_{RR} - K_{RC}\,K_{CC}^{-1}\,K_{CR}\right] q_R = g_R \;\Leftrightarrow\; K_{RR}^{*}\,q_R = g_R$$

The superelement method is available in most commercial finite element codes. The reduced system size is highly lowered compared to the original one, the ratio n_C / n_R being the order of 100 to 300. The performance of the superelement is very impressive even for large models. For the 4-cylinder engine block illustrated on figure 3, the time required to create the superelement is about one hour on a standard PC (Athlon with 1.5Gb of physical memory) with the finite element code Samcef.

3.2 Linear problems

Once the reduced system is obtained, we still have to solve it for the n load cases. For linear problems for which boundary conditions consist in constraints and linear stiffnesses, the reduced stiffness matrix can be inverted. We use either the Gauss method or the Cholesky factorization. For this last one the CPU time needed is almost an hour for the case of the 4-cylinder engine block, the matrix size being $[7125 \text{ x } 7125]$.

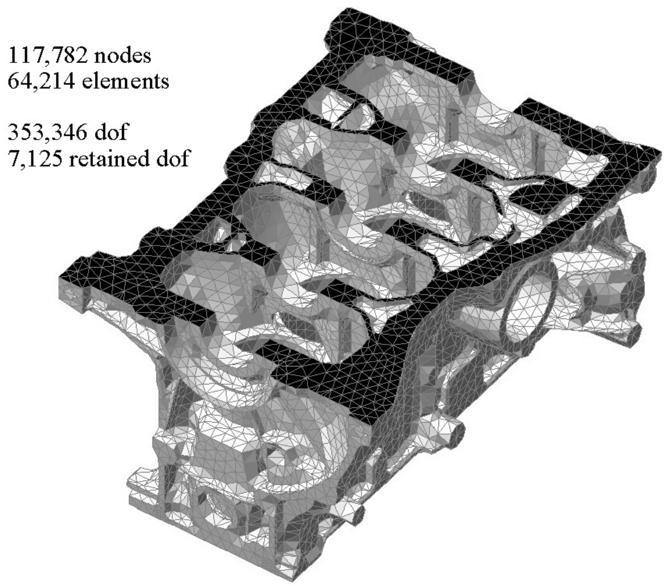

Figure 3: 4-cylinder engine block mesh with machined surface in black (courtesy of Renault)

Once the stiffness matrix K_{RR}^{*} is inverted, the solution is obtained within a few seconds since we solve a explicit system

$$q_R = K_{RR}^{*^{-1}}\,g^{(l)} \;\; l = 1 \cdots n$$

For a given set of boundary conditions, numerous simulations may now be achieved by changing the tool, the tool path or the cutting conditions.

3.3 Contact problems

In several machining operations, it is not possible to ensure a permanent contact between the workpiece and the supports. In practice, this happens when the part geometry prevent from using clamps such as for transverse turning operations on brake disks. More generally, contact is not ensured when the contacting surfaces of workpiece and/or support are not perfect.

Such *one-sided* contacts are modeled by setting a upper value on the normal degrees of freedom located in the contact zones between the part and the supports. The solution is obtained by minimizing the total system energy with upper bounds on some degrees of freedom,

$$\begin{cases} \min\limits_{q_R} \ \frac{1}{2} q_R^{\ T} K_{RR}^{*} q_R - q_R^{\ T} g_R^{(l)} \quad l = 1, n \\ \text{with } q_j \le U_j \quad j = 1, J \end{cases}$$

The minimization problem is well-known: a quadratic function submitted to bounds. The difficulty comes from the great number of load cases and the system size. A scheme similar to the superelement one is adopted to solve the problem. The degrees of freedom of the reduced system are split between the J bounded ones q_b and the other ones q_o to obtain a new system,

$$\begin{cases} \begin{bmatrix} K_{bb} & K_{bo} \\ K_{ob} & K_{oo} \end{bmatrix} \begin{bmatrix} q_b \\ q_o \end{bmatrix} = \begin{bmatrix} 0 \\ g_o \end{bmatrix} \\ q_o = K_{oo}^{-1} \left[g_o - K_{ob} \, q_b \right] \qquad (a) \\ \left[K_{bb} - K_{bo} K_{oo}^{-1} K_{ob} \right] q_b = - K_{bo} K_{oo}^{-1} g_o \qquad (b) \\ \Leftrightarrow K_{bb}^{*} q_b = g_b^{*} \end{cases}$$

so that the minimization problem is now limited to the J bounded degrees of freedom,

$$\begin{cases} \min\limits_{q_b} \ \frac{1}{2} q_b^{\ T} K_{bb}^{*} q_b - q_b^{\ T} g_b^{(l)^*} \quad l = 1, n \\ \text{with } q_b \le U \end{cases}$$

Practically, for a given set of boundary conditions, the first step consists in building part of the reduced system (b) by calculating $K_{oo}^{-1}, K_{bo} K_{oo}^{-1}$ and K_{bb}^{*}. This is the hardest part since it requires a matrix inversion. Then each simulation result is obtained by calculating g_b^{*}, solving the minimization problem for the n load cases and finding back the reduced system dof q_o thanks to equation (a). This takes only a few seconds since the number of contact dof is usually limited (the order of 50 at maximum).

4 APPLICATION

The case of the transverse turning operation of a brake disk is illustrated on figure 4. The disk is held in a three-jaws chuck and lies on three support located near the jaws. The machined surface presents a three-lobed error. The obtained axial run-out at 75 mm equals 14.6µm while the measured one equals 14.4µm.

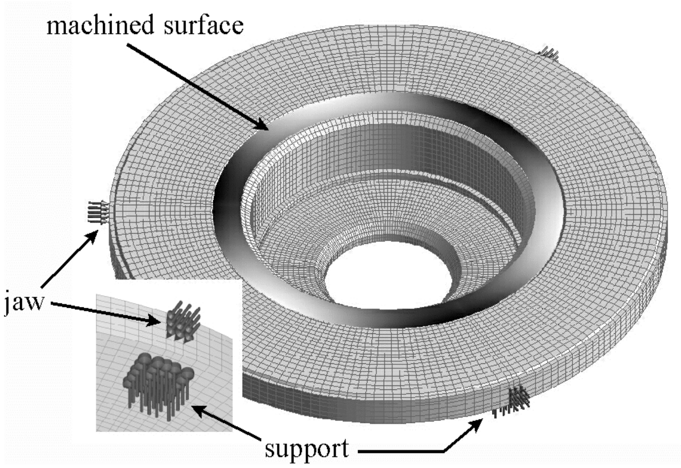

Figure 4: brake disk (courtesy of ACI)

5 CONCLUSION

Thanks to the introduction of contact, a certain class of applications can be treated successfully. Even for such non linear problems, the simulation tool remains very efficient.

Acknowledgments

The authors gratefully acknowledge the technical support of Walter Belluco at Renault and Auto Chassis International[1].

References

[1] Masset L., 2001, *Simulation of Face Milling and Turning with the Finite Element Method*, International Journal of Forming Processes, Vol. 4, pp. 481-498

[2] Schulz, H., Bimschas, K., 1993, *Optimization of precision Machining by Simulation of the Cutting Process*, Annals of the CIRP, Vol. 42/1, pp. 55-58

[3] Gu, F., Melkote, S. N., Kapoor, S. G., DeVor, R. E., 1997, *A Model for the Prediction of Surface Flatness in Face Milling*, ASME Journal of Manufacturing Science and Engineering, Vol. 119, pp. 476-484

[4] Liao G. Y., 2000, *Simulation of Engine Deck Face Distorsion During Fixturing and Milling Operations*, SAE Technical Paper, 01-1287

[1] ACI designs, develops, validates and manufactures front and rear-chassis and suspension systems.