Machining processes simulation: specific finite element aspects

L. Masset[1] , J.F. Debongnie

Manufacturing Laboratory, ASMA Department, Liège University, 1 chemin des chevreuils, B-4000 Liège, Belgium

Abstract

The paper presents a simulation tool designed to predict form errors of part surfaces obtained by face milling and turning processes. For these operations, the form error is often due to the flexibility of the workpiece and its supports. The finite element method is adopted to model the part geometry and to compute its deformations. Numerous load cases, up to a few thousands, are required to obtain the form error. Classical resolution methods are inefficient in this context. The *superelement method* is used to reduce the problem to a fraction of its initial size. Resolution requirements (CPU, memory and disk space) are drastically reduced so that large industrial applications can be solved.

1 Introduction

1.1 Machining simulation

Simulation tools proved to be essential during the design phases of manufacturing operations. Costs and time may be significantly reduced by avoiding try and error steps. In the field of machining, some commercial codes provide solutions for specific problems such as NC programming (Catia, NCSimul), dynamic aspects (CutPro), cycles optimization (Vericut), … Few codes are designed to predict the form error of machined surfaces although the form error prediction seems essential in process planning phase.

1.2 Form error prediction

In a general way, the part form error is due to the deformations of the whole kinematic system (machine-tool, tool, part) during the cutting operation. Most research works consider a rigid part although the part flexibility is often the main cause of form error. Among the few works taking the part flexibility into account, let us cite the following ones. Kops *et al.* [1] study the cylindricity error of bars. They use an analytical model which takes into account the part and the fixation system flexibility. Shulz and Bimschas [2], Gu *et al.* [3] use the finite element method to predict part form error in face milling and turning operations. They consider the static tool-workpiece-fixation system deformations. Liao [4] studies both static and dynamic workpiece responses.

[1] Corresponding author. E-mail: luc.masset@ulg.ac.be

Although this is not clearly stated by the authors, the main problem is the computational cost. In fact, the workpiece geometry has to be modeled with large 3D finite element meshes. The form error simulation requires a large number of load cases. For industrial applications, even a single simulation takes a long time. But the purpose of a machining simulation tool is to test several process settings (tools, fixation designs, cutting conditions, …). Therefore a special care must be brought to achieve the smallest possible simulation time. This paper focuses on the numerical methods adopted to meet this requirement.

2 Model of form error computation

2.1 Hypotheses

In the present work, the tool and machine-tool are supposed to be rigid. The form error comes only from the elastic part deformations due to the loads applied by the tool and the clamping devices. The finite element analyses are performed under linear static hypotheses. In milling, Liao [4] shows that, when the tooth entering frequency is sufficiently far from the natural frequencies of the workpiece-clamping system, the dynamic response contribution remains small compared to the static one.

Residual stresses and thermal effects are not considered in the present research. It has to be mentioned that, in cylinder boring operations, Kakade and Chow [5] demonstrate that thermal deformations greatly contribute to form error. However, for face milling and turning operations and for standard materials (steels, cast irons, aluminum, …), thermal effects are usually much smaller. Schulz and Glockner [6] measure a part temperature increase of only 20°C during the milling operation of a cast iron workpiece.

2.2 Principle

During the cutting process, the part is deforming under the loads applied by the tool and the fixation devices. The defect of any point of the machined surface is produced when the tool is cutting it. At that moment, if we denote u the point displacement normal to the machined surface, the point defect simply equals $-u$. The displacement of a surface point can be positive (towards the tool) or negative, the height of removed material being respectively greater and lower than the desired one. The form error is due to the defect variation along the whole machined surface.

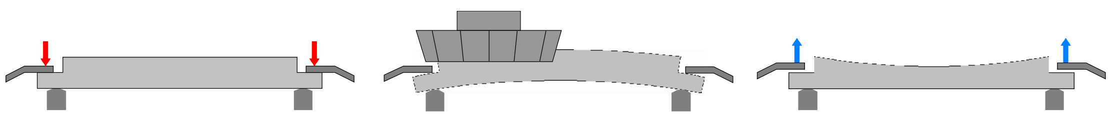

Figure 1: form error obtained after releasing the clamps

Figure 1 illustrates this principle. Before machining, the part is deformed by the fixation devices. The surface produced by the tool is plane. When the clamps are released, the surface is no longer plane due to the elastic spring back of the workpiece. Deformation and surface error go in opposite directions. In this work, this principle is adapted to a finite element method approach. The defect is computed at the n nodes of the mesh surface (the machined surface). The defect of a given node is obtained by the following way:

– determination of the loads acting on the part for this particular tool position,
– computation of the part deformation for this load case.

So, n load cases are applied on the finite element model to obtain the n nodal defects and finally the machined surface error (figure 2). For large models, the load case number can reach a few thousands. Classical resolution methods provided by finite elements codes are not well suited to solve such problems.

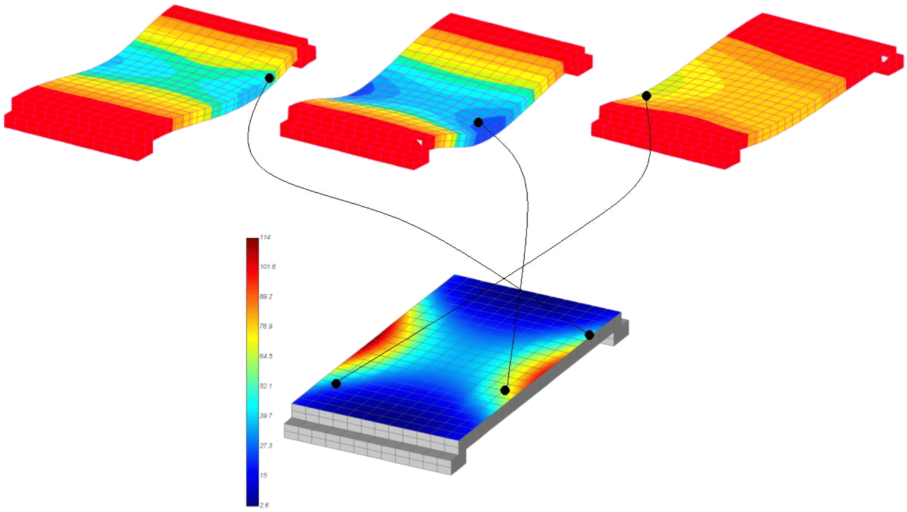

Figure 2: model principle - deformed structures and obtained machined surface

2.3 Model implementation

The part geometry is modeled with a volume element mesh. As a given amount of material is removed by the tool during the process, the geometry is constantly changing. Schulz and Bimschas [2] consider the exact geometry at each tool position using a CAD/automatic mesher procedure. We think this solution is far too complex and too expensive for an efficient simulation tool. So, we use a single finite element model corresponding to the workpiece geometry after the cutting operation. The resulting error is usually small since we mostly consider finishing passes where the depth of cut is small.

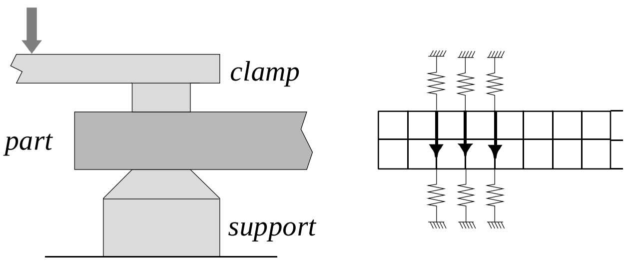

Figure 3: fixation model (clamp-support system)

The fixation device flexibilities are modeled with linear stiffness elements. The stiffness values can be obtained from experiments or by numeric simulations. The most usual fixation system is the clamp/support (figure 3). The loads applied by the fixation devices are taken into account by applying nodal forces or pressure.

In the present method, the following steps are carried out to obtain the form error of a machined surface for a given workpiece and for a given process setting (tool, cutting conditions, fixation design).

1. The n tool positions are computed.
2. For each tool position, the cutting forces are computed and applied on the surface mesh. The loads applied by the fixation devices do not vary with the tool position.
3. The n deformed structures are computed with a finite element solver (Samcef [7]).
4. The surface error is obtained by picking the displacements of the n cut nodes among the results.
5. Finally, the effective machined surface form error (flatness, cylindricity) is computed thanks to a specialized algorithm developed by Debongnie and Masset [8].

More details on the developed model can be found in previous works [9, 10].

3 Finite element analysis

3.1 Direct method

The direct method consists in solving directly the system with n load cases

$$K\,q = g \tag{1}$$

where K is the stiffness matrix, q are the degrees of freedom and g is the load vector. Three solvers are available in Samcef for linear problems: a classic frontal solver, a multi-frontal solver and an iterative solver. The iterative solver is obviously not adapted to multi load case problems. The performances of the two other ones are tested on a standard computer (Athlon 1.2 GHz with 1.5 Gb of physical memory) running Windows NT and Samcef version 8.1. Table 1 shows the direct method requirements with the frontal solver on two medium sized finite element models. It has to be mentioned that larger models (see figures 7 to 9) cannot be solved with the direct method because the required memory is too high.

	camshaft cover	suspension arm
time (s) [h m s]	1773 [29 m 33 s]	3957 [1 h 5 m 57 s]
memory (Mb)	402	530
disk space (Mb)	5717	7983

Table 1 : direct method requirements

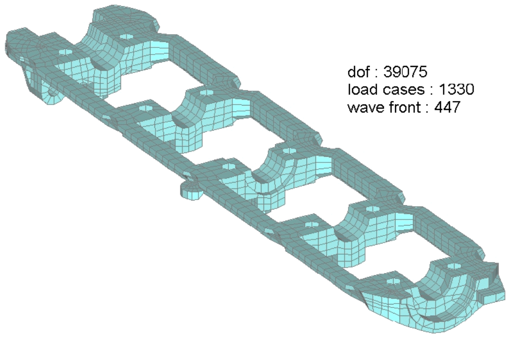

Figure 4: camshaft cover

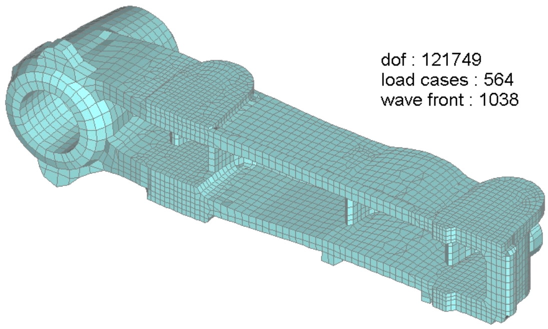

Figure 5: suspension arm

As expected, time and storage values are high. The finite element code is spending a lot of time in disk access. The file sizes depend on the system size and almost linearly on the load case number. For bigger models, storage size quickly exceeds the capacity of standard computers. Amazingly, the memory seems to increase a lot with the load case number. Figure 6 shows the memory required for the three computation steps of the frontal method as a function of the load case number. For scaling and condensation steps, the behavior looks quite normal: memory increases slowly with the load case number.

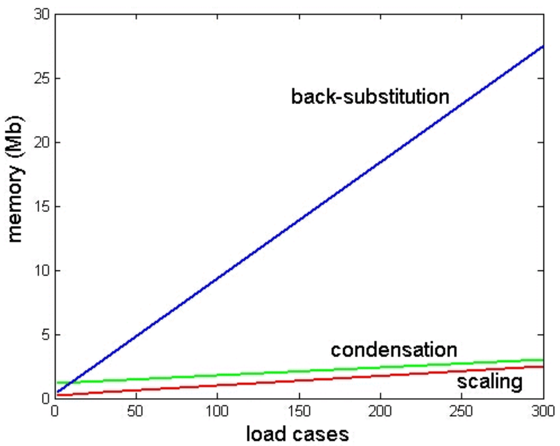

Figure 6: memory required by the frontal solver

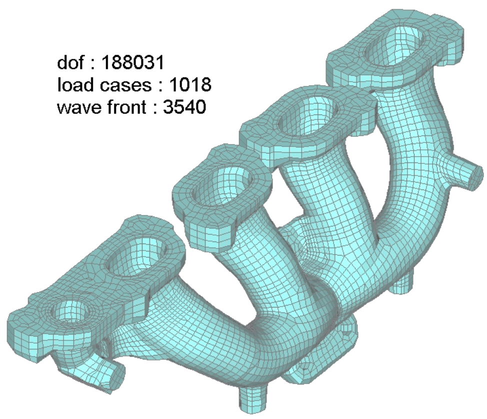

Figure 7: exhaust manifold

For the back-substitution phase, the slope is much higher. The reason is that the whole second member of equation 1 is stored in memory for this phase although there is no real reason for that, except a simplified programming scheme. For the classical applications of the finite element method, the load case number seldom exceeds a hundred. So, storing the whole second member is not a problem. For huge models with thousands of load cases, the required memory exceeds the limits of any standard machine. The multi-frontal solver exhibits the same problem. Obviously, Samcef code has not been designed to solve high load case number problems. The same statement probably applies to most commercial finite elements codes.

3.2 *Superelement method*

The problem to solve presents some specific characteristics:

– only the displacements of the machined surface nodes are necessary to obtain the surface error,
– loads are only applied on these nodes and a few additional nodes where the clamping forces act.

Most of the system degrees of freedom are not used in the frame of surface error prediction. The *superelement method* offers a convenient way to reduce the system size by condensing the *useless* degrees of freedom. If the n_R retained degrees of freedom are denoted q_R and the n_C condensed ones q_C, equation 1 can be written

$$\begin{bmatrix} K_{RR} & K_{RC} \\ K_{CR} & K_{CC} \end{bmatrix} \begin{bmatrix} q_R \\ q_C \end{bmatrix} = \begin{bmatrix} g_R \\ 0 \end{bmatrix} \quad \textbf{(2)}$$

since there are no loads applied on condensed degrees of freedom q_C. This leads to the following expression of the condensed degrees of freedom q_C

$$q_C = - K_{CC}^{-1} K_{CR} \, q_R \quad \textbf{(3)}$$

Finally the reduced system can be written as follow

$$\left[K_{RR} - K_{RC} K_{CC}^{-1} K_{CR} \right] q_R = g_R \quad \Leftrightarrow \quad K_{RR}^{*} \, q_R = g_R \quad \textbf{(4)}$$

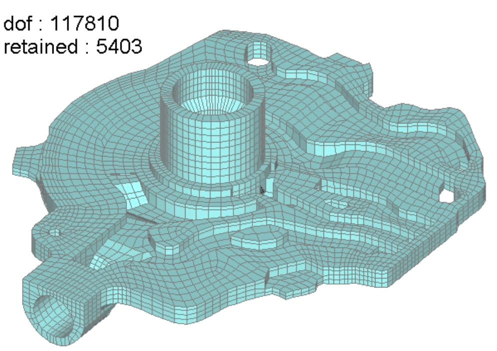

Figure 8: gear box cover

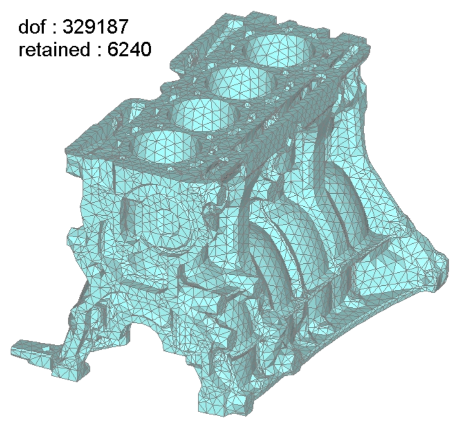

Figure 9: 4-cylinder block (D4 model)

The first step of the superelement method is the superelement creation, *i. e.* the creation of reduced system. The machined surface nodes and all the nodes where boundary conditions are specified (or could be specified in a future simulation) are retained. The characteristics of this phase with the multi-frontal solver are given by table 2. Even for very large models (4-cylinder blocks), the superelement creation works perfectly well.

	camshaft cover	suspension arm	gear box cover	exhaust manifold	4-cylinder block
total dof	39075	121749	117810	188031	329187
retained dof	2511	1947	5403	3141	6240
time (s) [h m s]	29	85 [1 m 25 s]	221 [3 m 41 s]	138 [2 m 18 s]	2,410 [40 m 10 s]
memory* (Mb)	151.6	309.4	755.7	521.5	3209 (987)
disk space (Mb)	138	364	567	604	1028

Table 2 : superelement creation requirements - * optimum value for the multi-frontal solver (value in brackets indicates the actual memory used, the optimal one being too high)

Once the superelement is created, the next step is the superelement utilization, *i. e.* applying the boundary conditions and solving the reduced system (4). Here, we find again the Samcef drawbacks of solving a multi load case problem but lowered. The characteristics of superelement utilization step with the multi-frontal solver are given at table 3. The storage space and computation time are acceptable. The memory required depends upon the load case number and the superelement size. In the frame of a superelement utilization in Samcef, the superelement is considered like any other element. Its stiffness matrix has to be stored completely in memory.

	camshaft cover	suspension arm	gear box cover	exhaust manifold	4-cylinder block
load cases	1330	564	1724	1018	2027
time (s) [h m s]	370 [6 m 10 s]	97 [1 m 37 s]	2287 [38 m 7 s]	450 [7 m 30 s]	3617 [1 h 0 m 17 s]
memory (Mb)	83	40.7	254.9	88.8	502.6
disk space (Mb)	368.8	134.7	1115	382	1508

Table 3 : superelement utilization requirements

When the specs of the direct and superelement methods are compared, one can clearly see the advantages of the superelement method. Another advantage of this method is that the boundary conditions can be modified without creating a new superelement. Several fixations designs can be tested with only the cost of the superelement utilization.

For a given boundary condition set, the improvement can still be greater. The size of the stiffness matrix K_{RR}^* is limited so the computational cost of its inversion is not too high. This way, we obtain an explicit form of the reduced system (4), *i. e.*

$$q_R = K_{RR}^{*\,-1} g_R \tag{5}$$

3.3 Stiffness matrix inversion

The stiffness matrix K_{RR}^* obtained by the condensation scheme (4) is a full matrix. So, there is no need to use complex inversion algorithm such as the *skyline method*. In the present work, we adopt the Cholesky factorization method. The algorithm comes from the *Lapack subroutine library* [11]. The inversion time is a cubic function of the matrix size. Two versions of the algorithm are available: symmetric and full storage, the memory being equal to $n(n+1)/2$ and n^2 respectively. The full storage version is adopted since it runs faster (the maximum matrix size to reach the machine limit of 1.5 Gb is about 14,000). Before inverting, the stiffness matrix is scaled to avoid numerical problems. The inversion time is plot on figure 10 (on logarithmic axis) as a function of the stiffness matrix size n_R.

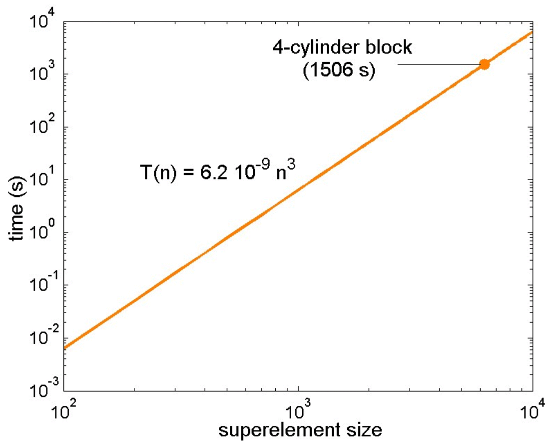

Figure 10: time for stiffness matrix inversion

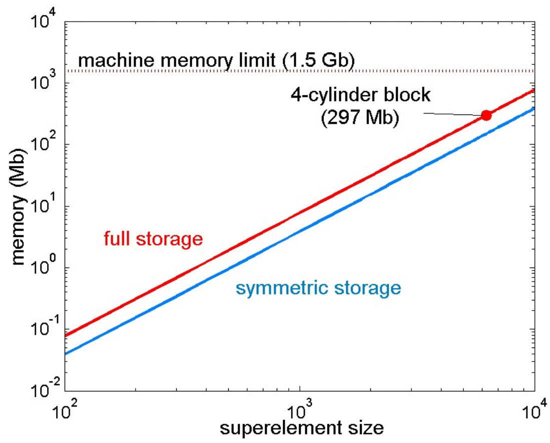

Figure 11: memory for stiffness matrix inversion

Once the stiffness matrix K_{RR}^* is inverted, the time to obtain one simulation result is very small (the order of a second). Actually, the solution requires only a matrix multiplication, which is very cheap if we take into account the numerous zeros of the load vector g_R.

4 Applications

4.1 Camshaft cover

The camshaft cover is made of aluminum A-S9U3Y40. Figure 12 shows the finite element model and the fixture design. The tool is a 100-mm mill with four carbide inserts. The aim of the simulation is to find the trajectory leading to the smallest form error among the two possible centered trajectories. The obtained results are illustrated on figure 13.

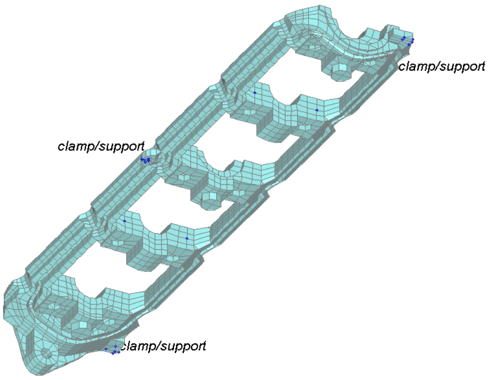

Figure 12: camshaft cover fixations

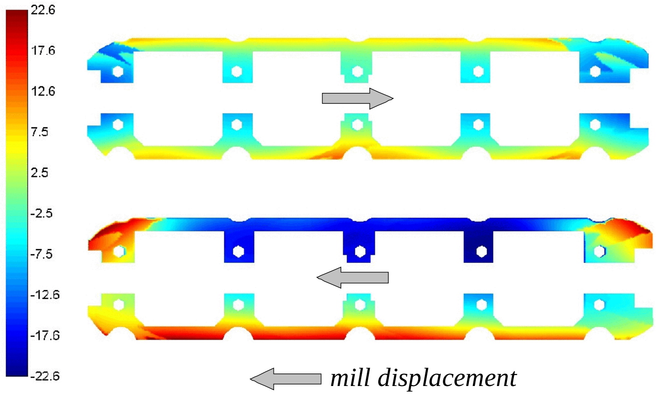

Figure 13: flatness obtained (respectively 27.4 and 47.2 µm)

4.2 4-cylinder engine block

The 4-cylinder block is made of cast iron GS-53. The top and bottom surfaces are face milled. Figure 14 shows a classical surface obtained by simulation. The surface topology depends on local stiffness and on loads applied by the tool. In some surface regions (in red on figure 15), it can be seen that several inserts are engaged, while in other regions (in blue), only one insert is cutting.

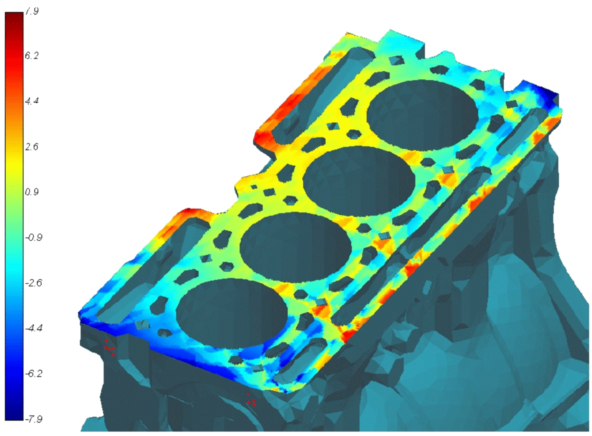

Figure 14: flatness of the top surface

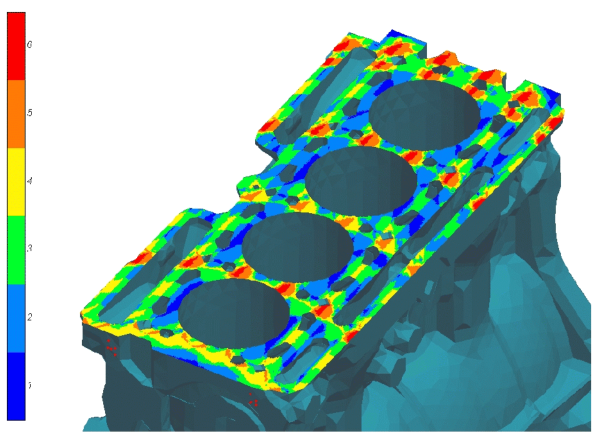

Figure 15: inserts engaged (mill going upwards)

5 Conclusion

In process planning phase, the ability to predict form errors of machined surface is mandatory to limit the experimental tests. Several process configurations need to be simulated in order to find the best possible ones. Therefore, simulation tools must be as fast as possible. When accounting the workpiece flexibility, a high load case number is unavoidable and the computation cost is potentially very high. The developed tool, based on the superelement method, exhibits a reduced computation time that makes the simulation of huge industrial applications feasible.

Acknowledgments

The authors gratefully acknowledge the financial and technical support of Renault.

References

[1] Kops L., Gould M., Mizrach M., Improved Analysis of the Workpiece Accuracy in Turning, Based on the Emerging Diameter, *ASME Journal of Engineering For Industry* **115** (1993) 253-257

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

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

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

[5] Kakade N. N., Chow J. G., Finite Elements Analysis of Engine Bore Distorsions During Boring Operation, *ASME Journal of Engineering For Industry* **115** (1993) 379-384

[6] Schulz H., Glockner K., 1994, Simulation of the deflection of workpieces due to the cutting forces, *Final report (study for Renault)*

[7] Samcef finite element code, *Système d'Analyse des Milieux Continus par la Méthode des Eléments Finis*, www.samcef.com

[8] Debongnie J.-F., Masset L., Controlling Form Errors From 3D Measures, *Proceedings of the 1999 ASME Design Engineering Technical Conferences*, Las Vegas Nevada, 12-15 september 1999

[9] Masset L., Debongnie J.-F., Foreau S., Dumont T., A Model for the Prediction of Form Errors in Face Milling and Turning, *Proceedings of the 1999 ASME Design Engineering Technical Conferences*, Las Vegas Nevada, 12-15 september 1999

[10] Masset L., Debongnie J.-F., Beckers P., Face milling and turning simulation with the finite element method, *Proceedings of the fourth international ESAFORM Conference on Metal Forming*, Liège, 23-25 april 2001, 627-630

[11] Lapack subroutine library (Linear Algebra PACKage), www.netlib.org/lapack/