Journées Scientifiques et Techniques en Mécanique et Matériaux
JSTMM 2008 – Usinage : Expérimentation et simulation
Monastir- Tunisie

Simulation of machining processes at global scale using SAMCEF for Machining

[1]L. Masset, [2]J.-F. Debongnie

Laboratoire de Méthodes de Fabrication, Département Aérospatiale et Mécanique, Bat. B52/3, 1
chemin des chevreuils, 4000 Liège (Sart-Tilman), Belgium
[1]luc.masset@ulg.ac.be, [2]jf.debongnie@ulg.ac.be

Abstract: SAMCEF for Machining is a professional solution dedicated to process engineers. This Computer Aided Engineering tool focuses on the simulation of the machined part behavior at a global scale. Several aspects of machining are taken into account such as form errors, chatter or thermal aspects. In this paper, we present a few industrial applications that demonstrate the benefits of simulation in production planning: reduction of setup delays, limitation of experimental validations, cost reduction.

Key words: machining / advanced modeling / finite element method / tool deflection analysis

Introduction

Production planning of automotive parts is known to be a very hard challenge. On one hand, the tolerances imposed on most parts (cylinder heads, gear box casings…) are very small because of the increase in quality constraints (increased product life, augmented delays for maintenance…). On the other hand, production rates should increase and unit cost should decrease in the mean time. In addition, new constraints appear such as environmental concerns, reduction of cutting fluids for example.

In this frame, process engineers face more and more issues meanwhile the delays for production planning are to be reduced. Simulation tools offer an efficient way to successfully achieve their objectives. Today, the most widely used software are the CAD/CAM suites such as CATIA, PRO-E or NC-Simul. These tools are well adapted to a large variety of tasks (detection of tool collision, writing of NC programs…). However, the mechanical system is always modeled as an assembly of rigid bodies (tool, spindle, machine components, part…) so that technical issues related to flexibility cannot be taken into account in these environments.

Since 1995, research works have been started at the University of Liège with the partnership of Renault. The purpose was to study the behavior of a machined part during various machining processes (turning, reaming and milling). The method is based on the Finite Element Method (Masset et al., 1999) and on a global approach of the system. A prototype software was developed at the University and used at the Powertrain department of RENAULT at Rueil-Malmaison. At the end of 2005, SAMTECH invested in this field (Samtech, 2008). A commercial tool, named SAMCEF for Machining (S4M), was launched at the end of 2007. It is designed to address several aspects of machining such as form errors, chatter prediction or thermal issues.

1. Geometric Errors

1.1 Principle

During any machining process, the mechanical system - machine-tool, tool, part and clamping devices - is submitted to several stresses, usually generated by the system itself. These stresses deform the system during the cutting process so that the obtained machined surface is always imperfect.

This fact is illustrated on the figure 1. A cylindrical bar is turned between centres (a). This simple case is modelled with a beam simply supported at both sides and submitted to a moving force (b). The deformation at a single instant of the turning process can be obtained with an analytical approach (c). At the end of the operation, the bar presents a defect that is minimum at the ends and maximum in the middle (d).

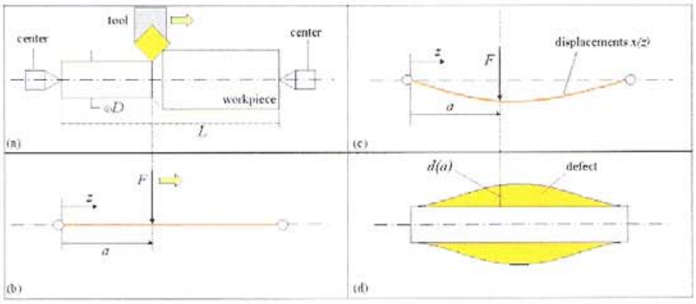

Figure 1: turning of a cylindrical bar

1.2 FEM approach

For industrial parts, the FEM method is used to compute the deformations of the workpiece during a machining operation, but the principle remains the same as for the analytical approach (Masset, 2001). For each node of the machined surface, we have to:

- compute the tool position and the corresponding cutting forces,
- perform the FEA analysis to obtain the whole displacement field,
- extract the displacement of the node and take its opposite to obtain the node defect.

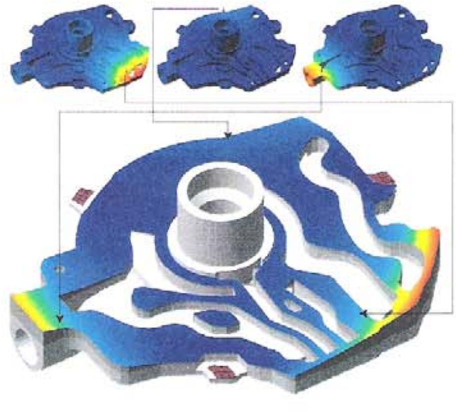

Figure 2: FEM approach

Figure 2 illustrates the method. A deformed structure is computed for several tool positions (three of them are drawn) and the final form of the machined surface is obtained by the superposition of the defects at nodes.

1.3 Numerical aspects

The particularity of the FEA is that there are a great number of load cases, up to several thousands for industrial applications. Standard FEM algorithms are not well adapted in this situation, mainly because of a memory issue (Masset, 2004). The first step of the adopted scheme is to use the superelement method. It consists in condensing the great majority of the degrees of freedom (dof) of the system while keeping only the interesting ones, i.e. the dof corresponding to the machined surfaces and to the clamping zones (respectively in blue and orange on figure 3).

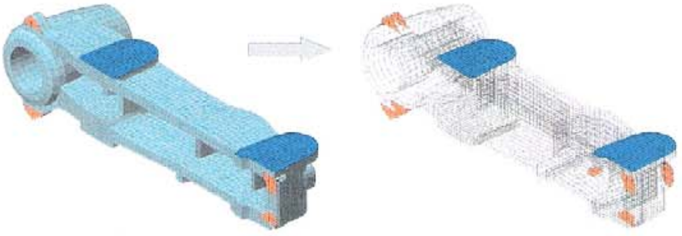

Figure 3: retained zones for the superelement method

The superelement method is implemented in the linear solver of SAMCEF. The gain is that the system to solve is much smaller than the original one (the order of 100 to 1000). Typical times for a superelement creation are given at the table below[1].

4-cylinder crankcase, 330000 dof, 839 retained nodes
515 sec.

gear box case, 858000 dof, 2664 retained nodes
2605 sec.

After the superelement is created, we have a reduced system characterized by its stiffness matrix K_R and submitted to m load cases,

$$K_R \; q_R = g_R^{(i)} \quad (i = 1, m) \tag{1}$$

The second step is to invert the stiffness matrix in order to obtain an explicit system,

$$q_R = K_R^{-1} \, g_R^{(i)} = S_R \, g_R^{(i)} \quad (i = 1, m) \tag{2}$$

where S_R is the flexibility matrix. Before inverting, we impose a set of boundary conditions on the system (constraint, linear stiffness…) corresponding to the actual clamping (support, clamps, chuck jaws…). Typical times for the matrix inversion are given at the table below.

---

[1] On a Pentium IV 3GHz with 2Gb of RAM running SAMCEF V10.1-02

	4-cylinder crankcase	gear box case
retained dof	2577	7539
time (sec.)	31	808

The final step is to solve the system (2). Each load case is composed by the cutting forces (depending on the tool position) and the clamping forces. The cutting forces are computed thanks to various analytical models (Kienzle, Gu) in function of the tool trajectory, the cutting conditions and the material properties. The solution of system (2) is obtained in a few seconds since the load cases are full of zeros and only one dof has to be computed for each load case.

Thanks to the adopted scheme, we are able to solve industrial applications efficiently. Several situations (position of clamping devices, tool trajectories, cutting conditions…) may be studied and compared in a very short time and this renders the method very attractive for an industrial use.

1.4 Tool deformation

In addition to the mechanical deformations of the part and the clamping system, we also compute the deformation of the tool. In turning, for ISO tool holders, a simple beam model is used to compute the flexibility at the tip of the tool using the tool holder section and its length outside the lathe turret. For special turning tools, we can import the flexibility computed with external FEM software.

In face milling, the behavior of the cutter is more complex to model. Moreover, it depends on the spindle flexibility.

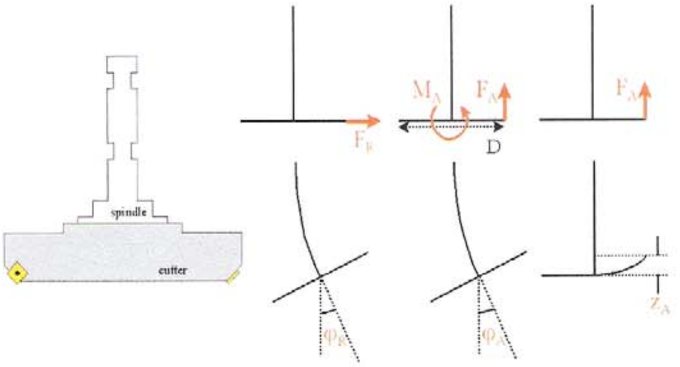

Figure 4: flexibility model of the spindle/cutter

The displacement is characterized by three deformation modes of the spindle/cutter: a flexion of the spindle due to a radial force FR, a flexion of the spindle due to the moment developed by an axial force FA and the proper deformation of the cutter body zA. With this simple model, we can compute the axial displacement at any insert for any position of the milling cutter since we know the forces applied on it.

This effect is important for cutters with large diameter - a small rotation of the spindle axis induces a huge displacement at the insert level - and for roughing operations where the cutting forces are huge (several tons).

1.5 Mill axis tilt and back-cutting

In milling, the cutter axis is often tilted to prevent back-cutting, i.e. the inserts cutting the machined surface a second time (figure 5). Back-cutting is usually avoided because it produces a rapid wear of the cutting inserts due to friction.

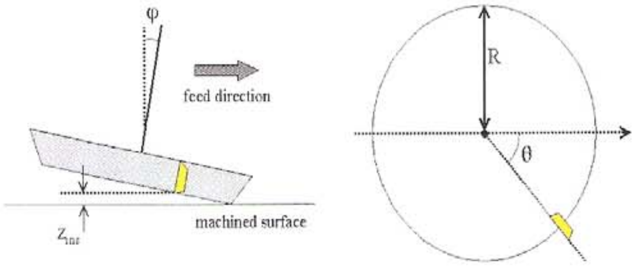

Figure 5: cutter axis tilt

The direct effect of axis tilt is to generate a channel-shaped surface (see figure 6). The height of an insert located at an angle θ is given by the following equation:

$$z_{ins}(\theta) = R\,(1-\cos\theta)\sin\varphi \tag{3}$$

where φ is the axis tilt angle and R is the cutter radius. From equation (3) we can easily obtain the surface produced by a given axis tilt.

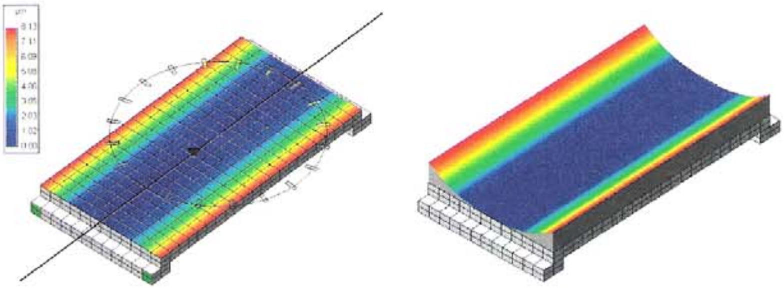

Figure 6: axis tilt effect; part wide 120mm; cutter diameter 160mm; axis tilt angle 0.3 mm/m

In addition, we also take into account the fact that the inserts of the mill may cut the machined surface when they pass for the second time at a given point. The adopted model is based on the same principle than for the form error prediction except that we look what happen at the back insert level. For a node that has already been cut by the front inserts of the mill, we just have to compare its height with the height of the mill at the node position (Masset, 2004). For back cutting, the height of a node depends on the deformation of the part at this particular instant (cutting forces) plus the defect that was produced previously on the node.

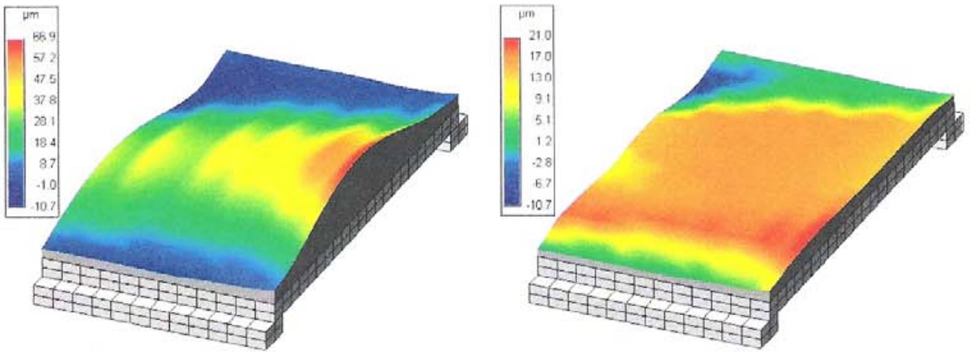

Figure 7: machined surface obtained without back-cutting (left) and with back-cutting (right)

To illustrate the principle, the figure above shows the machined surface that we would have obtain without taking into account the back-cutting effect and the surface that we obtain when we consider the back-cutting effect. In this example, the axis tilt angle is not sufficient enough (0.1 mm/m) so the back inserts of the milling cutter removes a given amount of material on the machined surface. Consequently, the surface is flattened in the middle. Axis tilt angles can be set properly with this type of simulation: just avoiding back-cutting without setting a too big angle.

2. Thermo-mechanical approach

2.1 Principle

Cutting always produces a huge amount of heat. The great part of this heat flows within the chips, another part in the tool and the rest goes in the machined part, causing dilation of the workpiece. This issue is especially crucial for aluminum parts and/or for processes for which the heat doesn't flow out easily (drilling, reaming and boring for example).

After the successive operations, when the part temperature decreases and returns to the ambient, the machined surfaces are distorted. For example, in drilling, holes are not correctly placed on the part.

2.2 FEM approach

The effect of cutting heat on the machined part is modeled in two steps: a thermal analysis followed by a mechanical analysis. In the thermal analysis, we compute an approximation of the heat produced by the cutting as a fraction α of the cutting power

$$H = \alpha P_c \tag{3}$$

with the cutting power P_c that may be obtained with the data supplied by the cutting tool manufacturer. The fraction α is more difficult to obtain. It depends on the type of process. For example, it should be less in face milling than in drilling because the chips are more easily evacuated. Experimental tests (figure 8) should be performed in order to get a good approximation of the data to use for simulations (Segurajauregui, 2007).

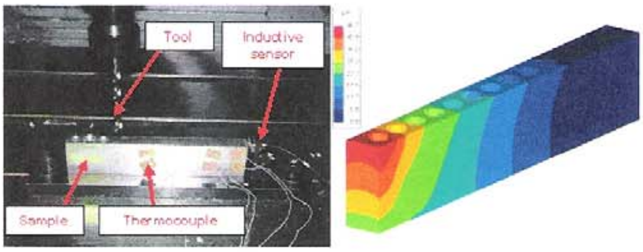

Figure 8: experimental setup for temperature and displacement measurements on aluminum samples (left); displacement obtained with simulation (right)

We can compute the heat flows generated during the various operations as a function of time and apply these thermal loads to the finite element model of the part. The SAMCEF non-linear solver MECANO is used to perform the thermal analysis. The result is the variation of the temperature in the workpiece in function of time (figure 9).

Figure 9: evolution of temperature from a thermal analysis when drilling respectively 2nd, 11th and 15th holes in an aluminum sample

The second step is to compute the mechanical deformations of the part clamped and submitted to the temperature variation. This step is performed with the same solver. Thus we obtain the evolution of the displacements in function of time (see figure 10).

Figure 10: evolution of displacements from the mechanical analysis (same holes as figure 10); the part is clamped at each of the four corners

2.3 Compensation of thermal deformations

After a thermo-mechanical analysis, we may modify parameters such as cutting conditions, drilling order or part clamping in order to improve the precision. Another possibility is to compensate the deformations using the computed displacement fields.

For the sample of figure 11, we know when each hole is drilled and how great the displacements at this particular instant are. So it is possible to modify the reference position of the holes according to the computed displacements in order to compensate the part dilatation (see figure below).

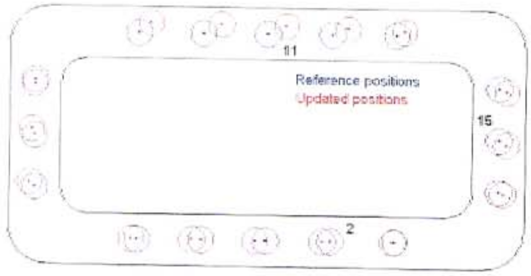

Figure 11: reference and updated hole positions

Here, the hole displacements of holes 2, 11 and 15 are respectively (-10,0), (24,10) and (10,-6) in micrometers.

3. Application

The operation is the face milling operation of the bottom face (crank shaft) of the cylinder crankcase of the Renault 2.0 dCI engine. The part is made of cast iron GL04.

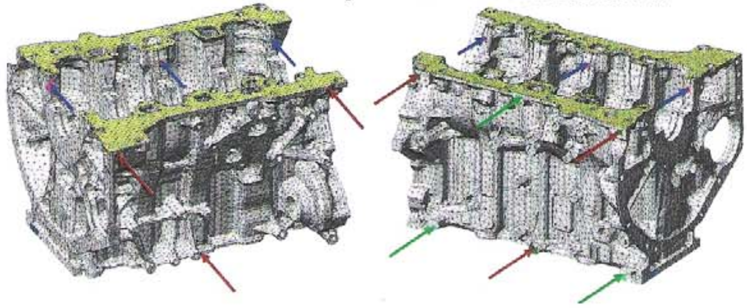

Figure 12: Clamping sequence (machined surface colored in yellow)

The clamping is made in three steps (see figure 12). First, the crankcase is put on three supports and maintained with three clamps (in red). Then three hydraulic supports act on the lateral face (in green). Finally, three double clamps maintain the part at the level of the crankshaft bearings number 1, 3 and 5 (in blue).

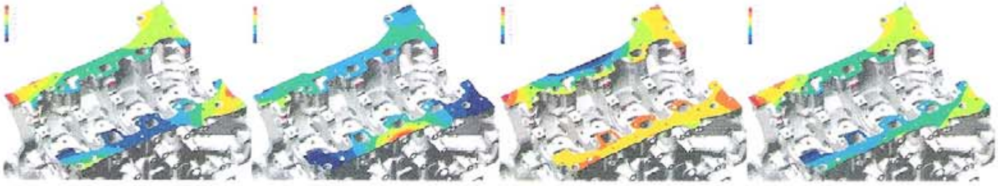

Figure 13: displacements obtained for steps 1, 2 and 3 and for the whole clamping operation (right)

In the simulation, we consider each step separately (see figure 13). The displacements of the machined surface at the end of the clamping phase (after step 3) are obtained by summing the displacements computed for the three steps (linear behavior). The tool is a face milling cutter of diameter 500 with 60 inserts. The trajectory is centered on the cylinders (see figure 14). The axis tilt angle is equal to 0.3 mm/m.

Once the part is clamped, we compute the defect due to milling, composed of the effect of cutting forces and the effect of the axis tilt (figure 14). The defect obtained at the end of the whole operation is the sum of the clamping defect and the milling defect. The flatness of the face is equal to 16.7 µm.

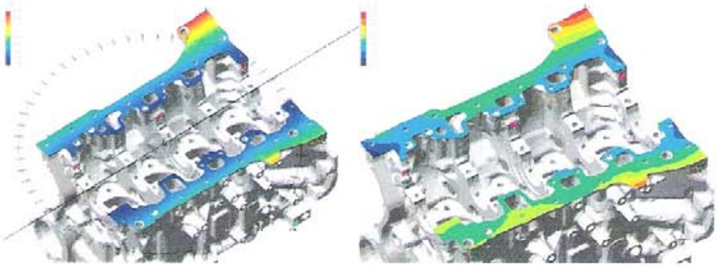

Figure 14: cutter, tool trajectory and axis tilt effect (left); flatness error (right)

Acknowledgments

The authors and SAMTECH s.a. acknowledge the contribution of RENAULT Powertrain Department (Rueil-Malmaison, France) and RENAULT DIESC (Le Mans, France) to this work. We also acknowledge the work of Unai Segurajauregui and Pedro José Arrazola (University of Mondragon, Spain) on the thermo-mechanical aspects.

References

Masset L., Debongnie J.F., Foreau S., Dumont Th. (1999) 'A model for the prediction of form errors in face milling and turning', ASME Design Engineering Technical Conferences, September 12-16 1999, Las Vegas, United States.

Masset L. (2001) 'Simulation of face milling and turning with the finite element method', International Journal of Forming Processes, vol. 4/3-4, pp. 481-498.

Masset L. (2004) *Analyse de gammes d'usinage par la méthode des éléments finis*, thèse de doctorat, Université de Liège, Belgium.

(http://www.ltas.ulg.ac.be/cmsms/uploads/File/PhD_Masset.pdf)

SAMTECH s.a. (2008). CAE Professional Solutions. www.samcef.com.

Segurajauregui U., Masset L., Arrazola P. J. (2007) 'Improving Quality in Machined Automotive Parts with the Finite Element Method', 10th ESAFORM Conference on Material Forming, April 18-20 2007, Zaragoza, Spain.