05A2035

The CAE design chain concept applied to automotive engine blocks

N. Gramegna, R. Merlo - EnginSoft Srl, Padova, Italy
L. Pirola - Fonderie Mario Mazzucconi SpA, Ponte San Pietro, Italy

Summary

High-performance engine block development necessarily requires high level CAE tools and a complete virtual loop in order to analyse any steps of “automotive component life” meaning manufacturing process, heat treatment, machining, assembling and life test.
In order to perform an accurate fatigue lifetime prediction of automotive component it is necessary to take several effects into account. Performance analyses of new aluminium alloy part should follow manufacturing phase as material mechanical properties depend on.

Simulation of metal casting processes and structural behaviours using computer programs which include flow and phase change effects, heat treatment and machining actions, has gained attention during the recent years.
In spite of many advancements in this field during recent years, simulation of engine block remains as one of the most challenging cases due to complexity of its geometry and the need for accurate prediction of manufacturing defects, mechanical properties of treated and machined components.

In this paper we present our recent experiences gained in modelling of an cylinder head using the integration MAGMASOFT-ANSYS in order to simulate heat treatment stress-strain effects after casting process.

Keywords

CAE integration, casting simulation, heat treatment simulation, residual stress

0. Introduction

A “Concurrent Engineering” technology for product industrialisation by means of CAE tools efficiently belongs in the design procedure for defining the component’s geometry and optimising the production cycle. The initial idea should become a production design which will account for the raw material’s advantages and disadvantages and for the facilities needed for the manufacturing process. Checking the process parameters at start-up and at steady state ensures the best variable set-up and defines their variation range so as to guarantee turning out components of the specified quality.

The practical simplifying assumptions that are usually accepted in the product-development chain are due to lack of knowledge of a component’s life cycle, from production to practical use, to neglect the stress and strain effects caused by the manufacturing process or heat treatment and to considering mechanical properties that do not reflect the material’s actual characteristics as depending on its microstructure and defects [1, 2, 3].

The currently available software tools for the design of new engine components allow the designer efficiently and profitably to integrate the structural analysis (relevant for component design) and the thermal and fluid-dynamic analysis (relevant for manufacture), thereby building up a virtual closed loop between the CAD/CAM and CAE environments and drastically slashing industrialisation and start-up times.

The objective is the link between studying the performance of the finished component and process simulation so as to optimize the component’s geometry and consequently achieve weight reduction by making use of an aluminum alloy’s actual properties.

The information gathered during process simulation can be used to:

1. Check the component’s feasibility in terms of smooth running and cost efficiency of the production process;
2. Compare several solutions, say, different casting processes or different process layouts, to assess the convenience of running certain machining jobs before or after the heat treatment.
3. Study the stress-and-strain status due to heat gradients which the part undergoes during casting and the successive heat-treatments.

1. The Integration Procedure

Integrating the simulation technologies calls for linking two or more efficient numerical tools, which should ensure reliability, computing speed, ease of use and result completeness [5].

The work done on a light-alloy (AlSi9Cu1) cylinder head of new design is a useful example of the above.

The cylinder head under consideration is that of a new direct-injection diesel engine whose most innovative feature is its small size.

The study follows the steps taken by “Fonderia Mario Mazzucconi” to manufacture and deliver the cylinder heads.

After several years of experience, casting process simulation is by now a routine tool for the engineering office. The innovation is the examination of stress conditions at process end and in considering such conditions as an initial constraint in all the successive heat-treatment and finish-machining operations.

The heat treatment (T6) was broken down into quenching and ageing.

The MAGMASOFT code (Fig. 1) was used to simulate the casting process (filling the cavity and solidification) and the induced stress status, as well as to evaluate the thermal history paying special attention to those transients having the highest temperature gradients during the entire heat-treatment. The stress-and-strain progress during quenching and aging was studied with the ANSYS code, which takes into account the creep effect so as the scheduled machining operations before or after T6.

As mentioned, the calculation methodology consists of two analyses: thermal and structural. To cut response time of the individual calculations as well as the overall time to complete the analyses, each analysis is carried out with the most time-efficient software. Indeed, today one has available the right tools to move information (geometry, mesh models and results) from one to another system, reliably and in real time (MAGMAlink module).

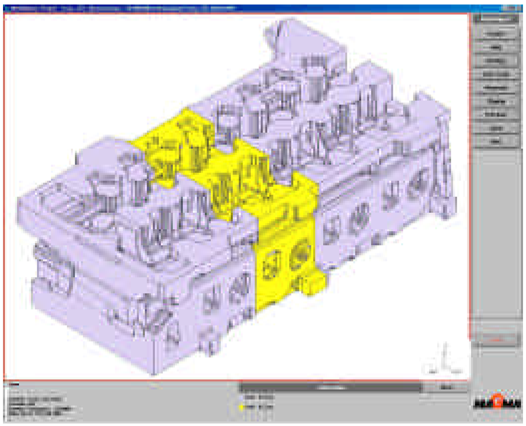

Fig.1 *CVs numerical model*

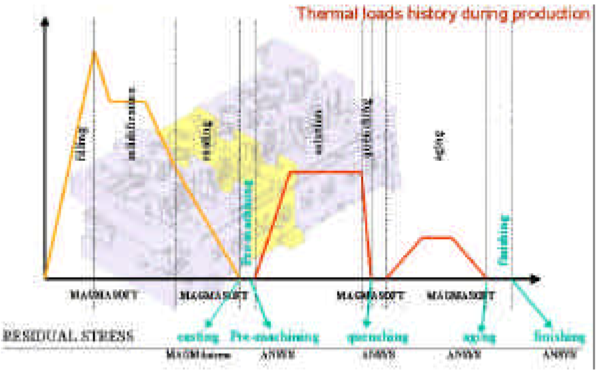

Fig.2 *Temperature history of cylinder head production*

Both the thermal and structural analyses are of the transient type, i.e., they permit simulating the typical thermal transients of the simulated processes.

The temperature profile in a generic point of the casting is qualitatively represented for all the simulated steps in the temperature-time plot of Fig. 2.

2. The Casting Process

The first part of the study was entirely carried out with one control-volume software (MAGMASOFT), using a suitable solver as required. In particular, the thermal-range calculation provides temperature distribution in space (Fig. 3) and the corresponding stress status is found using the MAGMAstress module. Fig. 4 represents such status across a section in terms of Von Mises' equivalent stress.

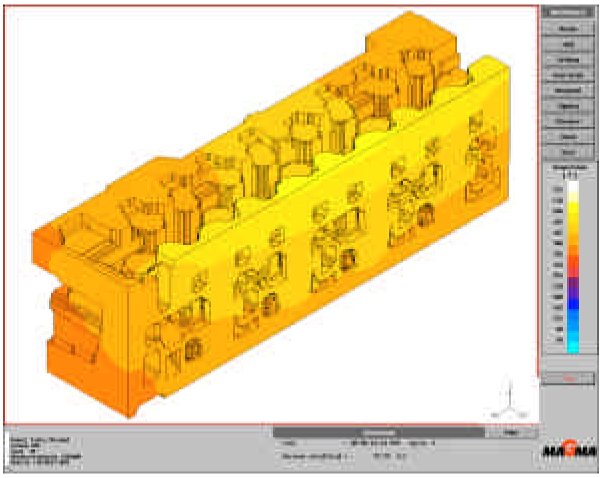

Fig.3 *Temperature distribution of cooling phase*

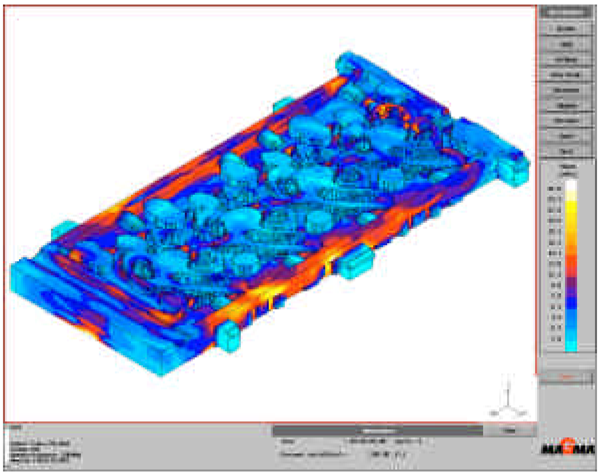

Fig.4 *Von Mises stresses distribution after cooling phase.*

Process simulation results are used to carry out the usual analysis of the casting's defects, according to the criteria available in the software [6] and, in this specific case, to check which parts of the casting are submitted to the highest residual stress and strain and whether plastic deformation occurs, which might cause the component's mechanical properties to decay after casting.

The defect levels found in the casting and its microstructure are evaluated by destructive or non-destructive tests (Fig. 5) and can be checked against the results of simulation.

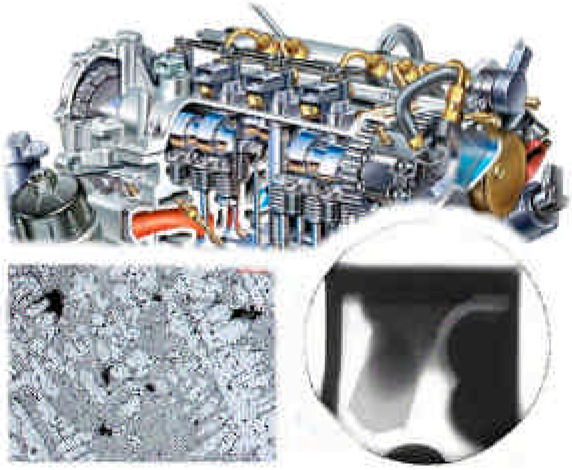

Fig.5 *experimental investigations*

3. Heat-Treatment and Finishing

As in solidification and cooling, although significantly more simply for the lack of liquidus/solidus transformation, heat-treating too causes stress and strain due to the difference in cooling time from one spot to another [4] and consequent expansion and contraction when the casting is dipped in water at room-temperature.

Determining the trend of stress and strain is the purpose of the identification of any crack initiation at the various local temperatures during the entire transient stage.

Solution-Treatment, when properly carried out, will remove the residual stresses altogether and provide stress-free castings for the successive heat-treatment.

The quenched casting can deform and shrink freely, without constraints. When submitted to severe thermal gradients, thin areas tend to shrink earlier and increase their mechanical strength, which opposes the shrinking of massive, slower-cooling areas. During quenching too, the stress-and-strain trend permits identifying any crack initiation at different local temperatures across the entire transient stage.

The thermal analyses, following T6 procedure, are identical to the preceding one (also the control-volume numerical model is unchanged), but the result, in terms of temperature distribution in space, is exported to the ANSYS code, within which the stress analysis is performed.

Defining the boundary conditions during quenching is based on experimental data; these made it possible to evaluate by inverse computing and to distinguish the heat-transfer coefficients at the outer, water and oil surfaces. Such distinction provides a sound although approximate method to consider the actual heat loss due to the water flowing in the cylinder head's cavities, with an evident temperature rise of the quenching medium in the more narrow ducts.

The mesh of the FE model is composed of parabolic tetrahedral elements (10 nodes per element); it is manually defined by the user in the singled-out critical zones where high quality of the model has to be guaranteed (Fig. 6). The entire model consists of 427,248 nodes, for 1,281,744 degrees of freedom (DOF).

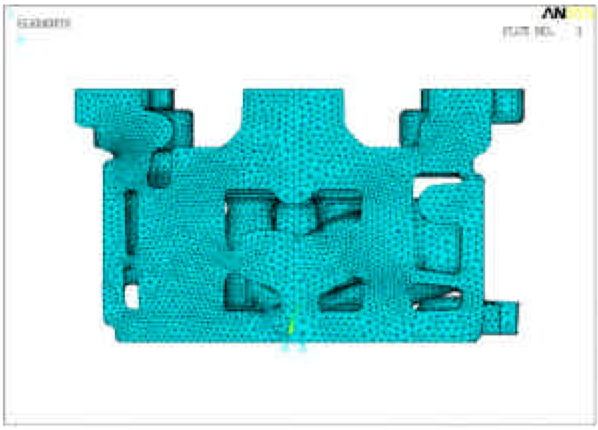

Fig.6 *FEM model*

Fig. 7 and Fig 8 show the temperature distribution in the considered head part and the results in terms of Von Mises' equivalent stress, as calculated during and at the end of the quenching process.

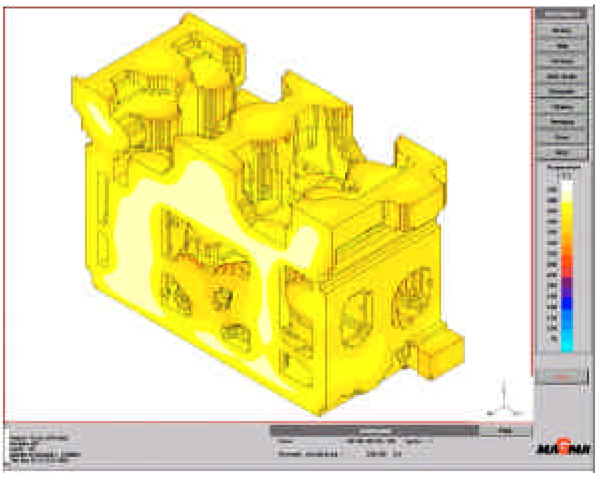

Fig. 7 *Temperature distribution during quenching phase*

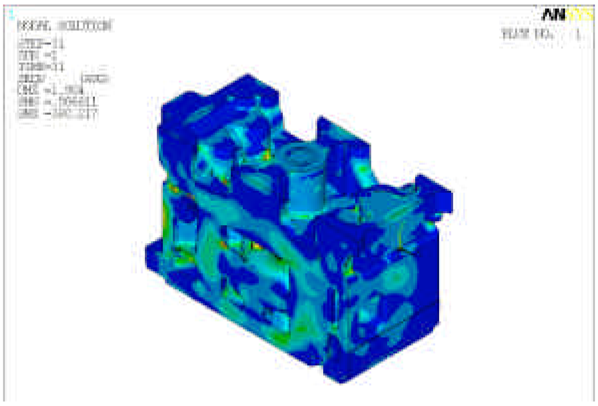

Fig. 8 *Von Mises residual stress distribution after quenching process*

Calculation of the elastic and plastic components of the strain curve permits obtaining the material's plastic-strain level at the end of the quenching process when the stress induced by the heat gradient exceeds the material's strength reference value (Fig. 9).

From this point, the numerical calculation proceeds with a new thermal gradient, typical of the ageing treatment, starting from the stress-and-strain conditions at the end of the quenching process, which have been calculated by relying on ANSYS' capability to handle pre-stress statuses. At this stage, the material's analytical model permits accounting for the creep phenomena that characterise the process with a new visco-plastic component due to the creep (Fig. 10).

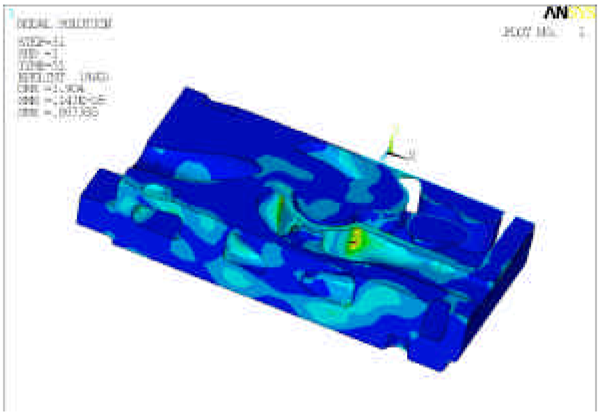

Fig.9 *Elastic strain after quenching*

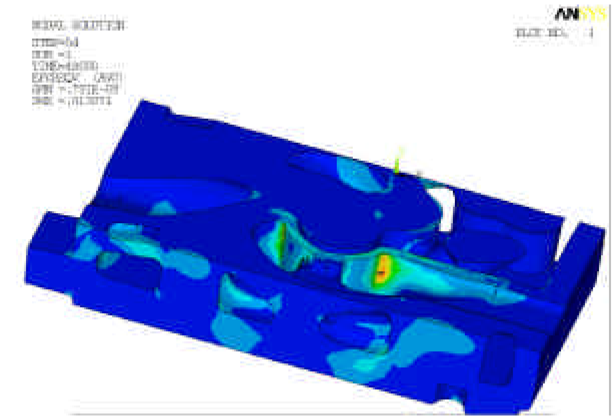

Fig.10 *Visco-plastic strain after aging*

The thermal analyses simulate heating to the pre-set temperature according to the upwards ramp of the heat-treatment furnace, holding at temperature and air-cooling.

The calculation for the 4th design stage, when the stress-and-strain status modifications induced by machining are evaluated, starts from previous deformation condition of the automotive component.

The "Birth & Death" technique, with the elimination of the material machined-away, is a non-linearity consisting of the elimination or ex-novo creation of some Finite elements, in the sense of contributing to overall stiffness, during the loading process.

In this specific case, the option was used to simulate material removal by machining (Fig. 11).

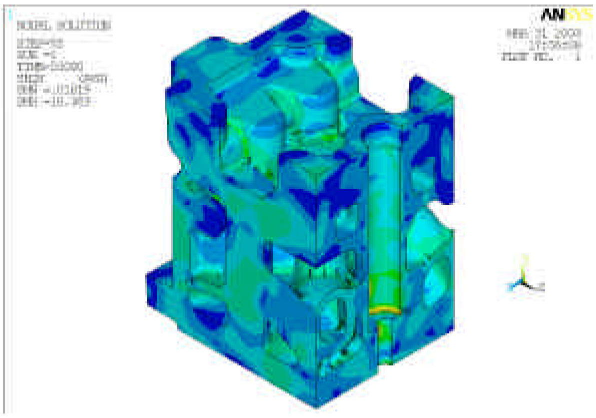

Fig.11 *Von Mises stress distribution after final machining*

Conclusions

Intelligent Digital Prototyping (IDP) in a CAE environment is an innovative design approach, which ensures that the required quality standards are attained and the production process is optimised [7, 8, 9]. During the prototyping stage, the simulation strategy should rely on efficient tools, which can dialogue with each other so as to tackle each stage of production and use of the component step by step.

The case described, on the study of a diesel-cylinder head, follows this approach so as to identify the parts of the cylinder head that are the most stressed during the manufacturing process and which are the residual stress-and-strain levels that will be unavoidably added to the operating load.

At this moment, design engineers will be able to take this pre-stress status into account and check the component's current safety coefficients.

By comparison of the several possible solutions, the one guaranteeing, say, for an equal weight, the highest safety coefficients and the best process conditions can be selected.

Full integration calls for reliable competence and technologies in order to blend the several CAE disciplines - combustion, hydrodynamics, vibro-acoustics, kineto-dynamics, machining, the production process and thermo-mechanical fatigue - into one procedure only.

References

[1] G. Hartmann, J. C. Sturm: "Integrated Numerical Optimization of Highly Loaded Aluminium Cylinder Heads".

[2] A. Egner-Walter, G. Hartmann, M. Kothen: "Integration of manufacturing process simulation into the process chain", 20th CAD-FEM Users' Meeting 2002, October 9-11 2002.

[3] J. C. Sturm, E. Hepp, A. Egner-Walter,: "Integration of casting simulation into crash simulation", 20th CAD-FEM Users' Meeting 2002, October 9-11 2002.

[4] John Campbell: "Castings", Butterworth-Heinemann.

[5] F. Bonollo, S. Odorizzi: "Numerical Simulation of Foundry Processes", SGEditoriali.

[6] I. Dall'Argine: "Porosity of a Cylinder Head Cast into a Permanent Mold: Study and Solution of the problem through a Process of Analysis", Engin Soft Conference and Users' Meeting 2002.

[7] G. Hartmann, C. Sturm: "Integration of Casting Simulation into the Design-Process-Chain of Structural Components", TCN-CAE 2003, International conference on Cae and Computational Technologies for industry.

[8] F. Linares, Z. Zamperin, N. Gramegna, L. Furlan: "iDP approach to quality assessment and improvement of diecasting process", TCN-CAE 2003, International conference on Cae and Computational Technologies for industry.

[9] F. Bonollo, N. Gramegna: "Magnesium diecast steering wheel: from simulation to mechanical behaviour", TCN-CAE 2003, International conference on Cae and Computational Technologies for industry.