

WHITE PAPER

Mechanical
optimization of the
injection system in
a **COMPRESSION
MOLDING MACHINE**

The machine under study is a Continuous Compression Moulding (CCM) hydraulic press for plastic cap manufacturing with extrusion of dry-blend granulated compounds. The machine performs the metering, the pelleting and the insertion in the cavities of the molded resin in order to obtain the final shape which is a semi-finished cap.

The CCM machine is made of the following components: controller, extrusion, electric box, cooling box, molding module. The machinery has two carousels: one for completing the molding of the resin and one for executing the injection of the cavity. The latter performs the portioning of the melted pellets, the transportation and the compression inside the mold. In the injection phase, a pneumatic piston will insert the melted pellet into the mold.



Fig. 1 - a) Original 12 mm Piston. b) Broken Piston

The pneumatic piston is the component which we will investigate in an optimization approach that we present in this article.

A critical aspect of the problem is that the compression phase to complete the piston movement only lasts fractions of a second, two milliseconds for ascending and two milliseconds for descending - this allows a maximum production capacity of 600 Pieces per Minute.

The main consequences are high speed results and huge accelerations which can lead to potentially harmful collisions.

The first optimization procedure was completed by manually changing the shape of the piston in the configuration with a 12 mm diameter, in order to prevent occurring fractures as shown in Figure 1.

The design activity was carried out using Solidworks (from Dassault Systèmes) as geometry modeler and ANSYS as Finite Element Method tool.

The manual optimizations followed the mechanisms of intake and exhaust valves inside endothermic engines, where the right shape configuration of tapered surfaces is essential for improving performances. In this way, the piston region where fractures generally occur, was modeled through a tapered geometry (see Figure 2). The solution for the design problem

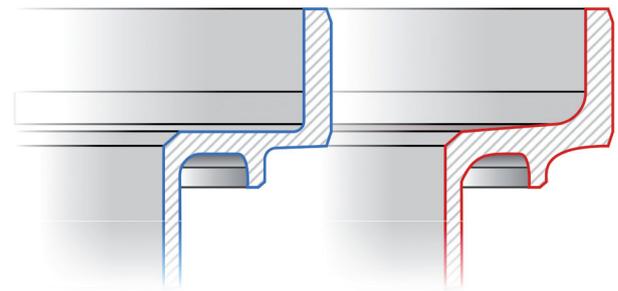


Fig 2 - Tapered Geometry (RED)

required a trade-off between the reduction of weight and an increased safety factor, both could be achieved with a stress analysis. The increase of the weight of a high-speed moving component led to damages in some surrounding areas. For this reason, we decided to design a light yet strong piston. After 20 working days, the manual optimization had to be terminated due to difficulties we experienced in achieving further improvements in the performances of the part. The results are summarized in Table 1 and in the Before and After Geometry comparisons (Figure 3).

In the next phase of the study, the new component to be optimized was a 16 mm diameter piston that was initially sketched with the same original geometrical configuration (not tapered) of the 12 mm piston (before the manual shape optimization). In this configuration, the piston features some potentially harmful regions for the fatigue analysis as illustrated in Figure 4. In addition to the geometry modeler (Solidworks) and the FEM tool (ANSYS), an optimization tool has been adopted (modeFRONTIER). The idea is to use modeFRONTIER for schematizing the design process and for defining an optimization strategy. The first step was to prepare

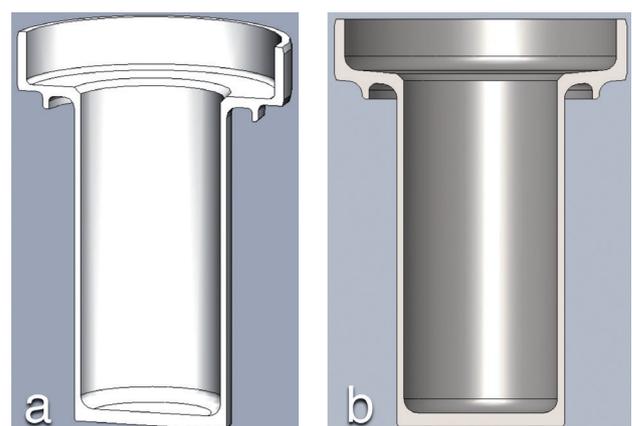


Fig. 3 - a) Original geometry b) Manual optimized geometry

a parametric geometry of the piston in order to transform the most relevant model entities into design variables that can be managed by the optimization Environment. Figure 5 shows the two-dimensional draft sketches and the nine parameters that control all the possible piston shapes. The problem is solved in two dimensions because its axial-symmetric characteristics also allow a consistent reduction of simulation time.

Geometry	Minimum Safety Factor
ORIGINAL (before)	0.9
MANUAL OPTIMIZED (after)	1.2 (+33.3%)

Table 1 - Comparison Before and After the manual optimization

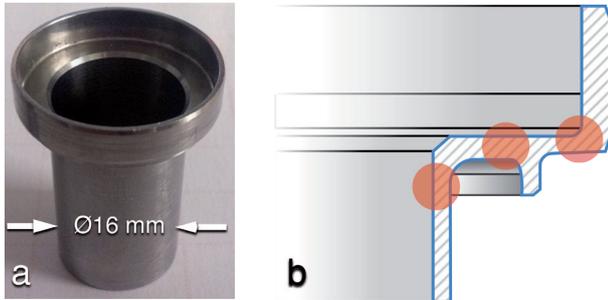


Fig. 4 - a) 16 mm piston with original geometry b) Critical regions (RED)

The second step of the automatic optimization set-up was to create a FEM model starting from the parametric geometry. Here, Workbench from ANSYS has been used as it allows the designer to lay-out the simulation process. In our present case, two simulations had to be performed (see the scheme in Figure 6): the first should solve the stress generated by the collision of the piston (on the left), while the second (on the right) represents the final run of the analysis that will supply the simulation results to modeFRONTIER.

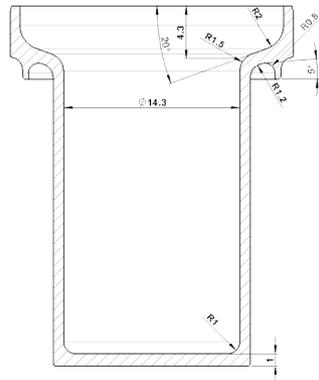


Fig. 5 - Free geometrical free parameters

The final step of the optimization procedure is to define the objectives and constraints along with the strategy to be used for the search of the optimum. The modeFRONTIER workflow shown in Figure 7 reflects the entire process in a single diagram: input variables (on the top), process integration and optimization strategy (in the middle), output variable, objectives and constraints (at the bottom).

With regard to the adopted strategy, an initial set of configurations was needed for initializing the optimization algorithm. The wizard of the optimization tool provided the proper choice of methods in order to pursue the objectives in a time frame of 24 hours. The system selected the SOBOL Design of Experiments of 50 starting points (designs). Then a Multi-Objective Genetic Algorithm was used for achieving the two design objectives, the Maximum Principle Stress (X axis, see Figure 8) and the Minimal Principal Stress (Y axis, see Figure 8). The constraints were provided by the acceptable weight tolerances (+ - 10%) towards the original design.

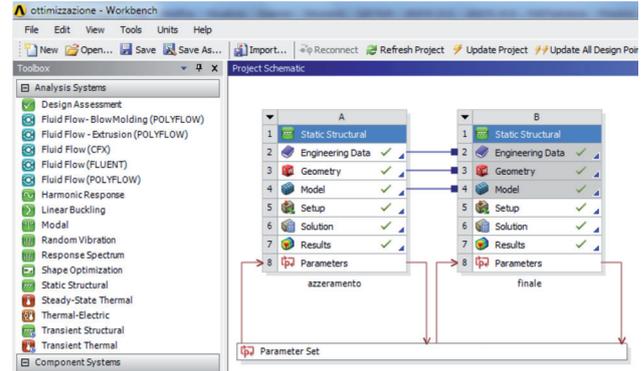


Fig. 6 - ANSYS Workbench simulation layout

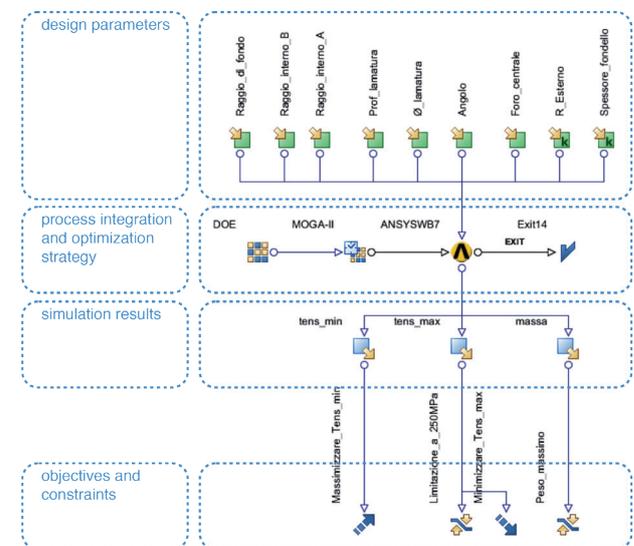


Fig. 7 - modeFRONTIER design optimization workflow diagram

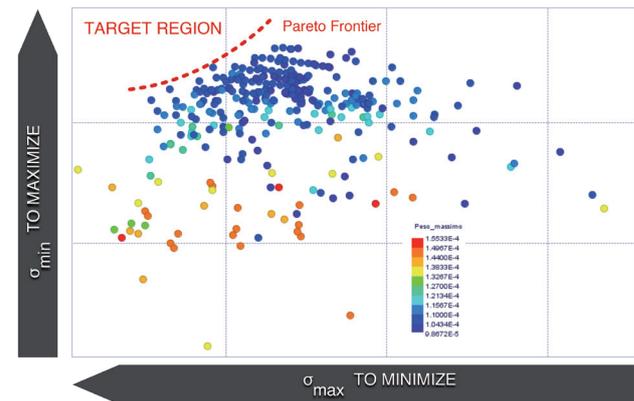


Fig. 8 - Scatter chart displaying the 800 results of the automatic optimization process

The optimization successfully completed 800 different piston configurations (which are graphically represented by a bubble in the scatter chart in Figure 8, the color scale is associated with the weight of the piston (from blue to red).

The scatter chart also presents the Pareto Frontier which contains all the optimal designs (Pareto designs). It is possible to operate the trade-off between the piston configurations within the designs. This is a subjective choice that has been carried out in order to select the final design obtained through

Geometry	Minimum Safety Factor
ORIGINAL (before)	0.9
MANUAL OPTIMIZED (after)	1.3 (+44.4 %)

Table 2 - Comparison Before and After the automatic optimization on the 16 mm piston

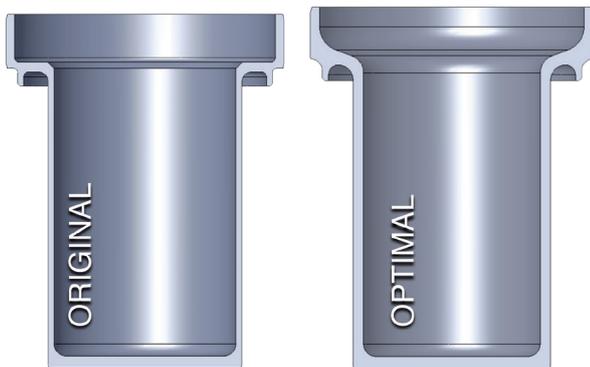


Fig. 9 - Original 16 mm piston (LEFT) and Optimal piston as result of automatic optimization process

	MANUAL OPTIMIZATION	OPTIMIZATION WITH modeFRONTIER	OVERALL OUTCOMES
VIRTUAL ANALYSES TESTED	about 30	800	+2667% better exploitation of simulation resources
PERFORMANCES IMPROVEMENT	+33.3 %	+44.4 %	+33% better results
TOTAL TIME TAKEN TO COMPLETE THE TASK	20 days	4 days	500% better exploitation of design time

Table 3: Comparison between manual and automatic optimization.

an automatic optimization procedure (see Table 2). The optimal piston selected for the comparison with the original one is shown in Figure 9. With modeFRONTIER it is also possible to obtain statistical analysis results across the design variables. The statistical plot in Figure 10 shows the direct and inverse relationship between the problem variables. More specifically, it features in colors (from red to blue) and numbers (from 1 to -1) the correlation between the input parameters and

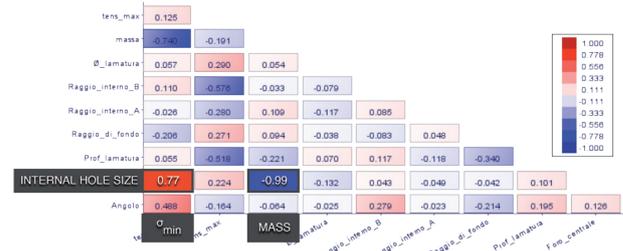


Fig. 10 - Correlation Plot

the output measure. For example, by increasing the internal diameter of the piston it leads to a consistent (0.77 means high direct relationship) increase of the Minimal Principal Stress and an even greater (-0.99) reduction of the total mass of the piston.

Conclusion

In conclusion, we can say that we have obtained a sufficient improvement on the 16 mm piston while substantially saving time for completing the tasks.

Moreover, by adopting a design framework as the one provided by modeFRONTIER, the designer can focus more on the pre and post-processing of the optimization results. It is no longer necessary to change, again and again by hand, each parameter at a time for increasing the safety factor of the component under study. This leads to a better exploitation of the design cycle time (as illustrated in Table 3) and to an optimal solution, both through objective optimization and subjective selection of the best compromise among the available results.

Andrea Minardi - SACMI Group

Francesco Franchini - EnginSoft

EnginSoft is a premier consulting firm in the field of Simulation Based Engineering Science (SBES) with a global presence. It was founded in 1984, but its founder and initial employees had been working in SBES since the mid '70s. Throughout its long history it has been at the forefront of technological innovation and remains a catalyst for change in the way SBES and CAE technologies in general are applied to solve even the most complex industrial problems with a high degree of reliability.

Today, EnginSoft is comprised of groups of highly qualified engineers, with expertise in a variety of engineering simulation technologies including FEM Analysis and CFD, working in synergic companies across the globe. We are present in Italy, France, Germany, the UK, Turkey and the U.S.A. and have a close partnership with synergetic companies located in Greece, Spain, Israel, Portugal, Brazil, Japan and the U.S.A.

EnginSoft works across a broad range of industries that include the automotive, aerospace, defense, energy, civil engineering, consumer goods and biomechanics industries to help them get the most out of existing engineering simulation technologies.

