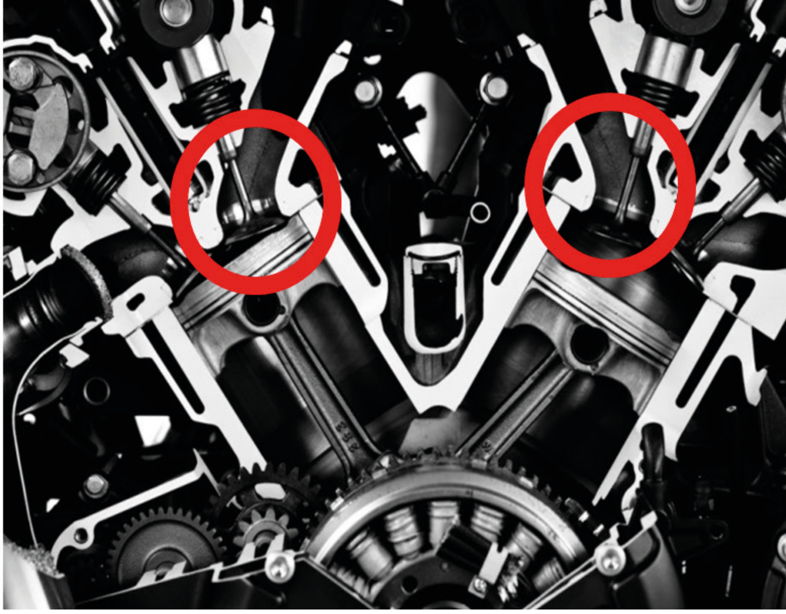


Fluid Dynamics Optimization of Racing Engine Inlet Ducts at Aprilia Racing

Racing engines are experiencing continuous evolution, allowing them to achieve extraordinary levels of performance and complexity. However, at the same time, regulations are restricting engine development by constraining some of the main design parameters. Consequently, it has become increasingly difficult to improve the engine's performance with traditional design methods, therefore an ever-increasing adoption of new CAE methods is required.
This paper presents a method to optimize the fluid dynamics for the intake valves and ports of the Aprilia RS-GP motorbike. The aim is to develop a procedure in which parametric geometry design, automatic mesh generation and 3D CFD analysis are coupled within modeFRONTIER in order to maximize the efficiency of the valve and port, while guaranteeing design feasibility.
As a result, the discharge coefficient at maximum valve lift of both valve and port has been improved by 1.5% and 2% respectively. These results have been validated through physical experiment with measured improvements of 1.2% and 1.6%, respectively.

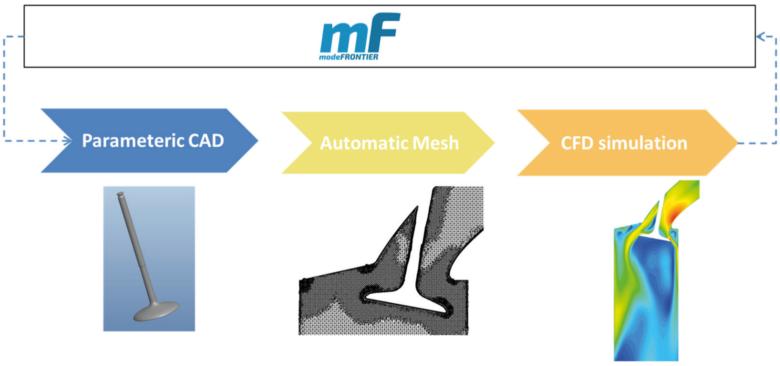


Fig.1 – Optimization process of intake valve and port

1. Improve fluid dynamic efficiency of Intake Valves and Intake Ports of MotoGP motorbike

Every year, MotoGP teams try to enhance the performances of their bikes by working on handling, traction and engine performances. Within this last aspect, effort is focused on several aspects, such as maximum generated power or maximum efficiency and durability.
Due to MotoGP regulations or structural limits, several engine design parameters are fixed or cannot be modified by engineers during development stages. For this reason, the best way to increase power is to enhance the volumetric efficiency of the engine, minimizing pressure losses and thus maximizing the intake mass flow rate.
A two-step optimization procedure was performed starting from the current engine configuration. In the first step, the inlet valve geometry was optimized while in the second, the terminal part of the inlet port was optimized using the valve geometry obtained in the first study.

Both optimizations have been performed using modeFRONTIER, integrating different software tools in an automated process, and applying the available multi-objective optimization algorithms. The parametric geometry has been built with the PTC Creo Parametric. The ANSYS ICEM CFD has been used to generate a structured mesh of tetrahedral elements using the octree algorithm. Finally, to quantify the fluid flow performances of each candidate solution, a full 3D steady state simulation model has been defined in ANSYS CFX. The fluid domain has been modeled in order to replicate the experimental bench used for

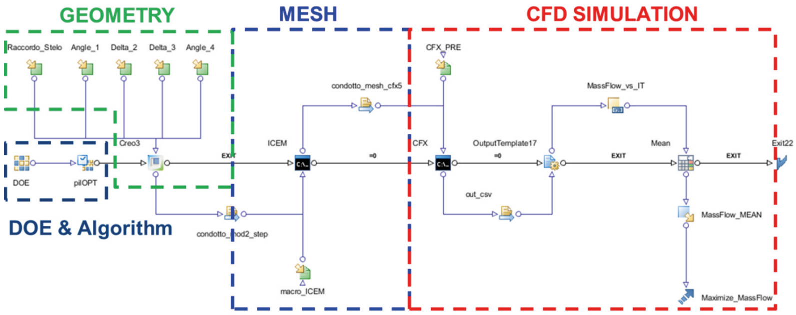


Fig.2 – Optimization workflow in modeFRONTIER

results validation, considering a fixed valve position at the maximum lift and throttle wide open. The two equations k-omega SST model is used to model the turbulent flow inside the cylinder.

2. Intake Valve Optimization

The valve geometry determines the fluid flow characteristics of air entering in the cylinder chamber. The first step is the optimization of the valve's head. In fact, the valve's stem design is fixed due to structural constraints and distribution configuration. The fillet radius and four angles defining the valve's head geometry are considered as input variables, while the maximum valve radius is fixed to avoid interference with the exhaust valve. In fig.2, the workflow created in modeFRONTIER to drive the optimization task is reported, highlighting the sequential phases of the process automation.

Performing 3D CFD simulation is quite demanding in terms of computational resources and time, hence, classical optimization techniques may not be feasible. In this specific case, pilOpt and Simplex algorithms have been combined sequentially to find the best solution. PilOpt is a proprietary hybrid multi-strategy algorithm that combines both local and global search algorithms, speeding up the convergence rate, balancing real and RSM-based designs. Multiple RSMs are automatically trained, validated and used to reduce computational time and achieve better solutions. After running 100 designs using pilOpt, the search has been refined using the Simplex algorithm, with additional 60 design evaluations (fig.3).

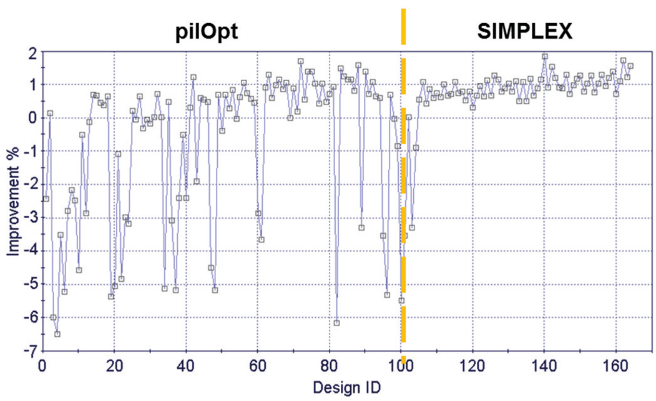


Fig.3 – Optimization results with modeFRONTIER: % improvement of intake overall mass flow rate

The results of the optimization lead to a 1.5% increase of the overall mass flow rate considering the optimized geometry. Looking more in detail to the velocity profiles and fluid streamlines (fig.4), it is possible to note how the optimized geometry enables less deviation and smoother transition of the fluid from the port to the cylinder chamber.

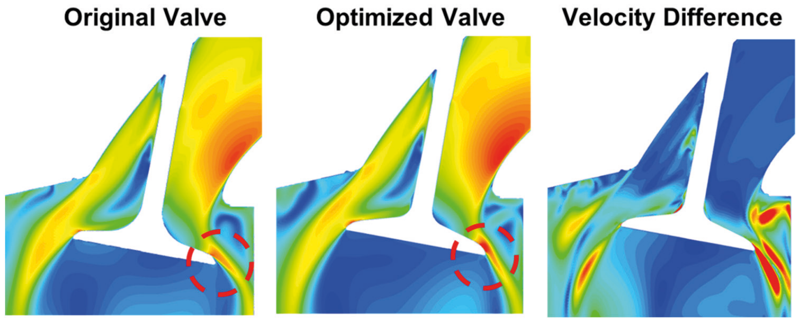


Fig.4 – Velocity field around baseline and optimized valve; velocity difference of baseline and optimized valves

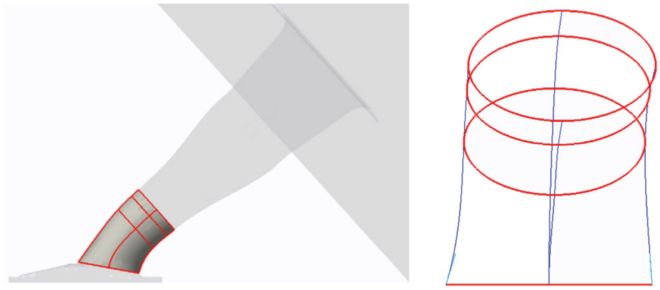


Fig.5 – Sections and splines used for the geometrical parameterization of the intake port

3. Intake Port Optimization

The second activity considered in this work is the optimization of the intake port geometry. The intake geometry is strongly constrained in its position from the intake valve actuation mechanism, the crankshaft displacement and the engine displacement. The major challenge is to define an efficient parameterization of the intake tube limiting the number of input variables, obtaining at the same time robust and accurate geometry variations, while respecting constraints coming from the limited free space.

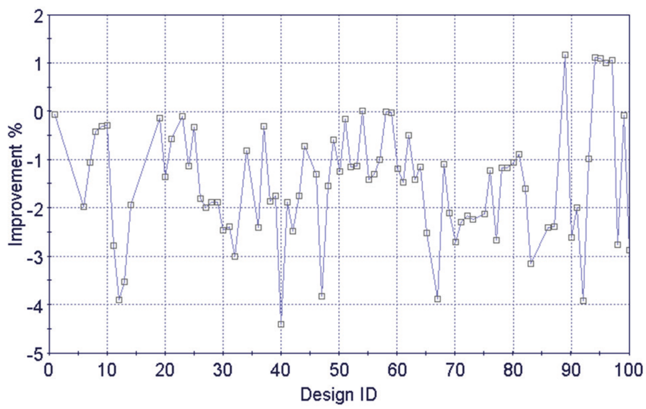
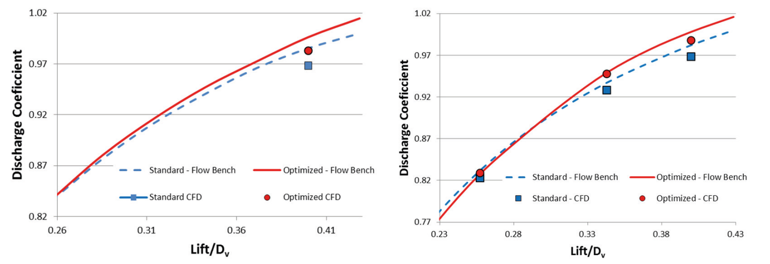


Fig.6 – Optimization results with modeFRONTIER: % improvement of intake overall mass flow rate



Optimized Valve Massflow		
h/D	CFD	Flow Bench
0.4	+ 1.5 %	+ 1.2 %

Optimized port Massflow			
h/D	RSM	CFD	Flow Bench
0.34	-	+ 2 %	+ 1.3 %
0.4	+ 1.45 %	+ 2 %	+ 1.63 %

Fig.7 – Comparison of baseline/optimal solutions found by numerical optimization (left: valve; right: port) with experimental measurements at the bench –discharge coefficient is measured in function of lift/diameter valve ratio

For these reasons, the port is described by four sections defining the frontal area and four splines defining the longitudinal expansion of the port and the position of the sections (fig.5). The spline shapes have been defined directly in modeFRONTIER using the Bezier curves. This function is implemented in the calculator node and allows the generation of smooth and continuous profiles using a limited number of input variables, namely control points. These are then imported in the CAD software where the 3D geometry is generated. A constraint is defined on minimum volume and, in cases where the constraint is not respected, the CFD simulation is not performed in order to save time.

As for the previous optimization, the Pilopt algorithm has been used with setting a total number of 60 iterations (fig.6). The best design has allowed a 1% improvement of the total mass flow rate compared to the baseline design. This is a satisfying result considering the limited possibility of variation of the port geometry.

4. Experimental validation of optimized results

The optimal results obtained by the numerical optimization have then validated using an experimental test bench, showing good agreement between simulation and experimental results (fig.7). It is interesting to note that the result obtained by the valve optimization approach has carried improvements at even lower valve lift ratios, while the optimization has been carried out only at fixed high valve lift ratio (0.4 h/D). This is essential to keep low computational effort and time and ensure a robust solution at other engine operating conditions.

The results illustrated in this article represent only a preliminary study to adopt proper optimization strategies to improve engine performances. Further steps would consider multiple valve lift values and involve combustion simulations

Alberto Clarich, Giulio Cassio
ESTECO