

Coupled mode FRONTIER/FIRE Approach for I-C Engine Intake Port Optimization

Author: Giuseppe Tesolin University of Trieste

Aiders: Peter Tibaut AVL List GmbH

Paolo Geremia ES.TEC.O s.r.l.

Francesco Pinto University of Trieste

Email: giuseppetesolin@tiscali.it

Summary

I-C Engine combustion efficiency and emissions characteristics are governed by the particular geometrical shape of the intake ports and the combustion chambers of the engines. Commonly we can distinguish two parameters depending on the particular intake port and combustion chamber shape: the *coefficient flow* or *discharge coefficient*, calculated through an open cross section located on the cylinder head, and the *swirl/tumble* level called also charging movement, that is responsible for the particular flow rotational motion within the cylinder. All these parameters influence in a relevant way the engine performances. In fact an augmentation of the discharge coefficient leads to an increased power of the engine. On the other hand increasing the swirl/tumble level it is possible to obtain a reduction of the emissions characteristics. Unfortunately they are conflicting parameters. It means that it is not possible to get a particular intake port shape with an optimum value for both the here described parameters. Hence the necessity to provide a methodology aimed at achieving global optimum values for both the discharge and the swirl/tumble level. This is the reason why performing manual optimizations on intake port surfaces. But a CFD manual optimization usually requires a large number of trials, without necessarily leading to a global shape optimum. So, the present work is aimed at achieving a global shape optimum for intake port surfaces using an automatic optimization tool, according to the customers' strategy of optimizing the discharge coefficient, targeting the swirl number (typical swirl intake port).

The coupling between three different packages, *ProEngineer* (CAD tool), *FIRE* (CFD tool) and *modeFRONTIER* (optimization tool) has been provided on the basis of scripts and macros. Three different steps are performed in each optimization loop:

- surface shape regeneration of the last evaluated intake port geometry;
- CFD simulation of the new intake port surface;
- analysis of the obtained results (*coefficient flow* and *swirl* level).

The optimization tool keeps under control, on the basis of different types of mathematical algorithms i.e. *Genetic* algorithms, the other two packages leading to the generation of a geometry satisfying the prefixed goal. More than hundred surfaces have been simulated with *modeFRONTIER*. By comparison between numerical best results and measurements (AVL test bench facilities), it has come out a good fitness.

Keywords

Coupling, **ProEngineer/FIRE/modeFRONTIER**, Automatic Optimization

Coupling ProE, FIRE and modeFRONTIER

The first step to perform, in order to provide an automatic intake port optimization, is establishing the coupling between our three different packages: ProEngineer, FIRE and modeFRONTIER. In other words, our purpose is to carry out a methodology aimed at running these packages in a loop just by clicking a push-button. In each of this loop modeFRONTIER, on the basis of different types of mathematical algorithms (i.e. Genetic algorithms), creates a new input file for the parametric 3D CAD tool ProE. Now this one can provide a regeneration of the previous intake port geometry and then with FIRE, it is possible to start up a CFD calculation that gives out the results for the discharge coefficient and the swirl number. Finally modeFRONTIER, on the basis of the FIRE results, will generate a new parametric input file for ProE and a new loop will be run.

ProEngineer: Intake Port surface regeneration

As already mentioned in the previous section, in each loop modeFRONTIER creates a new input file for ProE. Now, in order to run ProE in batch to provide a new intake port shape, it is required to perform a *ProE macro*. The following steps are here recorded:

- importing the new parametric file;
- performing a surface regeneration;
- saving a *STL* file of the new intake port geometry.

The *STL* format is necessary because FIRE is able to import only this kind of files. In each parametric file of ProE have to be modified three parameters: *Separation Edge*, *Side Contour* and *Low Contour*. There are then other eight parameters depending on these three inputs. So by modifying our three input variables, a total number of eleven parameters are varied. This is the

reason why they can not be shown directly, but in their for example minimum value, keeping fixed the other two parameters in their average value (see *Figure 1*).



Figure 1. Separation Edge, Side Contour and Low Contour (minimum values).

FIRE: CFD calculation

Now that a new intake port geometry has just been provided by ProE, a CFD simulation has to be run in order to calculate the discharge coefficient and the swirl number. The first one is defined as a sort of pressure loss through an open crossed section located on the cylinder head, on the other hand the swirl, is the flow rotational motion within the cylinder. Both of them depending on the particular shape of the intake port surface. To run in batch this CFD calculation, the following steps are required:

1. ***FIRE macro 01***: with this macro it is possible to import the *STL* file and then preparing it to the meshing process (performing edges and selections on the intake port surface for the mesh refinements).
2. ***Fame Generator tool***: to provide in batch a new mesh file on each new imported geometry.
3. ***FIRE macro 02***: a second macro is required to perform on the volume mesh a new selection called *Paddle Wheel*. This is a ring shaped selection within the cylinder and we need it to calculate the speed of the rotational motion, or in other words, the swirl number.
4. ***CFD solver***: starting up the FIRE solver to calculate the coefficient flow and the swirl number.

Our surface is a typical *swirl* intake port. This means that the *tumble* number is very small and we can not consider it in our calculation.

modeFRONTIER: achieving the best design configuration

The purpose of the here presented work is to carry out the particular intake port shape with highest discharge coefficient, in a given swirl interval [1.95; 2.05]. To achieve the best design goal have been adopted two different strategies of optimization. In the first one we have tried to optimize both the objective variables, running many simulations with different types of algorithms: *MOGA (Multi Objective Genetic Algorithm)*, *Full Factorial* and *Random Sequence*, but obtaining only four feasible designs, within the given constraints for the swirl. So, to find out many other points in the given swirl interval, have been performed simulations with the *RSM (Response Surface Method)* tool. These surfaces are some sort of interpolations. So, starting with all the “*real*” designs, calculated in the first part of the optimization and

interpolating them with the RSM, many other “virtual” points have been carried out. After choosing only the best virtual feasible points between all the virtual designs, we have to validate them with a CFD calculation. In *Figure 2* are shown all the results. The four “blue” points have been obtained in the first part of the optimization. All the “red” ones after running the RSM and post CFD validation.

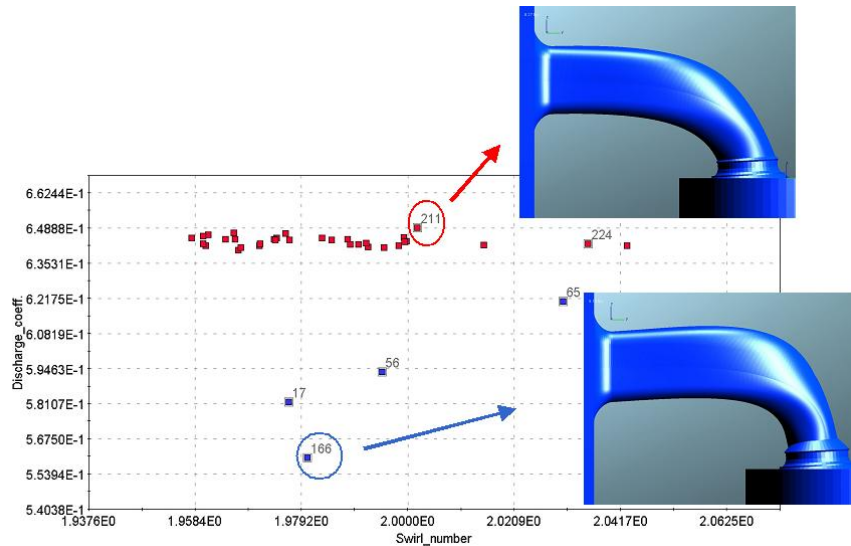


Figure 2. modeFRONTIER results

Design	Separation Edge	Low Contour	Side Contour	Swirl	Discharge
166	0.750	1.000	0.710	1.980	0.5603
211	0.974	0.165	0.971	2.001	0.6492

Table 1. 166 and 211 intake port results

There is an improvement in terms of discharge coefficient of around 6% between “red” and “blue” designs. The best result is the design 211 and the worst the 166 and always in *Figure 2* are shown the corresponding intake port surfaces. In *Table 1* are shown the same results as in *Figure 2* with, in addition, the corresponding input variables for the found best and worst objectives.

Flow Results

In this last section are shown the flow results in terms of *velocity* field and *streamlines* for both the design 166 and 211. For each different intake port geometry, we have different distribution of *velocity* and *streamlines* in particular in the area around the valve seat, situated on the cylinder head (see *Figure 3*).

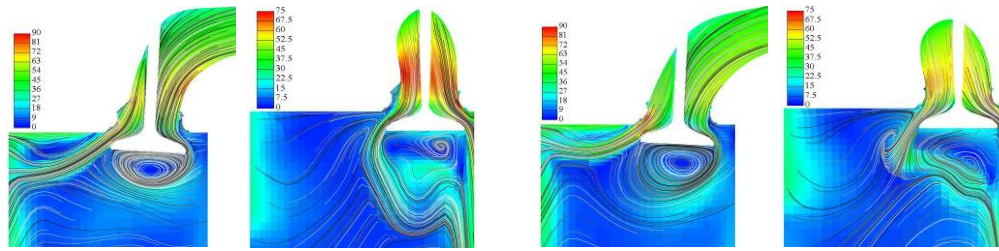


Figure 3. Velocity & Streamlines respectively for the design 166 and 211 (x and y cut)

Conclusions

Starting with the coupling, it has been successfully achieved in a quite short time and the optimization goal to maximize the discharge coefficient, targeting the swirl number in the interval $1.95 < \text{swirl} < 2.05$, has been successfully provided. Looking now at the results, it has not been possible to perform the modeFRONTIER simulations with a fine mesh, but only with a coarse one (4×10^5 cells). In fact in that case the calculation would have taken too much time (remember that just one FIRE calculation in MPI, with four CPUs, takes something like three days for a fine mesh of 1,5 million of cells). What could be quite innovative, is to perform the optimization running the FIRE solver with a fine mesh. This could be possible performing the calculation for the whole mesh just the first time. Then, after performing a new geometry and hence a new mesh, keeping always the same mesh settings, performing the CFD simulation only in that domains in which the mesh is different from the previous one. In that case this coupled approach will become quite more powerful in performing optimizations.

References

- [1] ESTECO 2004. ESTECO, modeFRONTIER 3.1 user manual
- [2] Graz AVL LIST GmbH. FIRE v8.3 Manuals
- [3] M. Ciofalo, G. Comini, G. Croce, E. Nobile and C. Nonino: “Fondamenti di TERMOFLUIDODINAMICA COMPUTAZIONALE” SGE editoriali, Padova, 2004
- [4] Corgard D. and Reitz R. D.: “Effects of Alternative Fuels and Intake Port Geometry on HSDI Diesel Engine Performance and Emissions” submitted for publication, SAE 2000 Congress
- [5] Giancarlo Ferrari: “MOTORI A COMBUSTIONE INTERNA 3edizione aggiornata” EDIZIONI il capitello, Torino, 2001
- [6] John B. Heywood: “INTERNAL COMBUSTION ENGINE FUNDAMENTAL” McGraw-Hill Book Company, Padova, 2004.
- [7] Senecal P. K., Montgomery D. T., and Reitz R. D.: “Diesel Engine Optimization Using Multi-Dimensional Modeling and Genetic Algorithms Applied to a Medium Speed, High Load Operating Condition” ASME-ICED 2000 Fall Technical Conference
- [8] D. T. Montgomery and R. D. Reitz: “Optimization of Heavy-Duty Diesel Engine Operating Parameters Using A Response Surface Method” SAE 2000-01- 962, 2000