Impact of inlet coherent motions on compressor performance

Jacopo Forlese\textsuperscript{1,2} and Giovanni Spoleti\textsuperscript{1,3}

\textsuperscript{1}GM Global Propulsion System Europe S.r.l., Corso Castelfidardo 36, 10129 Torino, Italy.
\textsuperscript{2}jacopoforlese@gm.com, \textsuperscript{3}giovannispoleti@gm.com.

Abstract. Automotive engine induction systems may be characterized by significant flow angularity and total pressure distortion at the compressor inlet. The impact of the swirl on compressor performance should be quantified to guide the design of the induction systems. In diesel engines, the presence of a valve for flow reduction and control of low pressure EGR recirculation could generate coherent motion and influence the performance of the compressor. Starting from experimental map, the compressor speed-lines have been simulated using a 3D CFD commercial code imposing different concept motion at the inlet. The swirl intensity, the direction and the number of vortices have been imposed in order to taking into account some combinations. Finally, a merit function has been defined to evaluate the performance of the compressor with the defined swirl concepts.

The aim of the current work is to obtain an indication on the effect of a swirling motion at the compressor inlet on the engine performance and provide a guideline to the induction system design.

1. Introduction

3D CFD simulation of the air induction system in internal combustion engines design is a process largely used at every production level. The addition of a centrifugal compressor leads to more long and complex analyses that have to deal with the needs of the time frame of design and production.

This work come from the necessity to find a correlation between coherent motions at the compressor inlet and the compressor behavior that affect the engine performance itself.

The maximization of boost and the turbo-compressor efficiency plays a very important role in the engine optimization in terms of performance and reduction of pollution. Bent and contorted under-hood pipes and EGR recirculation could lead to a very complex inlet system with negative effects on the compressor. These systems typically produce one or more changes in flow direction before the compressor inlet. Some systems produce strong swirl distortion able to influence the flow through the wheel and the stability. \cite{1}

Compared to the other works available in literature, the peculiarity of this project is to study the compressor behavior modification due to theoretical shapes of inlet flow without referring to a specific induction system design. \cite{2} \cite{3}

The case of study is a mid-size diesel engine for automotive traction.

2. Discussion

2.1. Classifications of swirl motion
The flow angularity generated by inlet systems may have both radial and circumferential velocity components. In particular conditions is possible recognize coherent motions as bulk swirl or paired swirl.

![Bulk swirl and Paired swirl](image)

**Figure 1.** Typical swirl pattern.

The bulk swirl consists of an inlet flow that is all rotating in the same direction, either with or opposed to the compressor rotation. In the figure 1 is shown a typical bulk swirl pattern. In addition to the axial component the predominant velocity component is the tangential.

The second type of swirl pattern is called paired swirl and consists of two or more paired vortices rotating in opposite directions. In the figure is possible to see also two example of paired swirl. Nevertheless, the vortex pairs do not have to be of equal magnitude: one or more vortices can dominate a smaller set of vortices.

There are also other types of swirl beside bulk and paired, such as in a rectangular pipe, but typically, upstream the compressor wheel, the most common motions are the bulk swirl generated by bend and twin swirl generated by flap to reduce the flow.

### 2.2. Swirl description

In order to evaluate and compare the motion generated by the inlet system the swirl angle has been calculated as a function of the circumferential position at constant radius at the compressor inlet section. The swirl angle is defined as

\[
\alpha = \tan^{-1}\left(\frac{v_{\text{tangential}}}{v_{\text{axial}}}\right)
\]  

The swirl description used in this document has been implemented in the commercial CFD 3D software in order to easily evaluate the value for each cell of the fluid domain and plot it for an external circumferential crown as in the figure 2 [4]. In the example, a T-junction with a paired swirl along with a plot showing how the swirl pattern look like a sinusoidal wave when the swirl angle is plotted out at a constant radius.
The coherent structure could be represented by the average and the standard deviation of the swirl angle value in the crown:

- A bulk swirl will have a high average value representing the intensity of the swirl;
- A paired swirl will have an almost zero average and the standard deviation will represent the intensity of the motions.

The flow has been considered positive if with the same rotation direction of the compressor wheel, otherwise negative.

2.3. Analysis setting and CFD model definition

The aim of the current work is to obtain an indication on the effect of a swirling motion on the engine performance and provide a guideline to the induction system design.

The work is based on the 3D Computational Fluid dynamic simulation of the operation of the compressor stand-alone. The idea is to compare the compressor maps of the compressor with and without a coherent motion imposed at the inlet.

The analysis has been led with the commercial software StarCCM+ by CD-Adapco - Siemens. A coupled implicit scheme has been choose for the steady-state simulations, the turbulent model was a SST-$k$-$\omega$, the fluid is compressible and the properties were temperature dependent. The fluid domain was divided into almost 5 million of polyhedral cells, the mesh was generated using the StarCCM+ tools. A mesh refinement is present at the shroud, at the diffuser and at blades paying particular attention to catch the curvature with very good accuracy.

As the wall treatment was used the All $y^+$ Wall Treatment that is a hybrid approach that seeks to recover the behaviors of the other two wall treatments in the limit of very fine or very coarse meshes. It is a design goal that this wall treatment should give results similar to the low-$y^+$ treatment as $y^+$ close to 0 and to the high-$y^+$ treatment for $y^+$ greater than 30. It gives reasonable results also for intermediate meshes where the cell centroid falls in the buffer layer [5].

In our model 12 prismatic cell layers has been used: with this settings the $y^+$ is in all the flow conditions below 30 (mainly about 1) in order to take advantage of the more accurate low-Reynold approach. [6]
The impeller motions was modelled with a moving reference frame: it is a method commonly used in proprietary CFD codes, where it is referred to as frozen rotor. This approach resembles a snapshot of the flow, frozen in time, thus failing to predict the unsteadiness of the flow between the diffuser vanes. Thus different relative positions result in different solutions, which is a major disadvantage of this approach. It is a fast preliminary method, however, and gives satisfying results and represent a good compromise between accuracy and computational effort giving a solution that represents the time-averaged behavior of the flow. [7]

The choice of the MRF is mainly due to the high number of the simulation done in this project and the analysis points selections has included only stable points of the compressor map (not affected by evident surge and choke phenomena).

Inlet was set as a stagnation inlet and the outlet was set as a pressure outlet in the high mass flow conditions and as a mass flow outlet at the left part of the map.

This is due mainly to stability: setting the mass flow rate the simulations are more stable in low flow conditions, on the other hand setting the pressure ratio is more stable at the right part of the map. It is possible to notice that in this way reply an experimental point is more accurate because close to the choke the speed-lines are almost vertical (for example a small error setting the mass flow close to choke condition could lead to ambiguity in determine the correspondent pressure ratio).

The experimental compressor map has been used as starting point to set the boundary condition for each speed-line. To reduce the simulation time has been chosen to consider only 3 of the 8 speed-lines of the map as the most representative condition for the engine (100k, 150k and 200k rpm). Then, for each speed-line up to 7 point were simulated with more than 50 k iterations each one. Also the points characterized by incipient surge and choke phenomena are still in the stable portion of the map and they have been analyzed in order to consider the range of the map.

![Figure 3. Setting of the boundary condition type depending from the map.](image_url)

2.4. Develop Concept
In order to consider all the typical swirl, generalized coherent motions were set at the compressor inlet. The simple theoretical motions considered have been:
The bulk swirl, where the entire flow field is rotating either in the same direction as wheel rotation (co-swirl, positive swirl angle) or in the opposite direction of wheel rotation (counter-swirl, negative swirl angle); both the direction has been considered.

The twin swirl that is composed of two vortices rotating in opposite directions. One vortex is a co-swirl vortex while the other is a counter-swirl vortex; the two vortices are considered symmetric.

For each type of swirl and direction two level of intensity has been considered: low swirl angle value (6°) and high swirl angle value (10°).

### Table 1. Definitions of coherent motions.

| Number of swirl | Direction     | Swirl angle value | Definition |
|-----------------|---------------|-------------------|------------|
| 1               | Co-rotating   | 6°                | 1L +       |
| 1               | Co-rotating   | 10°               | 1H +       |
| 1               | Contra-rotating | 6°          | 1L -       |
| 1               | Contra–rotating | 10°           | 1H -       |
| 2               | 1a            | 6°                | 2L         |
| 2               | 1b            | 10°               | 2H         |

* Changing the direction of the twin swirl does not provide any difference in the results.

The 3D CFD software used allow to set the flow velocity components at the inlet through field functions. Flow velocity components are set in a cylindrical coordinate system integral with the compressor inlet system. [4] In the bulk swirl, the tangential velocity has been as proportional to the pipe radius and the radial velocity is zero. In the twin swirl, the radial component is dependent from the angular coordinate $\theta$:

$$v_{radial} = \cos^3(\theta) \quad (2)$$

The tangential component is dependent from the angular coordinate $\theta$, the y-coordinate and the radial coordinate $r$. The tangential component has a maximum absolute value equal to 1 in order to be compatible with the radial component.

$$v_{tangential} = \begin{cases} 
1 & \text{if } w > 1 \\
-1 & \text{if } w < -1 \\
\text{else} & w 
\end{cases} \quad (3)$$

where:

$$w = \begin{cases} 
[\sin \theta (0.4 - \cos(\pi y))] + r & \text{if } y > 0 \\
[\sin \theta (0.4 - \cos(\pi y))] + r & \text{if } y < 0 
\end{cases} \quad (4)$$

Setting the value of the axial component (greater than 1) is possible to define the swirl angle value.
2.5. Swirl Merit Function

In literature [2], [4] and [1] it’s possible to find indication about the impact of swirl on compressor performance: a co-rotating swirl could reduce the flow and the pressure ratio but could increase the efficiency and the stall margin. Otherwise a counter-rotating swirl could have the opposite effects. Thus, in order to quantify the effects of swirl on compressor map, a merit function has been introduced called Speed Line Efficiency Factor.

The SLEF is calculated for each compressor speed-line and is defined as:

\[ SLEF = R \cdot E_p \cdot E_s \cdot E_c \]  

Eq. 5

where:

- \( R \) is the map range (calculated as mass flow difference between the near-choke and the near surge point)
- \( E_p \) is the peak of efficiency,
- \( E_s \) is the efficiency at the surge point,
- \( E_c \) is the efficiency at the choke point.

The calculated merit function can take into account the extension of the map and the increasing of efficiency at the same time.

3. Results

In the plots figures are plotted the 3D-CFD results in terms of compressor maps and Speed Line Efficiency Factor. The co-rotating swirl maximize the efficiency at all the swirl angle values simulated, but decreasing the mass flow rate: this lead to the highest value of Speed Line Efficiency Factor. The counter-rotating swirl can maximize the mass flow rate and the pressure-ratio values, but with a lack in terms of efficiency. The counter-rotating flow with low swirl angle (6°) value represent a good trade-
off between the efficiency and the extension of the compressor map (same SLEF of the single swirl co-rotating with low swirl angle). The twin swirl leads to a mass-flow-rate/pressure-ratio map very close to the baseline but with a loss of efficiency growing with the increasing of the swirl angle.

In the plot below it is possible to see the compressor map, the efficiency map for each imposed swirl type and the Speed Line Efficiency Factor calculated for each speed-line simulated.
3.1. Engine impact evaluation
A complete compressor map obtained from the optimal swirl concept simulation has been put in a GT-Power 1D model in order to understand the impact on the engine performance as an additional verification.

Typically, 1D Diesel performance simulation are run with compressor boost and brake torque fixed with the aim of have the same combustion condition. Thus, the comparable output are the air-fuel-ratio and the brake specific fuel consumption (BSFC) at the full-load condition.

The result is that a co-rotating swirl at the compressor inlet with a high swirl angle value (10°) in our case could lead to a reduction of BSFC up to 1%.
Figure 9. Compressor map comparison.

Figure 10. Efficiency map comparison.
4. Conclusion
In the compressor inlet design for automotive engine, there is the necessity to create a correlation of the possible coherent motions with the performance of the compressor and the engine itself. This paper presents a case of study of optimization of the induced swirl in order to maximize the overall performance. The merit function shows that the co-rotating swirl can increase the compressor performance with positive effect on the engine efficiency and the behavior rise increasing the swirl angle value.

The limitation of this study should be taken into account evaluating the impact of swirl margin. Surge is typically an oscillating phenomenon and the steady state analysis cannot give an accurate forecast of the complex behavior. Nevertheless this work can be a starting point to transient analysis of the impact of coherent motions on compressor surge margin.

References
[1] Galindo J, Tiseira A, Navarro R, Tari D and Meano C, 2017, Effect of the inlet geometry on performance, surge margin and noise emission of an automotive turbocharger compressor, Applied Thermal Engineering, Elsevier
[2] Kyrtatos N and Watson N, 1980. Application of aerodynamically induced prewhirl to small turbocharger compressor, J. Eng. Power, ASME
[3] Nikpour B, 2004, Turbocharger compressor flow range improvement for further heavy duty diesel engines. Proceedings of Thiesel - 2004 Conference. Thermo and Fluid Dynamic Processes in Diesel Engines
[4] Bouldin B and Sheoran Y, Inlet Flow Angularity Descriptors Proposed for Use With Gas Turbine Engines, 2002 SAE Technical Paper 2002-01-2919
[5] STAR-CCM+ version 10.06, User Guide, Cd-Adapco Siemens
[6] Versteeg H K and Malalasekera W, 2007, An Introduction to Computational Fluid Dynamics, Second edition, Pearson, 58-62
[7] Petit O and Nilsson H, Numerical investigations of unsteady flow in a centrifugal pump with a vaned diffuser, International Journal of Rotating Machinery 2013
[8] SAE S-16 Committee, ARP 1420, 1978 Gas Turbine Inlet Flow Distortion Guidelines, Society of Automotive Engineers
[9] Galindo J, Serrano J R ; Margot X, Tiseira A. Schorn N and Kindl H, 2007, Potential of flow pre-whirl at the compressor inlet of automotive engine turbochargers to enlarge surge margin and overcome packaging limitations, International Journal of heat and fluid flow, Elsevier