On the effects of pulsating flow in turbocharger turbine

Roberto Mosca

rmosca@kth.se

KTH CCGEX

Two main phenomena characterize the flow field inside a turbocharger turbine: 1) an intensive heat transfer from the turbine (hot side) to the compressor (cold side) and the surrounding due to the high temperature of the exhaust gases 2) pulsating flow due to the periodic movement of the valves to which the turbine is directly connected through a series of bend pipes (exhaust manifolds).

The project aims at understanding of the mutual interaction of the two phenomena and the effects of different geometrical designs of the hot side as a system.

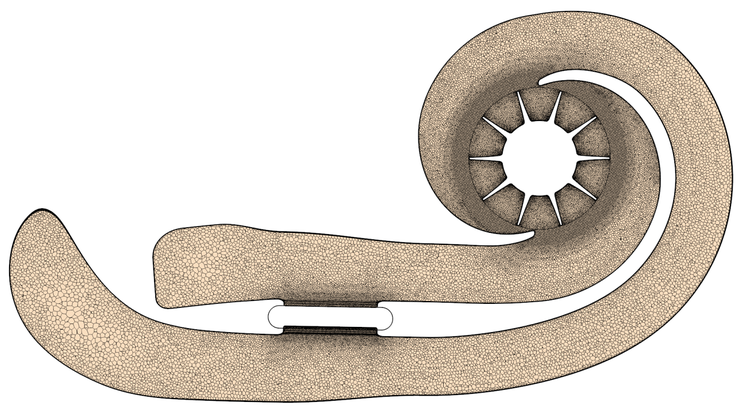
Introduction and Motivation:

On 17th April 2019, the European Parliament and Council stated the new CO_2 emission performance standards for passengers cars. The target of reducing the emissions by 15% for 2925 and 37.5% for 2030 (with reference to 2021) respectively, has the aim to inspire automotive companies in researching and developing new solutions to fulfill the required standards.

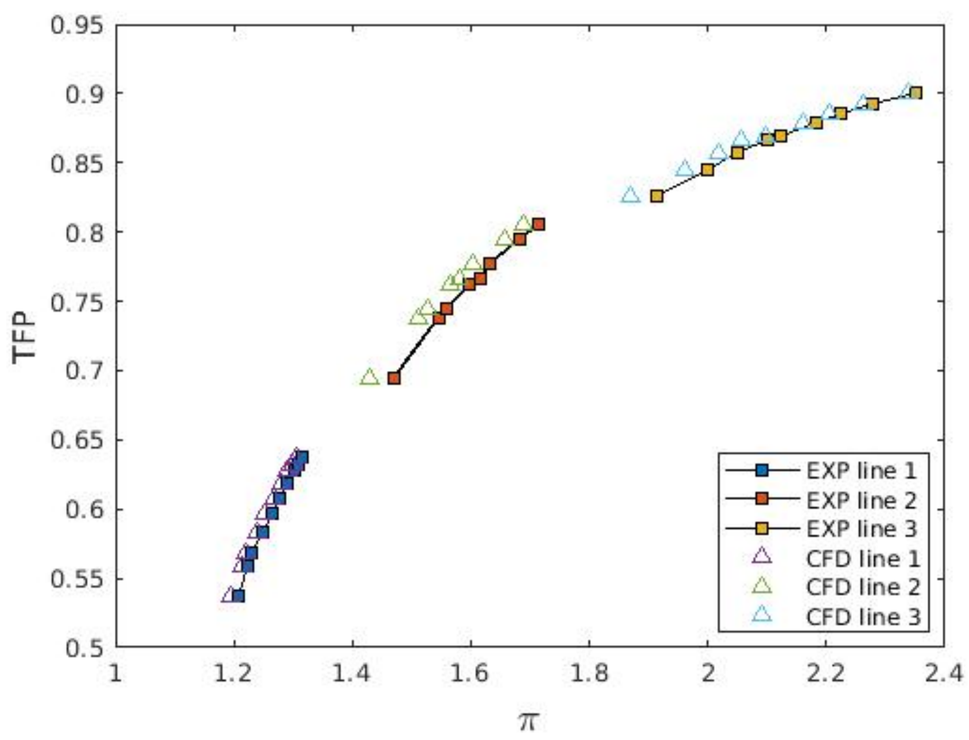
Despite the increasing interest in hybrid and electric transportation, internal combustion engines (ICEs) still represent the 95% of the recirculating vehicles, so that big are the efforts in finding new technologies to break down the emission produced. Among them, turbocharging has been receiving a lot of interest both from industry and academia and it has been demonstrated n effective solution in conjunction with the so-called engine downsizing. A deeper understanding of the flow mechanism inside turbocharger and the development of new design techniques are crucial to improve the efficiency of the system and reduce emissions.

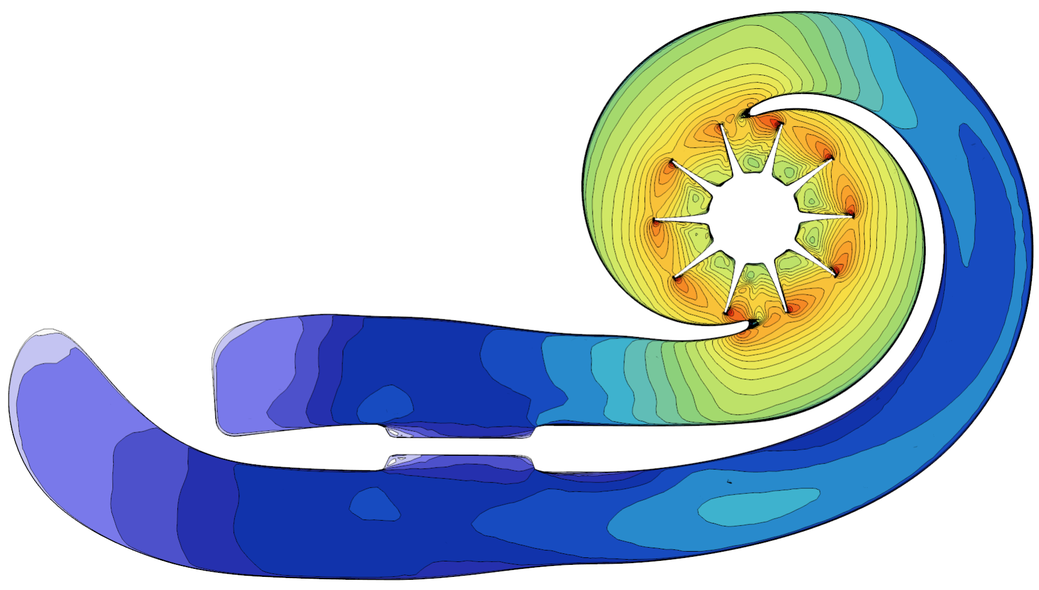
Setup:

- ❑Geometry: dual volute turbocharger turbine with VCV and wastegate open
- ❑Methodology: RANS, URANS and DES with k-ω SST turbulence model
- ❑Mesh: polyhedral mesh given by 2, 5, 10, 13 millions cells



Results:





Summary and Conclusion:

The complete performance map calculation has been performed for a dual volute turbine configuration under both full and partial admission load in continuous flow scenario.

A steady-state simulation has been performed to cover the performance map, where the motion of the rotor has been simulated through a multi reference frame (MRF) technique on a mesh composed by 3 millions polyhedral elements. The set up was able to correlate the experimental data with an accuracy of 99%.

Acknowledgement: