DTU Library

Adjoint Optimisation of the Turbulent Flow in an Annular Diffuser

Gotfredsen, Erik; Agular Knudsen, Christian; Kunoy, Jens Dahl; Meyer, Knud Erik; Walther, Jens Honore

Published in:
Proceedings of the 30th Nordic Seminar on Computational Mechanics (NSCM-30)

Publication date:
2017

Document Version
Peer reviewed version

30th Nordic Seminar on Computational Mechanics
NSCM-30
J Høsberg, N.L. Pedersen (Eds.)
2017

ADJOINT OPTIMISATION OF THE TURBULENT FLOW IN AN ANNULAR DIFFUSER

ERIK GOTFREDSEN*, CHRISTIAN AGULAR KNUDSEN*, JENS DAHL KUNOY‡, KNUD ERIK MEYER* AND JENS HONORE WALTHER*†

*Department of Mechanical Engineering, Technical University of Denmark, Building 403, DK-2800 Kgs. Lyngby, Denmark

†Computational Science and Engineering Laboratory, ETH Zürich, Clausiusstrasse 33, CH-8092 Zürich, Switzerland

‡ MAN Diesel & Turbo, Teglholmsgade 41, DK-2450 Copenhagen SV, Denmark

Key words: Fluid Mechanics, Optimisation, Sudden Expansion, Adjoint Solver, CFD.

Summary. In the present study, a numerical optimisation of guide vanes in an annular diffuser, is performed. The optimisation is preformed for the purpose of improving the following two parameters simultaneously; the first parameter is the uniformity perpendicular to the flow direction, a 1/3 diameter downstream of the expansion. The second parameter is the pressure loss introduced by these guide vanes. The optimisation yields an improvement of the uniformity of 1.5% and a 28% reduction in the over all pressure loss.

1 INTRODUCTION

The flow in an axisymmetric expansion (circular diffusor) is used in many different engineering applications, such as heat exchangers, catalytic converters and filters. These applications require a relatively uniform flow just after the expansion. To minimise the pressure loss in the expansion, an ideal solution would be to use a quite long expansion, but this is often not possible due to space restrictions. Therefore, a short expansion combined with e.g. guide vanes is often used potentially leading to an inhomogeneous flow distribution. The present study will use a Selective Catalytic Reduction (SCR) system for large marine diesel engines as a test case. The catalyst is designed for a specific flow rate at the inlet. A non-uniform inflow to the catalyst will severely reduce the efficiency of the process. The SCR system is placed on the high-pressure side of the turbocharger and in order to maintain the efficiency of the engine, the pressure losses has to be minimised. The present study analyses the flow using the commercial computational fluid dynamics (CFD) software STAR-CCM+[1] and the geometry with guide vanes in the inlet pipe[2] is optimised using the adjoint capabilities of the software.

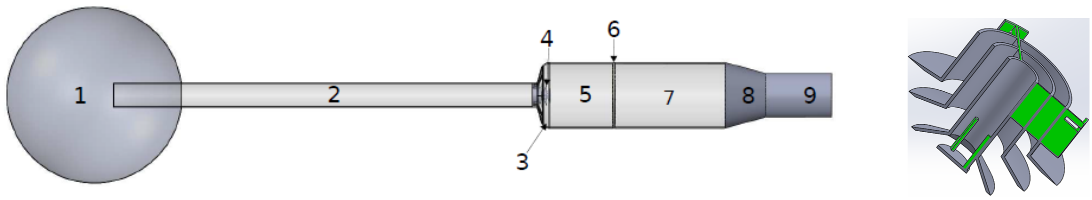

Figure 1: Left: Geometry of the model[4], where the numbers refer to: 1) inlet surface 2) straight pipe 3) expansion 4) guide vanes 5) upstream reactor 6) catalytic element 7) downstream reactor 8) diffuser 9) outlet straight. Right: A cross section of the experimental guide vanes where the connection plates is highlighted in green.

The uniformity is based on the velocity spatial flow variation in sub parts of the area weighted with the total area and mean velocity.

2 MODEL

A three-dimensional CFD model is created and shown in figure 1. The geometrical model corresponds to the experimental model described in the technical report[3]. An hemisphere is added to the inlet pipe to represent the laboratory facilities regrading the inflow. The catalytic element is modelled as a porous element. The physics in the CFD model consisted of a flow solver where the turbulence is modelled with Reynolds-Average Naiver-Stokes (RANS) equations with a realizable $k - \varepsilon$ turbulent model with two layer all y^+ wall treatment. The no-slip condition is imposed at solid walls and the flow is imposed by applying a mass flow inlet and pressure outlet. The models are chosen such that the adjoint method can be applied on the guide vanes.

3 MODEL VALIDATION

The CFD models are validated with experimental results obtained from a downscaled experimental model[3] of the catalytic converter. The Reynolds number is 10^5 upstream the expansion. The experiments are performed at laboratory conditions, with lower pressure, temperature and velocity than the full-scale catalytic converter. The results consist of different velocity planes obtained with Particle Image Velocimetry (PIV) and velocity along different lines obtained with Laser Doppler Anemometry (LDA). A measured and simulated out-of-plane velocity field upstream the catalytic element are shown in figure 2. The tendencies from the numerical results of the flow show high velocities near the reactor wall, while a triangular shaped backwards flow appears in the center. This is in good agreement with the experimental results A difference can be seen radially outwards from the connection plates for the guide vanes. From the simulation, it is seen that vortices are created at the connection plates. The impact of these vortices could be decreased but not removed by refining the mesh around the connection plates[4].

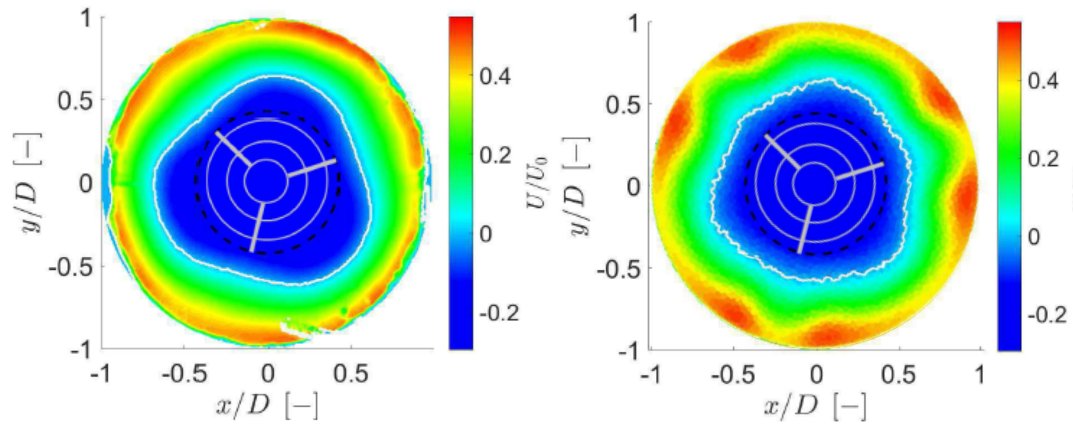

Figure 2: Visual comparison of the experimental and numerical results[4]. Left: experimental, right: simulated. The white line indicated 0. A cross section of the guide vanes is projected down on the measuring plane and showed as gray.

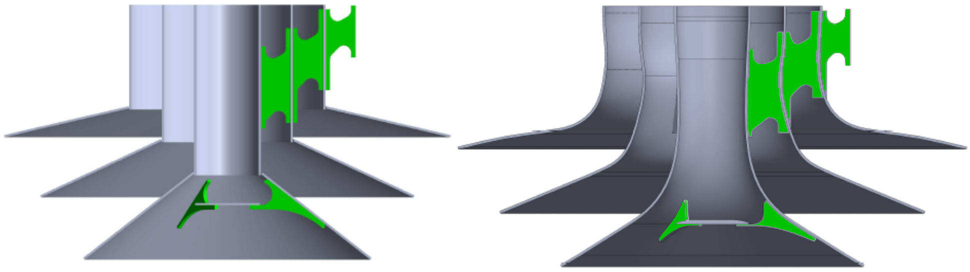

Figure 3: The guide vanes[4]. Left original, right optimised, connection plates are highlighted in green.

4 OPTIMISATION OF GUIDE VANES

The validated numerical model is applied on an updated geometry, where the catalytic element is moved closer to the expansion. Furthermore is the guide vanes changed to the orginal design[2]. This geometrical update is done to achieve a more realistic down-scale model of the SCR-system. The calculation time is reduced by removing the inlet hemisphere. The connection plates are highlighted in green and shown in figure 1 and 3. The connection plates are not optimised an remain unchanged. The experimental guide vanes is thicker and larger than the two, shown in figure 3. The adjoint method is used to see tendencies for the optimisation and then manually applied the changes to the design, in order obtain a geometry that can be produced without excessive costs. The result in figure 3 indicates that the pressure loss and uniformity are improved by changing the cross sectional area and smoothing out the sharp bends. A velocity field for both cases are showed in figure 4. This indicate a small change in the uniformity, but as stated a pressure drop is observe. The reduction in pressure loss is 28%.

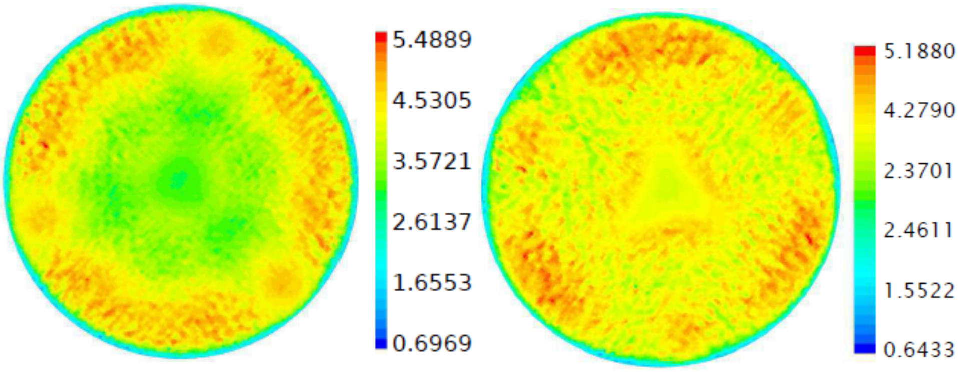

Figure 4: Colours indicate axial velocity field before the catalyc element. Left: the original guide vanes with a pressure drop of 316 Pa. Right the optimised guide vanes with a pressure loss of 226 Pa[4].

5 CONCLUSION

The present study shows that it is possible to achieve a uniform velocity distribution of $\gamma > 0.95$, just 1/3 large diameter downstream of the sudden expansion. It shows also that the initial design of guide vanes could be improved, such as the pressure loss reduces by 28% for the entire system.

REFERENCES

[1] *STAR-CCM+ Documentation,* (CD-Adapco, 2015).

[2] Kunoy, J. D. SCR WP-05: Guide vanes in SCR reactor inlet. Confidential internal MAN Turbo and Diesel report.

[3] Gotfredsen, E. & Meyer, K. E. Measurements in a model of an SCR system for large ships. Tech. Rep., Technical University of Denmark (2017).

[4] Knudsen, C. A. *Optimization of an Annular Diffuser*. Master's thesis, Tecnical University of Danmark (2017).