Faculty Engineering Technology
Program Sustainable Energy Technology

UNIVERSITY OF TWENTE.

COLLOQUIUM

Group: **Engineering Fluid Dynamics**

As part of his MSc thesis assignment

Ni Wang

will give a presentation, entitled:

Automatic Adjoint Optimization of a Venturi Mixer

Date: Tuesday August 18, 2015

Time: 14:00

Room: Horst Building NH.115

Summary:
Computational Fluid Dynamics (CFD) is used for prediction of the flow field, but CFD can also be used for optimization of the flow field with respect to design inputs. This shape optimization can be achieved based on CFD in order to obtain the optimum value of performance variables which are of engineering interests.

The domestic wall-mounted boiler department of Bosch Thermotechnology, Deventer wants to reduce the pressure loss of the venturi mixer by shape optimization. This in order to increase the ratio of maximum power output and minimum power output (modulation range) of the wall-mounted boiler. The optimization method used is an adjoint method, because of its parameter-free optimization and low computational cost. The adjoint solver of ANSYS Fluent has been used in this study.

The first part of the study focused on the venturi geometry without gas injection. A validation study involving four turbulence models has been carried out first, by comparing results with results from literature. These models are: k-ω SST model, low-Reynolds k-ε model, Spalart-Allmaras model and realizable k-ε model. The focus has been on their abilities in predicting the flow separation point and the recirculation zone. It turned out that the k-ω SST model performs the best in the comparison with results from literature.
Then a parameter study of the diffuser and nozzle angle has been carried out, including a comparison with results from literature. A design chart in the form of a contour plot of the pressure losses as function of diffuser angle and nozzle angle has been produced. Finally, an adjoint optimization has been conducted and a substantial pressure loss reduction has been achieved.

The second part of the study focused on the venturi geometry with gas injection. Two basic types of venturi mixers used by Bosch have been investigated. First a parameter study of the gas injection pipe has been carried out. The investigated parameters were gas injection pipe radius and the injection angle. Then, an adjoint optimization of an existing venturi has been pursued. This has resulted in a substantial pressure loss reduction. Based on the reduction of the pressure loss, an estimation of the increase in modulation range has been determined.

In conclusion, the adjoint method is a powerful method for parameter-free shape optimization. However, there are obstacles to conduct the shape optimization using the ANSYS Fluent adjoint solver stand alone. The two main obstacles are lack of robustness in convergence and the difficulty to export the optimized geometry for further processing. External applications like RBF Morph or the adjoint solver in other CFD codes like Openfoam can be investigated in the future.

Assessment committee: **Chairman:**

Prof.dr.ir. C.H. Venner	(chairman)
Prof.dr.ir. H.W.M. Hoeijmakers	(mentor)
Dr.ir. N. Beishuizen	(mentor)
Dr.ir. D. de Kleine	
Dr. G.G.M. Stoffels	
Dr.ir. S.H. Jongsma	d.d.