NUMERICAL RANS SIMULATION OF CAVITATION IN THROTTLES - APPROACHES AND FIRST RESULTS -

M. Fuchs[1], M. von Dirke[2], M. Macdonald[3], W. Waidmann[1]*

[1]Aalen University of Applied Sciences, Aalen, Beethovenstrasse, 73430, Germany.
[2]L'Orange GmbH, Stuttgart, Porschestrasse, 70435, Germany.
[3]Glasgow Caledonian University, Glasgow, Cowcaddens Road, G4 0BA, Scotland, UK.

* *Martin.Fuchs@htw-aalen.de*

ABSTRACT

This paper provides an insight into the latest findings of a research project aiming to obtain a reliable and resource-friendly prediction of cavitation in fuel injections systems via the use of Computational Fluid Dynamics (CFD) software.

Modern diesel injection systems are characterised by fast opening and closing valves and very high pressure drops of up to 2000 bar. Due to the geometric design or rapidly changing boundary conditions, the pressure drops locally below the steam pressure which induces cavitation [1]. In regions of higher pressure values, steam condenses and the collapsing steam bubbles can erode metals over a period of time. As a result of this, the cavitation may finally cause failure of the hydraulic components [2].

The research is split into different phases, where the first phase deals extensively with preliminary investigations. Therefore, the behaviour of the Ansys CFX standard cavitation model is investigated on throttles as well as on fuel injectors. In this context, important discoveries were made and are presented in this paper. Due to their great influence the results show the significance of such preliminary examinations. They are highly recommended when starting a new simulation to avoid and distinguish bewteen numerical and geometrical errors. In this paper, attention is mainly paid to the influence of the geometry of the investigated part.

NOMENCLATURE

Symbols

L	=	Length	[mm]
dp	=	Pressure difference	[Pa]
$\dot{m}$	=	Mass flow rate	[kg s^{-1}]
p	=	Pressure	[Pa]
p_{vap}	=	Vapour Pressure	[Pa]
Δp	=	Pressure difference	[Pa]
t	=	Time	[s]
T	=	Temperature	[K]
u	=	Velocity	[m s^{-1}]
α	=	Angle	[°]
μ	=	Viscosity	[Pa s]
μ_{turb}	=	Turbulence Viscosity	[Pa s]
ρ	=	Density	[kg m^{-3}]
$Ø$	=	Diameter	[mm]

Abbreviations

2D	=	Two dimensional
3D	=	Three dimensional
CAD	=	Computer-Aided-Design
CFD	=	Computational Fluid Dynamics
CFX	=	Commercial CFD Program
CT	=	Computer Tomography
CTP	=	Cavitation Transition Point
ICEM CFD	=	Ansys Meshing Software
RANS	=	Reynolds-Averaged-Navier-Stokes
Re	=	Reynolds Number
RPE	=	Rayleigh-Plesset Equitation
RT-1	=	Throttle II
sqrt	=	Square Root
SST	=	Shear Stress Transport Turbulence Model
ST-1	=	Throttle I - 1° Section
ST-1 360°	=	Throttle I - Full Geometry (360°)
Voxel	=	Volumetric Pixel

Subscripts

in	=	Inlet
out	=	Outlet

1. INTRODUCTION AND MOTIVATION

To perform a reliable CFD simulation, detailed and structured preliminary examinations are advisable. This paper presents selected examinations which were observed within the investigation of cavitation in fuel injection systems. Due to their great influence, the results show the significance of such preliminary examinations. They are highly recommended when starting a new simulation to avoid and distinguish numerical and geometrical errors. In this paper, attention is mainly paid to the influence of the geometry.

2. NUMERICAL METHOD AND MATHEMATICAL DESCRIPTION

To discretise the three-dimensional Navier-Stokes equation, the Ansys CFX solver (Release 13) is used. As the numerical background is not the primary focus of this study, references are made to literature, e.g. Ferziger et al [3]. The CFX solver is a pressure-based, implicit, unstructured solver [10]. Most of the calculations were made steady-state, whereas transient simulations were also performed. The structured meshes (hexagons only) were created with ICEM CFD. Due to a Reynolds Number (Re) of greater than 10E4, a turbulent flow is assumed and therefore a turbulence model is required. The goal of any turbulence model is to provide a method for calculating the influence of turbulence fluctuations on the mean flow field. For this study, the SST turbulence model is used [5], [6].

Cavitation is realised by using a two-phase flow approach. The interphase mass transfer is described by a simplified Rayleigh-Plesset equation (RPE). Thereby an incompressible liquid with constant density ρ and molecular viscosity μ is assumed. The turbulent effects are described in the framework of the turbulent viscosity approach (μ_{turb}) Gravity is neglected, because this effect is very small in comparison to the kinetic energy of the fluid [1].

3. EXPERIMENTAL SETUP AND PROCEDURE

To investigate the behaviour of cavitation in a restricted fluid flow, the behaviour of two different throttles is presented within this paper.

3.1. Throttle I

For the examination of the numerical errors, measurement data is not required. Instead, the deviation caused by the variation of parameters will be investigated.

Depending on their geometry, throttles can be treated as two dimensional (2D), which leads to more resource friendly calculations. The flow through the throttle ST-I is assumed to be planar and symmetric about a middle plane, thus a resource efficient calculation should be realisable by using a 2D calculation.

Since the CFX solver has only a three-dimensional (3D) code, an appropriate 3D mesh is needed. To gain the performance benefit of 2D simulations only a cut section of the throttle with an angle of $\alpha = 1°$ as shown in figure 1 is used. The thickness of the 3D mesh is reduced to one rotatory layer consisting out of hexahedral elements. The mesh treatment can be called "quasi 2D".

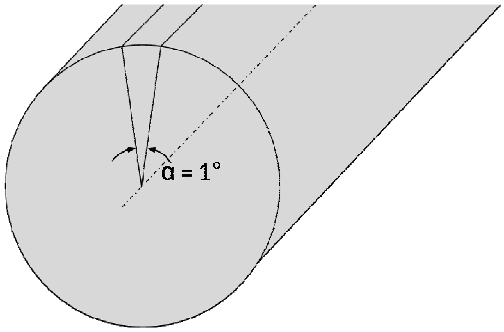

Figure 1: Cut section of Throttle "ST-I-360°"

The quasi-2D approach will be confirmed by three dimensional calculations with the circular throttle ST-I-360°. The size of the test throttle is based on the investigations of Schmitt [4]. The sharp-edged throttle used has a ratio of the restriction diameter with respect to the restriction length of 2:1 as shown in Figure 2.

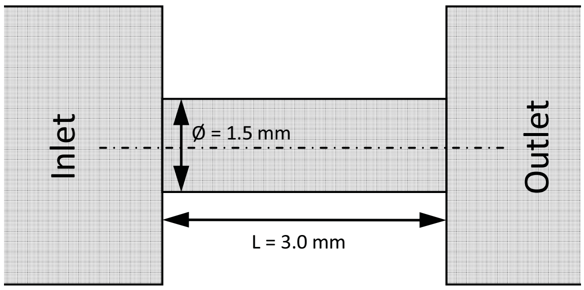

Figure 2: Throttle "ST-I"

Within the dissertation of Schmitt an empirical equation describing a flow through a throttle was formulated. These empirical measurements and the theoretical formula can be used for further examinations.

3.2. Throttle II

Moreover, measurements of a throttle used in a fuel injector are available. This throttle, called RT-I, is non symmetric and has to be treated fully 3D. The restriction diameter is less than 400 microns. Due to confidential reasons, this geometry can not be published in detail.

To gain a constant mass flow rate over time, the edge of the restriction is rounded by doing abrasive flow machining. By the use of this technique the geometry is changing, inter alia in the restriction region. When doing CFD simulations normally only the CAD geometry is available.

To investigate the influence of the change in geometry caused by abrasive machining, a simulation with the scanned geometry of a machined throttle was carried out.

3.3. Validation

The numerical simulations are validated by computing the flow through a throttle. The mass flow rate $\dot{m}$ is the observed quantity of interest. The inlet pressure p_{in} is kept constant, whereas the outlet pressure p_{out} is modified in order to account for different points of operation.

Choked flow, a limiting condition on the liquid flow through a restriction, occurs when the Venturi effect decreases the liquid pressure p to below that of the liquid vapour pressure p_{vap}. Then the liquid partially flashes into bubbles of vapour. The bubble formation in the restriction limits the flow from increasing any further. A decrease of the outlet pressure does not increase the mass flow rate.

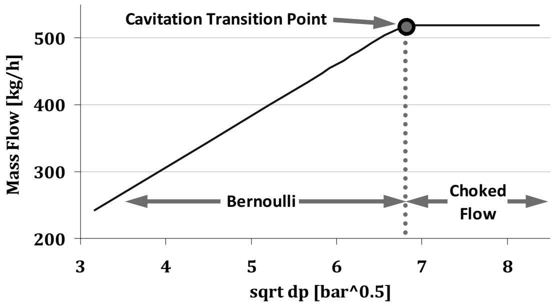

Figure 3: Mass flow curve

In Figure 3 the mass flow rate is shown as a function of the square root of the pressure difference $\Delta p = p_{in} - p_{out}$. At the inlet, the pressure $p_{in} = 100\ [bar]$ and the Temperature $T_{in} = 313\ [K]$ are fixed. The outlet pressure is varied in the range of $p_{out} = 100\ [bar]$ to $p_{out} = 20\ [bar]$. The graph can be split into two areas. First, a linear part with a positive slope following Bernoulli's Principle can be observed. Then the mass flow rate reaches a constant maximum which lasts even when decreasing the outlet pressure and is called choked flow.

4. RESULTS AND DISCUSSION

4.1. Influence of the Geometry

Within this paper the focus is on the influence of the variance between CAD geometry and the geometry of a real component. Therefore a industrial Computed Tomography (CT) of a throttle used in a fuel injector was carried out.

4.1.1. Industrial Computed Tomography Scanning

Industrial Computed Tomography is a method using X-rays to produce three-dimensional representations of components. It can be used in many areas of industry for both internal and external mappings of parts and assemblies. The main advantage of CT scanning is the non-destructive behaviour. Some of the key uses for Industrial CT are failure analysis, assembly analysis, metrology and reverse engineering applications.

Figure 4 depicts schematically the working method of an industrial CT. For the scan the part is placed on a rotary table between the X-ray source and a detector. As the part rotates the cone of X-rays produces 2D images which are collected by the detector. The 2D images are then processed to create a 3D Volume rendering of the geometry of the part.

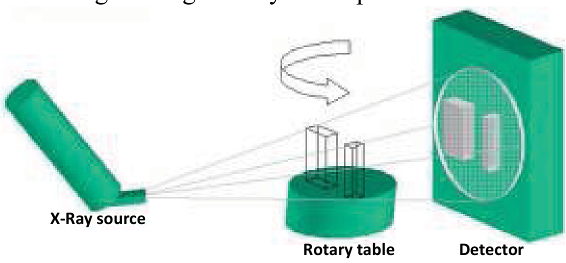

Figure 4: Working method of an industrial CT

The point cloud, consisting out of voxels (volumetric pixel), are then processed to create a usable digital model, using specialised reverse engineering software The resulting CAD file can be used for the flow simulations.

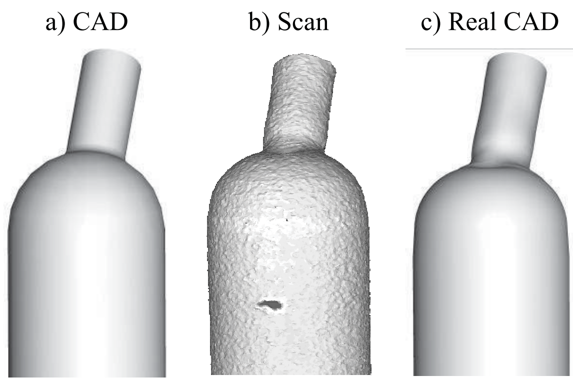

Figure 5: Different stages of the geometry of RT-I

Figure 5 shows the different stages of the geometry. First the CAD part was available (a), then a scan of the throttle was made (b) and after the reverse engineering, a CAD model of the real geometry (c) is generated.

4.1.2. Results of the CT-Scan

After the point cloud is reverse engineered, the deviation of the gained CAD model in respect to the point cloud has to be checked. For the RF-I throttle, the maximum deviation is 15 micron at an uncritical spot in the inlet area. In the restriction area the deviation is less than 10 microns, whereas the CT has a resolution of about 8 microns.

4.1.3. Results of the Flow Measurements

As described above, the mass flow rate can be used to observe the quantity of interest, in this case the influence of the geometry. In Figure 6 the calculated mass flow rate and some measurement data is shown. The pressure difference was varied between *0 [bar]* and *90 [bar]*, while the inlet pressure was kept constant at p_{in} = *100 [bar]*.

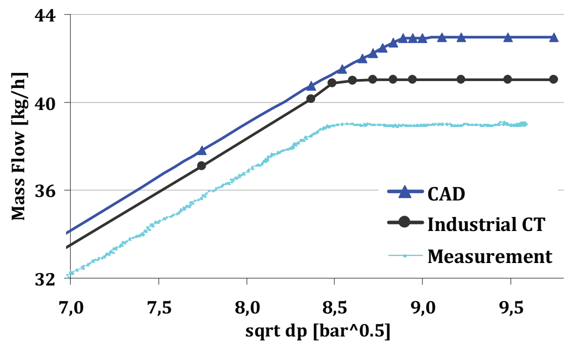

Figure 6: Mass Flow Measurements of RT-I

As can be seen in Figure 6, the choked mass flow rate of the CAD-geometry is more than 10% over the measured mass flow rate. Using the scanned geometry, the variation is halved and the value of the pressure difference at the Cavitation Transition Point (CTP) matches very well. However, the choked mass flow rate is still 5% over the measured mass flow rate. This variation is still unsatisfactory, and further investigations have to be carried out.

4.1.4. Outlook

As mentioned above, further approaches have to be investigated to get a more exact solution. As a next step, a mesh refinement, according to the best practice guidelines, e.g. [7], [8], [9] will be carried out, as well as simulations with transient boundary conditions.

4.2. Shape of the Outlet

Starting with a simulation of the RT-I throttle, the solution was not converging at high pressure drops. The post-processed results showed that the whole outlet area was filled with vapour as shown in Figure 7. In addition, a backflow of the vapour fraction was present at the outlet.

Figure 7: Volume fraction of diesel vapour (result of a not converged simulation)

A first approach to gain convergence was the extension of the outlet, giving the fluid the opportunity to settle down and condense. However no improvement was gained with this approach.

Next a nozzle was added, to accelerate the flow, which should eliminate the backflow. But this modification also did not fulfill the expectations. Figure 8 shows the geometric modifications at the outlet area.

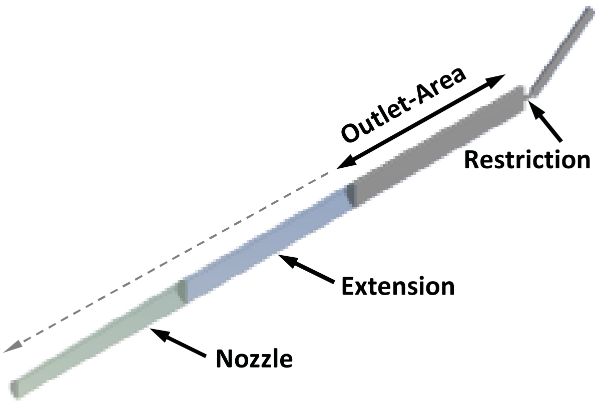

Figure 8: Modifications of the outlet-area

So far, the outlet geometry as the cause for the poor solution could be excluded. As mentioned above, the phenomena of the totally cavitating outlet area was only at high pressure drops. It was found, that the initialisation with the high pressure drop, brought the solver into an unsteady operating condition from where it could not reach convergence any more.

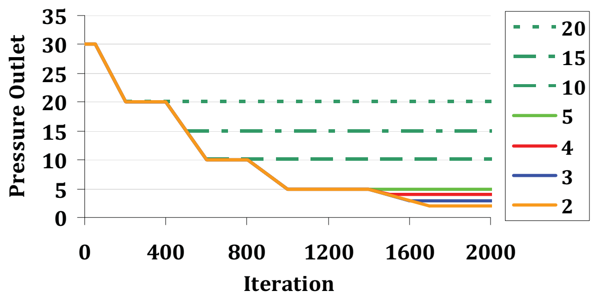

Figure 9: Pressure-Ramps

Since the solver is not able to reach convergence when starting at a high pressure drop, a so called pressure-ramp (see Figure 9) was defined and implemented as a user function. That way, the pressure can be lowered over time. The advantage of this approach is that the simulation can be initialised with a converged low pressure drop solution. Then, the pressure will be lowered gradually over time (or iteration number that corresponds to time). In addition, platoons with a constant pressure value are implemented. This provides additional stability and an output file can be written at the end of each platoon (when a fully converged solution is reached) for further examinations.

4.3. Cut Section vs. full Geometry

When investigating the influence of several coefficients or serious program manipulation, a resource friendly control case is needed. Therefore the ST-I throttle is used.

As described above, the whole throttle is not used for calculation, but only a slice with an angle of $\alpha = 1°$. This cut section is defined rotationally symmetric and has a thickness of one element layer with periodic nodes. The cut planes were defined as “symmetry” which corresponds to a slip wall condition. The section is illustrated in Figure 10.

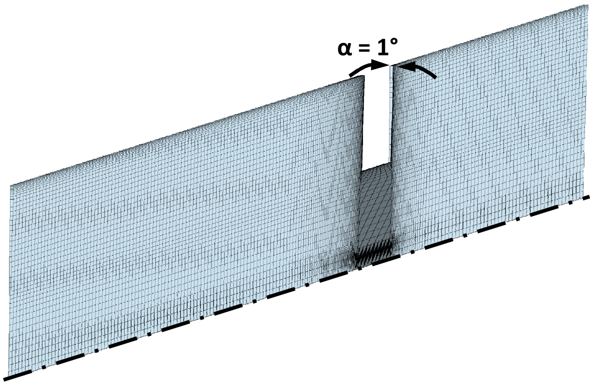

Figure 10: ST-I Throttle

The solution gained from this slice (e. g. mass flow rate) has to be multiplied by 360 to get a statement for the whole throttle.

The CFD simulation was performed stationary. Due to the convergence of the solution, transient effects can be neglected. In addition, a calculation of the whole geometry was performed and the mass flow rate was compared to that of a throttle slice. Figure 11 shows the geometry of the whole throttle ST-I-360°.

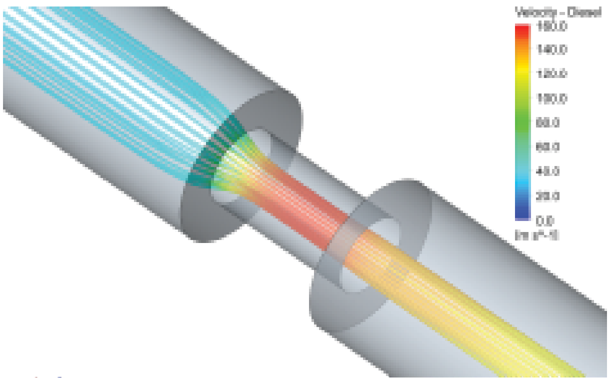

Figure 11: Streamlines in ST-I 360°

As expected the deviation in mass flow is marginal. For a defined pressure difference of $\Delta p = 60\ [bar]$ (choked flow condition) the deviation of the mass flow rate is smaller than *0.02 %* which can be treated equal to zero.

The results show, that the assumption of similarity between whole geometry and a quasi 2D treated slice is suitable and qualified.

5. CONCLUSIONS

Presented was the influence of the geometry when simulating throttled flows. It was shown, that not only numerical errors have to be taken in account, but also the influence of the variation and tolerance of the investigated part.

Depending on the manufacturing process significant variations between CAD geometry and the real geometry could be present. If so, the use of the real geometry should be taken into account. The measurements can be carried out as described in this paper.

Reaching no convergence when stating a new simulation could have many reasons; and the solution is not always recognizable at first sight. If so, structured approaches are recommended and a sensitive handling of the solver should be aimed. In the presented case, the use of a so called pressure-ramp could be helpful.

Moreover was shown, that depending on the geometry 2D simplifications are valid. Thereby a significant performance benefit is gained.

6. ACKNOWLEDGEMENTS

This study is part of a joint project with L’Orange GmbH, Glasgow Caledonian University, Ansys Inc. and the Aalen University. I would especially like to thank Dr.-Ing. Johannes Einzinger from Ansys Inc. Germany for his help and discussions related to the results presented in the paper. Special thanks also to Dr. von Dirke for the clarifying comments and discussions.

7. REFERENCES

[1] C. Brennen, Cavitation and Bubble Dynamics, Oxford University Press, 2005.

[2] W. Bauer, U. Iben and M Voß, Simulation of cavitating flow in injection systems, Numerical Analysis and Simulation in Vehicle Engineering, VDI-Berichte 1846, pp. 1029 - 1041, VDI-Verlag, 2004.

[3] J. H. Ferziger, M Perić, Computational Methods for Fluid Dynamics, Springer, Berlin, 2010

[4] T. Schmitt, Untersuchung zur stationären Strömung durch Drosselquerschnitte in Kraftstoff-einspritzsystemen von Dieselmotoren, Dissertation, Technische Universität München, 1966.

[5] F. R. Menter, Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications, AIAA Journal Vol. 32, No. 8, August 1994.

[6] F. R. Menter, Ten Years of Industrial Experience with the SST Turbulence Model, Turbulence, Heat and Mass Transfer 4, Begell House, Inc. 2003.

[7] Atkins Consultants and members of the NSC Sirehna HSVA FLOWTECH VTT Imperial College of Science & Technology Germanischer Lloyd Astilleros Espanoles, Best Practice guidelines for marine applications of computational fluid dynamics.

[8] NAFEMS Ltd, (04. August 2011 16:54) http://www.nafems.org/resources/cfd_guidance/

[9] Best practice guidelines for turbomachinery CFD (20. July 2011 15:00) http://www.cfd-online.com/Wiki/Best_practice_guidelines_for_turbomachinery_CFD

[10] Ansys Inc. CFX Release 12.0 User Manual. 2009.