Numerical simulation of flow metering system for liquefied natural gas

**J. Sluse1, R. Maury2,**

**J. Gersl1, A. Strzelecki2**

*1Czech Metrology Institute, Okruzni 31, 638 00 Brno, Czech Republic*

*2CESAME Exadebit, Poitiers, France*

*E-mail (corresponding author): jsluse@cmi.cz*

# Abstract

Increasing usage of liquefied natural gas (LNG) hand in hand with the decreasing price makes this „green“ fuel accessible. Higher consumption of LNG opened gate for development of new facility for measuring of flow rate with lower uncertainty. The new equipment for measuring the LNG flow rate developed at CESAME Exadebit uses a Laser Doppler Velocimetry technique. The principle of the equipment is based on the velocity measurement in one point behind convergence nozzle and then solving the flow rate. It is convenient to create nearly flat velocity profile behind the nozzle throat (like piston velocity profile) to reduce the shear region influence on the mass flow rate calculation. The aim of this article is validation of numerical models of LNG flow in the new flow metering system in OpenFOAM software by comparison with experimental data and analytic formulas. This is the first step for the nozzle shape optimisation.

The numerical simulation study is divided into following parts:

• The first part is dedicated to the flow through a long straight pipe and a standard Venturi tube (ISO 5167-4) [2]. Mentioned cases are simplified version of the LDV package system. In these cases some analytical and experimental results are known [1]. Mesh convergence and choice of a turbulence model are checked for these cases. The numerical validation is realized by comparing analytical, experimental and numerical aerodynamic behaviour of the flow.  Furthermore, the outlet solution obtained for the straight pipe case is used as inlet boundary condition for the LDV package system.

• In the second part the knowledge acquired from the previous calculations is used for the LDV package system (meshes, turbulence model and boundary conditions). Then the numerical simulation of the LDV package is compared with experimental data.

# 1. Introduction

The consumers of LNG want to use this fuel in location of their residence, not where the natural gas is extracted or liquefied. This requirement can be met by transportation of LNG. The transportation is provided by tanker ships and road tankers. Loading and unloading needs to be very fast and transported volume has to be measured with small uncertainty. These factors force manufactures of flowmeters to invest money into research. The flowmeters need to have satisfactory range of flow rate (saving time) and satisfactory accuracy (saving money). The developed device which is subject of this article fulfils these two conditions and will be used as a standard for calibration of flow meters.

The principle of the equipment is based on the velocity measurement in one point behind convergence nozzle and then solving the flow rate. It is convenient to create nearly flat velocity profile behind the nozzle throat (like piston velocity profile) to reduce the shear region influence on the mass flow rate calculation.

The aim of this article is first to simulate flow through simplified geometries to determine optimal settings for the simulation of the full LNG flowmeter mock-up which is developed by CESAME Exadebit. After that we simulate the flow in the mock-up itself. The result will be used for optimization of the nozzle shape of the mock-up.

This research is a part of a bigger project “Metrological support for LNG custody transfer and transport fuel applications” (LNG II) within the European Metrology Research Programme (EMRP).

# 2. Long straight pipe

*2.1 Introduction and geometry*

The long pipe was chosen because it is a canonical case where results are well known and the article continues step by step to increase the complexity. The aim of this part was to simulate the flow through the pipe and get a fully developed velocity profile. The results were used for choosing of the optimum mesh and also for choosing of the optimum turbulence model for other numerical simulations. The following turbulence models are tested: *k–*, realizable *k–*, *k- SST* and Reynolds Stress Model. The results were compared with experimental data from Princeton University (Zagarola and Smits) [1] and with analytical results [1].

The pipe is 12 meters long and diameter of the pipe is 0.08 m. The length was chosen to get fully developed profile at the end of the pipe.

*2.2 The mesh, boundary conditions, solver*

Meshes have been realized using a capability of OpenFOAM called blockMesh. The setup is more difficult than in commercial programs but the mesh is fully in the user’s hands. All the meshes included coarse part in the core of the stream and refinement in the direction to the wall. Table 1 below contains details about the meshes and one example of the mesh is shown in Figure 1. Six kinds of meshes were tested - one was 2D and rest were 3D. The meshes were created with another decomposition and numbers of cells. One after another was tested and the results were used to prepare the next one.

The meshes were created as a core and continuous layers. The cross section of the core was created as a square where vertices weren't connected by lines but by arcs.



Figure 1: The mesh structure of the straight pipe.

**Table 1:** Mesh property.

|  |  |  |  |  |
| --- | --- | --- | --- | --- |
| **Mesh number** | **Cells** | **Cells in core**  | **Cells in refinement layer** | **Typical size of cell on wall** |
| 1 / 3D | 3 168 000 | 10 | 15 | 0.170 mm |
| 2 / 3D | 24 768 000 | 19 | 30 | 0.085 mm |
| 3 / 3D | 5 328 000 | 10 | 30 | 0.036 mm |
| 4 / 2D | 36 000 | 20 | 40 | 0.013 mm |
| 5 / 3D | 7 872 000 | 15 | 30 | 0.036 mm |
| 6 / 3D | 4 992 000 | 13 | 30 | 0.007 mm |

The simulation was done for three fluids (air, liquid nitrogen and LNG).

The numerical problem was solved as a steady, viscous, turbulent, incompressible flow by a *simpleFoam* solver with several models of turbulence. Converged results were those results which had residuals lower than 10-12 for velocity in *z* axis and 10-10 for velocity in *y* and *x*axis.

*2.2 Mesh convergence*

The results show the dependence of velocity (*z* component of velocity) on position. The position of the line which was used to sample the data is 11.5 m behind the inlet and it was started in the centre of pipe and lays on the positive direction of *y* axis. The vertical coordinates in the graphs in Figure 2 and 3 are given as velocities in the middle of the cells divided by maximal velocity of all cells on the line. The horizontal coordinates in the graphs are given as radiuses of the middle of cells divided by maximal radius of the pipe (0.04 m in this case).

The results for *k–ε* model are presented first. As can be seen from the Figure 2 different velocity profile occurs only in mesh number 1. This mesh is not suitable for any other calculation. Other meshes have more or less the same results and will be subjected to another testing.

Figure 2: Velocity profile for k *– ε* turbulence model.

The Figure 3 shows the results from *k-SST* turbulence model.

Figure 3: Velocity profile for *k-SST* turbulence model.

The results from numerical simulation which used turbulence model k*–SST* aremore or less the same apart from the mesh number one. From the comparison of the meshes above it follows that the meshes number 5 or 6 are sufficient for the modelling. For further computations we use the mesh number 6.

*2.3 Turbulence models*

The comparison between turbulence models was made only on the mesh number 6.

Figure 4: Velocity profile – different turbulence models.

The results (Figure 4) of all models of turbulence aren’t the same. To choose optimal model it is necessary to compare these results with analytic results or with experiment.

Nevertheless literature recommends *k–SST* turbulence model for small diameter.

*2.4 The comparison of numerical simulation and the analytic approach*

The velocity profile in a pipe can be described in two ways. First way is the pow-law approach and the second way is logarithmic approach. In this article only logarithmic approach was used which is given:

For y+ < 5 (viscous sublayer)

$$u^{+}=y^{+}$$

For 5 < y+ < 30 (buffer layer)

$$u^{+}=11.5logy^{+}- 3.05$$

For 30 < y+ (logarithmic zone)

$$u^{+}=5.75logy^{+}+ 5.5$$

With:

$$y^{+}=\frac{yu\_{τ}}{ν} u\_{τ}=\sqrt{\frac{τ\_{w}}{ρ}}$$

$$u^{+}=\frac{u}{u\_{τ}}$$

Where:

*y+* is the wall coordinate – the distance *y* to the wall, made dimensionless with the friction velocity *u* and kinematic viscosity**

*u+* is the dimensionless velocity – the velocity *u* parallel to the wall as a function of *y* (distance from the wall), divided by the friction velocity *u* ,

w is the wall shear stress,

 is the fluid density,

*u* is called the friction velocity or shear velocity.

The numerical simulation for comparison was solved with the mesh number 6 and the smallest cell near the wall is smaller than y+ = 1.

Figure 5: Dependence of velocity profile normalized by the friction velocity on the wall coordinate - comparison of various turbulence models with analytic formula.

The comparison (Figure 5) between numerical simulation and analytic formula shows agreement for the *k- SST* model which use *k-* model near the wall and *k-* model in core.

*2.5 The comparison of numerical simulation and the experimental data*

The experimental data are based on the measurement in wind tunnel at Princeton University [3]. These experiments were carried out by Zagarola and Smits. The main aim of the experimental measurement was to describe the velocity profile near the wall for more velocities (from low Reynolds number to high Reynolds number). The experiments were done at atmospheric pressure for various velocities.

The curves from experimental data in the Figure 6 show decrease of u+ near the value of y+ = 50 000. It is caused by axis symmetry where the data are from opposite part of pipe.

The turbulence model *k- SST* has a good conformity.

Figure 6: Dependence of velocity profile normalized by the friction velocity on the wall coordinate - comparison with experiment.

*2.5 The summary for the long pipe*

The aim of this part was to choose the turbulence model which has a good agreement with the experimental data (air) and the analytic formula.

Six different meshes were created and tested. The meshes differed both in number of cells and decomposition of geometry to the cells. From all meshes the mesh number 6 was chosen as the optimal one. This mesh was used for search of the optimal turbulence model. The results from the simulation were compared with the analytical formulas and the experimental data. The *k–SST* model was chosen as the optimal turbulence model giving the best agreement in the comparison. Now it is clear which refinement of the mesh is optimal and which turbulence model should be chosen. This experience will be used for the next simulations.

The next step is to solve the velocity profiles for another fluids and then to use it as initial condition for the next geometry (Venturi tube, mock-up).

In view of the above, the simulation was done for other fluids (air, liquid nitrogen, LNG) and illustrated in the Figure 7.

Figure 7: Dependence of velocity on the position for various fluids other than air.

# 3. Venturi tube

*3.1 Introduction and geometry*

This part of the article is focused on simulation of flow through a Venturi tube. The fully developed profile was prescribed on the inlet. For the Venturi tube a pressure drop can be determined between a point upstream of the convergent part of the nozzle and the middle of nozzle. From the pressure drop the CD coefficient can be calculated. This coefficient is the main parameter of the Venturi tube and it depends on type of the Venturi tube, process of manufacture etc. The CD coefficient is stated in the standard ISO 5167-4 [2].

The aim of this part was to compare the CD coefficient from the simulation with the standard ISO 5167-4 and again to test sensitivity of the results to the turbulence model and mesh parameters.

*3.2 The mesh, boundary conditions, solver*

For the generation of a computational mesh the blockMesh tool was used again. All the meshes included coarse part in the core of the stream and refinement in the direction to the wall. The refinement of the cells was created also in the throat of the nozzle. Table 2 shows the details about the mesh (Figure 8).

**Table 2:** Mesh properties.

|  |  |  |  |
| --- | --- | --- | --- |
| **Mesh number** | **Cells** | **Cells in core**  | **Cells in refinement layer** |
| 1 / 3D | 431 904 | 10 | 15 |
| 2 / 3D | 3 376 704 | 19 | 30 |
| 3 / 3D | 2 041 728 | 13 | 30 |



Figure 8: Refinement to the centre of the Venturi tube.

*3.3 Mesh convergence*

The verification of correct set up of decomposition of geometry into cells was done by turbulence model *k-*. The simulation was carried out with fully developed profile on the inlet.

The velocity profiles do not change significantly for transition between the meshes number 2 and 3 indicating that the cell density in these meshes is already sufficient. The next comparison was done between CD coefficients of each mesh. The CD coefficient is the main parameter in the ISO standard.

The formula for calculating the coefficient CD is

$$C\_{D}= \frac{\dot{m}\sqrt{1-β^{4}}}{ε}\frac{4}{πd^{2}\sqrt{2∙ρ∙∆p}}$$

Where:

*m* is a mass flow rate,

** is a ratio of nozzle diameter and inlet diameter,

** is coefficient of expansion,

*d* is diameter of the nozzle,

** is density of the fluid,

*p* is pressure difference.

**Table 3:** CD coefficient for each mesh (*k – ε* model).

|  |  |  |  |
| --- | --- | --- | --- |
|  | **Mesh 1** | **Mesh 2** | **Mesh 3** |
| p (Pa) | 1323.83 | 1321.13 | 1328.79 |
| CD (-) | 0.973 | 0.974 | 0.971 |

The results (Table 3) of CD coefficients are very similar for all meshes but they are not in compliance with CD coefficient written in the ISO standard where the value is 0.984 or 0.995 depending on the production method. The ISO standard also states unchanging CD coefficient for a range of Reynolds numbers from 200 000 to 2 000 000 for the casted Venturi tube. The next step therefore is to focus on comparison of the CD coefficients in simulations with different Reynolds numbers.

*3.4 Comparison of CD coefficient for different Reynolds numbers*

Simulation was done with the mesh number 3 and only velocity magnitude on inlet was changed. The velocity was being chosen so that the Reynolds number was 750 000 and 1 500 000.

**Table 4:** CD coefficient for different Reynolds number.

|  |  |  |
| --- | --- | --- |
|  | **Re 750 000** | **Re 1 500 000** |
| p (Pa) | 18560.00 | 76379.13 |
| CD (-) | 0.990 | 0.994 |

From the results (Table 4) for the CD coefficient it can be seen that each Reynolds number has different CD coefficient.

*3.5 Turbulence models*

As it has been already seen on the simulation with the long pipe, the turbulence model has significant influence on the result. Also in this case the turbulence model was being changed and the results compared. The simulation (Figure 9 and 10) was done only for two turbulence models namely *k-* and *k- SST*.

Figure 9: Dependence of the velocity on the position – z axis.

Figure 10: Dependence of the pressure on the position – z axis.

**Table 5:** CD coefficients for different turbulence models.

|  |  |  |
| --- | --- | --- |
|  | ***k- SST*** | **k-** |
| p (Pa) | 1289.00 | 1323.00 |
| CD (-) | 0.984 | 0.972 |

From the Table 5 it can be seen that different turbulence models give significant difference in the CD coefficient. *K- SST* model is compliant with the ISO standard for the casted Venturi tube where the CD coefficient is 0.984. The conformity was achieved.

*3.6 The summary for the Venturi tube*

The smallest mesh where mesh converged results were obtained is the mesh number 3.

From the results it is apparent that Reynolds number has an influence on CD coefficient.

Selection of turbulence model has a significant influence on CD coefficient. It is the same as in case of the simulation with the long pipe where the result from *k- SST* model is in agreement with literature. In this case the CD coefficient modeled with the *k- SST* model agrees with the CD coefficient for casted Venturi tubes in the ISO standard.

# 4. LDV package system

The LDV package system is new equipment for measuring the LNG flow rate using Laser Doppler Velocimetry technique. This system is developed by the CESAME Exadebit company in France. The principle of this equipment is based on velocity measurement in one point behind the convergence nozzle and then calculation of the flow rate.

*4.1 Introduction and geometry*

The LDV package system consists of front part (inlet part) where the medium is seeded by particles (bubbles or another spherical material). The fluid flows through the convergent nozzle to the measuring space. For measurement by LDV it is necessary to make the equipment with windows. The LDV package system (Figure 11) includes special cavities with windows on the body of the measuring system. The medium continues to the divergent part of the nozzle and goes outside. For satisfactory accuracy of measurement and low uncertainty of the measurement it is convenient to create nearly flat velocity profile behind the nozzle throat (like piston velocity profile) to reduce the shear region influence on the mass flow rate calculation.

The aim of this part of the paper is a simulation of flow through the LDV package system. The results will be compared with measured experimental data. The measurement was done for air and also for liquid nitrogen and LNG. The velocity profile from experiment is available only for air. The measurement with liquid nitrogen was done only in one measurement point.



Figure 11: The sketch of LDV package system.

*4.2 The mesh, boundary conditions, solver*

The blockMesh tool was used also for generating a structured mesh for the LDV mock-up which is more complex since it contains several T-junctions of cylindrical cavities to the main pipe (Figure 12 and 13). Strong emphasis was put on location where the nozzle connects to the measuring space. The cavity with window for LDV measurement is also connected to this space. The refinement was made in the same way as in the Venturi tube in the direction to the measuring space and near the wall. In total four main meshes were created (Table 5).

#

Figure 12: The sketch of the LDV package system.

Table 5: Mesh properties.

|  |  |  |  |  |
| --- | --- | --- | --- | --- |
| **Mesh number** | **Cells** | **Cells in core**  | **Cells in refinement layer** | **Typical size of cell on wall** |
| 1 / 3D | 857 220 | 14 | 22 | 0.6 mm |
| 2 / 3D | 1 399 686 | 14 | 30 | 0.02 mm |
| 3 / 3D | 4 049 012 | 20 | 54 | 0.02 mm |
| 4 / 3D | 7 699 848 | 40 | 108 | 0.01 mm |

*4.3 Mesh convergence*

The verification of correct decomposition of the geometry into cells was done using the turbulence model *k-SST*. The simulation was carried out with fully developed profile on the inlet.

The simulations were carried out consecutively from mesh number 1 to mesh number 4 and conformity between velocity profiles and experimental data was recorded.

From these results it was concluded that the mesh number 4 is in the best agreement with the experimental data. The results strongly depend on the density of cells in the mesh.

*4.4 Turbulence models*

The simulation was done for two turbulence models, namely *k-* and *k- SST*. The velocity profile in the measuring space (6mm behind the nozzle outlet) for air is shown on Figure 14 for both turbulence models and for experimental data.

The turbulence model has not significant influence on the result. The results from numerical simulation and from the experimental data are in satisfactory agreement in the measuring section. Some differences between results are shown in the cavity.

Figure 14: The comparison between experiment and numerical simulation – 10 m/s, 10 bars, air.

*4.5 Simulation with liquid nitrogen*

Experimental data for liquid nitrogen are available only in one point inside of the measuring section (6mm behind the nozzle outlet). Therefore it is not possible to carry out comparison of whole velocity profile. Comparison (Figure 15) between the experiment and the simulation in one point shows difference of approximately 2% in velocity on the axis in the measuring space.

Figure 15: The comparison between experiment and numerical simulation – 10 m/s, 10 bars, liquid nitrogen.

*4.6 The summary for the LDV package system*

The simulated velocity profiles for both turbulence models show good agreement with the measured velocity profile for air.

Difference between simulation and experimental data for liquid nitrogen was around 2 % in the velocity in the middle of the nozzle. The optimal mesh for the simulation of air flow is the mesh number 4 but the quality and amount of cells between the nozzle and the measuring space have more significant influence on the velocity profiles in case of the liquid nitrogen.

# 5. Conclusion

For the long straight pipe fully developed profiles for air, liquid nitrogen and LNG were obtained. The turbulence model *k- SST* has a good conformity with analytical formula and experimental data.

In case of the Venturi tube the flow through the tube was simulated and the CD coefficient was determined for different Reynolds numbers, different types of mesh and different turbulence models. For *k- SST* model the conformity was also obtained. Another important conclusion was the dependence of CD coefficient on Reynolds number.

Last and the most important part of this article is dealing with LDV package system (mock-up). The conformity between simulation and experimental data was obtained for normalized velocity profile behind the nozzle in the air. The result between simulation and experimental data for liquid nitrogen had the difference around 2 % in the velocity in the middle of the nozzle. Comparison for LNG will be done in the future because for now experimental data are not available.

The result will be used for optimization of the nozzle shape.

#  Acknowledgement

The research leading to the results discussed in this paper has received funding from the European Metrology Research Programme (EMRP). The EMRP is jointly funded by the EMRP participating countries within Euramet and the European Union.

# References

1. MCKEON, B. J., J. LI, W. JIANG, J. F. MORRISON a A. J. SMITS. “Further observations on the mean velocity distribution in fully developed pipe flow”. *Journal of Fluid Mechanics*, **501**, 135-147, 1999.
2. ISO 5167-4: *Measurement of fluid flow by means of pressure differential devices inserted in circular cross-section conduits running full -- Part 4: Venturi tubes*, 2003.