JET Volume 17 (2024) p.p. 54-61
Issue 3, 2024
Type of article: 1.04
http://www.fe.um.si/si/jet.htm

VERIFICATION AND VALIDATION OF STUDENT'S CFD TOOL ON THE VENTURI FLOW CASE

VERIFIKACIJA IN VALIDACIJA ŠTUDENTOVEGA CFD-ORODJA NA OHIŠJU VENTURIJEVEGA PRETOKA

Tihomir Mihalić[1ℜ], Ante Zaninović[1], Srđan Medić[2], Filip Krsnik[1]

Keywords: CFD, venturi, turbulence, mesh, student

Abstract

The main goal of this study is to investigate the usability of the CFD software ANSYS Student 2023 R1 for students` scientific research. To carry out this investigation, a comparison was carried out of the analytically calculated flow rates through a standard venturi tube with the results obtained by simulations of the flow in the specified program. Given that it is a student, free version of the program, which comes with certain limitations, proving its usability by performing verification and validation on known geometry would be of great benefit. To ensure comparability, a 3D model of the venturi tube was created in accordance with the ISO 5167 Standard. The flow rate was calculated on that selected geometry of the venturi tube, using the well-known equations of fluid mechanics. The selected geometry of the venturi tube was discretised in the ANSYS Student 2023 program, respecting all CFD rules. Comparison of these two sets of results showed a match within 5%.

ℜ Corresponding author: Zagreb University of Applied Sciences, Mechanical Engineering Department, Brozova 6A, 10000 Zagreb, Croatia

1 Zagreb University of Applied Sciences, Mechanical Engineering Department, 10000 Zagreb, Croatia

2 Karlovac University of Applied Sciences, Mechanical Engineering Department, I. Meštrovića 10, 47000 Karlovac

Povzetek

Glavni cilj te študije je raziskati uporabnost programske opreme CFD ANSYS Student 2023 R1 za znanstveno raziskovanje študentov. Za izvedbo te preiskave je bila izvedena primerjava analitično izračunanih pretokov skozi standardno venturijevo cev z rezultati, pridobljenimi s simulacijami pretoka v navedenem programu. Glede na to, da gre za študentsko, brezplačno različico programa, ki ima določene omejitve, bi bilo dokazovanje njegove uporabnosti s preverjanjem in validacijo na znani geometriji zelo koristno. Za zagotovitev primerljivosti je bil izdelan 3D-model venturijeve cevi v skladu s standardom ISO 5167. Hitrost pretoka je bila izračunana na izbrani geometriji venturijeve cevi z uporabo dobro znanih enačb mehanike tekočin. Izbrana geometrija venturijeve cevi je bila diskretizirana v programu ANSYS Student 2023 ob upoštevanju vseh pravil CFD. Primerjava teh dveh rezultatov je pokazala ujemanje znotraj 5 %.

1 INTRODUCTION

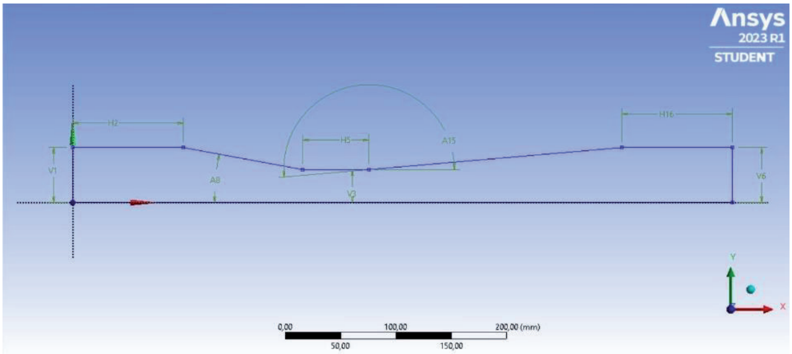
The EU's desire to popularise science leads to the need for students to get involved in scientific research as early as possible. In the past, scientific research was reserved for postdoctoral students. The approach that students get involved in the world of scientific research at the very beginning of their studies at undergraduate levels, or at least as part of their final theses, provides advantages to society, and to the Faculties themselves, because they get a better overview of which of these students have the research affinity *[1]*. Furthermore, it is the opinion of the authors of this paper that involving students in scientific researchs contributes to the development of civil society.

It has been shown with this approach that some students already have a hidden potential to produce scientific articles of the highest level *[2]*. The tendency to involve students in research is demonstrated by the example of some Faculties in Croatia, which give additional points for published scientific articles when they apply in graduate studies.

In order for students to be involved in scientific research, it is necessary to provide them with tools that they could use in their work *[3]*. In the field of CFD, these tools are various computer programs, some of which are commercial and some of which are open source. Considering the high price of commercial CFD tools, the idea appeared to test students` free versions of them, which the manufactures of these software packages themselves say that students may use in their research only with prior registration.

2 GEOMETRY GENERATION

The geometry and dimensions of the Venturi tube used for the purposes of this research were determined according to the ISO 5167-4:2003 Standard, *[4]*. According to the guidelines of the specified Standard, the geometry of the model was created in the ANSYS Student Design Modeler (Fig. 1).



Details View

Dimensions: 8	
A15	185 °
A8	10,5 °
H16	100 mm
H2	100 mm
H5	60 mm
V1	50 mm
V3	30 mm
V6	50 mm

Figure 1: *Venturi geometry according the ISO 5167-4:2003 Standard [3]*

The angle φ = 10° was chosen for the divergent conical part. The ratio of the diameter of the inlet part and the throat of the Venturi tube is:

$$\beta = \frac{d}{D} = \frac{60\ \mathrm{mm}}{100\ \mathrm{mm}} = 0.6 \qquad 2.1$$

After all the dimensions of the test venturi tube were determined, a 3D model was created, shown in (Fig. 2).

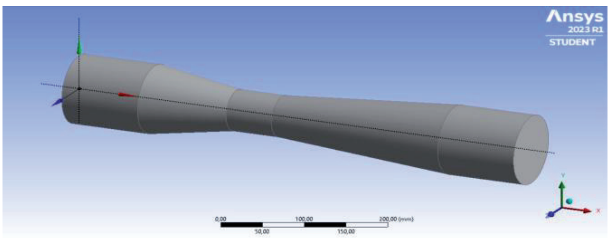


Figure 2*: 3D model of the Venturi pipe*

2 DISCRETISATION AND MESH GENERATION

The first step of creating a mesh of control vulumes is defining the boundary conditions, (Fig. 3). It is necessary to determine the fluid inlet and outlet from the venturi tube, and set the limits necessary for numerical calculations.

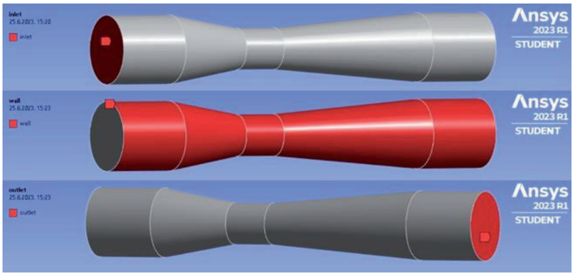


***Figure 3**: Definition of boundary conditions*

The next step is to set inflation conditions on the boundary region of the pipe wall. Inflation in a CFD simulation is a technique used to create mesh layers near the walls, to model the boundary zone between the fluid and the wall better. This technique is particularly useful in flow simulations, where it is important to model the fluid layer adjacent to the wall accurately [5]. Inflation is achieved by adding additional mesh layers near the wall. These layers have a higher density of nodes than the rest of the mesh, and extend within the fluid boundary layer. The goal of inflation is to create enough nodes within the boundary layer to capture and model the turbulent effects occurring at the wall`s surface. Inflation is usually applied in combination with turbulence models, in order to model turbulent flows near the wall more accurately, and in order to satisfy the value of the dimensionless parameter *y+*, the required value of which depends on the selected turbulence model. This technique enables a better resolution of the boundary conditions, and reduces the need for a very fine mesh of the entire computaional model, which would require large computing resources. The venturi wall is set as the inflation boundary, the number of inflation layers is set to 8, and the layer growth rate is set to 1.2 towards the centre of the pipe, (Fig. 4).

The discretisation itself was performed using tetrahedral control volumes. The tetrahedral mesh is flexible, and adapts well to geometries with curved surfaces, which is why they are one of the most commonly used elements in the analysis of turbulent flows, *[5].*

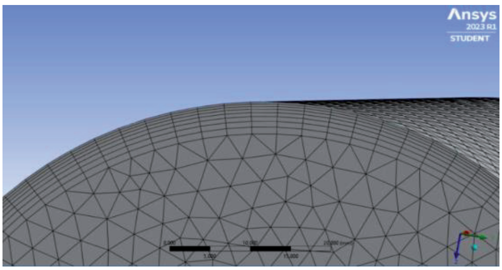


***Figure 4**: Inflation boundary layers*

For the purposes of this work, ANSYS Student was used, which limits the number of control volumes to 512000. Due to this fact and the addition of additional layers of inflation, the size of the elements was set to 3.5 mm, so that the number of elements was less than the allowed 512000, (Fig. 5).

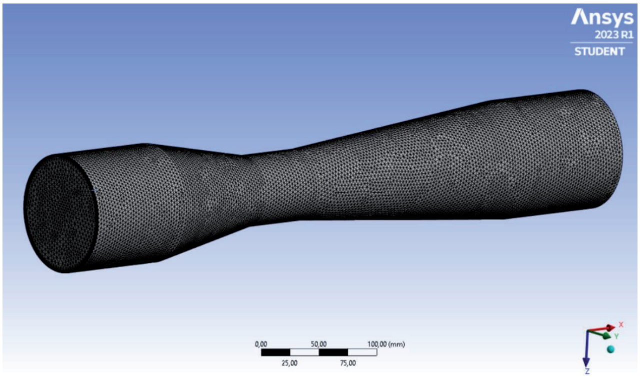


Figure 5: *Generated mesh*

3 PREPARATION AND SIMULATION RUN

The $k - \varepsilon$ model of tubulation was chosen, due to the limitations of the computing power of the computer used to create the simulation and the limitations of student licences. For the purposes of this work, water in liquid form was chosen as the working fluid. The inlet was defined as a velocity inlet (Velocity Inlet), with an initial velocity value of 2 m/s. The turbulence parameters were also defined: The turbulence intensity was set to 5%, and the hydraulic diameter was set to the pipe inlet diameter D = 100 mm. Atmospheric pressure was set at the outlet of venturi pipe.

Furthermore, a coupled (Fully Coupled Method) was used. This method uses fully implicit resolution of pressure and velocity, and allows full coupling between velocity and pressure at each iteration step, which is great for capturing small fluctuations due to turbulence, [6]. In this work, the convergence limit for all iteration parameters was set to the value 10^{-6}. This value brings sufficiently precise results considering the density of the mesh, the chosen discretisation method, and the mathematical and turbulence models.

4 MODEL VERIFICATION

Comparison of images obtained in known experimental flows through a standard venturi pipe with images of CFD simulated flows was used, to conduct verification of the built model and used student version of CFD software. The comparison was flow visualisation of the 2D contours of the pressure through the central plane of the Venturi tube in the axial direction (in the x-y plane), (Fig. 6).

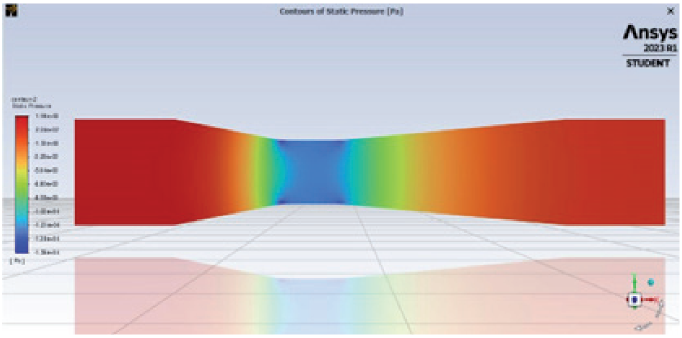


Figure 6: *Pressure distribution along the axial plane*

The pressure contours obtained by the CFD student package showed good agreement with the experimental images of flow through a standard venturi tube. This experimental images are well known and are in every book of fluid mechanics, so they are not presented here.

Furthermore, a comparison was made of the velocity distribution contours and flow velocity vectors with known experimental flow images, (Fig. 7 and 8).

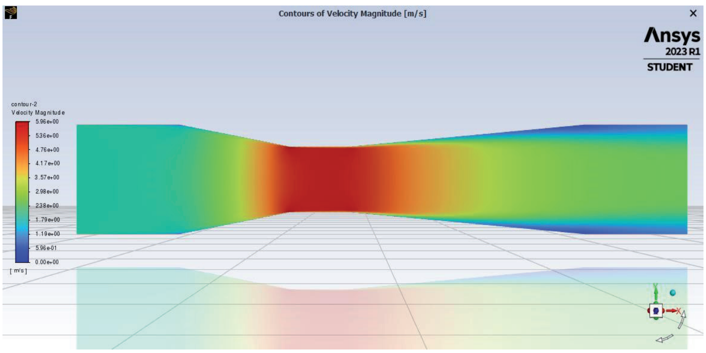


Figure 7 *Contures of velocity magnitude*

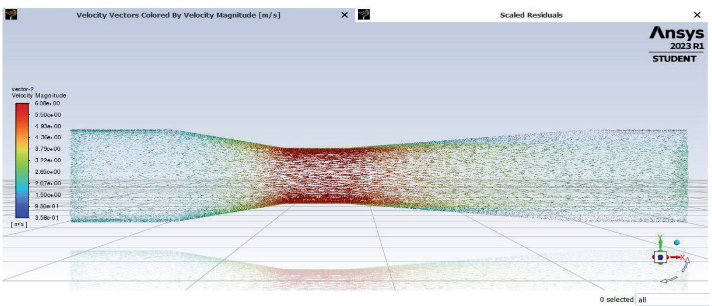


Figure 8 *Velocity vectors colored by velocity magnitude*

A comparison of the velocity field and velocity vectors obtained by the student CFD package with the existing experimental ones showed a good match. This proved that the student CFD package simulates well the physics of the flow itself and the phenomena in it. This proved the verification of the student CFD package.

5 RESULTS COMPARISON

The values obtained from the CFD simulation have a certain deviation with respect to the values obtained by analytical calculation, as shown in Table 1.

	Analytical calculation results	**CFD results**	**Relative deviation r [%]**
Δp [Pa]	13411.434	13977	4.22
q_v [m^3/s]	0.01571	0.01604	2.11
q_m [m^3/s]	15.6817	16.0111	2.11

Table 1: *Comparison of results gained with the analytical calculation and CFD*

The values obtained from the CFD simulation have a certain deviation with respect to the values obtained by the analytical calculation. It can be seen that this deviation was less then 5%.

6 CONCLUSION

From this work it can be concluded that the student’s version of Ansys Fluent can be used for students` researches and projects with satisfactory reliability.

Future work suggests finding a numerical correlation between the accuracy of the results of student’s cfd Ansys Fluent with geometry complexity.

References

[1] **T. Mihalić, J. Hoster, V. Tudić, T. Kralj:** *Concept Design and Development of an Electric Go-Kart Chassis for Undergraduate Education in Vehicle Dynamics and Stress Applications,* Applied sciences (Basel), 13 (2023), 20; 11312, 22. doi: 10.3390/app132011312

[2] **N. Šimunić, T. Jurčević Lulić, J. Groš, T. Mihalić:** *Analysis of Surface Curvature Influence on 3D Scanning Accuracy of Dental Castings,* Interdisciplinary description of complex systems, 19 (2021), 3; 357-456. doi: 10.7906/indecs.19.3.8

[3] **S. Medić, V. Kondić, T. Mihalić, V. Runje:** *Research of the Design Feasibility of a 3-Wheel Electric Vehicle with a Simplified Control System,* Tehnički glasnik, 14 (2020), 1; 32-35. doi: 10.31803/tg-20200124204834

[4] **ISO 5167-4:2003:** *“Measurement of fluid flow by means of pressure differential devices inserted in circular cross-section conduits running full — Part 4: Venturi tubes“*, 2003

[5] **T. Mihalić, Z. Guzović, A. Predin:** *Performances and Flow Analysis in the Centrifugal Vortex Pump,* Journal of fluids engineering, 135 (2013), 1; 011002-1-011002-7. doi: 10.1115/1.4023198V. Runje, T. Mihalić, T. Kostadin: IMPROVEMENT OF A WEEDHOPPER 2 ULTRALIGHT AIRCRAFT BY USING MODERN MATERIALS, Journal of energy technology, 11 (2018), 4; 29-40. doi: 2463-7815

Nomenclature

(Symbols)	(Symbol meaning)
CFD	Computational fluid dynamics