Using FEATool Multiphysics software for CFD Education

Alexander G. Kuzmin

Abstract— **Numerical simulations of two-dimensional viscous and inviscid flows with FEATool Multiphysics package are presented. Three examples under consideration are as follows: 1) the laminar compressible flow over a backward facing step, 2) the inviscid supersonic flow over a bump in a channel, 3) the heating of an incompressible laminar flow by a hot cylinder.**

Results of the simulations are in good agreement with those obtained with the commercial software ANSYS-18.2 CFX. The considered examples show the suitability and convenience of FEATool for educational purposes, as well as high accuracy of flow simulation in certain cases, e.g., in the case of inviscid supersonic flow. At the same time, the package needs improvements in setting boundary conditions at inflow and outflow boundaries for the opportunity of computation of wider variety flows, for example, transonic ones. Also, an asset would be the incorporation of up-to-date turbulence models for computation of viscous flows at high Reynolds numbers.

Keywords— **Heat transfer, inviscid compressible flow, numerical simulation, viscous incompressible flow.**

I. INTRODUCTION

FEATool Multiphysics is a user friendly software developed for numerical simulation of engineering problems concerned with fluid and solid mechanics, electromagnetics, acoustics, and heat transfer [1]. Simulations with FEATool can be started without having to learn extensive documentation or courses. The package is available for Windows and Linux OS, and it involves an integrated modeler, finite-element solver and graphical postprocessor. In particular, it provides a convenient environment for the numerical simulation of compressible and incompressible flows. The package does not require any skills in mesh generation or preprocessing, although basic knowledge in flow physics would be helpful. In addition, FEATool offers an opportunity to incorporate and use other freely available solvers, such as OpenFOAM [2], SU2 [3], and FEniCS [4]. A number of clear tutorials are available.

At the start of a numerical simulation, the user is

suggested to choose a system of governing equations.

The system of Navier–Stokes equations is used for modeling laminar flows of incompressible fluid; the equations are to be solved for the flow velocity field and static pressure p.

Simulation of compressible inviscid flow is based on the Euler equations for the velocity field, pressure p, and density ρ.

The equation governing heat transfer involves convection and conduction phenomena, as well as heat generation by volume sources.

Characteristic features of FEATool are as follows:

(*) Easy creation of the computational domain in two and three dimensions.

(*) Mesh generation with built-in software or Gmsh/Triangle tools. The generators allow for easy grid refinement, as well as accessing advanced grid generation settings. A choice of FEM shape functions is also available.

(*) Simple dialog boxes are available for the setting of boundary conditions for equations governing fluid flow and heat flux.

(*) Several built-in solvers allow for solving coupled stationary, time-dependent, and eigenvalue problems. The built-in solvers are monolithic, so that all equations are discretized and solved together in a large coupled system. This typically allows for faster and more robust convergence of solutions at the expense of higher memory usage.

FEATool switches to postprocessing mode after computation of a solution. The visualization options include choosing predefined or entering user-defined expressions, selecting or excluding which cells to display, and setting the minimum and maximum plot scale limits.

In this paper, we present three examples of flow simulation using a free trial version of FEATool and available documentation/tutorials.

II. FORMULATION OF PROBLEMS AND NUMERICAL RESULTS

Example 1. Laminar Flow over a Backward Facing Step

We consider a 2D channel whose upper wall is parallel to the x-axis: $y=10$, $0 \leq x \leq 120$. The profile of the lower wall consists of two horizontal segments $y=5$, $0 \leq x \leq 20$ and $y=0$, $20 \leq x \leq 120$, and the vertical segment $x=20$, $0 \leq y \leq 5$, see Fig. 1. Evidently, the segments compose a step of height $h=5$. Here and further in the paper, the Cartesian coordinates

(x,y) and geometric parameters h, d are dimensional and given in millimeters.

In a FEATool tutorial, the flow density is non-dimensional and equal to 1, and the dynamic viscosity is set to 0.0017. Instead, we choose standard parameters of atmosphere at sea level in SI units [5]: the density of air ρ_{in}=1.2754 kg/m^3, static pressure p_{in}=101,325 Pa, static temperature T_{in}=288.11 K, and dynamic viscosity μ=1.789×10^{-5} Pa·s .

At the entrance of channel x=0, $5 \leq y \leq 10$, the inflow velocity U_{in} is parallel to the x-axis and equal to 0.5 m/s. At the exit x=120, $0 \leq y \leq 10$, we set the static pressure p_{in}. On the walls of channel, the no-slip condition is prescribed.

The Reynolds number based on the height of channel $2h$=10, velocity U_{in}, and the dynamic viscosity μ is Re=$2h\rho U/\mu$=371.5.
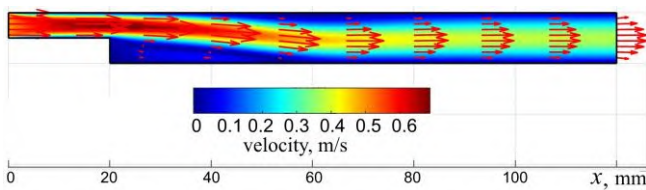

Figure 1. Velocity contours in the laminar flow over a backward facing step at U_{in}=0.5 m/s; computations with FEATool

Computations were performed on a mesh of approximately 1.2×10^4 triangular cells using a "stationary" option in solver settings. Figure 1 depicts the obtained velocity contours in the channel; as seen, there is a flow separation from the lower wall at the corner x=20, y=5, followed by a reattachment to the wall at x_r≈62, y=0.

For comparison, the same problem was solved with the package ANSYS-18.2 CFX. A hybrid computational mesh consisted of quadrangles in 30 layers on the walls and triangles in the remaining region; the total number of mesh cells was 246,068. Figure 2 demonstrates the flow reattachment to the lower wall at x_r≈60. This is in agreement with experimental and numerical studies [6, Fig. 3], which show that, when Re=371.5, reattachment occurs at a distance of 7.8h=39 downstream of the step, i.e., at x_r=59 in the case at hand.
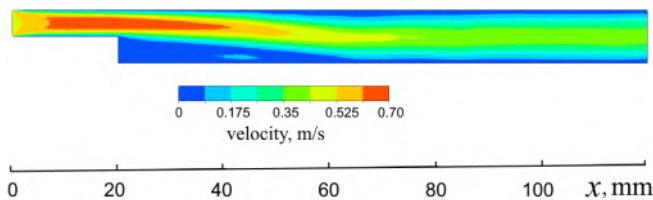

Figure 2. Velocity contours in the laminar flow over a backward facing step at U_{in}=0.5 m/s; computations with ANSYS

Although there is some discrepancy between the values of x_r inferred from Figs. 1 and 2, both flow fields show the same maximum velocity $U(x,y)|_{max}$=0.69 m/s in the flow;

the maximum is located over the corner of lower wall.

Example 2. Inviscid Flow over a Bump in a Channel

Another FEATool tutorial describes the computation of inviscid supersonic flow over a circular-arc bump using the system of Euler equations. Following this tutorial, we computed supersonic flow over a simpler bump whose profile is an isosceles triangle located on the lower wall of a channel whose dimensions are $-1000 \leq x \leq 4000$, $0 \leq y \leq 2000$. The height of triangle is h=50, and its endpoints lie on the wall at x=0 and x=1000, see Fig. 3. At the entrance of channel x=-1000, the flow is parallel to the x-axis and the Mach number is set to 1.6, whereas the static temperature, pressure and density correspond to the sea level. The free-slip condition was used on the walls and bump. The solution obtained on a mesh of approximately 3.6×10^4 triangular cells showed a good resolution of oblique shock waves, see Fig. 3.
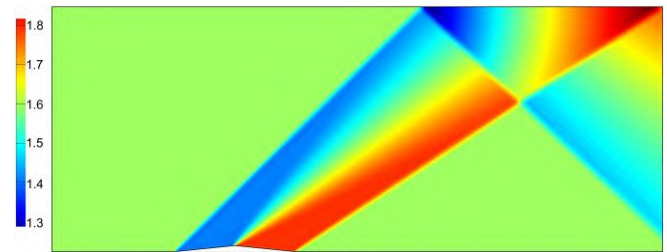

Figure 3. Mach number contours in a channel with a bump at M_{in}=1.6; computations with FEATool

The same problem was solved with ANSYS-18.2 CFX on meshes of approximately 1.9×10^5 and 9.6×10^5 triangular cells. The resolution of shocks on the first mesh was insufficiently good, whereas the results obtained on the second mesh (see Fig. 4) demonstrated excellent agreement with those shown in Fig. 3.
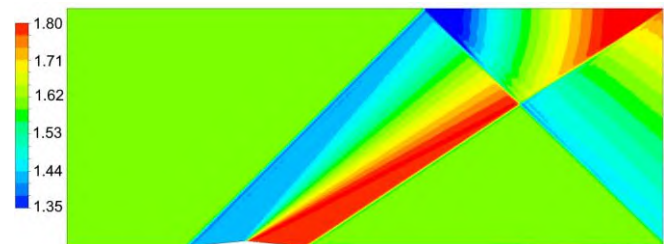

Figure 4. Mach number contours in a channel with a bump at M_{in}=1.6; computations with ANSYS

An attempt to compute transonic flow with a local supersonic region over the triangular bump or circular-arc one using FEATool was not a success. In this case, the flow is supposed to be subsonic at the entrance and exit of channel; meanwhile, the available version of the package had not provided the setting of standard conditions on subsonic boundaries (given density ρ and velocity components at the inlet and pressure p at the outlet).

Example 3. Heating a Water Flow by a Hot Circular Cylinder

For CPU savings, we assume the flow to be symmetric about the *y*-axis and consider a half of the channel, as shown in Fig. 5. The dimensions of channel and cylinder in this example are the same as in a FEATool tutorial [1]: $0 \leq x \leq 7.5$, $0 \leq y \leq 50$; the radius of cylinder is 3, the center is located at $x=0$, $y=20$. The flow enters the channel in the vertical direction at the inlet $y=0$, $0 \leq x \leq 7.5$, and inflow velocity is $V_{in}=0.04$ m/s. Following [1], we impose the temperatures of cylinder $T_{cyl}=330K$ and water at the inlet $T_{in}=300K$.

Instead of the density $\rho=22$ and dynamic viscosity $\mu=2.8 \times 10^{-3}$ Pa·s chosen in [1], we set the physical properties of water as follows: $\rho=997$ kg/m^3, $\mu=8.9 \times 10^{-4}$ Pa·s, heat capacity at constant pressure $c_p=4181.7$ J/(kg·K), and thermal conductivity $k=0.6069$ W/(m·K). The no-slip condition and given temperature T_{cyl} were imposed on the surface of cylinder, and the symmetry condition was used on the left and right walls of channel. A solution of the system of Navier–Stokes and heat transfer equations was obtained on a mesh of about 1.2×10^4 cells. The solution demonstrates a rise in flow temperature over the cylinder, see Figs. 5 and 6.
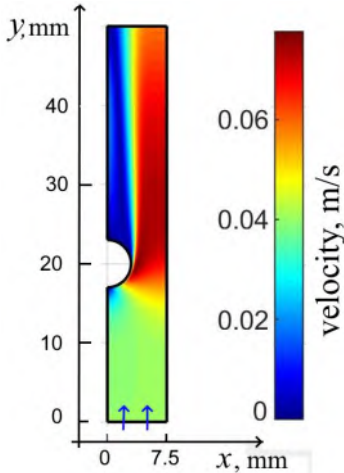

Figure 5. Velocity contours in the laminar flow over a hot cylinder at $V_{in}=0.04$ m/s; computations with FEATool
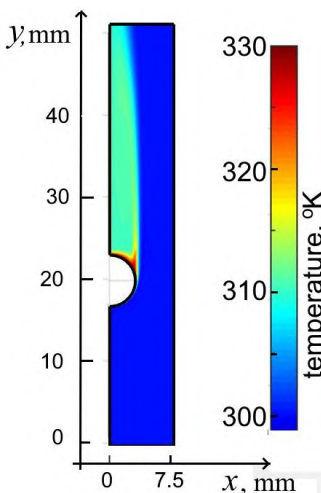

Figure 6. Temperature contours in the laminar flow over a hot cylinder at $V_{in}=0.04$ m/s; computations with FEATool

Figures 7 and 8 depict results of solving the same problem with ANSYS 18.2 CFX. As seen, the velocity and temperature fields are in qualitative agreement with those shown in Figs. 5 and 6. Some discrepancies near the exit are explained by different boundary conditions used in the solvers (given flux at the exit in ANSYS vs. given pressure in FEATool).

We notice that the only built-in turbulence model in FEATool is an algebraic one, which is not truly accurate for complex flows such as those involving separation and/or recirculation [7]. To employ other turbulence models, the user needs to incorporate either OpenFOAM or the SU2 package.
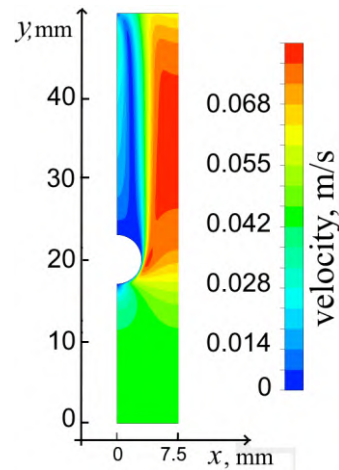

Figure 7. Velocity contours in the laminar flow over a hot cylinder at $V_{in}=0.04$ m/s; computations with ANSYS
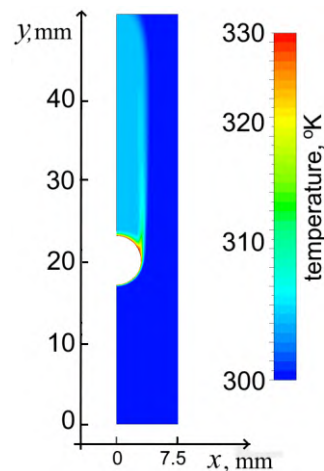

Figure 8. Temperature contours in the laminar flow over a hot cylinder at $V_{in}=0.04$ m/s; computations with ANSYS

III. CONCLUSION

In the considered examples, standard physical properties of air and water in SI units were used instead of unrealistic combinations of the density and viscosity presented in FEATool Multiphysics tutorials. The results of simulations confirm that FEATool is a handy software that can be recommended for educational purposes at engineering universities and colleges. Meanwhile, there is need in more options for the setting of boundary conditions at inflow/outflow boundaries, as well as the incorporation of various built-in turbulence models for simulation of flows at high Reynolds numbers.

REFERENCES

[1] FEATool Multiphysics, 2023. Available: https://www.featool.com
[2] OpenFOAM, 2023. Available: https://www.openfoam.com
[3] SU2 foundation, 2023. Available: https://su2foundation.org
[4] The FEniCSx computing platform, 2023. Available: https://fenicsproject.org
[5] Standard sea-level conditions, 2023. Available: https://en.wikipedia.org/wiki/Standard_sea-level_conditions
[6] M. A. Hossain, Md. T. Rahman, Sh. Ridwan, "Numerical investigation of fluid flow through a 2D backward facing step channel," *Int. J. of Engineering Research & Technology*, vol. 2, issue 10, pp. 3700–3708, 2013. https://www.ijert.org/research/numerical-investigation-of-fluid-flow-through-a-2d-backward-facing-step-channel-IJERTV2IS100716.pdf
[7] H. K. Versteeg, W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*. 2nd edition, Pearson Education Limited, Harlow, England, 2007. https://books.google.ru/books?id=RvBZ-UMpGzIC&printsec=frontcover&hl=ru#v=onepage&q&f=false