9th ANKARA INTERNATIONAL AEROSPACE CONFERENCE AIAC-2017-152
20-22 September 2017 - METU, Ankara TURKEY

VALIDATION OF OPEN-SOURCE CFD SOFTWARE FOR HIGH SPEED TURBULENT CAVITY FLOWS

Osman Mirza Demircan[1]
and Oğuzhan Demir[2]
Istanbul Technical University
Istanbul, Turkey

Seyfettin Türk[3]
and Kürşad Melih Güleren[4]
Anadolu University
Eskişehir, Turkey

ABSTRACT

Open-source software are very popular in engineering applications. They are freely available and their source code can be inspected, modified or enhanced according to its user's needs. SU² and OpenFOAM are two common open-source solvers used in the field of Computational Fluid Dynamics and this work investigates the reliability of these software in solving turbulent cavity flows for high speed regimes. For this task, cavity flows are analyzed using the aforementioned software and the results are compared with existing experimental findings. A well-known and trusted commercial software is also used as a numerical reference.

INTRODUCTION

Cavity flow is a complex type of flow that is encountered frequently in aerospace applications. First studies regarding cavity flows date back to 1940s and extensive research have hitherto been made on the subject. Pioneering experimental studies on cavity flow begin with the analysis of pressure oscillation modes [Rossiter, 1964] and the classification of cavity flows in subsonic and transonic regimes [Plentovich, 1993]. Early experiments on cavity flows in supersonic regimes are investigated by Stallings and Wilcox in 1987. These early experiments are mainly focused on clean cavities: empty rectangular prisms.

The cavity flow can briefly be described as the flow encountering a cavity on a surface that has its boundary layer separated at the cavity's leading edge. The separated flow may behave in three different ways: open cavity flow is defined as the separated flow going over the cavity (Figure 1), transitional flow happens when the flow enters the cavity without touching the floor and closed flow is defined when the flow interacts with the cavity's inner surfaces (Figure 2). These behaviors are mainly driven by the cavity geometry and the flow regime [Plentovich, 1993].

[1] M.Sc. Student in Department of Aeronautics and Astronautics Engineering, Email: demircano16@itu.edu.tr
[2] M.Sc. Student in Department of Aeronautics and Astronautics Engineering, Email: demirog16@itu.edu.tr
[3] Ph.D. Student in Department of Airframe and Powerplant Maintenance, Email: seyfettinturk@anadolu.edu.tr
[4] Assoc. Prof. in Department of Flight Training, Email: kmguleren@anadolu.edu.tr

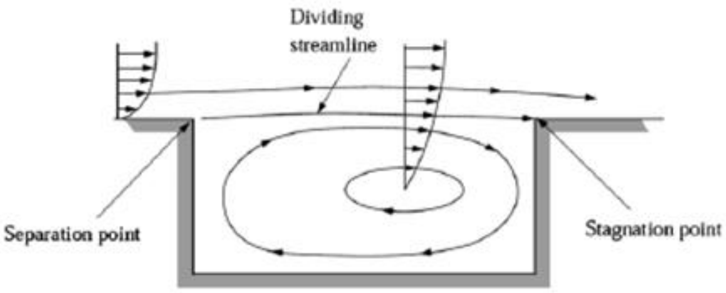

Figure 1: Open cavity flow [E.S.D.U. 2008]

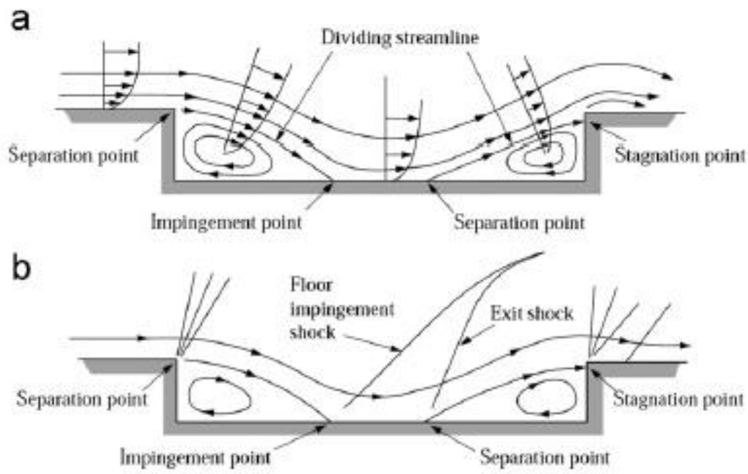

Figure 2: Closed cavity flow for (a) subsonic and (b) supersonic speeds [E.S.D.U. 2008]

A shear layer is formed between the separated boundary layer and the flow inside the cavity. This shear layer is an important aspect of the cavity flow as it drives its complexity by inducing acoustic waves when it interacts with the aft wall of the cavity (Figure 3).

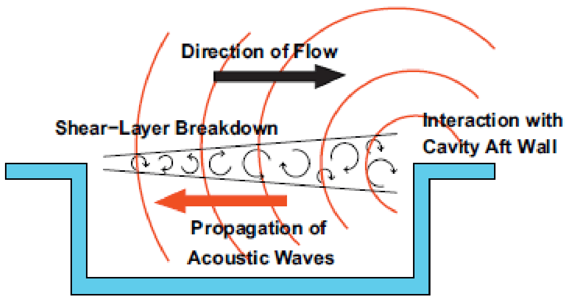

Figure 3: Schematic of a cavity flow problem [Lawson, 2011]

METHOD

In this work, two-dimensional RANS and URANS simulations are conducted. The classification of the analyzed cavity geometries based on Plentovich's guide [Plentovich, 1993] is made after results analysis using the pressure coefficient distribution at the cavity floor. Furthermore, the flow field is inspected in terms of Mach number and turbulence intensity in the vicinity of the cavity.

These analyses are expected to be verified using experimental data for validation. The transonic simulations are compared with Plentovich's results [Plentovich, 1993] and the supersonic simulations are compared with Stallings's findings [Stallings, 1987].

Two-dimensional analyses are conducted, neglecting the three-dimensional effects of the cavity flow. Experiments with widest cavities (cavities with highest width-to-depth ratio) are used as a reference when conducting these analyses in order to make a reasonable comparison.

A configuration matrix is defined in Table 1 for the intended analyses. These analyses are chosen such that all three types of cavity flows are analyzed.

Table 1: Selected cavity properties at various Mach numbers [Plentovich, 1993]

M_∞ \ L/D	6	10	14
0.9	o	t	c
1.5	o	o	c

o – open, t – transitional, c – closed

In Table 1, the reference experiments for the transonic configuration ($\boldsymbol{M_\infty = 0.9}$) have a cavity width-to-depth ratio of ($\boldsymbol{W/D = 8}$) and for the supersonic configuration ($\boldsymbol{M_\infty = 1.5}$) have a cavity width-to-depth ratio of ($\boldsymbol{W/D = 5}$).

Preprocessing

The fluid domain for each analysis is constructed using the sketches of the related experimental cavity models. Two experimental cavity models are used as a reference as shown in Figure 4 and Figure 5.

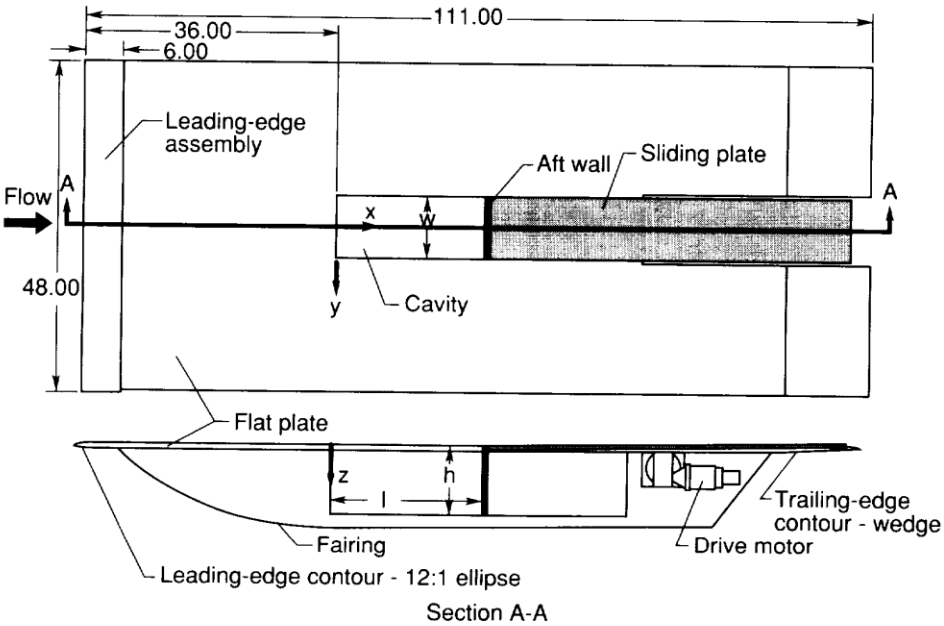

Figure 4: Sketch of variable cavity model in transonic regime [Plentovich, 1993]

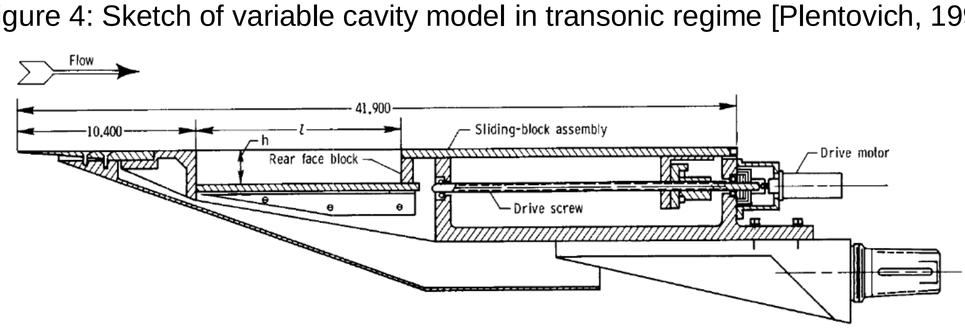

Figure 5: Sketch of variable cavity model in supersonic regime [Stallings, 1987]

The mid-plane section of these models is meshed using structured grids with quadrilateral elements for the numerical field shown in Figure 6.

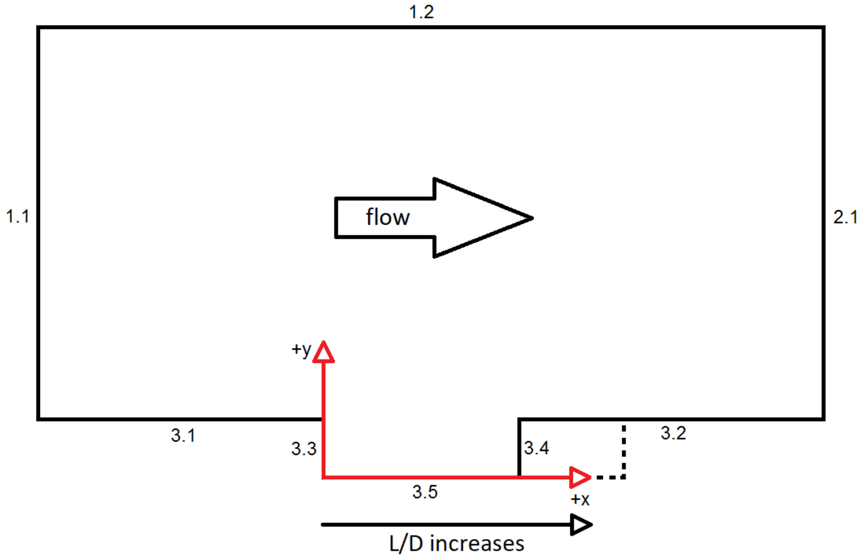

Figure 6: Representative numerical field for the two-dimensional cavity analyses

The generated mesh (Figure 7, Figure 8) have between 40000 and 77000 cells for the transonic analyses whereas a cell count between 30000 and 50000 is obtained for the supersonic analyses. The difference arises from the change in fluid domain dimensions. In both regimes, transonic and supersonic, the generated mesh has roughly the same quality with a dimensionless wall distance of $y^+ = 100$. The aspect ratio in the cavity fluid domain doesn't exceed 10 whereas high aspect ratio elements are generated away from the cavity in order to reduce the cell count.

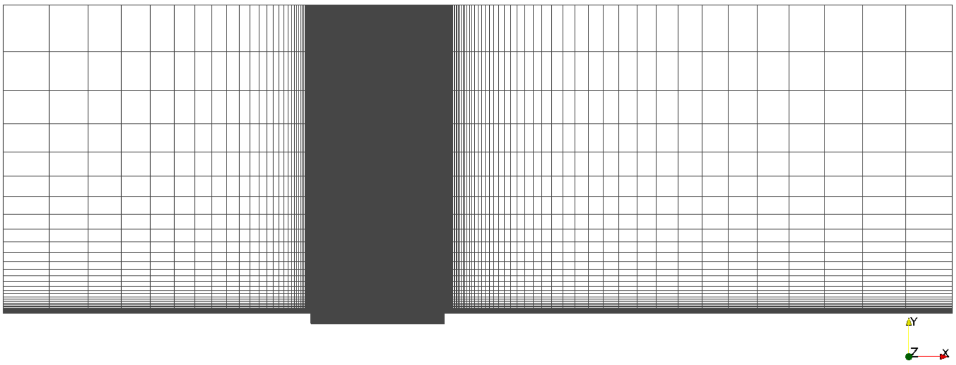

Figure 7: Generated structured grid for transonic speeds

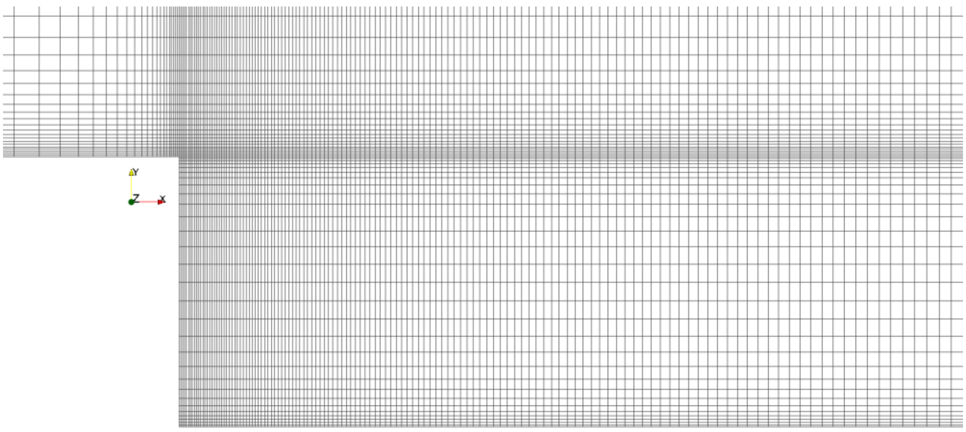

Figure 8: A closer view at the generated grid around the separation region

Solution

The generated mesh is imported into each solver for the simulation setup. As each solver has their own specific way of operating at this phase of the analysis, an explanatory table is constructed as shown in Table 2, referring to the defined numerical field in Figure 6.

Table 2: Numerical field definition and related parameters

#	Boundary	Applied Boundary Condition		
		ANSYS Fluent	SU^2	OpenFOAM
1.1	Inlet	Farfield	Farfield	Dirichlet Type*
1.2	Inlet	Farfield	Farfield	Neumann Type*
2.1	Outlet	Outlet	Outlet	Neumann Type*
3.1	Upstream wall	Adiabatic wall	Adiabatic wall	Adiabatic wall
3.2	Downstream wall	Adiabatic wall	Adiabatic wall	Adiabatic wall
3.3	Cavity front wall	Adiabatic wall	Adiabatic wall	Adiabatic wall
3.4	Cavity aft wall	Adiabatic wall	Adiabatic wall	Adiabatic wall
3.5	Cavity floor	Adiabatic wall	Adiabatic wall	Adiabatic wall

*wave transmissive boundary condition is applied for pressure in some cases.

The environmental conditions differ at different regimes. The experiments are carried out at varying absolute pressure and temperatures. However, relative to the freestream Mach number and cavity geometry, the environmental conditions have neglectable effect on cavity flow [Plentovich, 1993]. Consequently, ISA conditions are set for all analyses for the freestream as shown in Table 3.

Table 3: Freestream parameters at various Mach numbers

Freestream Mach Number	0.9	1.5
Freestream Velocity (m/s)	306.34	510.57
Static Temperature (K)	288.15	288.15
Static Pressure (Pa)	101325.0	101325.0

The analyses are conducted using the $\boldsymbol{k-\omega}$ turbulence model. RANS simulations are carried out using SU^2 and ANSYS Fluent whereas, URANS analyses are conducted with OpenFOAM. The pressure-based sonicFoam is preferred as a solver when using OpenFOAM.

RESULTS

The pressure coefficient Cp distribution on the cavity floor is retrieved at different configurations. The change along the longitudinal axis is compared with the experimental data at the mid-section of the related cavity configuration with a width to depth ratio of 8 for transonic analyses and 5 for supersonic analyses.

After inspecting the Cp distribution, Mach number and turbulence intensity contours in the vicinity of the cavities are plotted. These contours provide an insight on how the fluid behaves around a cavity at transonic and supersonic speeds.

The turbulence intensity contour is obtained with the following equation using the fluid domain parameters as inputs:

$$\boldsymbol{I} = \frac{\boldsymbol{u}'}{\boldsymbol{U}_\infty} \tag{1}$$

In equation (1), $\boldsymbol{u}'$ represents the root-mean-square of the turbulent velocity fluctuations and $\boldsymbol{U}_\infty$ is the freestream velocity in SI units. $\boldsymbol{u}'$ can also be computed as follows:

$$\boldsymbol{u}' = \sqrt{\frac{\mathbf{2}}{\mathbf{3}}\boldsymbol{k}} \tag{2}$$

In the latter equation, k represents the turbulent kinetic energy, which can be obtained from a solution using turbulence models. By substituting equation (2) into equation (1), one can calculate the turbulence intensity in the fluid domain as a percentage of the freestream velocity.

Pressure Coefficient Distribution

Cp distribution on the cavity floor is obtained from ANSYS Fluent, SU^2 and OpenFOAM for cavity various configurations listed in Table 1. The resulting plots are compared for each L/D configuration at different freestream Mach numbers in Figure 9 and Figure 10.

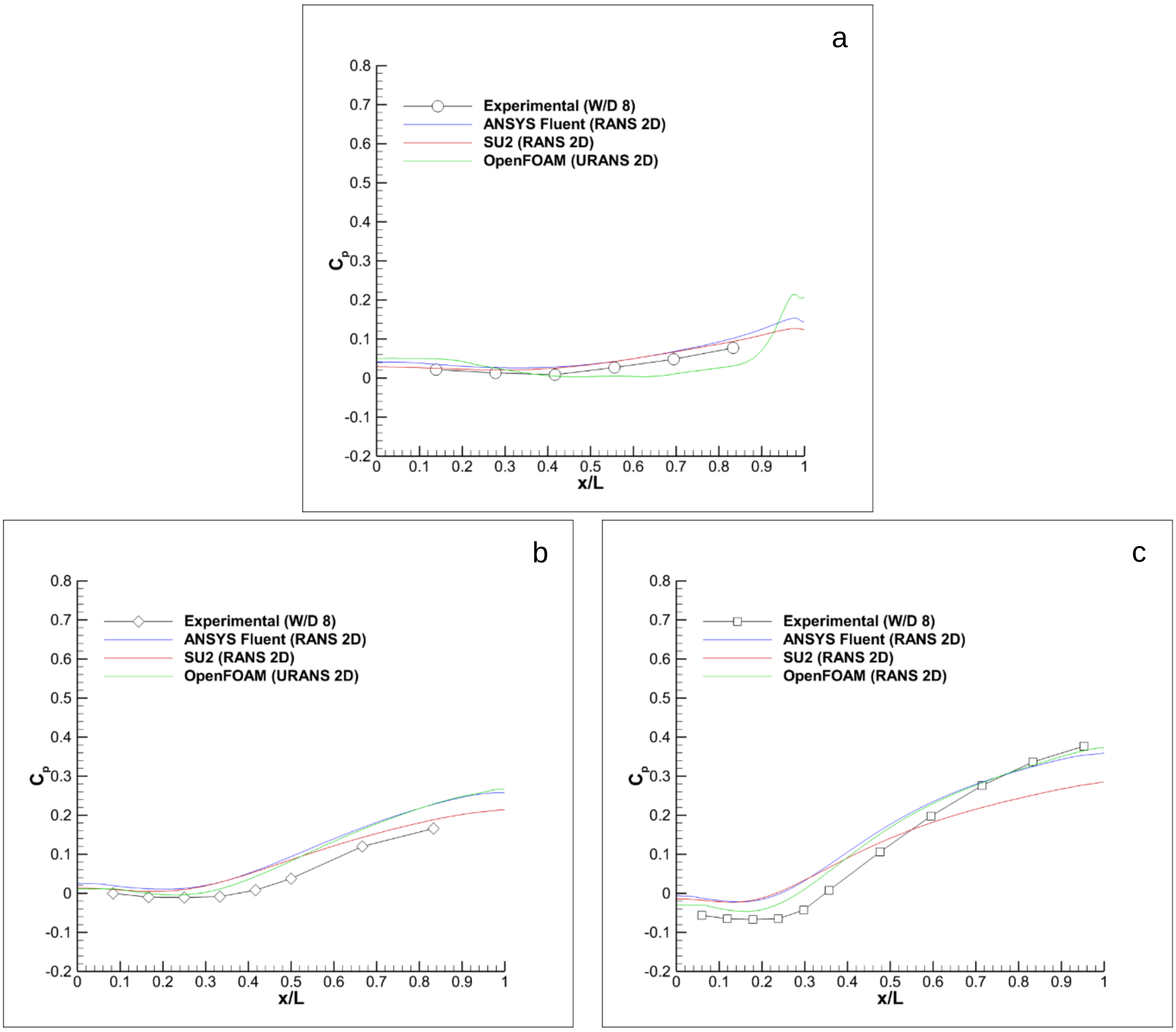

Figure 9: Transonic C_p distribution at cavity floor (a: L/D=6, b: L/D=10, c: L/D=14)

The results analysis in Figure 9 shows that URANS solutions with OpenFOAM gives better results compared to the RANS output of ANSYS Fluent and SU^2. Among the RANS solutions, it can be said that SU^2 data stray away from the experimental data as the length-to-depth ratio gets higher.

Considering these analyses are carried out as two-dimensional, the classification of wide cavities is possible with SU^2 and OpenFOAM with OpenFOAM being more accurate with the experiments.

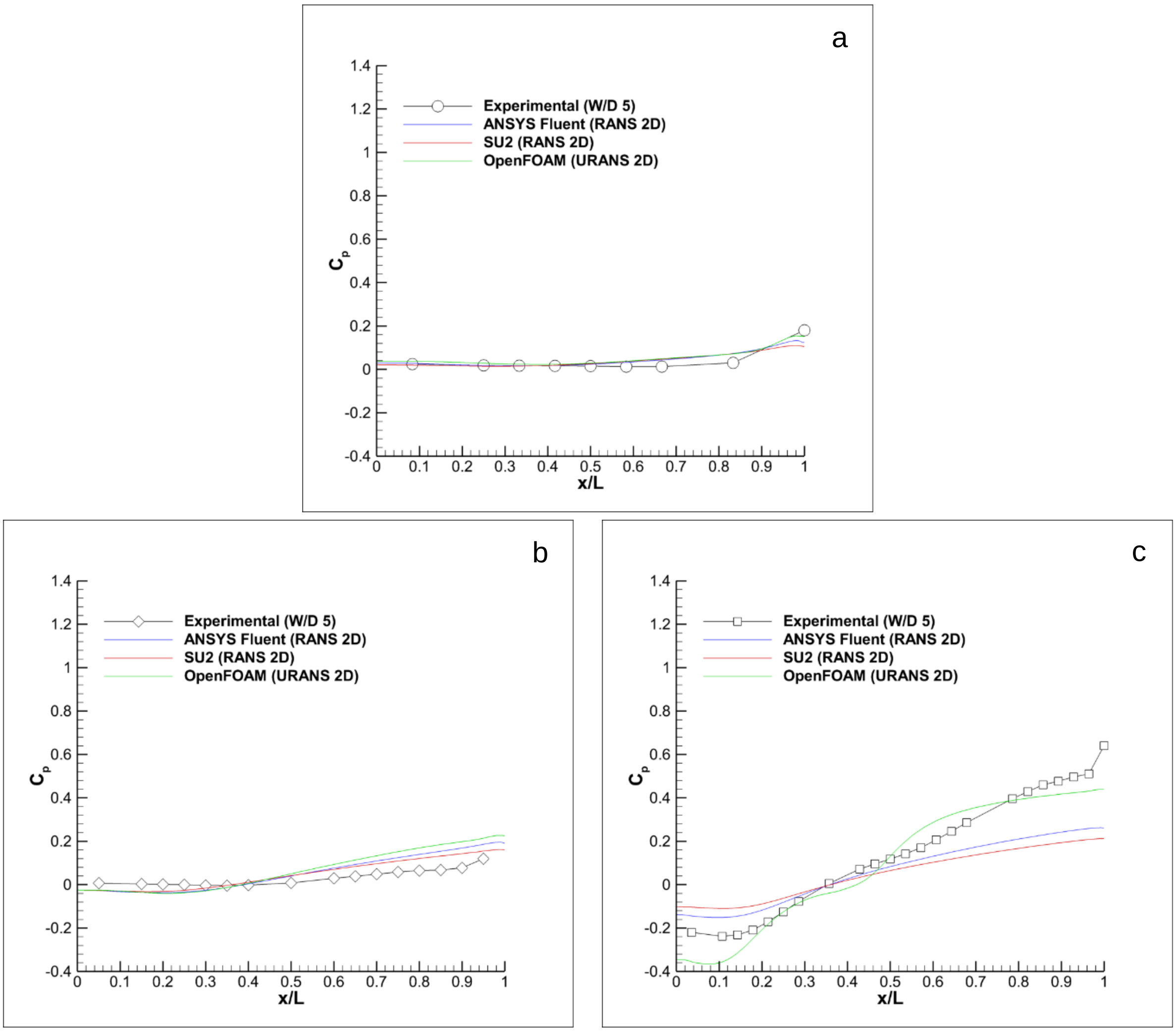

Figure 10: Supersonic C_p distribution at cavity floor (a: L/D=6, b: L/D=10, c: L/D=14)

Cp distribution on the cavity floor for the supersonic analyses are shown in Figure 10. According the latter, all solvers give acceptable results at L/D=6 and L/D=10. The last supersonic analysis results are quite intriguing as ANSYS Fluent and SU^2 fail to simulate a closed flow. However, OpenFOAM URANS simulation results in a closed cavity flow and gives somewhat reasonable results compared to the other solvers.

Finally, OpenFOAM URANS simulations in 2-D give the best results compared with the RANS simulations of SU^2 and ANSYS Fluent. As far as two-dimensional RANS simulations go, SU^2 lacks the accuracy of ANSYS Fluent and has trouble keeping up with the pressure coefficient increase in the second half of the cavity floor when compared with the experimental results.

Mach Number & Turbulence Intensity Contours

The Mach number and turbulence intensity in the vicinity of the cavity are analyzed in the following figures. Resulting contours for each solver are compared for various length-to-depth ratio configuration.

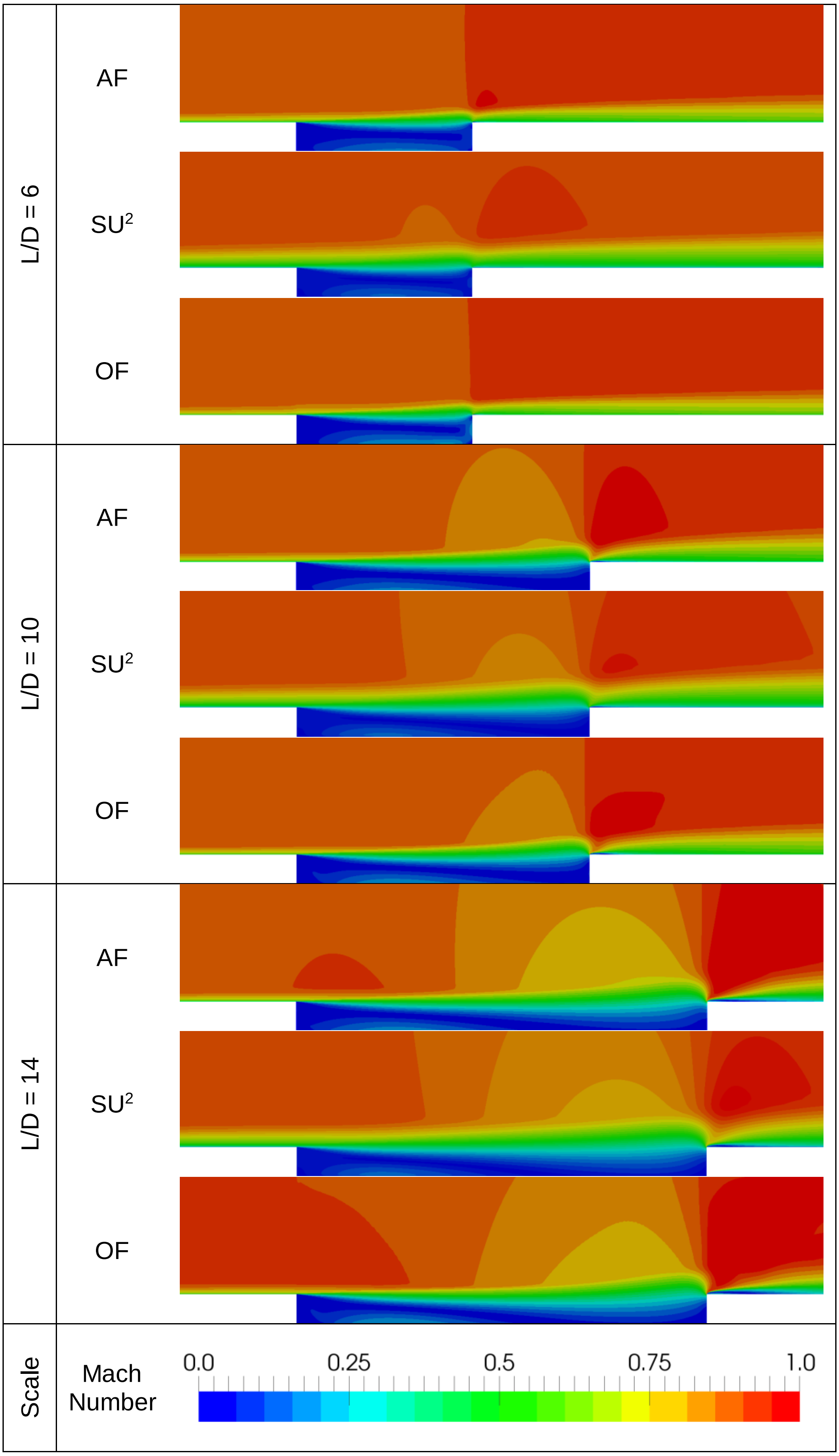

Figure 11: Transonic Mach number contours of ANSYS Fluent (AF), SU^2 and OpenFOAM (OF) at various length-to-depth ratios

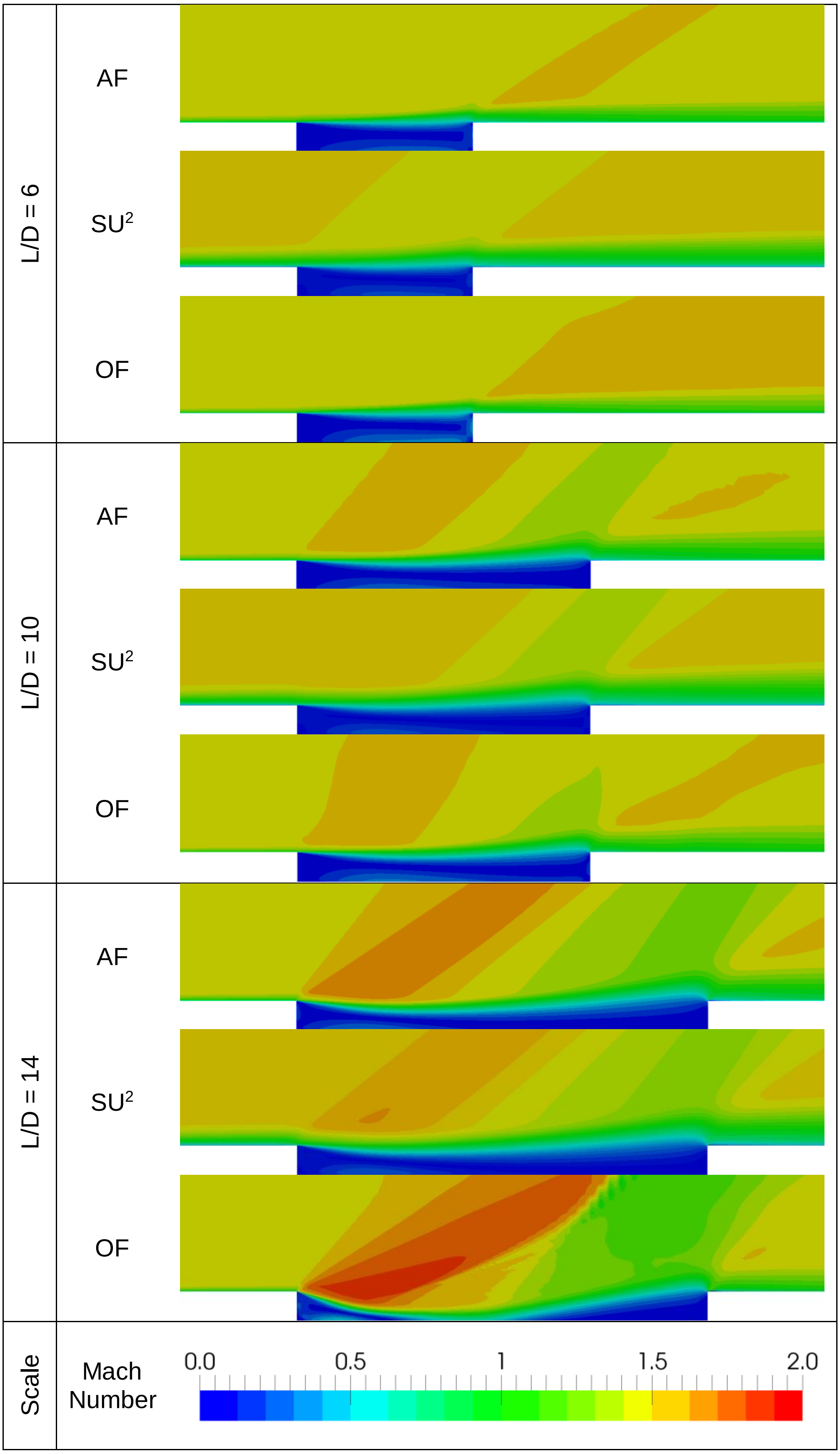

Figure 12: Supersonic Mach number contours of ANSYS Fluent (AF), SU^2 and OpenFOAM (OF) at various length-to-depth ratios

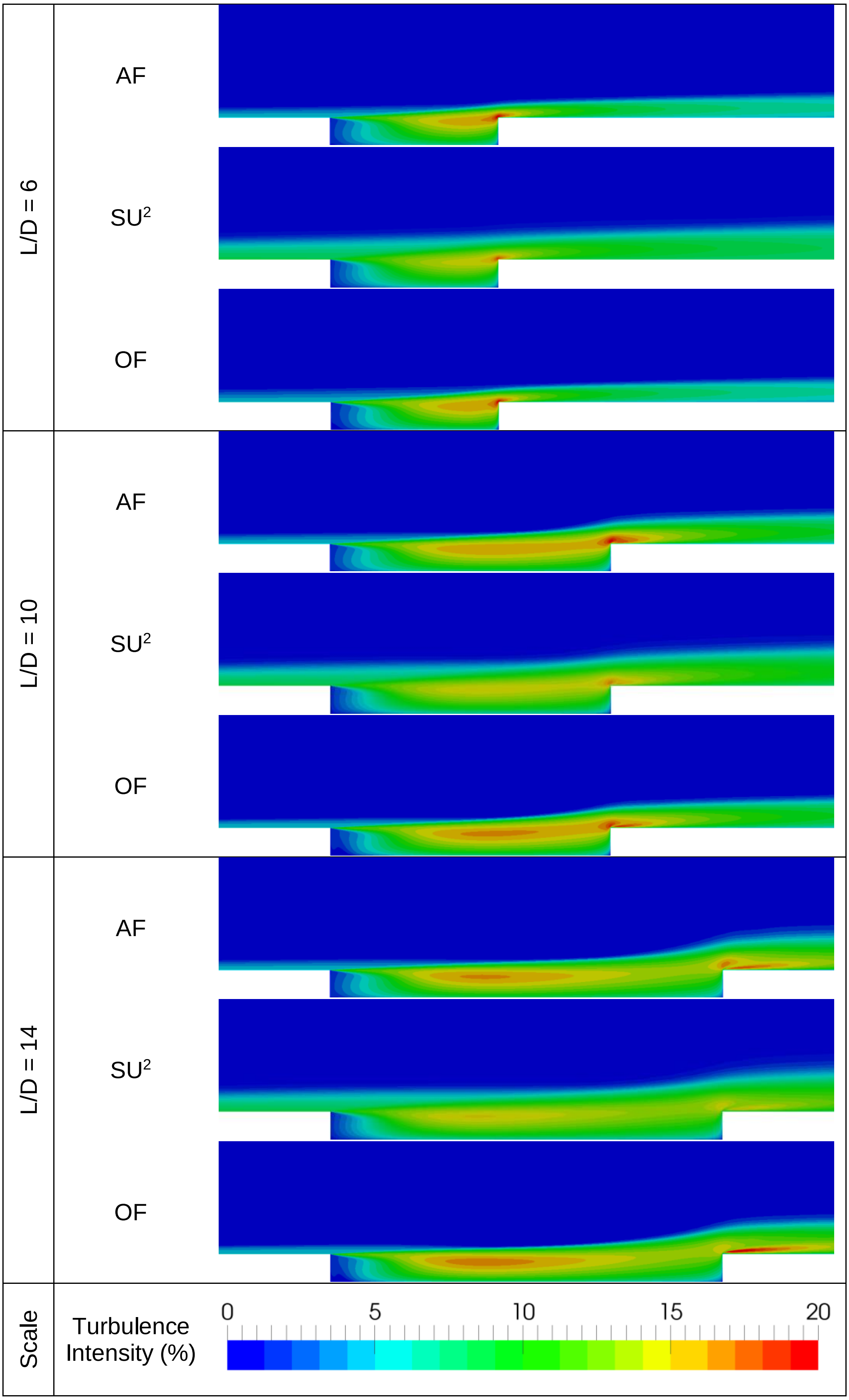

Figure 13: Transonic turbulence intensity contours of ANSYS Fluent (AF), SU^2 and OpenFOAM (OF) at various length-to-depth ratios

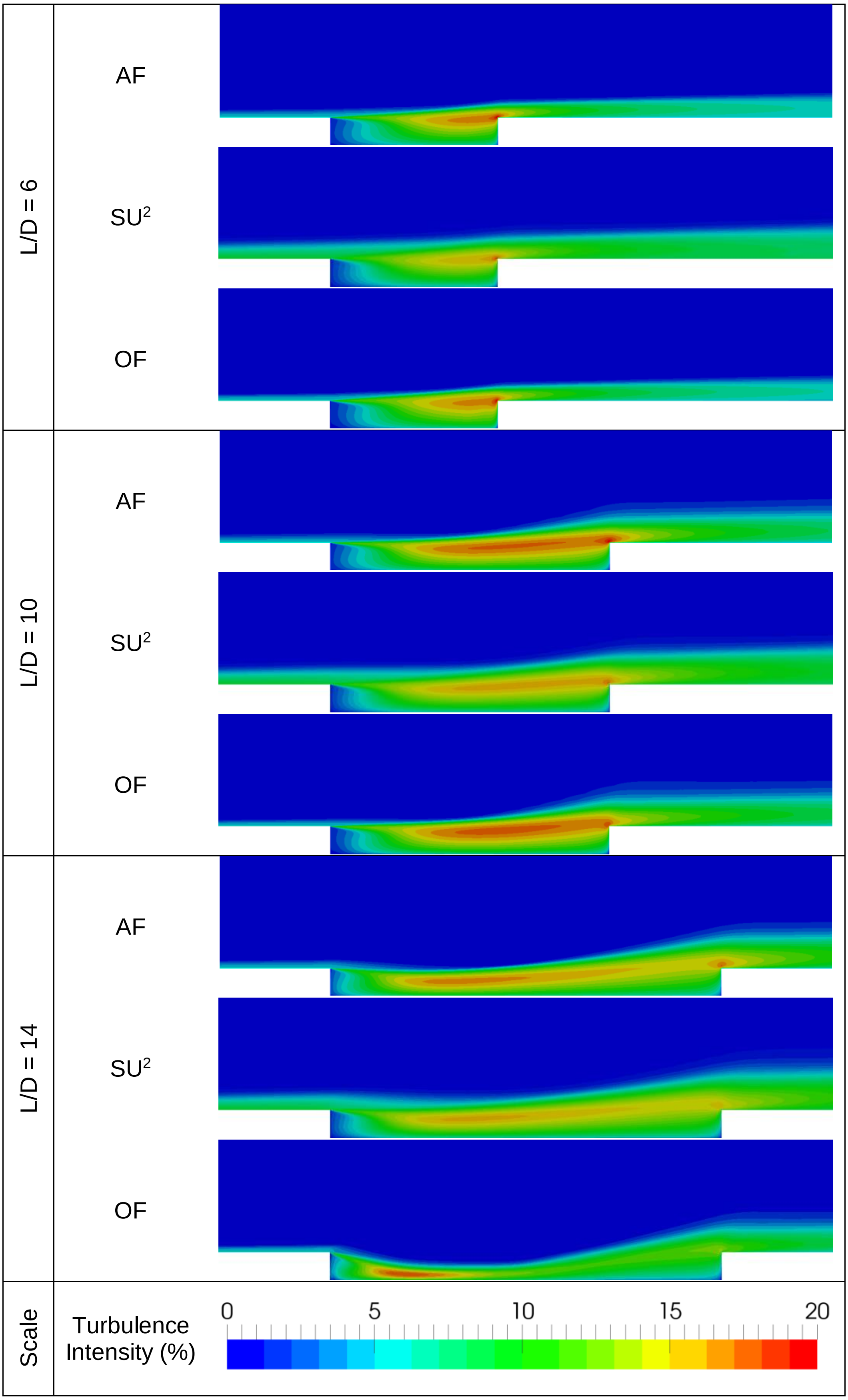

Figure 14: Supersonic turbulence intensity contours of ANSYS Fluent (AF), SU^2 and OpenFOAM (OF) at various length-to-depth ratios

CONCLUSION

In this study, cavity flows are analyzed numerically by the open-source software OpenFOAM and SU^2 for the Mach numbers of M=0.9, 1.5 and three different length-to-depth ratios of L/D=6, 10 and 14. Results are compared with the available experimental Cp data. It is seen that predictions are very in good agreement with the experimental data except for the M=1.5 and L/D=14. For this case SU2 fails the capture the trend of Cp, therefore the type of the flow is predicted as transitional, but the experimental data indicates that the flow should be closed. This is more clearly visible from the Mach contours in Figure 12 where the flow attaches the cavity flor for the OpenFOAM anlysis but not for SU2. The commercial code ANSYS Fluent was also tested for the same cases with the open-course codes, but it does not provide any improvement for Cp predictions. Generally, OpenFOAM code performs the best for all tested cases which is attributed mainly its URANS module. For the future work, the codes are planned to test for the same configuration but with a width-to-depth ratio of 1 to see the significant 3-dimensional effects.

ACKNOWLEDGEMENT

Numerical analyses in this study were run on the National Center for High Speed Computing UHEM (**http://en.uhem.itu.edu.tr/**). This study is supported by the Turkish Aerospace Industries (**https://www.tai.com.tr/en**) under the grant of TM-2031. The authors are grateful to the authorized staff for their support and assistance with this study.

References

Aerodynamics, E.S.D.U. (2008). Aero-acoustics of rectangular planform cavities. Part I: Time–averaged flow.

Lawson, S. J., & Barakos, G. N. (2011). Review of numerical simulations for high-speed, turbulent cavity flows. Progress in Aerospace Sciences.

Plentovich, E. B., Stallings Jr, R. L., & Tracy, M. B. (1993). Experimental cavity pressure measurements at subsonic and transonic speeds.

Rossiter, J. E. (1964). Wind tunnel experiments on the flow over rectangular cavities at subsonic and transonic speeds. Ministry of Aviation; Royal Aircraft Establishment; RAE Farnborough.

Stallings Jr, R. L., & Wilcox Jr, F. J. (1987). Experimental cavity pressure distributions at supersonic speeds.