

Effect of Inlet Velocity Profile in Simulation of Flow Development; Wind Tunnel Contraction

A. T. Teseletso, M. Namoshe, N. Subaschandar

Abstract— Contraction optimisation is one facet in designing an efficacious wind tunnel, and this vital step is usually accomplished by Computational Fluid Dynamics(CFD) where input data greatly affect the output. In this study, a 2D contraction derived from a sixth-order polynomial is simulated in two cases. One scenario involves the use of a User Defined Function (UDF) to set the velocity-inlet boundary condition, whilst the other is defined as constant velocity. These cases are compared to analyse how the different inlet-boundary conditions affect the flow development, the sizes of recirculation zones and boundary layer thickness for a 2D wind tunnel contraction. The study shows that the case with UDF gives a better representation of the simulation than the other case.

Index Terms—Boundary conditions, Computational Fluid Dynamics, Contraction, Inlet velocity profile, User Defined Functions, Wind tunnel.

I. INTRODUCTION

Computational Fluid Dynamics (CFD) software is attributed to cutting costs and simplifying optimisation and design of aerodynamic structures such as wing-body and aerofoils, which in pre-CFD era, their designs were based on data from experimental databases such as NACA [1]. Currently, computational fluid models development is poised at modelling complex fluid flows, and still experimental validation is a necessity [2]. In spite of the fact that the main objective of CFD is to eliminate costs associated with experiments in the design process, experimental validation is done to ensure that the modelling is consistent with the physics. Hence if the models are dependable then the model may err due to input data, for instance boundary conditions, not accurately representing the real scenario.

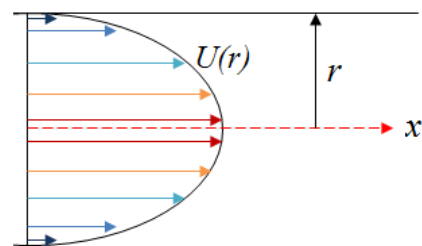
During optimisation study of wind tunnel contraction, only the contraction is modelled. In most cases reason being to reduce the computational domain; in turn, lessen the computational requirements and reduce complexity. Since the contraction is attached to other parts up and downstream, while setting the inlet boundary conditions it is worth noting that the part before it had effect on the fluid flow properties before reaching the contraction inlet. Not factoring in those dynamics

might result in incorrect boundary conditions being set, which can lead to misrepresentation of the whole scenario being modelled. Consequently, the use of a User Defined Function helps avert this problem. Also the contraction inlet side can be extended stream-wise by a factor of the hydraulic diameter or contraction length to enable the flow enough length to fully develop before reaching the contraction inlet.

In demonstration by [3], [4] the use of a UDF effectively represents the fluid flow as expected if experimented under the same scenarios.

The trend in contraction optimisation cases is that frequently, a constant one directional velocity magnitude is set to define inlet-velocity boundary conditions which does not satisfy the physics [5]–[8]. In consideration of the above cases, it is similar to a case of fluid flow in a duct [9]. The velocity profile has been developing in the inlet section and then reached the fully developed condition, which is characterized by the parabolic profile for laminar flow described by equation (1).

$$U(r) = -\frac{1}{4\mu} \left(\frac{dP}{dx} \right) r_0^2 \left[1 - \left(\frac{r}{r_0} \right)^2 \right] \quad (1)$$



$$U_{max} = -\frac{1}{4\mu} \left(\frac{dP}{dx} \right) r_0^2 \quad (2)$$

Where $\frac{dp}{dx}$ represents the change in pressure from inlet to outlet per unit length, which can sometimes be represented as;

$$\frac{\Delta P}{\Delta x} = \frac{P_2 - P_1}{x_2 - x_1} \quad (3)$$

In this paper, air flow through a 2D contraction developed from a sixth order polynomial [2] is simulated, with velocity-inlet boundary condition defined by a UDF and results compared to that of a constant velocity.

II. SIMULATION PRE-PROCESSING

A. Geometric Modelling

ANSYS 16 Package was used for geometrical modelling and virtual simulation. A 2D contraction cone of inlet height of 1m, outlet of 0.4m and length of 1 m was modelled. Its exit was

Manuscript received September 25, 2016.

A. T. Teseletso is with the Mechanical, Energy and Industrial Engineering Department, College of Engineering and Technology, Botswana International University of Science and Technology, Palapye, Private Bag 16

M. Namoshe is with the Mechanical, Energy and Industrial Engineering Department, College of Engineering and Technology, Botswana International University of Science and Technology, Palapye, Private Bag 16

N. Subaschandar is with the Mathematics and Statistical Sciences Department, College of Science, Botswana International University of Science and Technology, Palapye, Private Bag 16

extended to test section by 1m to visualise the fluid flow in the test section. A thousand(1000) key points were plotted to represent the sixth order polynomial as accurately as possible. Three stations named; contraction exit, mid-test section (1.5m from inlet)and exit (0.25m before outlet)respectively from inlet to outlet were positioned as shown below.

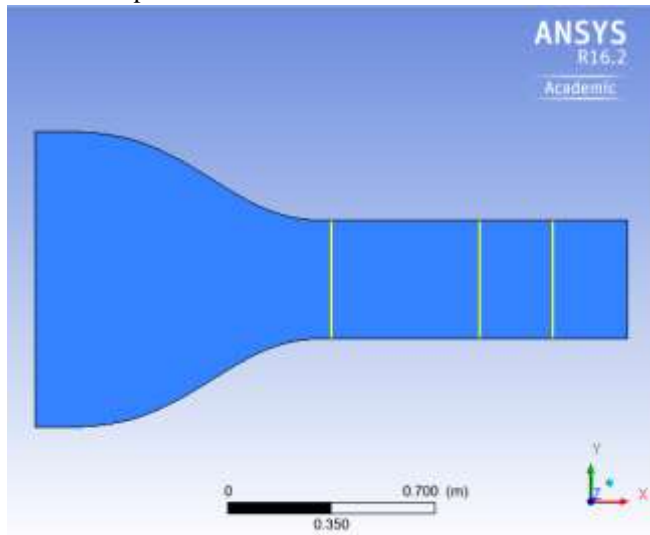


Fig. 1 Modelled geometry

B. Meshing

A Quadrilateral dominant structured -mesh of 147000 elements and 296000 nodes was generated on the geometry as illustrated inFig below. An inflation layer of 20 layers and growth rate of 1.1 was added to capture the flow closer to the wall.

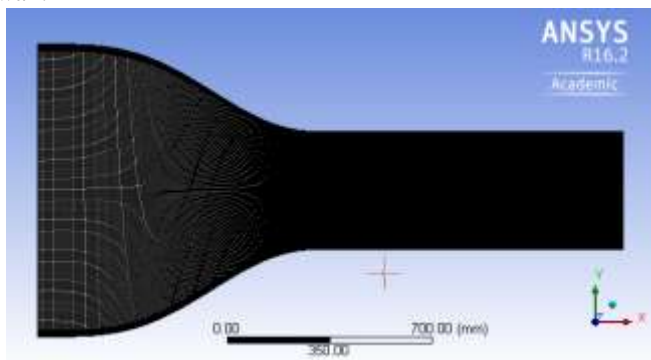


Fig. 2 Meshed Geometry

III. COMPUTATION

A. Model Selection

The flow was modelled using RNG k-epsilon model with enhanced wall treatment. Through the mathematical technique called “renormalisation group” (RNG) methods, this model was derived from the instantaneous Navier-Stokes equations to create a model which has laudable performance and accuracy for rapid strained flows, swirling flows and reliable for a wider class of flows compared to the standard k-epsilon model [10].

B. Boundary Conditions

TABLE I
BOUNDARY CONDITIONS

Zone	Type	Value
Outlet	Pressure-outlet	101325 Pa
Wall	Stationary wall	No slip
Inlet	Velocity-inlet	3.2 m/s
UDF defined		

For a case where the inlet velocity is set as constant, the magnitude is set as 3.2 m/s whilst for the UDF case a C code of equation (1) with $U_{max} = 3.2$ m/s defining the velocity x-component. Turbulent intensity and hydraulic diameter is set as 1% and 1m for both inlet and outlet zones.

C. Convergence

Computation converged after 4365 and 573 iterations for the case with a UDF and Constant defined inlet velocity respectively.

D. Mass Flow Rate

To validate the simulation the mass flow rate at inlet and out are compared, and if their difference is less than 1% then continuity is conserved.

TABLE II
MASS FLOW RATE (KG/S)

Zone	UDF case	Constant Velocity Case
Inlet	2.6133539	3.9200001
Outlet	-2.613354	-3.9200001
Net	-6.3895057e-08	9.23788e-09

E. Grid Independence

Two more mesh qualities were generated to enable grid independency checks, which are essential in determining if the results are reliant on the mesh or not. Mesh qualities generated one has 10% less elements and the other 10%

The results from the other two mesh qualities obtained were the same as the ones of the selected mesh quality, hence validates that the simulation results are independent of the mesh.

F. Uniformity

The deviation of the individual velocity magnitudes (u_i) along the mid-test section from the mean velocity (\bar{u}) gives the velocity uniformity at mid-test section, where the $U_\sigma = 0$ is the maximum uniformity and reduces as the value goes up.

$$U_\sigma = \sqrt{\frac{(u_i - \bar{u})^2}{N}} \quad (4)$$

The UDF case gives a $U_\sigma = 1.1$ and the other 1.2.

IV. RESULTS AND DISCUSSIONS

Fig below shows the velocity contour of the two cases (a) constant inlet velocity and (b) User Defined Function velocity. For all illustrations that follow these labels are used to differentiate between the results of the cases.

The same velocity magnitude from the wall to the centre is depicted by (a) at the inlet. This does not correctly represent the real life scenario as it does not account for the settling section.

The scenario (b) is consistent with the physics as the “fully developed” condition is well depicted at inlet. The xy-plot of

the velocity at inlet is shown in Fig below.

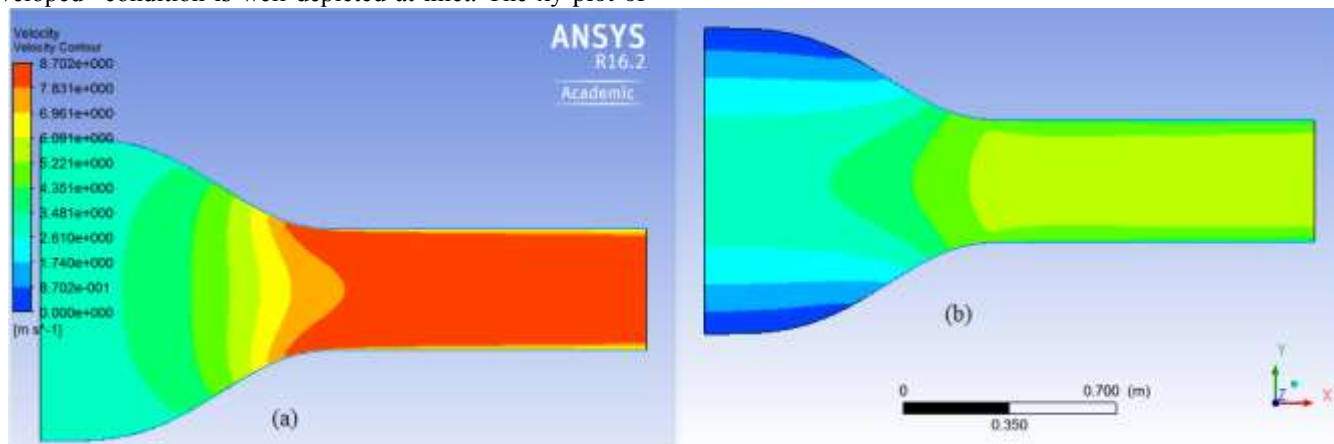


Fig. 3 Velocity Contours

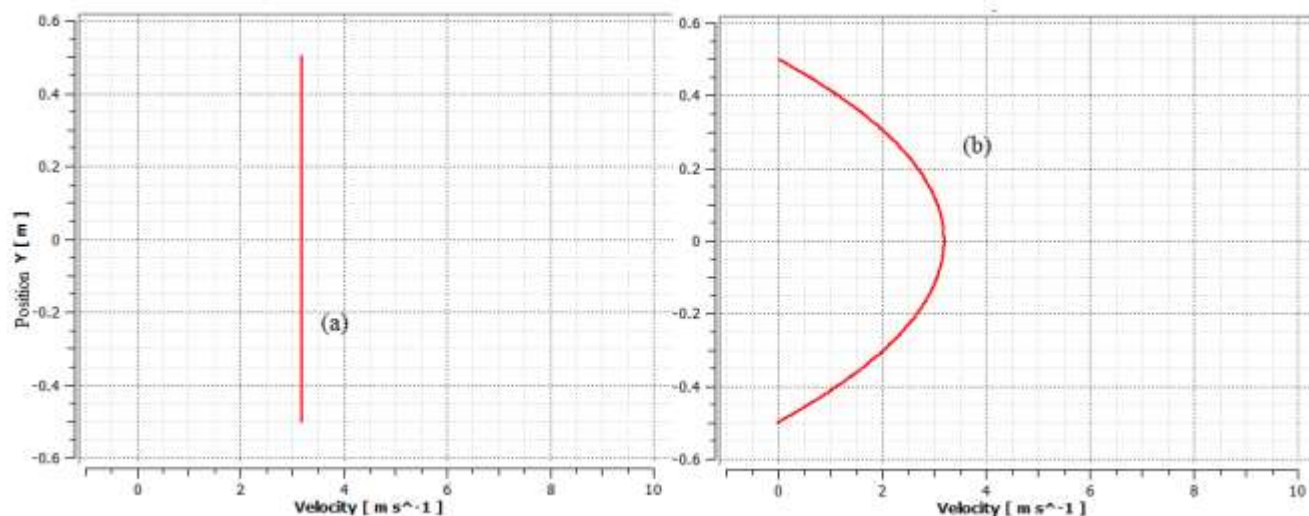


Fig. 4 Inlet velocity magnitudes at positions Y of Inlet

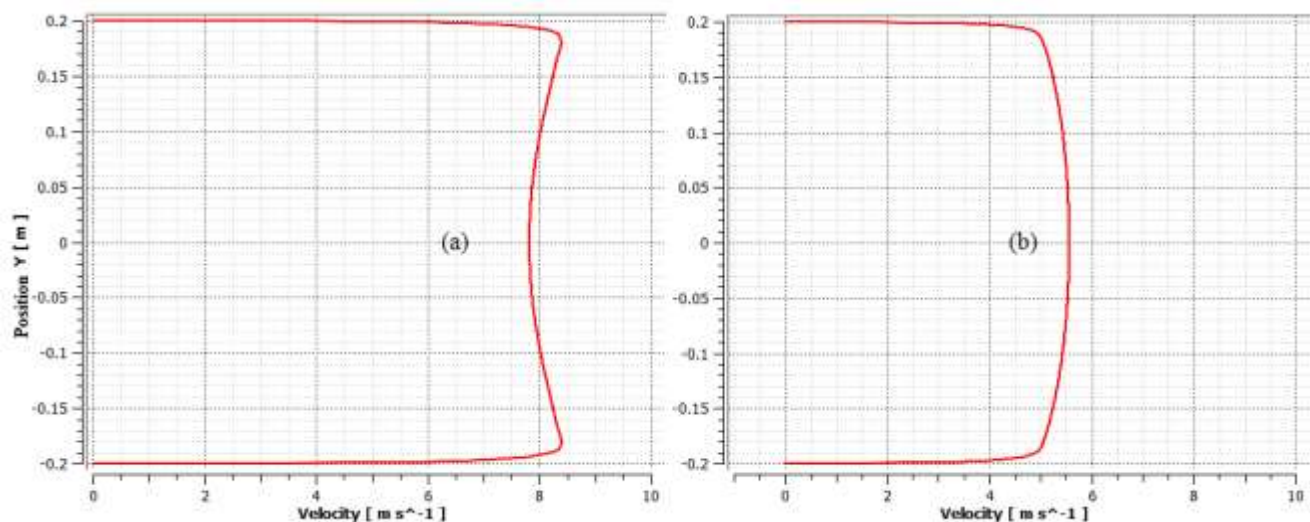


Fig. 5 Velocity magnitude at Contraction exit

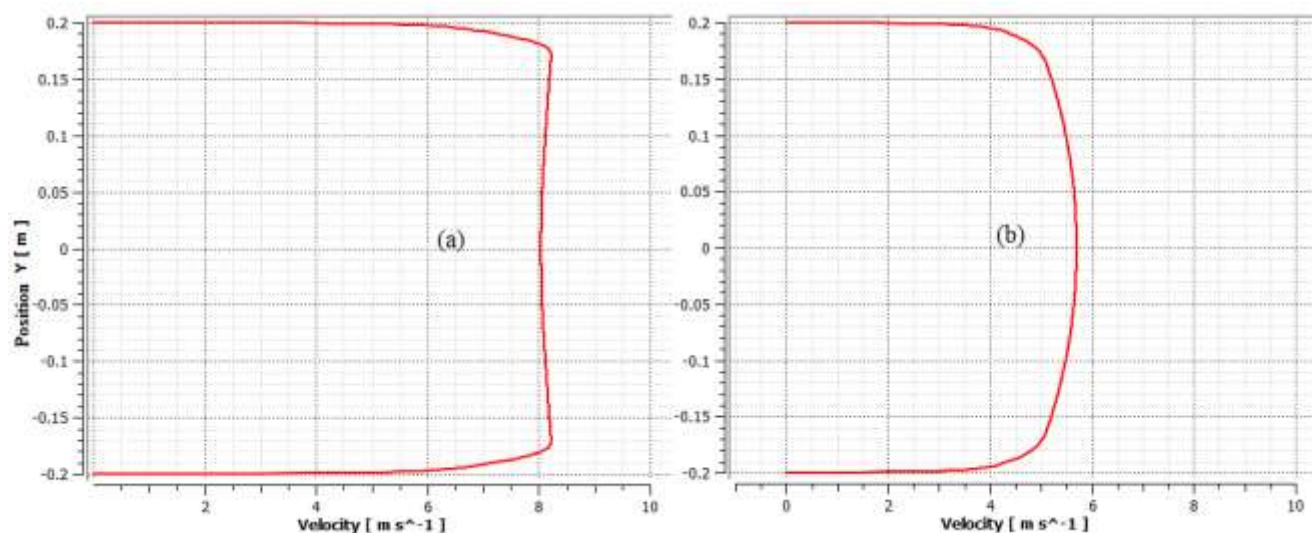


Fig. 6 Velocity magnitude at mid Test section

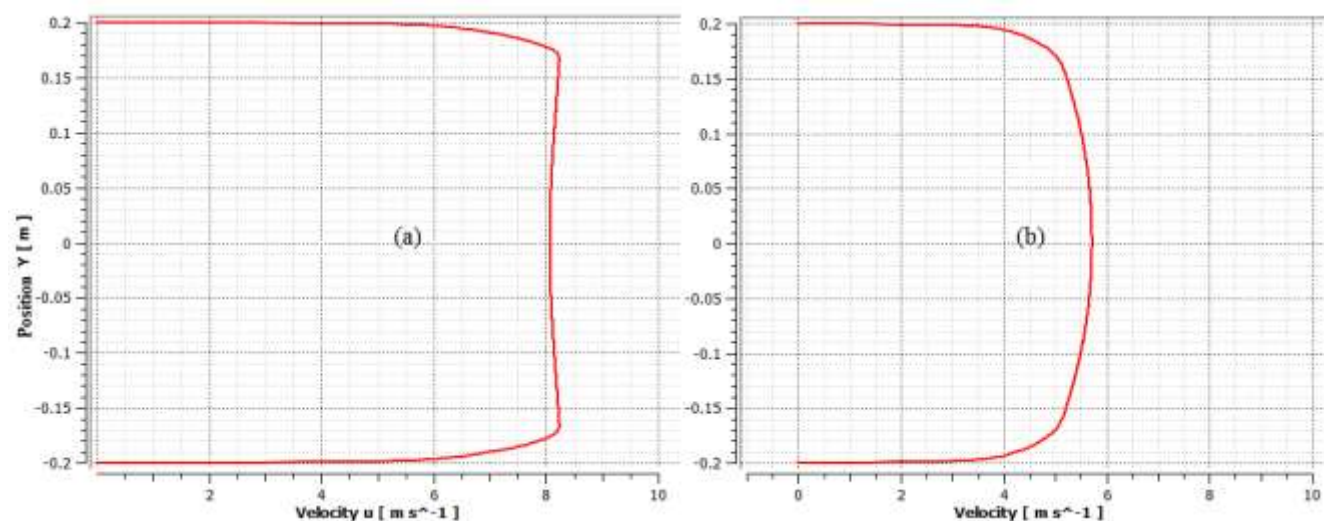


Fig. 7 Velocity at Exit station

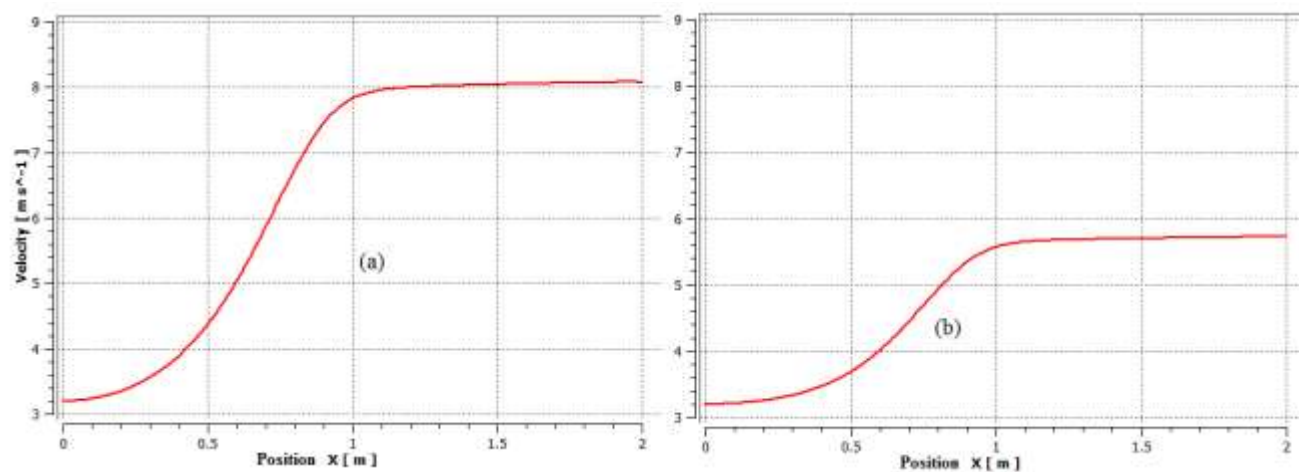


Fig. 8 Velocity along the symmetry

Contraction exit, mid-test section and exit velocity plots of the two cases are shown in figures above. The exit station is

placed 0.25m upstream to avoid disruptions at the outlet.

For the case with constant inlet velocity the profile is

unrealistic as the centre point where the velocity is expected to be the highest is even lower than that of the position close to the wall. The UDF scenario gives the expected results of the scenario.

The velocity along the symmetry of the contraction and test section rapidly increases to 8 m/s for case (a), while for the other case the velocity gently increases from 3.2m/s to 5.6m/s.

The pressure distribution of the two cases is shown in Figbelow. The first scenario shows a higher pressure gradient, the lowest being at the exit of the contraction which suggests boundary layer growth or flow separation.

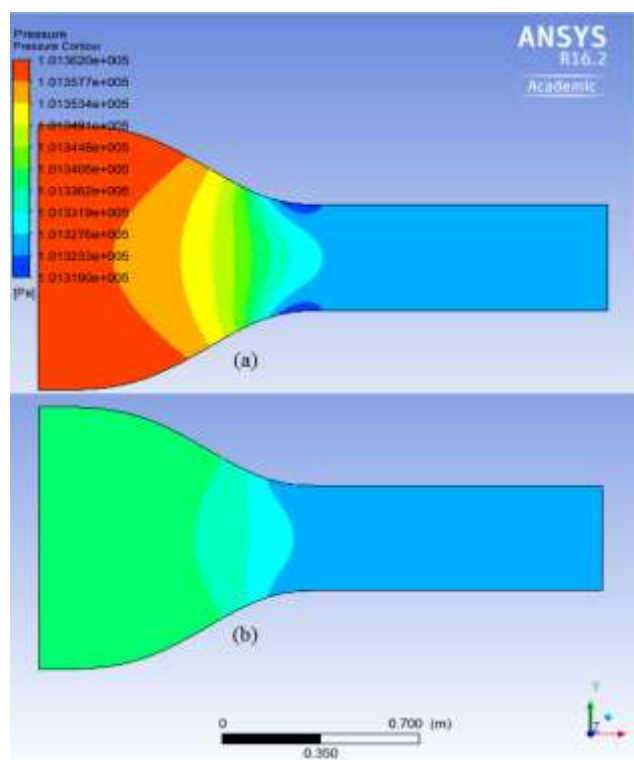


Fig. 9 Pressure Contour

V. CONCLUSION

The performance of the contraction for two inlet velocity profiles is studied using computational fluid dynamics. The use of a UDF to define the velocity inlet boundary in simulation of fluid flow through contraction greatly increases the chances of achieving an optimal contraction contour in case of optimisation, as it is a close representation of the actual flow when compared to constant magnitude defined velocity inlet boundary. This shows that the uniform inlet velocity does not result in a realistic solution inside the contraction. This work lays the foundation for optimisation of 3D arbitrary contraction profiles through CFD.

REFERENCES

- [1] S. S. Desai, "Relative roles of computational fluid dynamics and wind tunnel testing in the development of aircraft," *Curr. Sci.*, vol. 84, no. 1, pp. 49–63, 2003.
- [2] J. E. Sargison, G. J. Walker, and R. Rossi, "Design and calibration of a wind tunnel with a two dimensional contraction," *15th Australas. Fluid Mech. Conf.*, no. December, p. 4, 2004.
- [3] S. Majhi and G. Visavale, "Writing a User Defined Function (UDF) for CFD Modeling," 2016. [Online]. Available: <https://www.learncax.com/knowledge-base/blog/by-category/cfd/writing-a-user-defined-function-udf-in-ansys-fluent>. [Accessed: 23-Sep-2016].
- [4] R. Kobeissi, *CFD User defined function - UDF in Fluent tutorial*. England, 2014.
- [5] S. Kumar, V. K. Dwivedi, J. Kumar, and R. Goswami, "Design and simulation of Open Circuit Blowdown type Wind Tunnel," *Int. Congr. Comput. Mech. Simul.*, no. December, pp. 10–12, 2012.
- [6] D. E. Ahmed and E. M. Eljack, "OPTIMIZATION OF MODEL WIND - TUNNEL CONTRACTION USING CFD," *Int. Conf. Heat Transf. Fluid Mech. Thermodyn.*, no. July, pp. 87–92, 2014.
- [7] K. Javed and M. Ali, "Design & Construction of subsonic wind Tunnel focus on two dimensional contraction cone profile using sixth order polynomial," 2011.
- [8] a. S. Abdelhamed, Y. E.-S. Yassen, and M. M. ElSakka, "Design optimization of three dimensional geometry of wind tunnel contraction," *Ain Shams Eng. J.*, vol. 6, no. 1, pp. 281–288, 2015. <https://doi.org/10.1016/j.asej.2014.09.008>
- [9] Y. A. Cengel and J. M. Cimbala, *Fluid mechanics*. McGraw-Hill Science/Engineering/Math, 2004.
- [10] K. Sulaksana and P. Thamkeng, "Heat Transfer & Fluid Flow Simulation with ANSYS."



Atamelang T. Teseletso Postgraduate student, Mechanical, Energy and Industrial Engineering Department, College of Engineering and Technology, Botswana International University of Science and Technology, Palapye.



Molaletsa Namoshe Lecturer, Mechanical, Energy and Industrial Engineering Department, College of Engineering and Technology, Botswana International University of Science and Technology, Palapye.



N. Subaschandar Senior lecturer, Mathematics and Statistical Sciences Department, College of Science, Botswana International University of Science and Technology, Palapye.