Numerical Analysis of inlet vortex in the scaled engine

M Biglarian¹, A Najafi²*, and S M Negharchi³

¹ Department of Mechanical Engineering, Sharif University of Technology, Tehran, Iran
² Department of Wind Engineering, Flensburg University of Applied Sciences, Flensburg, Germany
³ Department of Mechanical Engineering, Babol Noshirvani University of Technology, Babol, Iran

* E-mail: ahmadreza.najafi@stud.hs-flensburg.de

Abstract. A jet engine that works near the ground and generates high thrust at low speed can experience the flow separation from the ground surface up to its inlet and, as a result, the formation of a famous vortex. This vortex extended from the ground surface to the fan inlet is known as the inlet vortex. In the present study, a jet engine model scaled down by 1/30 relative to the real jet engine with 3 million hexahedral elements is simulated using Computational Fluid Dynamics (CFD). The obtained results agree with empirical relationships, and the inlet vortex can be analyzed in scales much lower than the original prototype. Also, the inlet vortex gets weakened by increasing free stream velocity and completely degraded at higher velocities. By understanding how this phenomenon occurs and can be dealt with, the Foreign Object Damages (FOD) such as compressor surge, fan vibration, and particle ingestion into the engine core can be prevented.

Keywords: Computational Fluid Dynamics (CFD), Jet Engine, Inlet Vortex, Foreign Object Damages (FOD).

1. Introduction

Since the emergence of jet engines in airlines in 1950, the intake of foreign objects into the engine inlet has been one of the popular subjects of different studies [1]. When the engine works in static or pseudo-static modes (during taxiing and take-off), vortices may be formed between the engine and its inlet duct [2]. The inlet vortex has been well-known for six decades; however, its mechanism is not well understood, and different techniques are developing to prevent this phenomenon [1]. The collision of the inlet vortex and flow separation at the inlet lip significantly affects the engine performance [1]. In some cases, foreign object damage (F.O.D.) to the engine blades occur because vortex pitch debris into the intake section [3]. The stagnation pressure loss and flow deterioration at the fan inlet are among the effects of this phenomenon. So far, different tools have been invented to avoid or eliminate the inlet vortex. However, none of them are successful for all modes of external flow. The first system developed to prevent the inlet vortex formation was based on injecting excessive high-pressure air from the compressor to the ground surface near the stagnation point. By entering the stagnation point region near the ground, this airflow weakens and disappears the inlet vortex [4].
2. Geometry Specification
A 1/30 scaled engine model is simulated using CFD methods to recognize the inlet flow patterns in some regimes of free-stream flow in a real engine. This geometry is produced by CATIA software, and then it is imported to the ANSYS software environment. In this simulation, the inlet duct of the jet engine is considered two co-centric cylinders connected through an oval-shaped lip (Figure. 1). As shown in Figure 1, the inlet diameter (Di) and the height of the duct center above the ground (H) are 1/30 of the actual dimensions.

Since the effect of free-stream flow must be considered correctly; therefore, a cubic space around the jet engine is considered according to Figure 2 [5]. Dimensions of this cubic domain are 22 times as much as the characteristic length (inlet duct diameter, Di) in each direction. In Figure 2, the jet engine is located on the tailwind face, and other faces are named according to the flow passing through them; for example, the flow that comes from the engine upstream and goes downstream is named as follows: upwind and downwind. The inner and outer parts of the inlet duct of the jet engine and the ground, which are solid surfaces, are specified by black color in the Figure.

![Figure 1. Jet engine inlet modeled as a cylinder](image1.png)

![Figure 2. Applied computational domain and equivalent jet engine inlet like a cylinder](image2.png)

3. Jet engine scaling
All the dimensionless numbers must be constant for scaling the real model according to similitude and scaling laws. Two dimensionless groups which are obtained by similitude analysis are Reynolds number and Mach number. Reynolds number and Mach number are $5.6 \times 10^6$ and 0.388, respectively. If the real model is scaled down by 1/30, the flow velocity must be increased proportionally to keep the Reynolds number constant. In this case, the Mach number increases by 30 folds, and the flow regime transfers from the subsonic range to the supersonic one where Navier-Stokes equations are not valid anymore. Since Mach number is of greater importance than Reynolds number, Mach number is kept constant, and Reynolds number is multiplied by 1/30. Although Reynolds number changes, this simulation can be conducted because the flow regime is still turbulent.

4. Mesh generation and mesh independence study
The ICEM CFD software has been used to generate the structural mesh. In the present work, we first divided the geometry into smaller parts using the Multi-Block feature (Figure 3) and then, The mesh at the core of the inlet and around the lip was created using the O-GRID and C-GRID methods, respectively (Figure 4) [6]. To detecting better the effects of the inlet lip flow characteristics, the number of grids in this part is higher and gradually moving away from this area, the adjacent cells become larger (Figure 5). Also, the height of the generated meshes has been reduced all around the inlet pipe and near the ground to capture the effects of the boundary layer precisely (Figure 6) [7]. The number of generated mesh is about three million cells.
The location of minimum static pressure on the plane close to the ground must be examined for making solutions independent of the grid size. Figures 7 and 8 indicate the displacements in the x and z directions, respectively, which are non-dimensionalized relative to the inner diameter ($D_i$). As can be seen, two grids with 3 and 6 million have not significant changes relative to each other, and as a result, the grid of 6 million cells is selected as the simulation grid, and all analyses are conducted on this grid.

![Figure 3: Using the Multi Block method](image)

**Figure 3:** Using the Multi Block method

![Figure 4: generated mesh with O-GRID and C-GRID method around the inlet pipe](image)

**Figure 4:** generated mesh with O-GRID and C-GRID method around the inlet pipe

![Figure 5: generated mesh around the edge of the](image)

**Figure 5:** generated mesh around the edge of the

![Figure 6: small grid near the ground and around](image)

**Figure 6:** small grid near the ground and around
5. Numerical simulation

ANSYS Fluent software is used to simulate the fluid flow based on the finite volume method. The pressure outlet boundary condition is applied to the duct outlet. In this section, the engine inlet fan is not considered for simplicity in calculations. In pressure outlet boundary conditions, the static pressure is calculated by equation (1) as follows.

\[
\frac{P_0}{P_s} = (1 + 0.5 \times (K - 1) \times M^2)\frac{K}{K-1}
\]

Where \(P_0\) is the stagnation pressure, \(P_s\) is the static pressure, \(K\) is the ratio of specific heat at constant pressure to that at constant volume, and \(M\) is the Mach number. Based on the given Mach number at engine inlet and total pressure, the static pressure at the engine inlet is obtained. Pressure far filed boundary condition is considered for five faces of the cubic domain, and the flow Mach number is calculated based on the free-stream velocity in each case. In this study, three values of free-stream velocity are assumed as 0, 4, and 6 m/s. The no-slip and adiabatic conditions are also considered for domain walls. Air is assumed as a compressible fluid, and its density is obtained from the ideal gas law. \(k-\omega\) SST (Shear Stress Transport) model is used to simulate the turbulent flow[8,9]. This model applies the \(k-\omega\) method for viscous sublayer and the \(k-\epsilon\) method for free-stream flow; for this reason, it is suitable for analyzing the flow separation and predicting the vortex formation. Also, this method is in good agreement with experimental data of vortex detection [10]. The available high-performance RAM allows to use of the coupled technique for the pressure-velocity correlation; because this coupling method occupies a large portion of the RAM due to simultaneously solving all equations. All scalars other than pressure are discretized by the second-order upwind discretization scheme to achieve higher accuracy. Also, the pressure is discretized by the PRESTO method because this method is very efficient for high-pressure gradients. The pseudo-transient method with time steps of \(10^{-5}\) s is also applied to accelerate the convergence rate.

6. Results and discussions

In the case without outlet flow, the inlet vortex is formed, as shown in Figure 9. As can be seen, the vortex that separates from the ground surface and enters the inlet duct is observed clearly.

In order to specify the vortex center and the region where it is formed, a plane is created near the ground, and the curl characteristics of the flow field (fluid rotation) are examined in this place, as shown in Figure 10. This plane is parallel to the ZX plane (top view), and the inlet duct can be seen.
As it is observable, the generated vortex rotates around the y-axis (component perpendicular to the ground), the center of the vortex has the most significant rotational motion, and it is specified by black color. The tangent streamlines to this plane show that the flow taken from the surroundings separates from the ground right in the downside of the inlet lip and forms the inlet vortex.

Figures 11 and 12 indicate the local temperature variations from the top view (ZX plane) and side view (ZY plane). Based on Figure 11, the lowest temperature is attributed to the flow separation and inlet vortex formation; this reduction proceeds in Figure 12, and the temperature decreases at the inlet lip where the flow velocity reaches its maximum to Figure 9.

![Figure 9. Side view of the inlet vortex formation in the case without external flow](image1)

![Figure 10. Y component of flow field velocity](image2)

![Figure 11. Top view of temperature contour parallel to ZX plane](image3)

![Figure 12. Inlet vortex and temperature variation on it](image4)

When the relative humidity of air is high (close to 100%), or the vapor particles are rich in the air, the temperature can be reduced to the dew point temperature and cause the water droplets to be formed [11]. The formation of these droplets can detect the inlet vortex.

The formation of this vortex leads to the flow asymmetry at the inlet, which can cause the fan vibration and corrosion of compressor blades.

According to equation (2) [1], the boundary of the inlet vortex formation is specified by dimensionless velocity and height. $V_i$ and $V_\infty$ are the inlet velocity and external flow velocity, respectively. Based on this equation and for the dimensionless height of 2.6 (in this engine) and the inlet velocity of 132 m/s, the minimum free-stream velocity is 3 m/s to avoid forming an inlet vortex.
\[
\frac{V_i}{V_\infty} = 24 \times \frac{H}{D_i} - 17
\]  

(2)

Figure 13 shows the flow pattern at the engine inlet for a headwind flow with a velocity of 4 m/s. As can be seen, the inlet vortex is not created based on empirical relationships. In the following, the flow is indicated for the free-stream velocity of 6 m/s. In this case, all streamlines enter the engine core without separation from the ground surface.

Figure 13. Inlet flow pattern for a free-stream velocity of 4 m/s

Figure 14. Inlet flow pattern when the free-stream velocity is 6 m/s

7. Conclusion

In this study, the inlet vortex was analyzed by ANSYS FLUENT software for a model scaled down by 1/30 relative to the real model of a jet engine. The obtained results showed that the inlet vortex is formed in the absence of free-stream flow and can be extended to the inlet duct of the engine. The flow separation occurs under the inlet lip on the ground. Also, the evaluation of temperature variation indicated that the fluid temperature could be reduced to the dew point temperature in the inlet vortex core, and the high value of the relative humidity led to the formation of water droplets. According to the presented empirical relationship, which determines the threshold of the inlet vortex formation using the dimensionless velocity and height, the free-stream velocity to avoid the formation of inlet vortex was equal to 3 m/s in this jet engine. The numerical results for flow velocity higher than this threshold show that the inlet vortex is not formed. It should be noted that the proposed model is scaled down by 1/30 relative to the real model; however, it can result in solutions that agree well with the results of empirical relationships. This scaling level can decrease the computation costs and accurately predict the formation of the inlet vortex in different types of engines. Therefore, different methods to prevent vortex formation and the inlet flow behaviour can be analyzed.

8. Reference

[1] N. Horvath, Inlet Vortex Formation Under Crosswind Conditions, 3 (2013).
[2] J.P. Murphy, D.G. MacManus, Inlet ground vortex aerodynamics under headwind conditions, Aerosp. Sci. Technol. 15 (2011) 207–215. https://doi.org/10.1016/j.ast.2010.12.005.
[3] W.H. Ho, A Consolidated Study Regarding the Formation of the Aero-Inlet Vortex, Stud. Comput. Intell. 416 (2012) 345–364. https://doi.org/10.1007/978-3-642-28888-3_14.
[4] L.G. Trapp, R. Da Motta Girardi, Crosswind effects on engine inlets: The inlet vortex, J. Aircr. 47 (2010) 577–590. https://doi.org/10.2514/1.45743.
[5] J. Chen, Y. Wu, O. Hua, A. Wang, Research on the ground vortex and inlet flow field under the ground crosswind condition, Aerosp. Sci. Technol. 115 (2021) 106772. https://doi.org/10.1016/J.AST.2021.106772.
[6] Z. Ali, P.G. Tucker, Multiblock Structured Mesh Generation for Turbomachinery Flows, (2014). https://doi.org/10.1007/978-3-319-02335-9.
[7] Z. Ali, J. Tyacke, P.G. Tucker, S. Shahpar, Block topology generation for structured multi-block meshing with hierarchical geometry handling, Procedia Eng. 163 (2016) 212–224.
https://doi.org/10.1016/j.proeng.2016.11.050.

[8] R.W. Winfree, The Application of Computational Fluid Dynamic Analysis to Jet Engine Inlet Flow Quality, J. Phys. A Math. Theor. 44 (2011) 1689–1699. https://doi.org/10.1088/1751-8113/44/8/085201.

[9] M. Biglarian, M. MomeniLarimi, B. Ganji, A. Ranjbar, Prediction of erosive wear locations in centrifugal compressor using CFD simulation and comparison with experimental model, J. Brazilian Soc. Mech. Sci. Eng. 41 (2019) 1–10. https://doi.org/10.1007/s40430-019-1610-5.

[10] D.E. Glenny, N.G.T.E. Pyestock, Ingestion of Debris into Intakes by Vortex Action, Aeronaut. Reasearch Counc. CP No. 111 (1970).

[11] C.J. Johns, The aircraft engine inlet vortex problem, AIAA’s Aircr. Technol. Integr. Oper. 2002 Tech. Forum. (2002) 1–13. https://doi.org/10.2514/6.2002-5894.