

Experimental and numerical simulation of a moving carrier (FX63-137) through an air tunnel

W M Shibani¹, M F Zulkaffli¹, B Basuno¹, M F Abdul Ghafir¹, Z Subari¹

¹Department of Aeronautical Engineering, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia 86400 Parit Raja, Johor, Malaysia

E-mail: wanismustafa@yahoo.com

Abstract. This study investigated the movement of a carrier through an air tunnel using numerical simulation and experimental approach conducted in UTHM laboratories. The carrier was designed to have a particular trajectory moves along the air tunnel. The body of the main carrier has a shape of cube and rectangular wing of aspect ratio 2 is set in the middle of the body which has a shape of airfoil FX63-137 as its airfoil section. Experimental result of the carrier movement was investigated and compared with the simulation result obtained by Fluent's software. The experiment was conducted using a flow speed of 19.59 m/s at the inlet station. In the simulation, the flow is considered as an internal flow and has a constant temperature at the entry station. The pressure ratio between the inlet and outlet of the air tunnel is 235 N/m². In addition, that the flow is considered as turbulent flow and the present study used $k - \epsilon$ as its turbulence modelling in the numerical simulation. Comparison result between experiment and Fluent software in term of its trajectory and the speed of the carrier movement are found in a good agreement.

1. Introduction

Increasing human activities in the last two decades facilitated the significant growth of transportation of goods. Such a rapid increase in the demand for goods transport may seriously affect modern life. Present transportation systems, such as roads and railways, commonly involve route transportation. These transport systems involve problems such as traffic congestions, accidents, air and noise pollutions, and damage to roads and bridges. These issues suggest the necessity of developing an alternative way of transporting goods from one place to another. Transportation through air tunnels may provide a solution to this problem. This approach will not interfere with the existing transportation system.

The present study is to investigate feasibility of such transportation as an alternative way of delivering goods. In our approach, the goods are deliver using a carrier that float and move inside a straight air tunnel. The study of such carrier moving through an air tunnel is carried out experimentally and numerically, and their results are compared to each other. The carrier was designed as a cube-body attached with wings. In this respect, the flow problem in hand can be considered as the problem of a flow past through a wing-body configuration. As a result, their aerodynamics characteristics will be influenced by geometry parameter, such as aspect ratio, taper ratio, twist angle, wing incidence, the leading swept angle, etc.

In the numerical simulation, the movement of the carrier through air tunnel was evaluated by using Computational Fluid Dynamic (CFD) software, ANSYS Fluent. Here, the governing equations of fluid motion is treated in the form of Time Averaged Navier-Stokes Equations (TRANS). These



equations were supplied by additional equation called as turbulence modelling to represent the presence of the Reynolds stress term. There is various turbulence modelling embedded in the Fluent Software such as; one equation model Spalart-Allmaras, two equation models like Standard $k - \varepsilon$, RNG $k - \varepsilon$, Realizable $k - \varepsilon$, etc. The study on the use of various turbulence model can be referred to work done by Lakehal and Rodi [1], Iaccarino et. al [2] and Yakhot et. al [3]. On the other hand, the method of how the TRANS is solved numerically can be referred to Tannehill et. al [4]. The present work uses a Fluent software version 16.1 with standard $k - \varepsilon$ as its turbulence modelling. Gosman[5] indicated that this turbulence modelling represents the most well-known turbulent model, simple and has capability for capturing turbulence transport occurrence.

The use of airfoil section FX 63-137 in the present work is adopted from work provided by Kanyako et. al [6]. Their research work found that among three types of airfoils, namely, FX 63-137, E387, and S826, the FX 63-137 airfoil provides the highest C_L/C_D ratio compared with the others. The reason is probably the fact that this particular airfoil exhibits a gradual stall compared with the other airfoils [6]. Extensive researches mostly conducted in the aircraft manufacturer industries have the intention to obtain wing configuration with higher maximum lift to drag ratio [7, and 8].

2. Description of the model

Figure 1 shows the cube shape carriers with wings attached on both of its sides. The cube are measures $0.1 \text{ m} \times 0.1 \text{ m} \times 0.1 \text{ m}$. The wing has a chord length of 0.05 m and it is equal to its exposed half wing span. The wings are placed in the middle of the cube with its wing incidence angle of 18° . The total mass of the carrier is 100 grams. Figure 2 shows the air tunnel used in this study. It has a length of 3 m and a circular cross section with a diameter of 0.3 m . The carrier (object) was placed initially on the leftmost side of the tunnel.

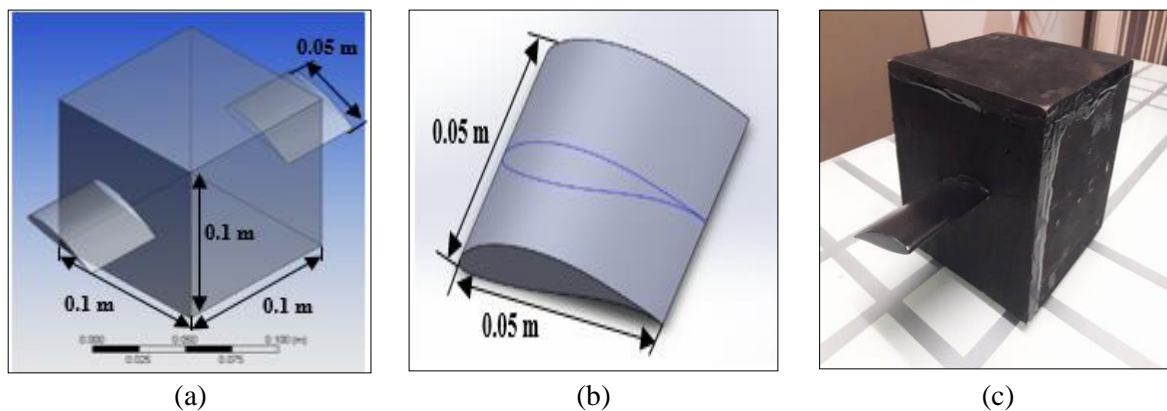


Figure 1. The models (a) The carrier, (b) The wing has a rectangular shape of FX63-137 airfoil section, and (c) The real carrier.

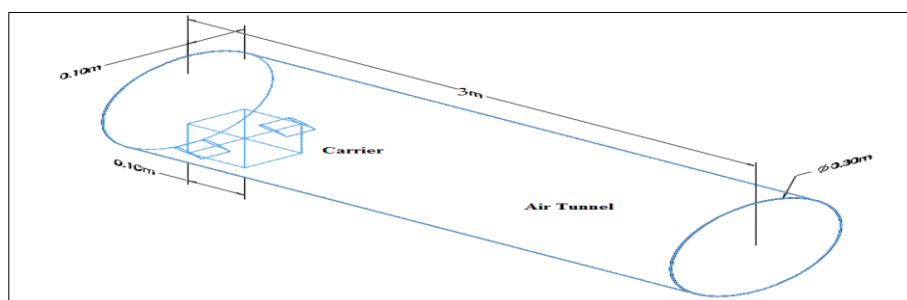


Figure 2. Three-dimensions view of the air tunnel and the carrier.

3. CFD methodology

The equations that govern the unsteady flow of an incompressible fluid are

$$\nabla \cdot \mathbf{u} = 0 \quad (1)$$

$$\rho \frac{D\bar{u}_i}{Dt} = F_i - \frac{\partial \bar{p}}{\partial x_i} + \mu \Delta \bar{u}_i - \rho \left(\frac{\partial \bar{u}'_i \bar{u}'_j}{\partial x_j} \right) \quad (2)$$

where ρ denotes density; F is force, μ is molecular viscosity, Δ is vector operator, \bar{u} is time-averaged velocity components, \bar{p} is time-averaged pressure, and u' is fluctuating velocity components and index i, j refer to 1, 2, 3 indices for the three components in a Cartesian coordinate system.

Equations (1) and (2) are the well-known Time-Averaged Navier-Stokes Equations (TRANS). ANSYS Fluent [9] has been used in the present study with turbulence model $k - \epsilon$ as proposed by Launder and Sharma [10]. Guan and Park [11], & Mohammadi and Pironneau [12] have choose the $k - \epsilon$ model after comparing with three models and they found that the $k - \epsilon$ model are more accurate and robust. This model was also used by Chern and Wang [13] in their investigation of the flow rate of the ball valve. Such approach had been applied to study the object movement along the air tunnel as presented in [14 - 16].

The governing equation of fluid motion presenting in the form of TRANS as in Eqn. (1) and (2) can be solved using various numerical scheme such as SIMPLEC, SIMPLER, SIMPLEST, CTSSIMPLE, PISO or SIMPLE. Each has its advantages and drawbacks [17 - 19]. The present work uses a SIMPLE scheme (Semi-Implicit Method for Pressure-Linked Equations).

3.1. Boundary conditions

The physical boundary condition of these problems are as follows:

- 1) Each end of the air tunnel is considered as pressure inlet and outlet, while the surfaces of the carriers and the air tunnel is considered as no-slip wall boundary condition, i.e., $u = 0$. (See Figure 3).
- 2) "Pressure inlet" was constantly set to 235 N/m² higher than the atmospheric pressure. This excess dynamic pressure produces a flow speed equivalent to 19.59 m/s at the inlet.
- 3) The right surface of the computational domain is assigned as the outlet. The boundary condition for "pressure outlet" is implemented at the exit boundary with zero-input value of the static gauge pressure, i.e. $\frac{\partial u}{\partial x} = 0$ of the Dirichlet-type pressure boundary condition.

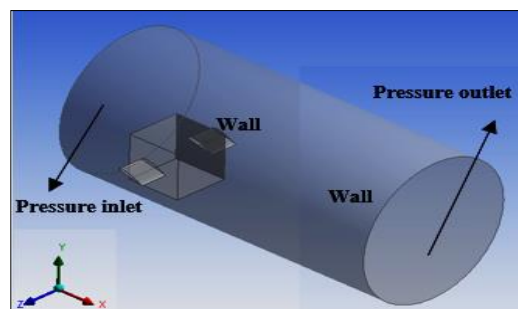


Figure 3. Boundary conditions of the air tunnel and carrier.

3.2. Mesh topologies

The carrier was designed using SOLIDWORKS software [20] and later imported into the ANSYS software [9]. A non-uniform unstructured mesh was used with a considerable fine mesh implemented in the regions of high gradients. A tetrahedron element was selected as the mesh of the carriers and the air tunnel. Tetrahedron mesh has the average orthogonal quality metric spectrum is 0.85 which is very good and the advantage of tetrahedral mesh is its flexibility. Figure 4 shows the layout of the cross-sectional plane of the mesh for the carrier. The number of mesh grids used in the simulation is 1811494.

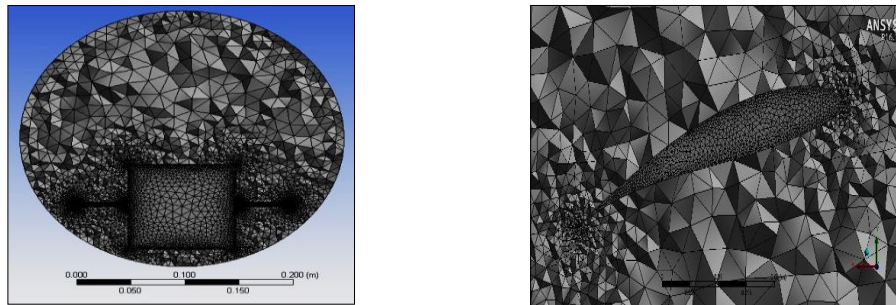


Figure 4. Grid section of the carrier (FX63-137) and circular cross section of the air tunnel.

4. Experiment methodology

The experiment was conducted in the Aerodynamic lab of Universiti Tun Huseein Onn Malaysia (UTHM). The length of the air tunnel is 3 m as shown in Figure 5.a. The source of air was located at 2.5 m before the test section as shown in Figure 5.b. The pipes were made from clear acrylic material and connected with clear acrylic flanges. There is a small hole from the top of pipe. It was made as a door to put the carrier on the bottom of pipe. Moreover, there is a small hole in the door to pull and push the bar inside the pipe in order to hold the carrier initially while the air is blowing. The bar was made from iron and welded with small valve for better carrier holding and less air obstruction. The net linked to the end of air tunnel is to contain the carrier before fall down and to avoid damaging the carrier. The pressure difference between outlet blower and entry station of the test section is set at 235 N/m^2 , same with condition set in the numerical simulation.



Figure 5. a. The real air tunnel and the position of the carrier.



Figure 5.b. The source of the air (blower pump) and the linked pipes (air tunnel).

5. Experimental and simulation results and discussion

Figure 6 show the movement of the carrier in the experiment taken by high speed camera. The time is written for each photo and then compared to numerical results. The dimensions shown in the photos are in meters' unit without scaling. The air tunnel showed here is 2 m in real length and diameter of 0.3 m. The results shown in the Figure 6 indicate that as the carrier floats and flew at different speed, the carrier also change in its aptitude. The presence of change of its attitude make the carrier at the time $t = 0.429$ s touch the ground.

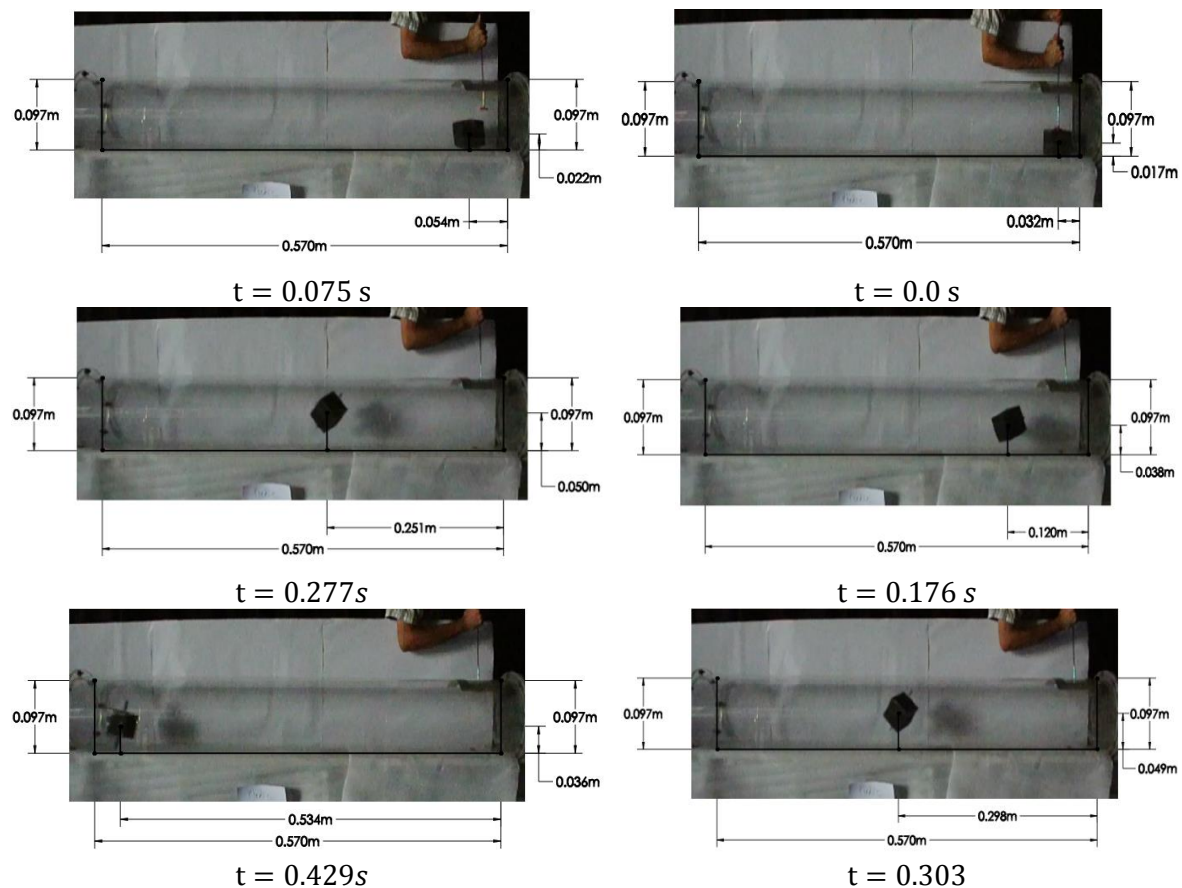


Figure 6. Experiment results for the carrier (FX63-137) captured by high speed camera.

Figure 7 shows the movement of the carrier (FX63-137) numerically. The flow pattern that surrounds the carriers at different stations is shown by the contours of pressure. The front view is shown in Figures 7, This Figure describes the flow conditions at various time steps.

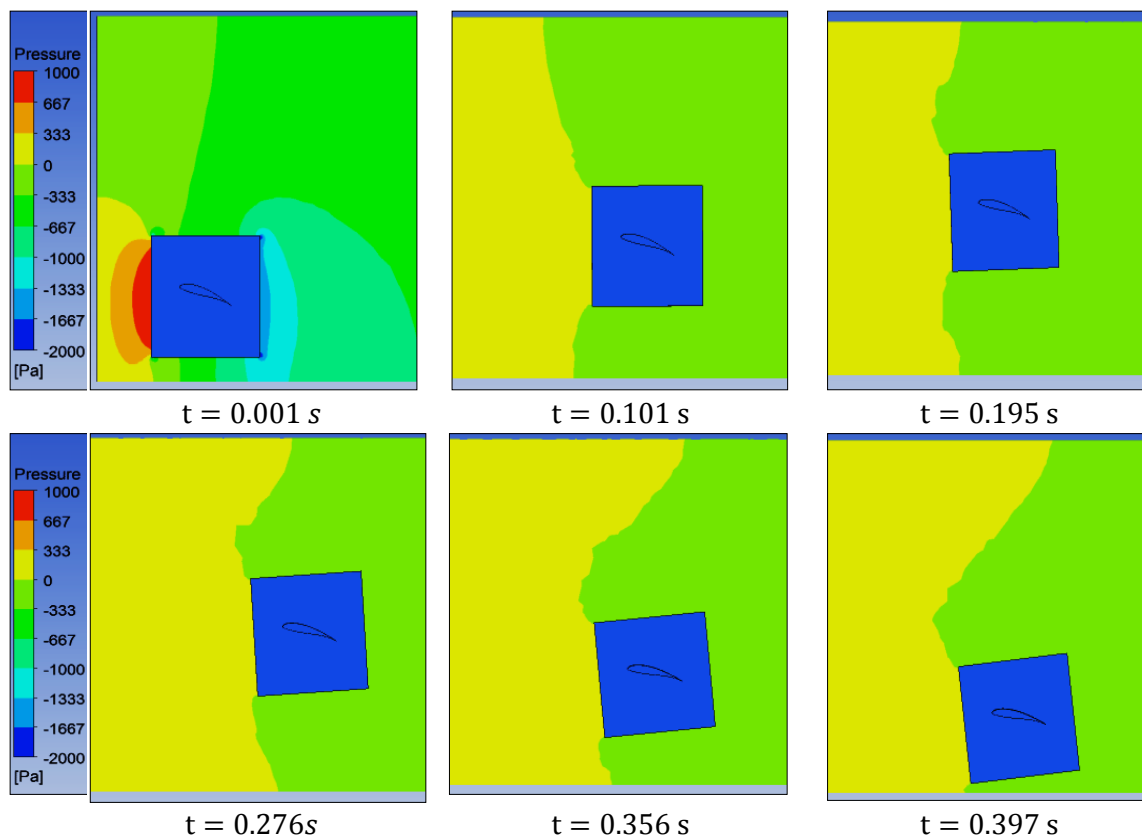


Figure 7. Pressure contours of the flow field around the carrier in various positions for XY-plane.

Based on the result shown in the Figure 7, the carrier reached speed of 0.6 m/s in 0.035 s with the lift force works on it is 5.05 N. As the time progresses, between time 0.035 s to 0.276 s, the orientation of the carrier is relatively similar to its condition at the beginning time the carrier moves. However, in the next following time, due to increasing movement speed, the aerodynamics moments work on the carrier able to rotate and change its orientation. Such condition makes the angle of attack increase gradually above stall angle, which make the lift force on the carrier less than its carrier's weight. As a result, the carrier starts to fall down. Such condition makes at time $t = 0.397$ s, the carrier touch the ground. At this time step, the carrier moved at speed of 4.44 m/s with the distance had been covered in the x-direction is 1.76 m.

Considering computational and experimental results indicate that the mechanism for transporting carriers through air tunnels are possible. Through the results presented in Figure 6 and Figure 7, the distance and speed experienced by the carrier can be defined. Figure 8 shows the comparison result between experimental and numerical data presented in term of velocity carrier against the distance. While Figure 9, shows their comparison in term of velocity plotted with respect to time. It can be observed from Figure 9 that the velocity magnitude of the carrier at the beginning was 0.83 m/s at time of 0.025 s for the experiment work which is higher compared to numerical result, while numerical results showed a bit high increasing after launching of the carrier during whole journey to reach 4.44 m/s at time of 3.97 s. Figure 10 compares the time taken for the carrier to travel through the air tunnel experimentally and numerically. It indicates knowing the carrier movement between both methods, where the travelling carrier experimentally was faster than numerically at each carrier positions in x-direction.

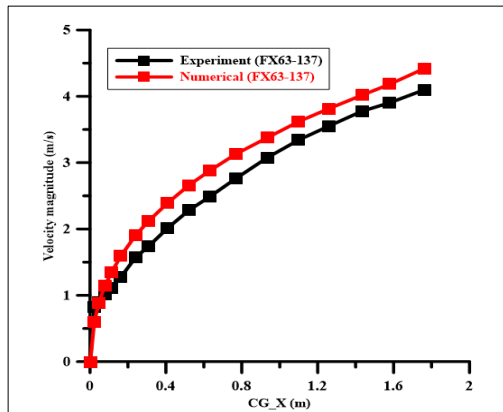


Figure 8. Distance travelled by the carrier in the X-direction versus velocity magnitude (u).

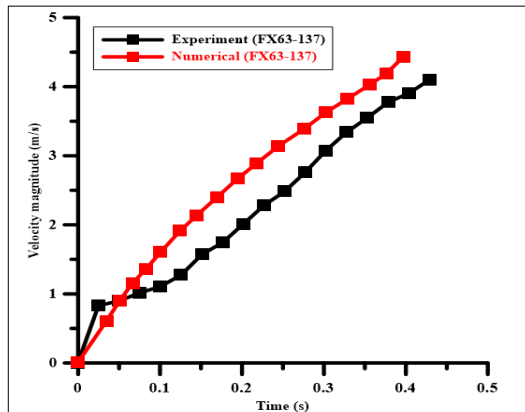


Figure 9. Velocity magnitude (u) of the carrier respect to time.

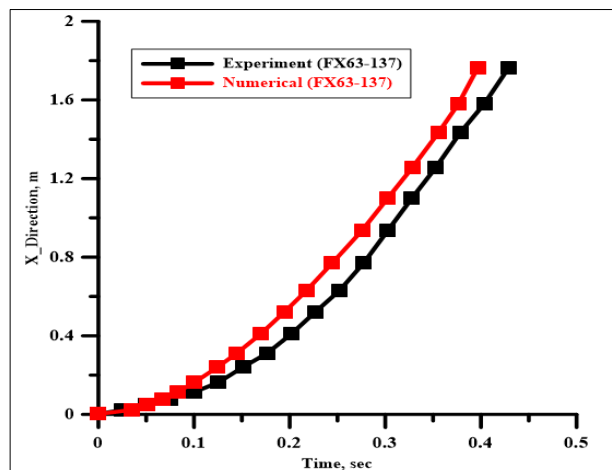


Figure 10. Distance travelled by the carrier in the X-direction versus the time taken for experiment and numerical results.

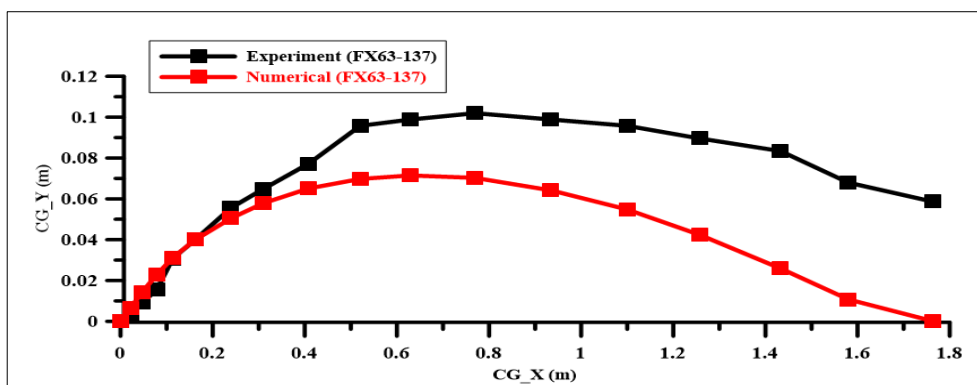


Figure 11. Carrier trajectory with characteristic Squares.

Figure 11 shows the movement comparison of the carrier along the X-direction, the motion of the carrier started from the same origin for both results. It is very clear from the graph that, good agreement with error less than 1% is obtained for both results until $x = 0.4$. As clear from the graph, by travelling carrier at distance of 0.53 m the error started increasing and limited to 7 % until landing position at distance of 1.76 m. At this time, new phenomena caused by the eddies started, which make the carrier roll a bit to the right side. This rolling can be seen clearly from Figure 6. Moreover, the carrier rolled to the right side and was not stable and unbalanced due to high force acted on the left side created by the continuous eddies. Furthermore, it can be seen that at a distance of 1.76 m in X-direction, the carrier has been landed probably for the numerical result, while the experimental result, the carrier has not been landed probably, because of the phenomena stated above and was very clear to see it from Figure 6, therefore, the landing was on the wing side that makes the error bigger reached to 5.8% between both results. The distance covered by the carrier is 1.76 m from 3 m which are 58.6% of the total length of the air tunnel.

6. Conclusion

An experiment and numerical study of a moving carrier (FX63-137) have been conducted. The movement of the carrier through the air tunnel for the current model had been found experimentally and numerically to have similar behaviour. The current model allows to float up to a distance of 1.76 m which is 58.6% of the air tunnel's length. As a good agreement has been found between results, there is a positive indication for the possibility of transporting goods by this carrier through an air tunnel in the nearest future.

Acknowledgment

The authors thank Universiti Tun Hussein Onn Malaysia (Grant Code: U416) for providing financial support for this research

References

- [1] Lakehal D and Rodi W 1997 Calculation of the flow past a surface-mounted cube with two-layer turbulence models *J. Wind Eng. Ind. Aerodyn.* **65**-78
- [2] Iaccarino G, Ooi A, Durbin P A and Behnia M 2003 Reynolds averaged simulation of unsteady separated flow *Int. J. Heat Fluid Flow* **24**(2) 147-56
- [3] Yakhot A, Liu H and Nikitin N 2006 Turbulent flow around a wall-mounted cube: A direct numerical simulation *Int. J. Heat Fluid Flow* **27**(6) 994-1009
- [4] Tannehill J C, Anderson D A and Pletcher R H 1997 Computational Fluid Mechanics and Heat Transfer, Series in *Computational and Physical Processes in Mechanics and Thermal Sciences* (New York: CRC Press, Taylor & Francis Group)
- [5] Gosman A D 1999 Developments in CFD for industrial and environmental applications in wind engineering. *J. Wind Eng. Ind. Aerodyn.* **81**(1) 21-39
- [6] Kanyako F and Janajreh I. 2013 Investigating blade performance of small horizontal axis wind turbine based on blade element momentum theory In Applications of information technology to renewable energy processes and systems (IT-DREPS) 2013 *1st International Conf. & Exhibition* on the 2013 May 29 (Amman, Jordan: IEEE) pp 85-90
- [7] Hossain A, Rahman A, Iqbal A K, Ariffin M and Mazian M. 2012 Drag analysis of an aircraft wing model with and without bird feather like winglet. *Int. J. Aeros. Mech. Eng.* **6**(1) 8-13
- [8] Saad M M, Mohd S B, Zulkafli M F and Shibani W M 2017 Numerical analysis for comparison of aerodynamic characteristics of six airfoils. In *AIP Conf. Proc. (Jogjakarta, 1-3 August 2016)* (Vol. 1831, No. 1) ed I Taib, N Asmuin and S A Osman (AIP Publishing) p 020004
- [9] Fluent, ANSYS. ANSYS Fluent User's Guide, (Canonsburg, PA, USA, 2015) release 16.2 pp 1-2718.
- [10] Launder B E and Sharma B I. 1974 Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. *Lett. Heat Mass Transfer* **1**(2) 131-7

- [11] Guan Song X and Park Y C 2007 Numerical analysis of butterfly valve-prediction of flow coefficient and hydrodynamic torque coefficient. *In Proc. of the world congress on Engineering and computer science* pp 24-26
- [12] Mohammadi. B and Pironneau. O 1993 Analysis of the K-Epsilon Turbulence Model. *Research in Applied Mathematics, John Wiley & Sons Ltd*
- [13] Chern M J and Wang C C 2004 Control of volumetric flow-rate of ball valve using V-port. *Transactions-American Society of Mechanical Engineers Journal of Fluids Engineering* **126** 471-81
- [14] Shibani E, Zulkafli M F, Basuno B and Ab Wahab A B 2016 Numerical Analysis of a Moving Object in the Air Tunnel. *International Journal of Mechanical & Mechatronics Engineering IJMME* **16**(03) 72-8
- [15] Shibani W M, Zulkafli M F and Basuno B 2017 A comparative study on the motion of various objects inside an air tunnel. *In AIP Conf. Proc. (Jogjakarta, 1-3 August 2016)* (Vol. 1831, No. 1) ed I Taib, N Asmuin and S A Osman (AIP Publishing) p 020005
- [16] Shibani W M, Zulkafli M F and Basuno B 2016 Numerical Study of a Winged Container Moving in an Air Tunnel *IOP Publishing* **160**(1) p 012041
- [17] Tao Wenquan 2001 *Numerical Heat Transfer* (Xi'an: Xi'an Jiaotong University Press)
- [18] Zhang Z S 2003 *Turbulence* (Beijing: National Defence Industry Press)
- [19] Norberg C, Sohankar A and Davidson 1995 Numerical simulation of unsteady flow around a square two-dimensional cylinder. *In Twelfth Australian Fluid Mechanics Conf.* pp 517-520
- [20] Tickoo S 2016 SolidWorks for designers (Schererville, IN, USA) 14th Edition. *Cadcam Technologies*