DOI: 10.13009/EUCASS2017-653

7th european conference for aeronautics and space sciences (eucass)

Comparison of Turbulence Models Effectiveness for a Delta Wing at Low Reynolds Numbers

David Monk[1,a] and Dr. Edmund A. Chadwick[2,b]
[1,2]University of Salford,43 Crescent, Salford, M54WT
[a]PhD Student, University of Salford
[a]d.p.monk@edu.salford.ac.uk
[b]Reader in Applied Mathematics, University of Salford
[b]e.a.chadwick@salford.ac.uk

With the process of aircraft design and aerodynamic testing relying more than ever on Computational Fluid Dynamics (CFD), queries into which turbulence model is most effective for delta wings are raised. This study will look into the case of flow passing over a moderately swept delta wing at low Reynolds numbers. Using the CFD solver, ANSYS FLUENT, three models were tested, the Spalart-Allmaras one-equation model, the k-omega two equation model and the Transition SST four-equation model. The Spalart-Allmaras adds a single additional variable for a Spalart-Allmaras viscosity and solves the entire flow field as opposed to using wall functions. Its advantages come in the form of stability and its reliability for results convergence. The k-omega model solves for the specific rate of dissipation of kinetic energy and uses wall functions. This model is most effective in flows that show strong curvature and separation. And finally, the Transition SST model, a hybrid of both the k-omega model at the walls and the k-epsilon model in the free stream. This turbulence model does not use wall functions and is most accurate when solving the flow near the wall. The different turbulence models are compared against wind tunnel experimental results.

1. Introduction

1.1 Motivation

As the science of aerodynamics looks to the future, it's reliance on computational methods is increasing, specifically Computational Fluid Dynamics (CFD). As the capabilities of computers increases, their ability to effectively replicate true to life conditions and produce accurate solutions also increases. The list of CFD software programmes available for aerodynamic studies is extensive; the software available to myself at the University of Salford is ANSYS, offering meshing tools and the solver FLUENT [1]. Within this solver it is possible to select one of several turbulence models.

However, the effectiveness of Computational Fluid Dynamics is still widely debated within the Aerodynamic community, with many claiming that wind tunnel testing is still the most effective method of replicating an aircraft in flight. It is not just an issue of accuracy, but also financial problems, including the relatively cheap running costs of CFD tests but their high time consumption per result, as opposed to the high running costs of wind tunnels but their ability to provide a large amount of data within a short space of time. This paper will conduct a study into which turbulence model can adequately reproduce results obtained from a wind tunnel test conducted in a moderately swept stealth configuration.

1.2 Objectives

The overall objective of this work is to provide some basis of confirmation towards the effectiveness of three different turbulence models in comparison to a model of the same geometry and dimensions undergoing tests in

DOI: 10.13009/EUCASS2017-653

a wind tunnel at a similar, low Reynolds number. The geometry chosen is an optimised UCAV 1303 configuration [2] provided by Dr. Joe Coppin of DSTL, Fareham. This can be split into the following sub-objectives:

- Conduct an alpha sweep test on the optimised 1303 configuration model in the University of Salford's Low-speed Wind Tunnel , observing the effects towards lift, drag and pitching moment
- Replicate the tests using ANSYS FLUENT as the solver in the freestream, using each of the three chosen turbulence models, Spalart-Allmaras, k-omega, and the Transition SST.
- Using a previous study conducted by myself and a colleague, the freestreams CFD results will be compared to wind tunnel corrected results.

As stated previously, a newly optimised UCAV 1303 geometry was provided in the form of CAD file. This allows for a surface model to be produced using the ICEM meshing tool in ANSYS. This CAD model can also be manipulated to allow for a wind tunnel model to be 3-dimensionally printed. A description of the geometry and dimensions, as well as details on the model production will be provided in a later section of this paper.

Both the computational and physical models will be put under an alpha sweep test, mimicking the take-off conditions of the aircraft [3].

1.3 Study Limitations

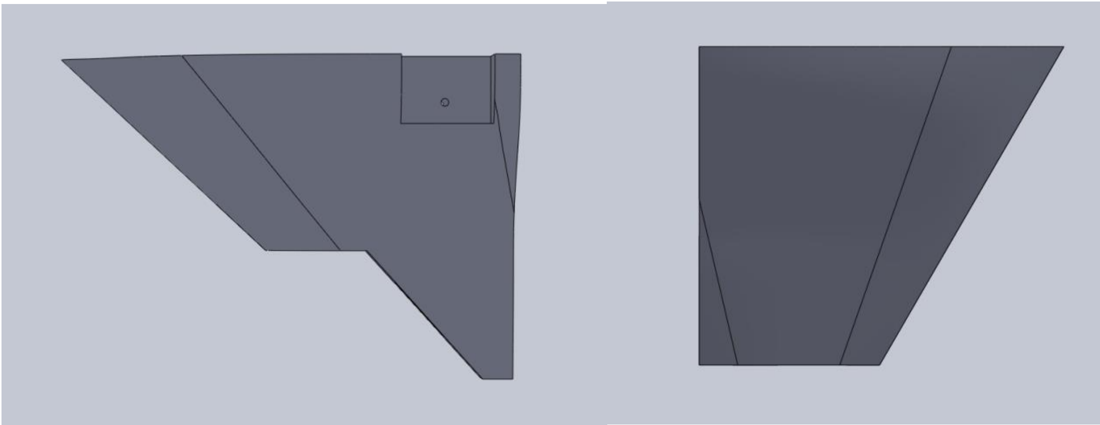
Due to the comparative nature of the study, it is essential that all of the CFD tests and the wind tunnel tests are conducted under the same condition, most importantly, the Reynolds number. The tests were run at a Reynolds number of approximately 0.56 million. The low Reynolds number nature of the study is a result of the limitation presented by the facilities at the University of Salford. This is a low-speed, closed section tunnel with a working cross-section of 850mm by 1150mm by 1500mm, restricting the maximum length of the model's Mean Aerodynamic Chord (MAC). With a fully serviced motor, the tunnel can attain speeds of up to ~40m/s, however, it would be unwise to push the motor to its limits and so a speed of 30.87m/s (0.09M) was chosen. This value was decided upon as it was the speed produced from running the tunnel motor at 70% with the effects of the blockage of the model taken into account. The atmospheric pressure and temperature cannot be controlled and so the viscosity cannot be modified. Due to these reasons the Reynolds number was low.

2. Optimised 1303 Configuration

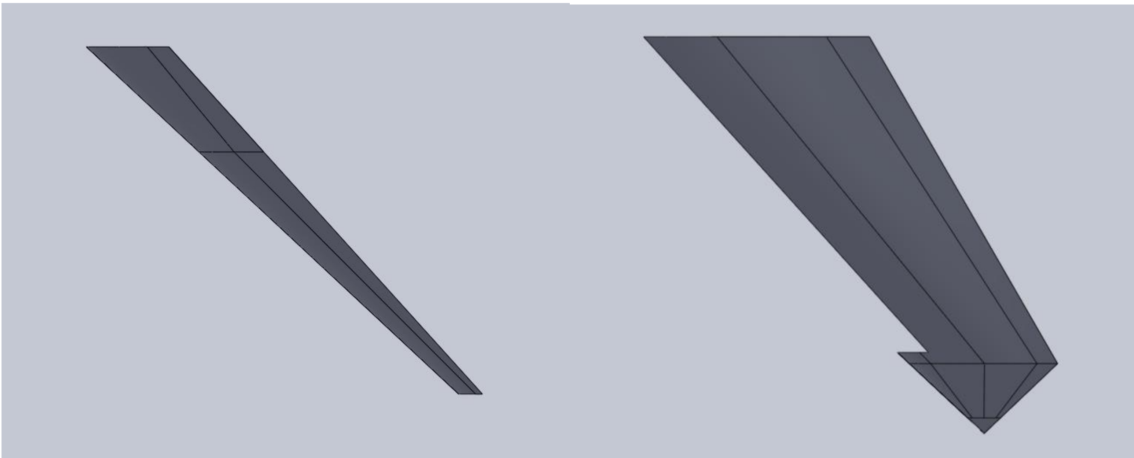
The 1303 UCAV was developed by Boeing and Air Force Research Laboratories (AFRL). It is based on the 1301 configuration, a design presented benefits of long subsonic range/mission radius, available options of controls and longitudinal balance considerations [4] . Some important characteristics of the 1303 configuration include a leading edge sweep angle of 47°, trailing edge angles of +/- 30° and an aspect ratio of 3.85. The aerofoils used in the wings to simplify the geometry consist of a NACA 64a0012 at the root, from which it reduces non-linearly to a NACA 64a008 at the first crank. This is the aerofoil used for the rest of the wing. The fuselage of the wing was created by increasing the thickness at the root (McLain 2009).

For the wind tunnel experiment, it was necessary to create a physical model of the optimised 1303 configuration. This was achieved by splitting the CAD geometry file used for the Computational Fluid Dynamics simulations into six pieces symmetrically along the root chord of the aircraft as depicted in the following figures:

DOI: 10.13009/EUCASS2017-653



Figures 1 and 2: The front and rear left of the inboard of the optimised 1303 configuration CAD model



Figures 3 and 4: The left leading edge and the left outboard of the optimised 1303 configuration CAD model

These six pieces are 3D printed and bonded together to create one complete model, with great care being taken to ensure that the surface finish of the model is as smooth as possible and the dimensions remain accurate. This is to reduce any potential aerodynamic error with in any results produced. It is important to note that this model will be used for a leading edge study also, and as such the leading edges are interchangeable. The leading edges used in this study are un-deflected and replicate those of the leading edges in the CFD simulations used in this paper.

3. Turbulence Models

The first turbulence model to be studied is the Spalart-Allmaras model. This is a one equation model that solves the modelled transport equation for the kinematic eddy viscosity [5]. Although a relatively new addition to this class of one-equation models, where it is not necessary to calculate a length scale related to the local shear layer thickness, the Spalart-Allmaras model was designed specifically for aerospace applications. With respect to simulations involving wall-bounded flows, such as those similar to wind tunnel tests, this model has produced very promising results. Another benefit comes from the model's ability to produce excellent results for boundary layers subjected to adverse pressure gradients. This is particularly useful for this optimised 1303 UCAV application, due to the adverse pressure changes that occur during vortex separation and turbulent flows over the surface of a wing [3].

In its standard from, the Spalart-Allmaras model is a relatively low-Reynold-number model and requires the viscous affected region of the boundary layer to be properly resolved. When the mesh is coarse, the

DOI: 10.13009/EUCASS2017-653

implementation of the Spalart-Allmaras model in FLUENT has been designed to include wall functions. Near-wall gradients of the transported variable in this model are far smaller than those found in the κ-ε or κ-ω models. This potentially makes the mode less sensitive to numerical errors when non-layered meshes are used near walls.

The k-omega model solves for two variables, these are k, the turbulent kinetic energy, and omega, the specific rate of dissipation of kinetic energy. It has difficulty converging and relies on an initial guess at the start of the solution, as such it is popular in industry to first run the k-epsilon model, in which the rate of dissipation of kinetic energy is solved, first. However, the k-omega model is far more capable of producing accurate results in many aerodynamic situations, specifically flows that exhibit strong curvature and separated flows.

The SST (Shear Stress Transport) k-omega turbulence model is a two-equation eddy viscosity model. It is a combination of both the k-epsilon and the k-omega turbulence models, using the former in the free-stream and the latter near the walls. With this implementation of the k-omega model, this allows for the model to be directly usable to the wall through the viscous sub layer. As a result, the SST k-omega model can be used at low Reynolds numbers without the need for extra damping functions. It has been noted by many of those that work with this particular model that promising results have been produced in situations with adverse pressure gradients and separating flows.

4. Experimental Procedure

4.1 Wind Tunnel

After checking that the wind tunnel was completely unlocked and all the necessary equipment was fully functioning, the following series of events took place:

1. Check the pylons are the appropriate distance apart to load the model into the wind tunnel. Once these are correct, tighten the nuts and bolts leaving the capability for the model to be pivoted. Do the same for the trailing point and pylon.
2. The incidence angle of the model is changed electronically by hand. In order to avoid the starting and stopping of the wind tunnel the angle is conveniently given as a digital reading of mV. It is necessary to use a clinometer to confirm the mV at each angle of incidence. By recording these values, the precise angles of incidence can be obtained without having to stop the flow of air over the model, this is beneficial to the Reynolds number reached during the experiment.
3. Before the wind tunnel is activated, values for the pitching moment at each angle of attack, as well as wind-off values for the Lift and Drag should be recorded.
4. Orientate the model at the first angle of attack to be tested (-10°). Ensure that the wind tunnel is completely cleared of any small/loose parts and securely seal it. Take readings of the initial temperature of the tunnel and the atmospheric pressure in the lab as these will both affect the Reynolds number.
5. Activate the wind tunnel, slowly increasing the motor speed until the desired wind speed is reached. In this case the motor was brought to 80% of its full power, providing a wind speed of approximately 30.87m/s
6. Allow the flow of air to pass over the model for long enough to reach turbulent conditions, then, using the PicoLog, take a series of 200 samples of Lift, Drag and Pitching moment across 10ms and record the average. Complete this for each incidence angle up to 30° at increments of 2°.
7. A reading from the Betz manometer must also be taken at each angle of attack as blockage will affect the wind speed as the experiment takes place.
8. Finally, with the mV values for lift drag and pitching moment at each incidence angle created, the results can be implemented into a spreadsheet where, with the implementation of the calibration values, the coefficients of each of the dynamic parameters can be calculated.

DOI: 10.13009/EUCASS2017-653

4.2 ANSYS FLUENT Simulations

In both 2-dimensional and 3-dimensional problems, FLUENT has the option to use a number of different grid-types (meshes). For 2D simulations, triangular and/or quadrilateral cells can be generated. For 3D simulations, tetrahedral, hexahedral, polyhedral, pyramid or wedge cells are available, either individually or a hybrid. The choice of which type of cell to use depends on the complexity of the simulation. As such, the following should be considered whilst selecting the mesh type:

- Set-up Time
- Computational Expense
- Numerical Diffusion

The most time consuming grid types to build are those that consist of quadrilateral/hexahedral elements. This is due to the fact that using these element types will result in a structured mesh. The generation of structured and block-structured grids is incredibly time consuming for complex geometries and in some cases, have to be transformed into unstructured before a solution can be achieved. Set-up time for complex geometries is the main motivation for creating unstructured grids, consisting of triangular and tetrahedral elements. This is only an issue with complex geometries, if the geometry is simple, and a large number of elements are required, there may be no difference in the time between generating structured and unstructured grids [6].

Computational expense is one of the most limiting factors with regards to what type of mesh to use, and how many elements can be permitted to use. These simulations are sensitive to the number of cells included in the set-up. Generally, the larger the number of cells, the more time dependant the simulation becomes to get a convergence of the results. This can be negated with adequate computing power. As this is the case, the quality of the mesh is proportional to the computing power available. When geometries are more complex, triangular/tetrahedral meshes should be considered if time issues are present. These grid types produce far fewer elements than the equivalent mesh composed of quadrilateral/hexahedral elements. Not only do triangular/tetrahedral meshes provide benefits with regards to generation time, but also in the ability to allow clustering of cells in areas of particular note, such as the model. Structured meshes place cells in regions where they are not required, adding to the amount of time required for convergence to occur, without further improvement to the mesh quality.

There are situations when structured meshes are more economical to apply, as they produce a much larger aspect ratio in their cells. If such aspect ratios were present in, for example, triangular meshes, skewed cells may be produced. This creates problems with accuracy and may impede the convergence of the solution. As mentioned earlier, converting a structured quadrilateral/hexahedral mesh to unstructured before solving in FLUENT will result in fewer cells than the original mesh, which will work to lower the computing power required for a solution.

The final consideration is Numerical Diffusion, which is one of the dominant sources of error in multi-dimensional problems, and is also labelled as false diffusion. It is known by this term due to the fact that this diffusion is not a real phenomenon. Its effect on a flow calculation is analogous to that of increasing the real coefficient. False diffusion involves the following:

- Most noticeable when the problem is convection-dominated as the real diffusion is said to be small.
- Implementing a second-order discretisation scheme during solving with FLUENT can work to reduce numerical diffusion effects.
- The effects of numerical diffusion can be negated further by refining a mesh, as it is directly related to the resolution of the mesh.
- It can also be negated by aligning the flow with the mesh, which is only possible with a structured mesh.

DOI: 10.13009/EUCASS2017-653

The mesh type used in the following simulations was a hexahedral mesh developed using the program ICEM in ANSYS Workbench 17.0. The model was placed directly in the centre of a cuboid domain approximately 20 times its root chord length in both the X and Y directions, with the domain being only 10 times the root chord length in the Z direction due to only half the planform being used as a means of reducing the number of cells as computing power available was limited. The boundary conditions used are Pressure-Far-Fields (PFF) on the five outer surfaces, with a plane of symmetry running along the face of the root chord. The corresponding X and Y components of flow are input into each of the PFF faces, effectively changing the angle of attack of the model against the flow.

Pressure-far-field boundary conditions are available for use in ANSYS FLUENT, allowing for the modelling of a free-stream condition at infinity. The information required for this boundary condition is free-stream Mach number and the static conditions. This boundary condition is only applicable when the ideal-gas law is used, and for effectiveness it is advised that the extent of the boundary be at least 20 times the chord length from the centre of the model. A popular shape for the boundary is spherical.

It is a non-reflecting boundary condition based on the introduction of Riemann invariants, for one-dimensional flows normal to the boundary. Subsonic flows, such as I will be dealt with in this paper, have two Riemann invariants, corresponding to incoming and outgoing flows.

The FLUENT solver was set up using some of the following parameters:

- Density Based
- Chose turbulence model
- Energy – on
- Ideal Gas
- Sutherland’s law of dynamic viscosity

Solution methods included:

- Implicit
- Gradient – Green-Gauss Node
- Second order upwind

The mesh created consisted of approximately 5 million cells, the largest, reasonable amount possible with the computing power available.

4.3 Wind Tunnel Corrections

In order to compare the free stream alpha sweep results obtained from those obtained of the three different turbulence models to the experimental results, wind tunnel corrections were applied. An initial CFD test was thus conducted using one of the turbulence models, the Spalart-Allmaras one equation model. The 1303 model was tested at free-stream and in a simulation modelling the width and height of the tunnel specified in **Section 4.1** of this report. The purpose of this is to observe the effects of the tunnel walls on the aerodynamic parameters exhibited on the optimised 1303 model. In a closed tunnel with solid walls, the model experiences, the model will experience an up-wash that will, as a consequence, increase the lift exerted on it. The model acts as a solid blockage within the tunnel, causing an increase in the tunnel flow velocity. These sets of results were compared and noticeable differences were apparent, for example an increase in the lift coefficient was noted (**Figure 6**). As such wind tunnel corrections were applied to the CFD wind tunnel tests[7].

DOI: 10.13009/EUCASS2017-653

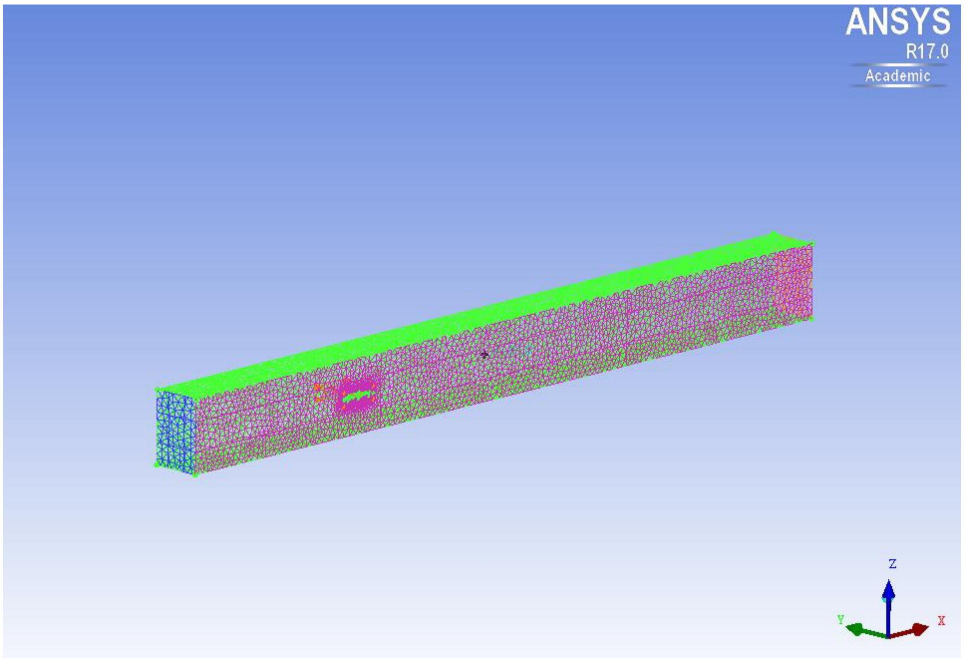


Figure 5: Mesh produced using the ICEM meshing tool, modelling the tunnel walls of the University of Salford's Low-speed wind tunnel

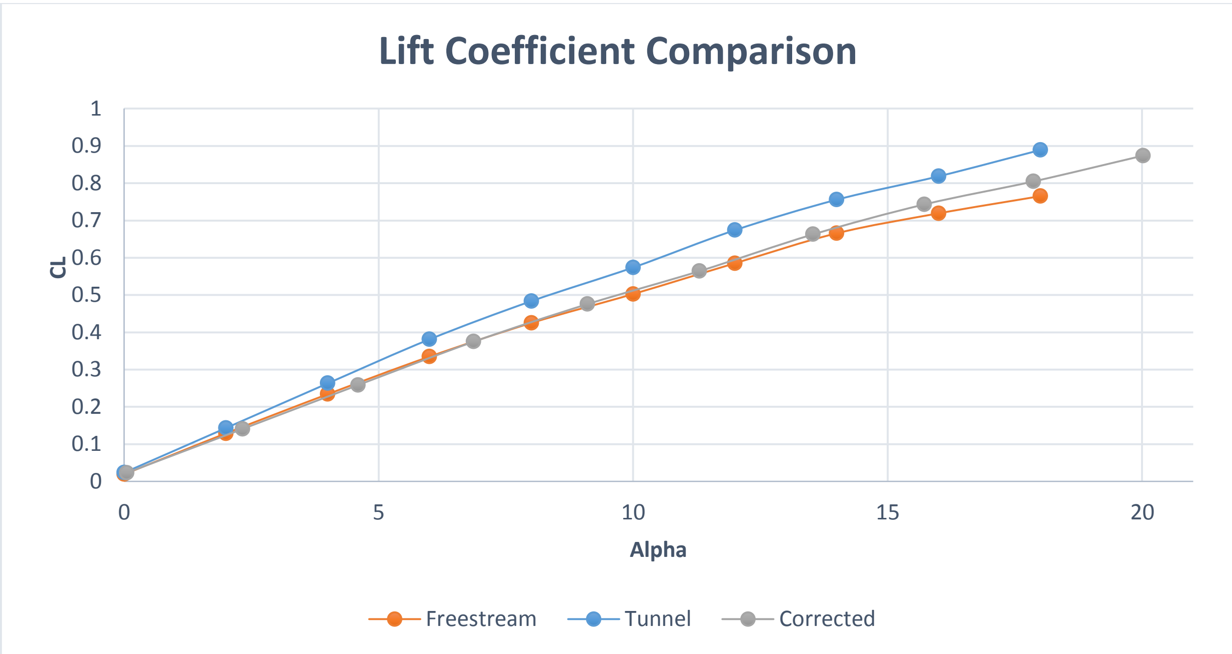


Figure 6: Lift coefficient results from the wind tunnel comparison study

With confirmation on the effects of the wind tunnel walls on the aerodynamic parameters, and further confirmation of the wind tunnel corrections improving the results making them close to the free stream results. With this knowledge the corrections can be applied to the results taken from the physical wind tunnel tests.

DOI: 10.13009/EUCASS2017-653

5. Results

The following tables and graphs present the results obtained from the CFD comparison studies. Alpha sweep tests were conducted first in the wind tunnel, with wind tunnel corrections being applied to those results in an attempt to negate the effects of the tunnel walls. Alpha sweep tests were then conducted using the CFD software ANSYS FLUENT, placing the optimised 1303 model in a free stream condition. Three tests were conducted in total changing the turbulence model in the FLUENT solver in each case. The first test was solved with the Spalart-Allmaras model, the second with the k-omega model and the third with the K-omega SST model.

Table 1: Aerodynamic values obtained from the Wind Tunnel Alpha sweep tests

Alpha	CL	CD	CM
0	0.000000	0.001304	-0.00900
1	0.054995	0.000000	-0.01387
2	0.108254	0.000303	-0.01854
3	0.170354	0.0025005	-0.02400
4	0.230062	0.0052562	-0.03001
5	0.286027	0.0077032	-0.03539
6	0.342740	0.0127815	-0.04062
7	0.397885	0.0177989	-0.04472
8	0.455444	0.0244523	-0.05103
9	0.502734	0.0309284	-0.05428
10	0.543039	0.041343	-0.05271
11	0.593613	0.0552432	-0.05483
12	0.649125	0.0747104	-0.05588
13	0.694795	0.1045605	-0.05435
14	0.733560	0.131571	-0.05301
15	0.767199	0.159165	-0.05068
16	0.807216	0.192197	-0.04862
17	0.839942	0.2231512	-0.04683
18	0.859136	0.2491182	-0.04230
19	0.846169	0.2778895	-0.04497
20	0.840922	0.3143839	-0.05856

Table 2: Coefficients of Lift taken from the CFD Alpha Sweep tests

	CL		
Alpha	SA	K-omega	SST
0	0.0191632	0.019872	0.017978
2	0.129064	0.12877	0.12538
4	0.23412	0.23319	0.22758
6	0.33492	0.33387	0.32547
8	0.42516	0.42565	0.41489
10	0.50282	0.50595	0.49311
12	0.58534	0.58451	0.57671
14	0.66578	0.6607	0.65754
16	0.71917	0.71816	0.7101
18	0.76532	0.7622	0.74836

DOI: 10.13009/EUCASS2017-653

Table 2: Coefficients of Drag taken from the CFD Alpha Sweep tests

	CD		
Alpha	**SA**	**K-omega**	**SST**
0	0.002774	0.002242	0.001824
2	0.00146	0.000861	0.000985
4	0.00407	0.003469	0.001691
6	0.010074	0.009482	0.007266
8	0.01995	0.019315	0.01685
10	0.037328	0.03629	0.034284
12	0.067509	0.065419	0.064499
14	0.10436	0.10265	0.10054
16	0.13976	0.13996	0.13625
18	0.18328	0.18223	0.1786

Table 2: Coefficients of Pitching Moment taken from the CFD Alpha Sweep tests

	CM		
Alpha	**SA**	**K-omega**	**SST**
0	-0.0056292	-0.0064303	-0.0052931
2	-0.0163324	-0.016716	-0.015549
4	-0.026494	-0.02656	-0.02437
6	-0.036314	-0.036269	-0.03324
8	-0.043077	-0.043488	-0.040148
10	-0.043896	-0.045443	-0.040816
12	-0.043438	-0.043407	-0.040409
14	-0.042444	-0.041647	-0.039392
16	-0.038659	-0.038144	-0.034633
18	-0.04235	-0.041756	-0.040159

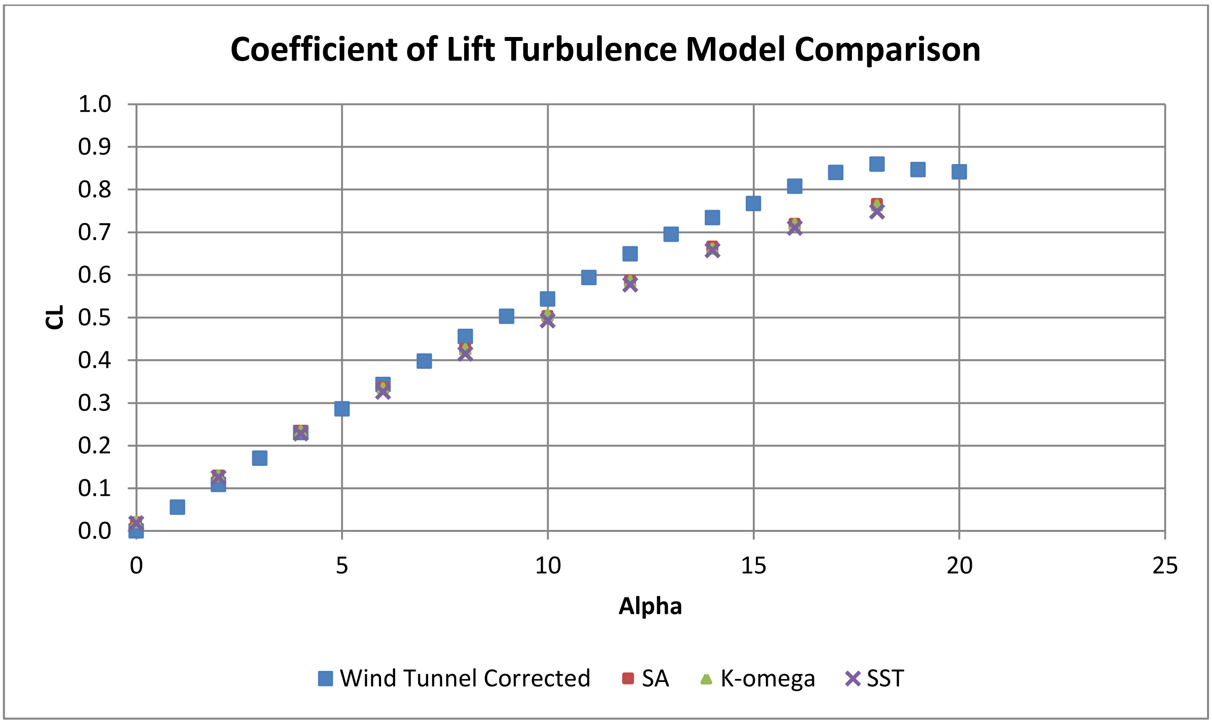


Figure 7: Coefficient of Lift plotted against angle of attack results comparison

DOI: 10.13009/EUCASS2017-653

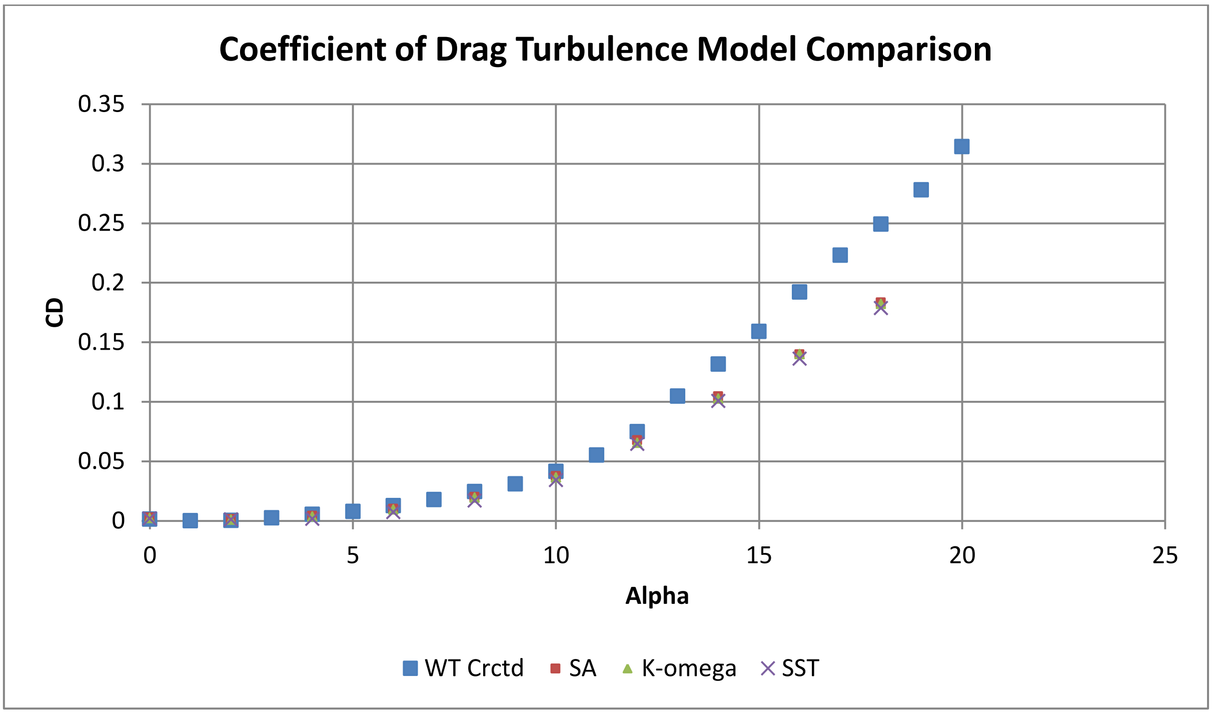


Figure 8: Coefficient of Drag plotted against angle of attack results comparison

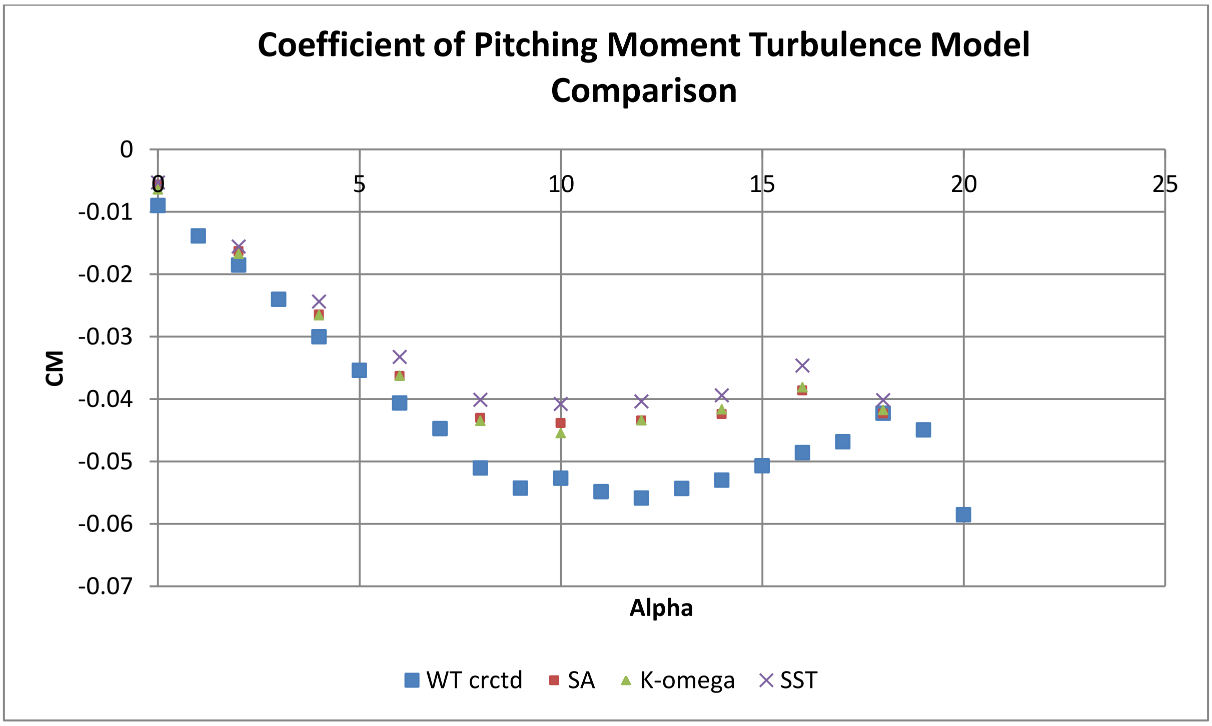


Figure 9: Coefficient of Pitching Moment plotted against angle of attack results comparison

DOI: 10.13009/EUCASS2017-653

6. Conclusions

The results produced from the series of Computation Fluid Dynamics studies were plotted against the wind tunnel results of the optimised UCAV 1303 physical model. What these results appeared to show was very little difference in their accuracy. In the case of the coefficient of lift, as depicted in Figure 7, all three turbulence models produce results of high accuracy from a range of 0° to 6°, and results of adequate accuracy up to 10°. There is very little difference in the accuracy of each of the separate turbulence models, however, with a keen eye it can be seen that the Spalart-Allmaras model produces fractionally more accurate results.

Observing the results of the coefficient of drag, similar conclusions can be reached. Spalart-Allmaras is the most accurate turbulence model, with the CD values closest to those of the wind tunnel study. The overall accuracy of the CFD results is higher in comparison to the other aerodynamic coefficients presented in this paper, with a range of 0° to 12°.

Finally, for the coefficient of pitching moment, none of the statements given in the previous examples can be applied. Overall the level of accuracy for all three turbulence models is poor, as presented in Figure 9. Interestingly, at low angles of attack the turbulence model producing the higher level of accuracy was the k-omega model, with angles of attack 14° and higher, the Spalart-Allmaras model becomes the better choice. The Transition SST model is by far the least accurate, showing for the first time in each of the three aerodynamic parameter tests a large difference in coefficient values produced compared to the other two models.

A final note to make with regards to the closeness of the values of the CFD results; this could be a result of the relatively low cell count present in the freestream mesh. With access to better computing power, a greater difference may possibly be seen. However, for those with limited resources, Spalart-Allmaras appears to be the marginally better choice.

DOI: 10.13009/EUCASS2017-653

7. References

1. Fluent, A., *12.0 Documentation.* Ansys Inc, 2009.
2. Coppin, J.a.T.B., *CFD Predictions of Control Effectiveness for a Generic Highly Swept UCAV Configuration.* AIAA, 2014. **32nd AIAA Applied Aerodynamics Conference**.
3. Breitsamter, A.K.a.C., *Vortex-Flow Manipulation on a Generic Delta-Wing Configuration.* Journal of Aircraft, 2015. **51**(5).
4. McLain, M.C.a.B.K., *Aerodynamic studies over a maneuvering UCAV 1303 configuration.* 2010.
5. Kim, S.-E., D. Choudhury, and B. Patel, *Computations of complex turbulent flows using the commercial code FLUENT*, in *Modeling complex turbulent flows*. 1999, Springer. p. 259-276.
6. Brett, J., et al. *Computational fluid dynamics analysis of the 1303 unmanned combat air vehicle*. in *17th Australasian Fluid Mechanics Conference*. 2010.
7. Wiriadidjaja, S., et al. *Aerodynamic interference correction methods case: Subsonic closed wind tunnels*. in *Applied Mechanics and Materials*. 2012. Trans Tech Publ.