7. ANKARA INTERNATIONAL AEROSPACE CONFERENCE
11-13 September 2013 - METU, Ankara TURKEY

AIAC-2013-017

THE EFFECT OF FILLET OF WING JUNCTION AND FRONT SHARPNESS ON THE FLOW CHARACTERISTICS OF AN X-45A TYPE UNMANNED COMBAT AIR VEHICLE

M.Murat Yavuz[1]
Osmaniye Korkut Ata University
Osmaniye, Turkey

ABSTRACT

It is known that the effect of sharp corners is one of the most important conditions on the planes and can affect both aerodynamics performance and mechanical robustness. Hence the sharp corners must be investigated carefully. Different experimental and numerical studies are available in the literature, but most of these studies are not up-to-date about new unmanned combat air vehicles (UCAV) and new investigations are needed for the development stage. In this study, the effect of fillet of wing junction and front sharpness are investigated for determination of flow characteristics around an x-45a type unmanned combat air vehicle. Computational fluid dynamics (cfd) analysis is used for investigating the optimum design parameters of the wing. ANSYS ICEM cfd modelling software is used for numerical modelling of wing and channel. FLUENT cfd package software is used for the solution of numerical models. The wing is placed in a channel and exposed to the free flow. Standard water properties are used as fluid properties. The angle of attack is selected as 13^0 and Reynolds number is taken as 20.000. K-epsilon turbulence model is selected in the modelling of turbulence and steady flow assumption is used. Determined values of iso-surfaces of vortices are illustrated around the wing. More knowledge is gained in 3D observations of flow characteristics and vortex formations. The results of the effect of fillet wing junction and front sharpness are given and they are examined in detail.

INTRODUCTION

The usage and developments of unmanned combat air vehicles increase with new investigations, especially developments of the last 20 years. New plane models are designed and their properties are developed. Automation system, manoeuvre capacity, self perception, time of flight and physical properties is the most critical and popular topics and they have been developed. Some researches focus on the engine and its performance [Turan, 2012; Hung and Gonzalez, 2012] about unmanned combat air vehicles. Most of the others studies are applied on the aerodynamics [Sahin et al., 2012; Yavuz 2012; Cummings et al., 2008; Cadogan et al., 2003; Mary, 2003] and mechanical design [Peng and Jinglong, 2012]. It is known that the flow characteristics [Brooks and Humphreys Jr, 2003] are directly influenced by the aerodynamic properties of the planes.

It is experienced that some failures of the aircrafts occur at the junction locations. The dynamic conditions and flow characteristics around the wing is a dominant source of these types of failure. The wing body junction [Fu et al., 2007] is also investigated at one of the numerical study in the literature. They compare the differences between different numerical simulations on the effect of wing junction and one of their results are given in fig. 1 for observing vortex formations around the wing. Intensive vortex formations occur at the further locations of the wing junction over the wing. The geometry of wing junction location influences the vortex formations. Hence, it must be investigated carefully. In this study, flow characteristics of a simplified geometry of an x-45a type unmanned combat air vehicle are investigated with adding fillet on the wing junction. The fillet radius is changed and flow around the wing is observed. Additionally, the effect of front sharpness of the wing is investigated and its effect on the vortex formations is observed.

[1]Research Assistant in Energy Systems Engineering Department, Email: muratyavuz@osmaniye.edu.tr

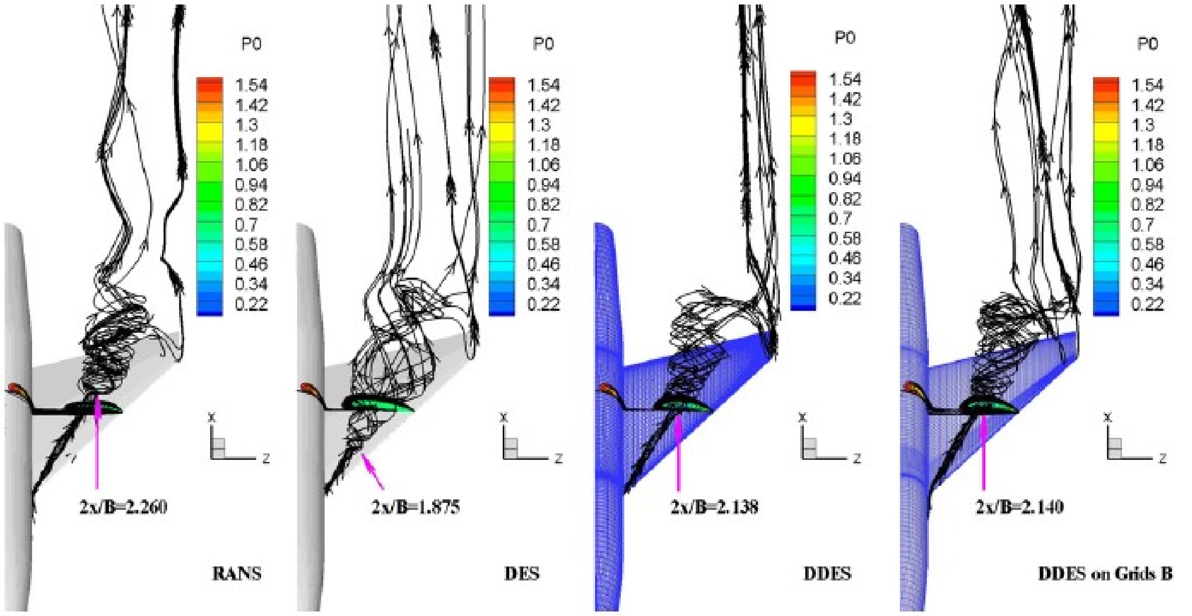

Figure 1: *Comparison of different simulation models at the wing body junction in the study of Fu et al. (2007)*

METHOD

On the contrary to use experimental method, computational fluid dynamic analysis is used for investigations of the effect of sharpness and fillet of the wing junction on the flow characteristics. The model of the wing is given in fig. 2. It has a length of 188 mm, a width of 251 mm and a thickness of 3 mm. Front and side edges of the wing is chamfered with an angle of 30^0. The edge is rounded with a radius of r and magnified view of wing junction location is also given in fig. 2.

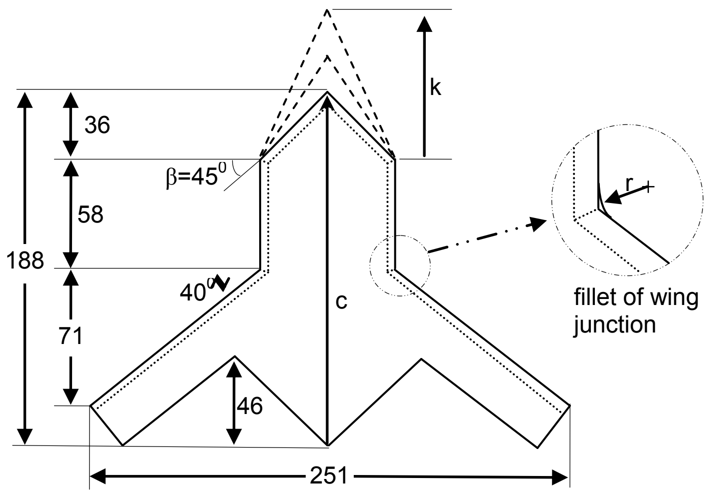

Figure 2: *Top view of x-45a type unmanned combat air wing with dimensions*

Front size/sharpness effect of the wing is investigated with chancing the tip length of the wing (k). The angle β is set for determination of k and the results are given in a ratio of k/c. Free-stream flow is calculated with constant Reynolds number of 20.000. Hence, free-stream flow will recalculate with respect to the change of length of the wing and it (U_0) is given in eq. 1, where ρ is density of fluid, c is length of the wing and μ is the dynamic viscosity of the fluid. Water is used as fluid and has a density of 998.2 kg/m^3 and viscosity of 0.001003 kg/m.s.

$$\mathrm{Re} = \frac{\rho U_0 c}{\mu} \qquad (1)$$

K-epsilon turbulence model is selected with respect to used Reynolds number in the analyses. This model is widely used and gives suitable results in the literature, especially solution of not complex geometries. A channel is modelled for providing the flow conditions. It has a length of 3000 mm, a width of 600 mm and a height of 600 mm. The wing model is placed at the centre of the wing with an angle of attack of 13^0. The geometry of the water channel and wing are shown in fig. 3.

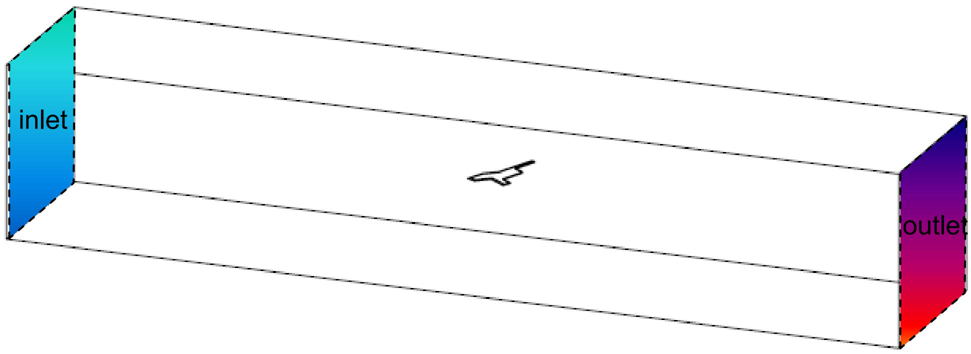

Figure 3: *water channel geometry and section plane*

The modelling of whole analysis and its solution is done with three main steps. First step is modelling, second step is solution and the last step is illustration of the results. Modelling of wing and channel is done with Solidworks modelling software and they are imported in ANSYS ICEM cfd for setting the numerical model of computational fluid dynamic analysis. The grid is formed with respect to analyse conditions. Grid density is increased around the wing for taking more sensitive results. Other far and/or less important locations of the analysis have less grid density for saving computational cost. Approximately 3.033.000 grid elements are used in the analyses. Also boundary conditions are determined in ANSYS ICEM cfd. The solution is applied in FLUENT software. The fluid and its velocity, determination of turbulence model and flow conditions are set before solution of the analysis. Further step is the post-processor step. In this step, the results are taken with suitable forms. TECPLOT software is used for post-processor step of iso-surfaces of the vortices.

Different investigations are applied on the front sharpness and rounding of wing junction. The changes are given in table 1. All dimensions are given in mm and degrees. Front sharpness of the wing is investigated in the first three tests and rounding effect of the wing junction location is investigated in other last three tests.

Table 1: *The used parameters in the analyses*

<table>
<tr><th>Test number</th><th>The angle, β</th><th>Chord length, c</th><th>k</th><th>k/c ratio</th><th>Radius of fillet of wing junction</th><th>Free-stream velocity</th></tr>
<tr><td>Test no-I</td><td>45</td><td>188</td><td>36</td><td>0.19</td><td rowspan="3">0</td><td>106.92</td></tr>
<tr><td>Test no-II</td><td>60</td><td>214.35</td><td>62.35</td><td>0.29</td><td>93.77</td></tr>
<tr><td>Test no-III</td><td>75</td><td>286.35</td><td>134.35</td><td>0.46</td><td>70.19</td></tr>
<tr><td>Test no-IV</td><td rowspan="3">45</td><td rowspan="3">188</td><td rowspan="3">36</td><td rowspan="3">0.19</td><td>10</td><td rowspan="3">106.92</td></tr>
<tr><td>Test no-V</td><td>20</td></tr>
<tr><td>Test no-VI</td><td>30</td></tr>
</table>

The calculations are done with steady flow assumption. In the eq. of 2, 3, 4 and 5, mass and momentum equations are given. Their dependency of time is ignored. Also same condition is valid for k and epsilon equations in the eq. 6 and 7.

$$\frac{\partial \rho}{\partial t} + div(\rho \mathbf{u}) = 0 \qquad (2)$$

$$\frac{\partial (\rho u)}{\partial t} + div(\rho u \mathbf{u}) = -\frac{\partial p}{\partial x} + div(\mu \ grad \ u) + S_{Mx} \qquad (3)$$

$$\frac{\partial(\rho v)}{\partial t}+div(\rho v\mathrm{u})=-\frac{\partial p}{\partial y}+div(\mu\ grad\ v)+S_{My} \quad (4)$$

$$\frac{\partial(\rho w)}{\partial t}+div(\rho w\mathrm{u})=-\frac{\partial p}{\partial z}+div(\mu\ grad\ w)+S_{Mz} \quad (5)$$

$$\frac{\partial(\rho k)}{\partial t}+div(\rho k\mathrm{U})=div\left[\frac{\mu_t}{\sigma_k}\ grad\ k\right]+2\mu_t E_{ij}\cdot E_{ij}-\rho\varepsilon \quad (6)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+div(\rho\varepsilon\mathrm{U})=div\left[\frac{\mu_t}{\sigma_\varepsilon}\ grad\ \varepsilon\right]+C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}\cdot E_{ij}-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad (7)$$

where, $C_\mu = 0.09$, $\sigma_k = 1.00$, $\sigma_\varepsilon = 1.30$, $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$ and μ_t is the eddy viscosity. μ_t is given in eq. 8.

$$\mu_t = C\rho v\ell = \rho C_\mu \frac{k^2}{\varepsilon} \quad (8)$$

RESULTS

In this study, the effect of fillet of wing junction and front sharpness is investigated on the flow characteristics of a simplified model of x-45a type unmanned combat air vehicle. The effect of two analysed conditions is examined separately and the flow behaviour around the wing is observed. It is identified that the front sharpness and fillet of wing junction has an important role on the formation of the flow characteristics. Vortex formation and magnitudes are searched in detail and optimum geometrical conditions of the wing are determined.

Table 2: *The magnitudes of vortices in different directions in the analyses*

	x-vortices		y-vortices		z-vortices		Vortices magnitude
	Min	Max	Min	Max	Min	Max	
Test no-I	-35.09	34.73	-102.18	82.14	-102.76	94.50	109.54
Test no-II	-29.63	36.36	-73.76	79.17	-82.01	77.03	82.39
Test no-III	-30.91	30.02	-60.59	56.76	-71.03	58.06	79.40
Test no-IV	-37.52	36.15	-83.60	88.30	-111.15	91.93	112.52
Test no-V	-36.61	38.67	-83.66	86.35	-113.22	92.51	101.32
Test no-VI	-31.87	26.99	-89.15	89.14	-96.50	100.09	100.85

The magnitudes of vortices in different flow directions are given in table 2. In test no I, II and III, the effect of front sharpness of the wing can be seen. When front sharpness of the wing or k/c ratio increases, it is seen that total vortex magnitude around the wing decreases. The decrease of total vortex magnitudes can be caused from decreasing of free-stream velocity with respect to constant Reynolds number. Similar characteristics can be seen negative and positive vortex values of x, y and z direction vortices. The change of x-direction vortex values is less than y and z direction vortices. Because of free-stream flow directly goes on x-direction.

Test no IV, V and VI have similar flow conditions with test no I. However the only difference is the effect of radius of fillet of wing junction. When adding the fillet on the wing at test no IV, a little increase of vortex magnitudes and vortices in x, y and z directions occur with respect to no fillet condition. But when increasing of radius of fillet of wing junction, it is seen that total vortex magnitude decreases. The least values of x direction vortices are found in test no VI in comparisons of test no IV, V and VI. However y direction vortices of test no IV are higher than test no V and VI results. When negative z direction vortices decrease, positive z direction vortices increase in test no IV, V and VI. The best suitable results are determined in test no VI results when comparing the effect of radius of fillet of wing junction. Also the best results occur at the highest k/c ratio results of the wing in test no I, II and III.

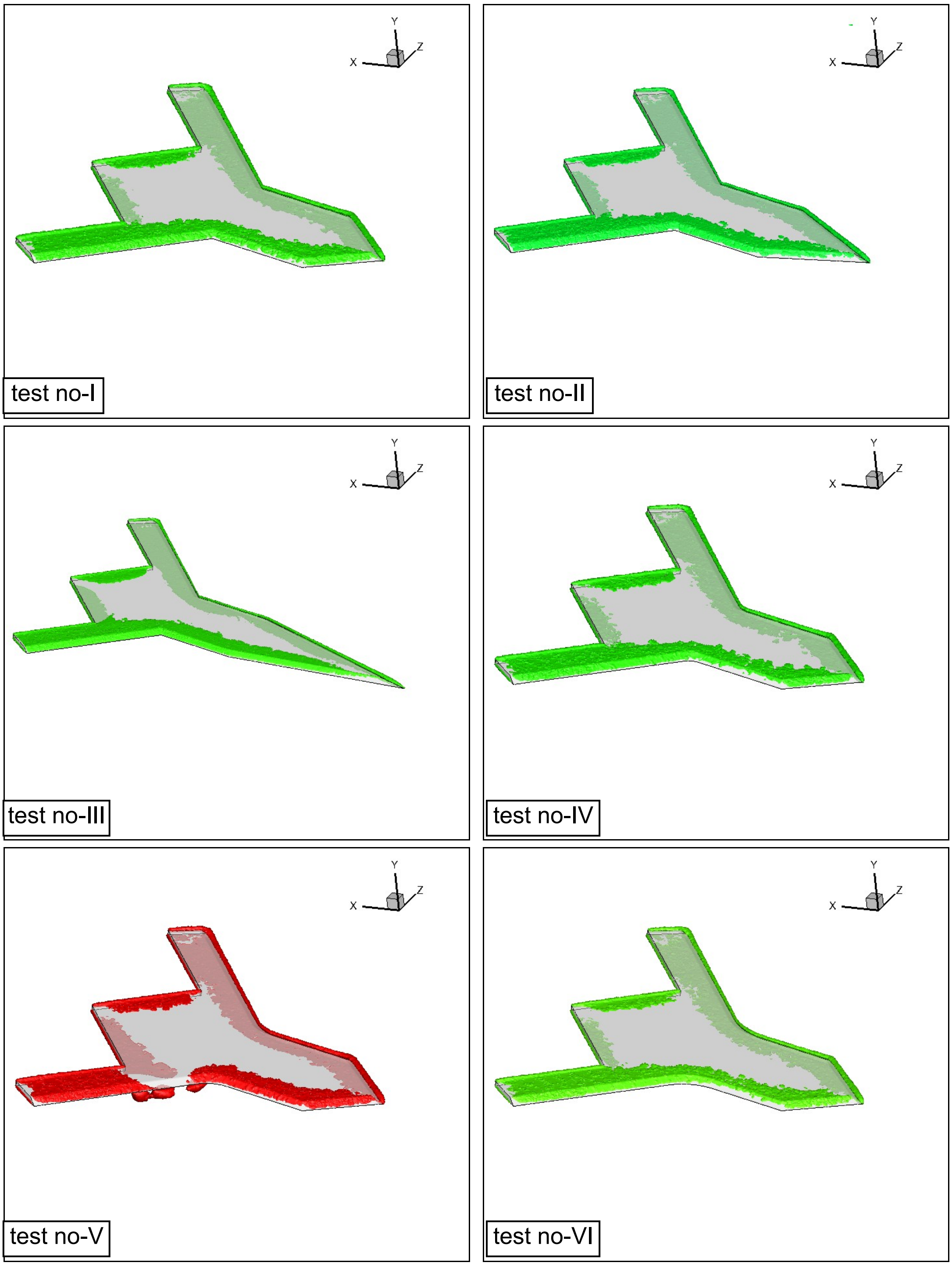

Figure 4: *Iso-surfaces of x vorticity at a value of 1.00*

3D flow topology of x vortices are given with it's a determined value by using iso-surfaces in fig. 4. The formation location of x vortices value of 1.00 is shown and it is seen that most of the locations are near side of edge of the wings. When increasing the k/c ratio, the vortex formations are decreasing. Fewer vortices occur at the tip of the wing by means of increasing of ratio of k/c. When comparing the results of test no IV, V and VI, vortices are decreasing especially at the fillet locations of wing junctions.

In fig 5, iso surfaces of z vorticity are given for a value of -3.00. The vortices are decreasing with both increasing of k/c ratio and radius of fillet of wing junction. The obtained vortices are decreasing especially upper surface of the wing from test no I to test no II and III. Also upper and side edge surface vortices are decreasing with creating and increasing radius of fillet of wing junction.

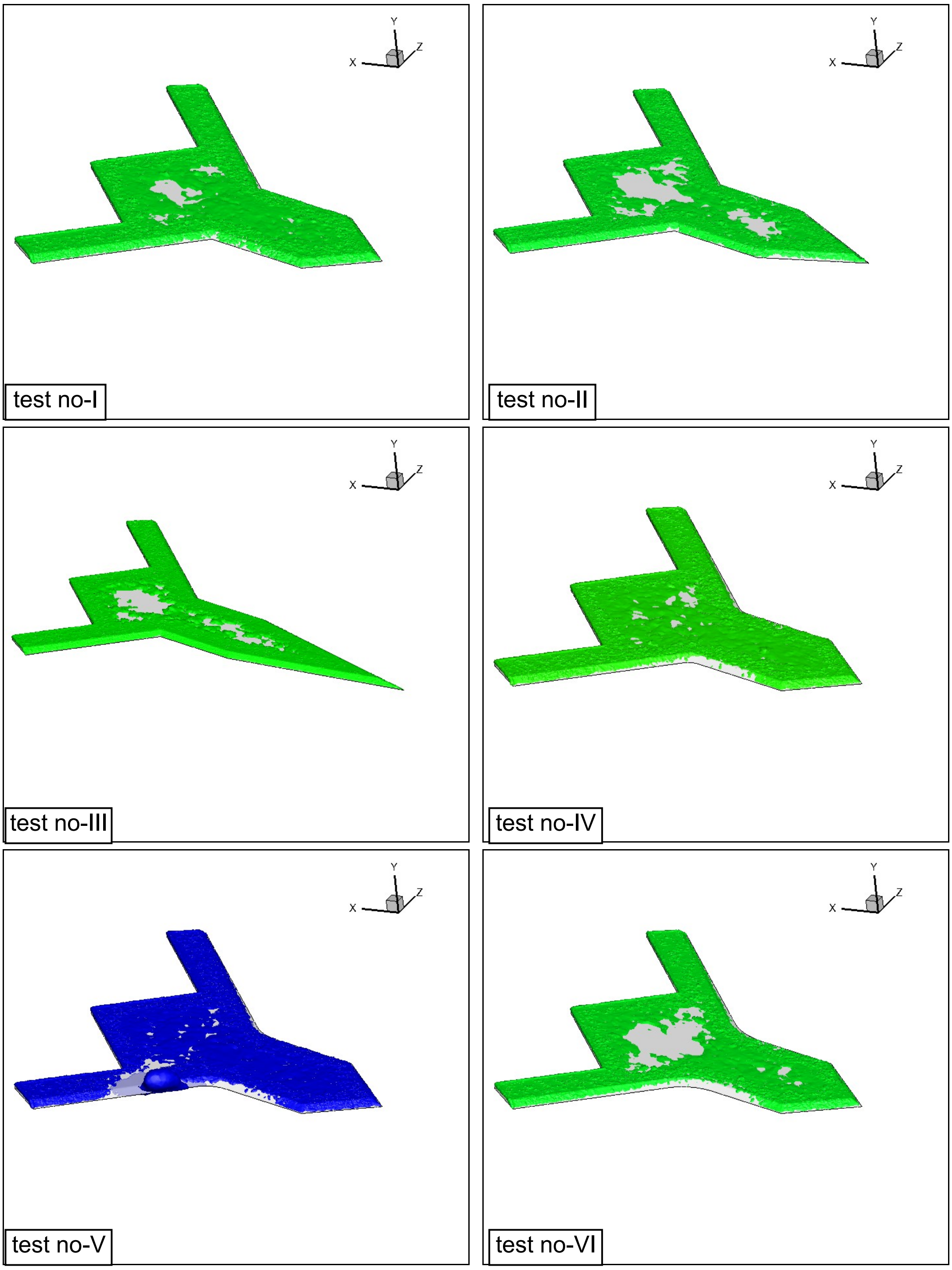

Figure 5: *Iso-surfaces of z vorticity at a value of -3.00*

Y direction of wall shear stress is given in fig. 6. Covered location of y direction shear stress value of 0.015 is decreasing with repect to increasing the ratio of k/c. Nearly only side edge locations of wing has y direction of wall shear stress at a value of 0.015 when ratio of k/c has the highest value. When comparing the results of test no IV and VI, it is determined that the covered locations of y direction wall shear stress is decreasing especially upper surface of the wing. Also more smooth formation of iso-surface of stress can be seen in test no IV rather than test no VI with the effect of increased radius of fillet of wing junction.

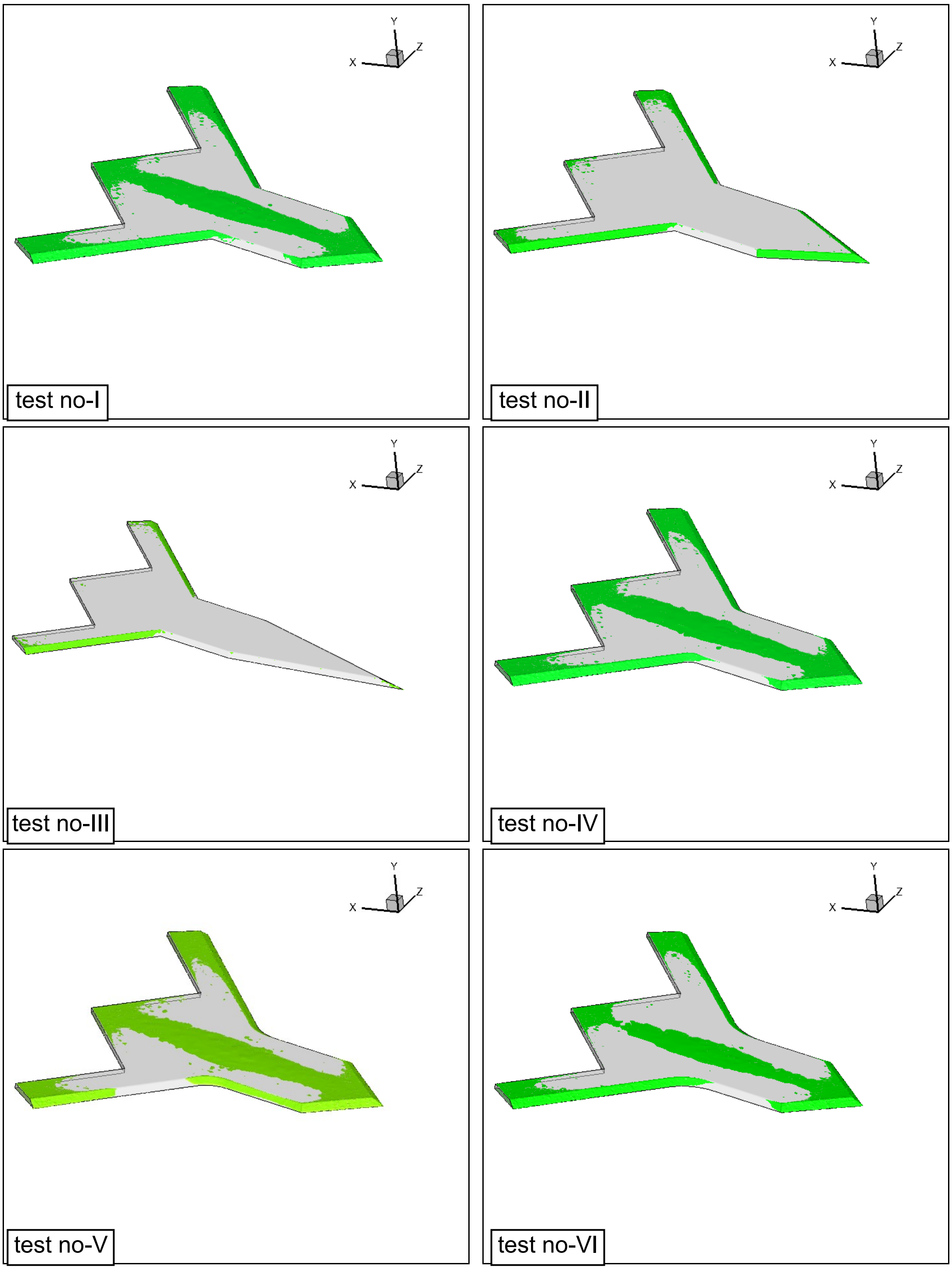

Figure 6: *Iso-surfaces of wall shear stress of y direction at a value of 0.015*

CONCLUSION

In this study the effect of fillet of wing junction and front sharpness are investigated on the flow characteristics around an x-45a unmanned combat air vehicle. Steady flow assumption is used in the analysis and k-epsilon turbulence model is selected for turbulence modeling. Some finding are obtained that

- Increasing the front sharpness cause to decrease vortices with respect to constant Reynolds number
- Decreasing of vortices initially begins on the tip edge and upper surface of the wing
- Increasing the radius of fillet of wing junction causes to decrease vortex formations and their magnitudes
- More smooth vortex formations occur with respect to addition of fillet on the wing junction

- Wall shear stress on y direction is decreasing with respect to increasing k/c ratio and radius of fillet of wing junction

It is seen that increasing the k/c ratio and rounding the wing junction give better vortex results and optimum performance.

References

Brooks, T. F. and Humphreys Jr, W. M. (2003) *Flap-edge aeroacoustic measurements and predictions*, Journal of Sound and Vibration, Vol. 261, p: 31–74, 2003

Cadogan, D., Graham, W. and Smith, T. (2003) *Inflatable and rigidizable wings for unmanned aerial vehicles*, 2nd AIAA “Unmanned Unlimited” Systems, San Diego, CA, p: 1-9, Sept. 2003

Cummings, R. M., Morton, S. A. and Siegel, S. G. (2008) *Numerical prediction and wind tunnel experiment for a pitching unmanned combat air vehicle*, Aerospace Science and Technology, Vol. 12, p: 355–364, 2008

Fu, S., Xiao, Z., Chen, H., Zhang, Y. and Huang, J. (2007) *Simulation of wing-body junction flows with hybrid RANS/LES methods*, International Journal of Heat and Fluid Flow, Vol. 28, p: 1379–1390, 2007

Hung, J. Y. and Gonzalez, L. F. (2012) *On parallel hybrid-electric propulsion system for unmanned aerial vehicles*, Progress in Aerospace Sciences, Vol. 51, p: 1–17, 2012

Mary, I. (2003) *Large eddy simulation of vortex breakdown behind a delta wing*, International Journal of Heat and Fluid Flow, Vol. 24 p: 596–605, 2003

Peng, C and Jinglong, H. (2012) *Prediction of flutter characteristics for a transporting with wingtip devices*, Aerospace Science and Technology, Vol. 23, p: 461-468, 2012

Sahin, B., Yayla, S., Canpolat, C. and Akilli, H. (2012) *Flow structure over the yawed nonslender diamond wing*, Aerospace Science and Technology, Vol. 23 p: 108–119, 2012

Turan, O. (2012) *Exergetic effects of some design parameters on the small turbojet engine for unmanned air vehicle applications*, Energy, Vol. 46, p: 51-61, 2012

Versteeg, H.K. and Malalasekera, W. An introduction to computational fluid dynamics, the finite volume method, Longman Scientific & Technical, 1995, 1st edition

Yavuz, M. M. (2012) *Transformation of flow structure on a delta wing of moderate sweep angle during pitch-up maneuver*, Journal of Fluids and Structures, Vol. 33, p: 59–69, 2012