DOI: 10.11776/j.issn .1000-4939.2025.01.003

WEI Ziyan, LI Jie, YANG Zhao

2025-02-13

ChinaRxiv

Abstract

Junctions between aircraft components readily induce corner flow structures, particularly for T-tail aircraft, where corner flow separation occurring at small sideslip angles can cause aircraft directional static instability. The k-ω SST method based on the quadratic constitutive relation (QCR) is employed to compute the Rood airfoil standard model for validating the computational accuracy of the k-ω SST-QCR approach, and to fair the horizontal tail-vertical tail junction of a validation aircraft to mitigate corner separation, thereby suppressing flow separation induced by complex interactions as the boundary layers of the horizontal and vertical tails merge at the junction, and achieving attached flow under small sideslip conditions. Results indicate that separated flow on the leeward side of the vertical tail is the main cause of directional static instability; through fairing cone modification at the horizontal tail-vertical tail junction, corner flow separation is significantly suppressed, restoring directional static stability at small sideslip angles.

Full Text

DOI: 10.11776/j.issn.1000-4939.2025.01.003

Numerical Computation and Optimization of Corner Separation on a Validation Aircraft

WEI Ziyan, LI Jie, YANG Zhao

(School of Aeronautics, Northwestern Polytechnical University, 710072 Xi' an)

Abstract: At the junctions of aircraft components, corner-flow structures are readily induced. In particular, for T-tail aircraft, under small sideslip angles, separation of the corner flow may lead to directional instability of the aircraft. The k-ω SST method based on the quadratic constitutive relation (QCR) is used to compute the Rood airfoil standard model to verify the computational accuracy of the k-ω SST-QCR method. It is then applied to rectify the corner separation at the horizontal-tail/vertical-tail junction of a validation aircraft, suppressing the flow separation caused by the complex interaction after the boundary layers of the horizontal and vertical tails merge at the junction, so that no separated flow occurs at the horizontal-tail/vertical-tail junction under small sideslip angles. The results show that the separated flow on the leeward side of the vertical-tail dorsal surface is the main cause of directional instability. By adopting a fairing-cone modification at the horizontal-tail/vertical-tail junction, corner-flow separation is significantly suppressed, and directional stability is restored under small sideslip angles.

Keywords: corner separation; quadratic constitutive relation; wing/body junction; flow control; directional stability

Chinese Library Classification No.: V211
Document Code: A
Article No.: 1000-4939(2025)01-0029-11

Numerical simulation and optimization of junction separation for a verification aircraft

WEI Ziyan, LI Jie, YANG Zhao

(School of Aeronautics, Northwestern Polytechnical University, 710072 Xi' an, China)

Abstract: The junction flow separation structure can be induced at the junction of aircraft components. Especially for a T-type tail aircraft, the junction separation flow may lead to directional instability under small sideslip angle. The k-ω SST-QCR method is used to analyze the corner separation at the T-type tail junction of a verify aircraft, and to suppress the flow separation caused by the complex interaction of the boundary layer. Results show the separation flow on the leeward side of the vertical tail causes the directional instability. The corner flow separation can be suppressed by using the fairing cone at the junction, so that the aircraft can maintain the directional stability at a small sideslip angle.

Key words: junction separation; QCR; wing/body junction; flow control; directional stability

Corner-separation flows exist widely in various flows commonly encountered in daily life, such as in turbomachinery, compressors, wings, bridges, and heat sinks for electronic equipment. Corner flow occurs because the upstream boundary layer collides with an obstacle, producing a relatively complex three-dimensional separated flow. Corner flow usually occurs when a boundary-layer flow encounters an obstacle protruding from a surface; this phenomenon occurs not only for blunt obstacles

---

Date received: 2022-12-19
Foundation item: National Natural Science Foundation of China (No. 11972304; 91952302)
Corresponding author: Li Jie, Professor. E-mail: lijieruihao@nwpu.edu.cn
Citation format: Wei Ziyan, Li Jie, Yang Zhao. Numerical computation and optimization of corner separation on a validation aircraft[J]. Chinese Journal of Applied Mechanics, 2025, 42(1): 29–39.
WEI Ziyan, LI Jie, YANG Zhao. Numerical simulation and optimization of junction separation for a verification aircraft[J]. Chinese Journal of Applied Mechanics, 2025, 42(1): 29–39.

there, streamlined obstacles can also produce such physical phenomena. This is caused by the sudden change in the pressure gradient induced by the obstacle and by the three-dimensional effects arising from horseshoe-vortex separation caused by the obstacle. Except at very low Reynolds numbers, whether in a laminar boundary layer or a turbulent boundary layer, it is prone to occur over an extremely wide range of Reynolds numbers[^1].

For aircraft, corner flow generally occurs at junctions of components such as the wing–fuselage, nacelle–wing, and horizontal tail (therefore corner flow is also called junction-region flow). The flow sweeps over or beats against the connected components, affecting the operational safety of aircraft components; at the same time, the junction also generates additional interference drag, affecting the aerodynamic performance of the aircraft, and in severe cases may even affect the longitudinal and lateral stability of the aircraft[^1]. Therefore, the design of junctions such as the wing–fuselage, nacelle–wing, and horizontal tail is very important. Especially for laminar-flow aircraft, the airfoils used for the wing and horizontal tail have relatively small leading-edge radii, and the position of relative thickness is relatively far aft, making them more prone to flow separation in the junction region[^2].

The flow interference in the wing–fuselage junction region is essentially produced by the convergence of the fuselage boundary layer and the wing boundary layer at the junction. Because the two boundary layers merge and interfere with each other, the junction flow has strong anisotropic characteristics; moreover, this flow has a relatively strong unsteady effect[^1–^2]. At the same time, this physical phenomenon changes substantially with factors such as flight condition and wing leading-edge radius; thus the flow generated by such boundary-layer interaction is very complex[^3]. A better understanding of the flow mechanisms of this mutual interference between the two boundary layers is of great help for aircraft design and for improving aircraft aerodynamic performance.

At present, studies of junction-region flow mainly focus on flow characteristics such as the leading-edge spatial separation point, the double-peak instability of the leading-edge flow, and the leading-edge horseshoe vortex[^2]. Early research on corner flow was primarily experimental. SIMPSON[^4] conducted wind-tunnel experiments on a 3:2 elliptical NACA0020 wing section with a leading edge; the experiments showed that the generation and shedding of the leading-edge horseshoe vortex on the plane wing section at the bottom of the corner region lead to low-frequency oscillations in the leading-edge region of the wing section. FLEMING et al.[^5] measured the velocity in the wing wake and tail traces and studied the transverse flow velocity pattern caused by the transverse adverse pressure gradient. LIEN et al.[^6], ÖLÇMEN et al.[^7–^9], and ANDERSON et al.[^10], in order to study the anisotropy of near-wall turbulent flow, measured the strong transverse flow at the saddle point in experiments.

For corner separation flow, especially separation flow located near the trailing edge, relatively few studies have been conducted so far. The corner separation flow generated by the mutual interference of the two boundary layers belongs to the second type of secondary flow[^1]; the flow has extremely strong anisotropy.

Fig. 1 Junction flow

Figure 1: Fig. 1 Junction flow

For general RANS methods, especially the S-A (Spalart-Allmaras) model and SST model based on linear eddy-viscosity models, which are relatively commonly used in engineering, although they have good accuracy in solving the flow in regions other than the interference region, within the influence range of the horseshoe-vortex system—especially in the separation region—it is almost impossible to obtain relatively ideal results.

Because nonlinear terms are lacking, the predicted separation region deviates greatly from the experiment. This includes the SST model, which captures the flow details of other types relatively well; the deviation in separation size and separation location can be as large as 100%[^2]. APSLEY et al.[^11] used 12 turbulence models to numerically simulate the wing-fuselage junction; none of the 12 turbulence models used in the computations could capture the separation location and separation size well, and some models could not capture the leading-edge saddle point and horseshoe-vortex structure well either. PARNEIX et al.[^12] used the V2F model to numerically simulate wing-fuselage corner flow and captured the three-dimensional separation line, the leading-edge horseshoe vortex, and its strength, but failed to capture the separation flow in the junction region. LIEN et al.[^6] developed an improved V2F method to compute wing-fuselage corner separation; although separation was successfully captured, the correct separation location and size were not obtained.

Because there is insufficient understanding of junction-region flow separation, and because turbulence models commonly used at present based on the linear eddy-viscosity assumption have insufficient accuracy in capturing junction-region separation flow, since 2016 NASA has carried out a series of studies on the corner separation flow produced by wing-fuselage interference at the trailing edge[^13-^17]. On the one hand, these studies were intended to investigate the flow characteristics of the corner region; on the other hand, they were also intended to form a computational fluid dynamics (CFD) standard case to assess whether current CFD technology has higher solution accuracy, especially for flow details such as Reynolds stresses. Experiments were conducted so that more evaluation quantities would be available, and at the same time clearer directions for improvement and enhancement would be provided.

图 1　结合区流动

Fig. 1　Junction flow

To explore the causes of the influence of junction-region separation flow on the directional static stability of a certain validation aircraft and the corresponding improvement measures, this paper first introduces the complexity of junction-region flow and the insufficiency of currently mature engineering RANS methods in capturing junction-region flow. Then, the k-ω SST-QCR method is used to verify the capture accuracy for junction-region flow. The paper performs computations on the Rood standard model[^1], named Rood, and compares and validates the results against experimental data. At the same time, the turbulence model is used to compute a validation aircraft at small angle of attack and under different side-slip-angle conditions, and the results are compared with wind-tunnel experiments to verify the reliability of the method and infer the causes of the phenomenon that leads to directional instability. Finally, through a streamlined-strake flow-control method, flow control is carried out for the horizontal-tail junction region to suppress flow separation in the junction region and thereby···

analyzing the causes leading to longitudinal static instability and proposes improvement measures.

1 Numerical Method

1.1 Finite-Volume Method and Spatial-Temporal Discretization Method

Let Ω denote an arbitrary finite control volume, with the corresponding boundary surface S. The integral form of the N-S equations can be expressed as

$$\frac{\partial}{\partial t}\iiint_{\Omega} Q\mathrm{d}\Omega + \iint_{S} F\mathrm{d}S = \iint_{S} F^{\mathrm{v}}\mathrm{d}S \qquad (1)$$

where Q is the conservative variable, F is the inviscid flux, and F^{v} is the viscous flux, specifically

$$F = \begin{bmatrix} \rho u & \rho v & \rho w \\ \rho uu + p & \rho uv & \rho uw \\ \rho vu & \rho vv + p & \rho vw \\ \rho wu & \rho wv & \rho ww + p \\ \rho h_t u & \rho h_t v & \rho h_t w \end{bmatrix},$$

$$F^{\mathrm{v}} = \begin{bmatrix} 0 & 0 & 0 \\ \tau_{xx} & \tau_{yx} & \tau_{zx} \\ \tau_{xy} & \tau_{yy} & \tau_{zy} \\ \tau_{xz} & \tau_{yz} & \tau_{zz} \\ \varphi_x & \varphi_y & \varphi_z \end{bmatrix}, \qquad Q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho e_t \end{bmatrix}.$$

Common spatial discretization schemes mainly include central schemes and upwind schemes. Compared with central schemes, upwind schemes do not require the addition of artificial viscosity, have better computational stability, and are suitable for numerical solution of complex flows. Upwind schemes are mainly divided into flux-vector splitting and flux-difference splitting. In this study, the inviscid terms are spatially discretized using a flux-difference-splitting scheme, while the viscous terms are spatially discretized using a second-order central scheme. This study adopts a dual-time implicit approximate-factorization method for time advancement. This method obtains the unsteady solution in physical time by solving a steady solution in a pseudo-time domain. The physical-time step is selected to be relatively large, while the stability-limited step is relatively small. The sub-iteration method in the pseudo-time direction, i.e., $\tau - TS$ iteration, makes the temporal discretization accuracy reach second order. To improve the computational efficiency of the numerical simulation method, acceleration measures such as local time stepping and multigrid are applied to accelerate convergence [18-20].

1.2 Boundary Conditions

This study mainly uses far-field boundary conditions and no-slip wall boundary conditions. The far-field boundary condition consists of one-dimensional Riemann invariants R^+ and R^-, from which the local speed of sound a_{far} and boundary-normal velocity $\bar{u}_{\text{far}}$ can be obtained as

$$a_{\text{far}} = \frac{\gamma - 1}{4}\left(R^+ - R^-\right), \quad \bar{u}_{\text{far}} = \frac{1}{2}\left(R^+ + R^-\right) \tag{2}$$

By decomposing the tangential and normal velocities, the velocity components can be expressed as

$$\begin{cases} u_{\text{far}} = u_{\text{ref}} + \dfrac{\xi_x}{|\nabla \xi|}\left(\bar{u}_{\text{far}} - \bar{u}_{\text{ref}}\right) \\ v_{\text{far}} = v_{\text{ref}} + \dfrac{\xi_y}{|\nabla \xi|}\left(\bar{u}_{\text{far}} - \bar{u}_{\text{ref}}\right) \\ w_{\text{far}} = w_{\text{ref}} + \dfrac{\xi_z}{|\nabla \xi|}\left(\bar{u}_{\text{far}} - \bar{u}_{\text{ref}}\right) \end{cases} \tag{3}$$

The reference velocity for the inflow boundary is the freestream velocity, while the reference velocity for the outflow boundary is the velocity at the interior point adjacent to the boundary. When $\bar{u}_{\text{far}} > 0$, the boundary is an outflow, and the entropy is determined by the interior-point value; when $\bar{u}_{\text{far}} < 0$, the boundary is an inflow, and the entropy is determined by the inflow value. The boundary density and pressure are determined from entropy and the speed of sound. For the no-slip wall boundary condition, the fluid velocity on the wall is defined as 0, i.e., $u_{\text{w}} = v_{\text{w}} = w_{\text{w}} = 0$. For an adiabatic wall, the normal temperature gradient is 0, i.e., $\partial T / \partial n = 0$.

1.3 k-ω SST-QCR Turbulence Model

Because of the anisotropy of corner-separation flow, conventional RANS methods have difficulty in accurately capturing the location and size of corner separation. Devenport et al. [21] pointed out that the inherent problems of RANS methods limit their ability to correctly capture the separation location and the evolutionary development of the corner-separation structure. Therefore, the turbulence model adopted in this study is the k-ω SST model modified by considering the quadratic constitutive relation, namely k-ω SST-QCR.

The k equation and ω equation in the conservative form of k-ω SST are

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho u_j k)}{\partial x_j}=P-\beta^*\rho\omega k+\frac{\partial}{\partial x_j}\left[\left(\mu+\sigma_k\mu_t\right)\frac{\partial k}{\partial x_j}\right] \quad (4)$$

$$\frac{\partial(\rho\omega)}{\partial t}+\frac{\partial(\rho u_j\omega)}{\partial x_j}=\frac{\gamma}{\nu_t}P-\beta\rho\omega^2+\frac{\partial}{\partial x_j}\left[\left(\mu+\sigma_\omega\mu_t\right)\frac{\partial \omega}{\partial x_j}\right]+2(1-F_1)\frac{\rho\sigma_{\omega 2}}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \quad (5)$$

where ρ is the density; k is the turbulent kinetic energy; $\sigma_k=0.5$; the eddy viscosity is $\nu_t=\mu_t/\rho$; and μ_t is the molecular-transport viscosity [19].

$$P=\tau_{ij}\frac{\partial u_i}{\partial x_j} \quad (6)$$

$$\tau_{ij}=\mu_t\left(2S_{ij}-\frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right)-\frac{2}{3}\rho k\delta_{ij} \quad (7)$$

$$S_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \quad (8)$$

$$\mu_t=\frac{\rho a_1 k}{\max\left(a_1\omega,\Omega F_2\right)} \quad (9)$$

Spalart et al. [20] proposed a QCR method based on the S-A model. This method adjusts the linear eddy-viscosity model; therefore, this method

can be used for a series of turbulence-model modifications based on the linear eddy-viscosity assumption. In the QCR method, only τ_{ij} is modified, namely

$$\tau_{ij,\mathrm{QCR}}=\tau_{ij}-C_{r1}\left(O_{ik}\tau_{jk}+O_{jk}\tau_{ik}\right) \quad (10)$$

$$O_{ik}=\left(\frac{\partial u_i}{\partial x_k}-\frac{\partial u_k}{\partial x_i}\right)\Big/\sqrt{\frac{\partial u_m}{\partial u_n}\frac{\partial u_m}{\partial u_n}} \quad (11)$$

where $C_{r1}=0.3$, and the remaining terms are left unchanged.

2 Numerical-Method Computational Validation

To verify whether the k-ω SST-QCR method is applicable to capturing corner-region separated flow, a relatively simple Rood wing-body junction model with comparatively abundant research data is first selected for simulation. The capability and accuracy of this method in capturing the separation position and flow details at a simple wing-body junction under experimental conditions are verified. Subsequently, it is used to compute the yawing-moment characteristics of the validation aircraft and is compared with experiments, so as to ensure that the method is accurate and reliable.

All computations in this study use the multi-block structured-grid solver developed by the research group. In the following computations, a steady calculation is first performed to obtain a stable initial flow field, and then an unsteady calculation is carried out. The unsteady calculation is run for 50 unsteady periods based on the freestream velocity and characteristic length, $\Delta T = U_\infty/L$, and the time step for the unsteady calculation is $\Delta T/20$. Here, U_∞ is the freestream velocity, and L is the characteristic length (the characteristic length is the one used for calculating the Reynolds number). On this basis, another 20 unsteady periods are computed, and the resulting flow fields are averaged to obtain the mean flow field. Similarly, for the aerodynamic-performance parameters, 20 unsteady periods are averaged. The flow-field parameters and aerodynamic characteristics in what follows are all averaged results.

2.1 Computational Validation of the Rood Wing-Body Junction

At the present stage, a commonly studied wing-body junction model is the Rood model, which consists of a Rood airfoil extending straight to form a simple wing section mounted vertically on a flat plate. The Rood airfoil is composed of a NACA0020 airfoil and a $3:2$ ellipse, with the region from the leading edge of the NACA0020 airfoil to near the maximum thickness modified into a $3:2$ ellipse. The Rood airfoil has occupied an important position in current studies. DEVENPORT et al.[21] conducted a large number of experiments on the Rood airfoil. The experimental data are rich and varied; moreover, the experiments observed many typical flow features and details of junction-region flow, such as spatial separation points, closed separation lines, and low-shear lines in the leading-edge region, and small-scale separation flow and fishtail-like streamlines in the trailing-edge region. Therefore, many scholars have carried out numerous improved experiments and numerical-simulation studies based on these experiments[21–25]. SIMPSON[1] pointed out that the separated flow at the leading edge of the junction belongs to the first type of Prandtl secondary flow, whereas the separated flow at the trailing edge belongs to the second type of Prandtl secondary flow; both flows have strongly anisotropic characteristics. Therefore, for general turbulence models based on the linear eddy-viscosity assumption, their inherent deficiency of lacking nonlinear terms makes it impossible for them to capture this type of separated flow well in this region.

Fig. 2 Depiction of the Rood experiment

Figure 2: Fig. 2 Depiction of the Rood experiment

Fig. 3 Depiction of the dense mesh

Figure 3: Fig. 3 Depiction of the dense mesh

This study mainly carries out computations for the wing-body junction model and experimental conditions used in Ref. [21]. The Rood airfoil model used in the experiment has a chord length of 305 mm, a maximum thickness of 71.7 mm, namely a maximum relative thickness of 24%, and a height of 229 mm. This Rood airfoil is inserted vertically into a flat-plate floor. The wind-tunnel freestream velocity is 27.5 m/s, and the experiment is performed at an angle of attack of 0° and a sideslip angle of 0°. A schematic of the Rood wing-body junction model is shown in Fig. 2.

图 2　Rood 实验示意图 [21]

Fig. 2　Depiction of the Rood experiment[21]

For this Rood model, three sets of multi-block structured grids are generated based on a C-type topology, with γ^+ kept near 1.0 for all of them. The grid counts of the three sets are 2.60 million, 6.30 million, and 13.60 million, respectively. The detailed grid-node distributions for the junction region and the overall grid are shown in Table 1, and a local view of the flat-plate region at the bottom of the dense-grid junction is shown in Fig. 3.

表 1　3 套网格分布

Tab. 1　Detail of three different grids

Grid	Number of grid points	Streamwise × wall-normal × spanwise	Growth rate
Coarse grid	2.6×10^6	$241 \times 129 \times 81$	0.20
Medium grid	6.3×10^6	$341 \times 153 \times 121$	0.10
Dense grid	13.6×10^6	$621 \times 181 \times 121$	0.05

图 3　密网格示意图

Fig. 3　Depiction of the dense mesh

The computation uses the k-ω SST-QCR method for comparison, in order to verify

calculation method and mesh, as well as the calculation accuracy. The incoming-flow inlet, outlet location, and model are kept consistent with the experiment

. Far-field boundary conditions are used for the inlet and outlet boundaries, and no-slip wall boundary conditions are used for the bottom flat plate and the wing.

Fig. 4(a) comparison of surface pressure distributions at $Y/T = 0.132$

Figure 4: Fig. 4(a) comparison of surface pressure distributions at $Y/T = 0.132$

Fig. 4(b) comparison of surface pressure distributions at $Y/T = 1.726$

Figure 5: Fig. 4(b) comparison of surface pressure distributions at $Y/T = 1.726$

The k-ω SST-QCR turbulence model is used to carry out RANS calculations on the coarse, medium, and fine meshes, and to analyze whether it can satisfactorily capture typical flow features in the junction region, such as the spatial separation point of the junction-region flow, the separation line on the bottom flat plate, and the low-shear line.

First, the surface pressure coefficient C_p on the wing section in Ref.

is compared (Fig. 4). It can be seen that, because the number of simple models of this type is relatively large in the above three meshes, and because the computational state is 0° angle of attack and 0° sideslip angle, at locations a certain distance away from the bottom flat plate the wing-section surface flow is not complicated. Even for the coarse mesh, the mesh in the wing region contains about 2.6 million cells. Therefore, the C_p values obtained from the three meshes at the two section locations differ little and agree well with the experimental results. This indicates that, in terms of macroscopic quantities at locations far from the bottom junction region, the mesh size and computational method are suitable for the calculation of the Rood wing-body junction-region model.

(a) Comparison of surface pressure distributions at $Y/T = 0.132$

(b) Comparison of surface pressure distributions at $Y/T = 1.726$

Fig. 4 The calculated surface C_p based on three grids compared with experiment

Similarly, this also shows that the k-ω SST-QCR method captures the surface pressure distribution on the wing in the wing-body junction-region model well. However, in order to capture the detailed flow features near the bottom, all subsequent analysis results are obtained from calculations based on the fine mesh.

Because the flow on the wing surface far from the bottom flat plate is not the most complex part of the flow field in the wing-body junction region, whereas the part of the leading-edge region of the junction region close to the bottom surface is the most complex part of the junction-region flow and also the part with the strongest anisotropy of the flow in all directions, the flow in this part belongs to the first type of Prandtl secondary flow. At the same time, turbulence models based on the linear eddy-viscosity assumption have strong limitations in simulating Prandtl secondary flow. Therefore, it is necessary to compare the flow-field details such as streamlines and pressure distributions in the leading-edge region of the Rood wing, so as to further judge the capability of the k-ω SST-QCR method to capture flow details in the junction region.

First, the streamlines calculated by k-ω SST-QCR at the bottom of the wing are compared with the experimental results in Ref.

21

. Fig. 5 shows the oil-flow experimental results from Ref.

21

and the bottom streamlines and enlarged leading-edge region calculated by k-ω SST-QCR. By comparing Fig. 5(a) with Fig. 5(b), it can be found that the k-ω SST-QCR method based on the linear eddy-viscosity assumption can clearly show that the spatial saddle point it captures is located at $X/T = -0.47$, consistent with the location measured in the experiment. In addition, the separation line begins from the three-dimensional saddle-point location and closes after moving downstream and meeting the low-shear line; the overall trend of the features captured in the experiment is also basically consistent. Moreover, the results calculated by the k-ω SST-QCR method also clearly capture the low-shear line after the leading-edge-region separation line, and its starting position is also relatively close to that in Ref.

21

.

However, in the leading-edge region, the streamline on the bottom plane calculated by k-ω SST-QCR has a pseudo-saddle point in front of the low-shear line, as shown in Fig. 5(c). LEE et al.

22

pointed out that this pseudo-saddle point is caused by the k-ω SST method itself, and the use of QCR correction in the present study did not significantly improve this pseudo-saddle point. According to the conclusion of LEE et al.

22

, the existence of this pseudo-saddle point does not have a significant influence on the overall macroscopic flow field or the flow details in other regions.

Fig. 6 shows the pressure-coefficient distributions on the bottom of the wing leading-edge region for the experimental results and the k-ω SST-QCR calculation results. It can be seen from the figure that the overall distribution trend and magnitudes of the surface pressure coefficient in the bottom region of the wing leading edge obtained by the calculation are basically consistent with the pressure-coefficient distribution measured in the experiment. However, near the elliptical low-shear line shown in Ref.

21

, the k-ω SST-QCR calculation results exhibit a local low pressure in the region indicated by the black dashed line in Fig. 6, causing the pressure-distribution contour lines to bend near the low-shear line. In the experiment, no such band-like low-pressure region is clearly observed at the same location, and this band-like low-pressure region divides the high-pressure region near the leading edge into two parts, whereas in the experiment there is only one complete local high-pressure region near the wing leading edge. By comparing the computational results of six different models, including linear

⋯nonlinearity turbulent models; it was found that all RANS models exhibit this phenomenon. Therefore, it may be caused by the inherent deficiencies of the RANS method, and the QCR correction cannot alleviate the occurrence of this phenomenon.

(a) Oil-flow experimental results from Ref. [21]

(b) Calculated bottom-surface streamlines

(c) Surface streamlines near the pseudo-saddle point

Fig. 5 Streamline plot on the bottom plane from experiment and computation

Fig. 5 The streamline at the bottom plane of the experiment and calculation

This problem not only appears in the RANS computational results. According to PAIK et al. [23] and SRIKUMAR et al. [24], similar phenomena also occurred in the results obtained using DES and wall-modelled LES. Although the use of the above models can significantly alleviate this problem, the cost is also relatively high. The computational results obtained by RYU et al. [25] using wall-resolved LES did not show a localized band-shaped low-pressure region around the low-shear line, and they pointed out that, because the RANS, DES, and wall-modelled LES methods all use modeled solutions when solving the bottom-wall boundary layer, the surface pressure distributions obtained by such model computations all have the above characteristics, while the basic flow-field characteristics and macroscopic quantities can still be obtained in a basically satisfactory manner.

Fig. 6 Comparison of the calculated bottom-plane surface C_p with experiment

Fig. 6 The calculated bottom plane surface C_p compared with experiment

In addition, by observing the near-wall surface region near the symmetric plane $X/T = -0.02$ at the leading edge of the wing, DEVENPORT et al. [21] found that reattachment lines and separation lines exist there; in the corner region between the bottom plane and the wing, a small secondary separation region was observed. ALBERTS [26] captured secondary separation lines using both

Fig. 8

Figure 6: Fig. 8

k-ω SST and wall-resolved LES, and in the wall-resolved LES simulation results, it was also observed that the third separation line corresponded to the third corner vortex in the space; however, the third line was not observed in the results obtained using the k-ω SST method. RYU et al. [25] also observed similar results. In the k-ω SST-QCR method used in this study, a relatively clear third separation line was captured (Fig. 7), indicating that this method has improved capability in capturing flow details compared with the SST model.

Fig. 7 Streamline plot of the secondary and tertiary separation lines

Fig. 7 The streamline of the secondary and tertiary separation lines

Figure 8 shows the computation performed in this study using the k-ω SST-QCR method···

The streamlines obtained for the wing–body junction region on the bottom plane at the trailing edge of the wing are shown. By comparing with Fig. 1, it can be seen that the wake in the trailing-edge region calculated in this study exhibits the same flow trend as the wake observed in Ref. [21], namely a fishtail-like flow trajectory, indicating that the overall trend of the computation is relatively accurate. From the flow details shown in Fig. 8, it can still be seen that there is a very small recirculation region in the trailing-edge region in the present calculation. In addition, Ref. [21] pointed out that a small and indistinct separation region exists at the trailing edge, as shown in Fig. 1, indicating that the k-ω SST-QCR method captures the overall trend and the flow details well.

图 8 计算所得翼型后缘底部附近流线

Fig. 8 The calculated streamline at the tailing edge region at the bottom plane

In summary, although the k-ω SST-QCR method has certain deficiencies in capturing the flow details in a small range of the leading-edge region due to the inherent defects of the RANS method, its simulation accuracy for the macroscopic quantities of the wing–body junction region, the overall flow trend, and the location of junction-region flow separation is already good overall, satisfying the accuracy requirements for the macroscopic quantities and overall flow-trend capture needed in aircraft design. Therefore, all subsequent computations adopt the QCR-corrected k-ω SST method.

2.2 Computational Validation of the Verification Aircraft

Wind-tunnel tests were conducted for a certain verification aircraft at small angles of attack and small sideslip angles. The tests were carried out in the FL-8 low-speed wind tunnel. In this low-speed experiment, a belly-mounted balance

Fig. 9

Figure 7: Fig. 9

Fig. 10

Figure 8: Fig. 10

was used for force measurement, and a symmetric-balance measurement experiment was used to quantitatively subtract the influence of the aerodynamic shape of the balance apparatus on the aircraft flow field and aerodynamic forces. A $1:3.25$ scaled model was used in the experiment. The incoming-flow Mach number was 0.2, and the Reynolds number (Re), based on the mean aerodynamic chord length of the scaled-model wing, was approximately 2 million. The yawing moment coefficient obtained in the experiment is shown in Fig. 9.

From the experimental data, it can be seen that the complete verification aircraft exhibits directional static instability at small angles of attack and small sideslip angles, but recovers the characteristic of directional static stability at large sideslip angles. Based on the characteristics of junction-region flow separation and the fact that the vertical tail is the main component providing directional static stability for the aircraft, analysis in combination with the experimental data suggests that the cause of directional static instability may mainly lie in the junction-region separation flow generated at the junction of the horizontal and vertical tails.

图 9　验证机偏航力矩系数曲线

Fig. 9　Yawing moment coefficient of laminar verifying aircraft

When the aircraft flies at small sideslip angles, the yawing moment coefficient generated by its vertical tail is relatively small. Meanwhile, the yawing moment caused by the junction-region separation flow generated at the junction of the horizontal and vertical tails is also a small quantity, so that the verification aircraft exhibits directional static instability at small sideslip angles. When the aircraft enters a large-sideslip flight state, the yawing moment coefficient provided by its vertical tail is relatively large. Even if a corner-separation flow structure is generated between the horizontal and vertical tails, it cannot exert an essential influence on directional static stability. Therefore, the aircraft again exhibits the characteristic of directional static stability at large sideslip angles. This may be the reason why the verification aircraft exhibits directional static instability at small sideslip angles. The horizontal-tail model of the verification aircraft is shown in Fig. 10.

图 10　平垂尾模型

Fig. 10　Vertical and horizontal tails of experiment model

To verify the above preliminary analysis, computations were first carried out for the experimental model, verifying that the computational method and grid used can meet the analysis accuracy requirements for the macroscopic quantities and flow details of the verification aircraft, and providing data support for the subsequent analysis of the directional static instability phenomenon of the verification aircraft and its improved design scheme.

The Mach number used in the computation was 0.2, and the Reynolds number was 2 million, consistent with the experimental values. The yawing moment of the whole aircraft was computed and validated for the 0° and −4° angle-of-attack conditions. The whole-aircraft grid adopted a multi-block structured ...

mesh, with the far field taken as $\pm 40L$, where L is the aircraft span. The vertex at the aircraft nose is located at the origin. The total number of grid cells is about 30 million, and the surface mesh in the horizontal tail region is shown in Fig. 11.

Fig. 11 Depiction of vertical and horizontal tails mesh

Figure 12 shows the comparison between the yawing-moment coefficient measured in the experiment and that calculated using k-ω SST-QCR. It can be seen from Fig. 12 that the results obtained with the currently used mesh and numerical method agree well with the experimental results, and can very well capture the directional static instability of the aircraft at small angles of attack and small sideslip angles. This indicates that the present mesh and computational method are suitable for capturing the overall macroscopic quantities of this validation aircraft.

Fig. 12 Yawing moment coefficient of calculation and experiment

3 Analysis and Optimization of Corner Separation on the Horizontal Tail

In the previous section, the experimental model of a certain validation aircraft was computationally verified, and the directional static instability exhibited by the aircraft in the experiment under small sideslip angles was captured. To verify whether the conjecture in the previous section is correct, it is necessary to analyze the streamlines and C_p on the horizontal-tail surface. Figure 13 shows the surface streamlines on the windward and leeward sides of the vertical tail, respectively, at an angle of attack of 0° and a sideslip angle of 2°. From the streamlines, it can be seen that there is a relatively large separation region on both the windward and leeward sides of the vertical tail. The main cause leading to vertical-tail stall, whether on the leeward side or the windward side, requires further analysis. Since the separated flow on the windward side is more readily suppressed than that on the leeward side, flow-control measures that weaken as much as possible, or completely eliminate, the separated flow on the windward side are adopted to control the flow in the horizontal-tail junction region; this is more conducive to analyzing the specific causes leading to directional static instability.

Fig. 13 Streamline of origin vertical tail model

According to the viewpoints proposed in related studies [2, 27 − 29], corner separation is mainly related to the boundary layer. The larger the boundary-layer thickness and the higher the energy, the smaller the relative corner separation, and it may even be completely suppressed. At the same time, because there is no shielding in front of the horizontal tail of the validation aircraft and the leading-edge positions of the horizontal tail and vertical tail almost coincide, the boundary layer at this location is relatively thin, making flow separation very likely to occur at the boundary-layer junction. According to the rectification methods proposed in Ref. [1], the rectification methods applicable to the junction-region configuration of this validation aircraft generally include swept/fillet transition, leading-edge sweep, and fairing cones. The methods of swept transition or leading-edge sweep require relatively large modifications to the horizontal-tail configuration of this validation aircraft, and the effects of these two rectification methods are not obvious when the angle of attack/sideslip range varies greatly. Therefore, a fairing cone is adopted in the junction region to increase the thickness and energy when the two boundary layers merge, thereby reducing the separation size in the junction region. This type of flow-control measure may be more suitable for the horizontal-tail configuration of this validation aircraft.

Therefore, two fairing cones with different radii are selected for rectification. Their radii are respectively 40% and 80% of the maximum thickness of the vertical-tail airfoil at the intersection of the horizontal and vertical tails. Figure 14 shows the comparison of the two fairing-cone configurations. In the figure, T is the maximum thickness of the local airfoil of the vertical tail, C is the chord length of the local airfoil of the vertical tail, and R_L and R_S are the radii of the large and small fairing cones, respectively. The white dashed line represents the rotation axis of the fairing cone. The fairing-cone configurations with fairing-cone radii of 40% and 80% of the maximum thickness are referred to as the small-radius and large-radius fairing-cone configurations, respectively. The rectified models are computed at an angle of attack of 0° and sideslip angles within the range of ±10°; the Mach number and Reynolds number are kept the same as in the previous section.

Fig. 14 Two different fairing cones model

Figure 15 shows the yawing-···

The comparison of yawing moment coefficients shows that, compared with the original configuration, the fairing-cone configuration with the smaller radius provides a slight improvement in directional stability near a sideslip angle of 0°, but it still exhibits directional static instability within the small-sideslip-angle range, with no essential difference from the original configuration. In contrast, the fairing-cone configuration with the larger radius completely eliminates the phenomenon of directional instability, maintaining directional static stability over the range of sideslip angles from -10° to $+10^\circ$.

Fig. 15 The yawing moment coefficient of three different models

Figures 16–17 show the surface C_p and streamlines on the windward and leeward sides, respectively, for the small-radius and large-radius fairing-cone models at a sideslip angle of 2°. By comparing the two fairing-cone models with the original configuration at a sideslip angle of 2°, it can be seen that, for the fairing-cone configuration with the smaller radius, separation in the junction region on the windward side is well suppressed. However, a relatively large separation region still exists on the leeward side, so that a directionally statically unstable region remains within the small angle of attack and small sideslip-angle range. This also demonstrates that corner separation on the leeward side is the primary cause of directional static instability under small-sideslip-angle conditions.

Moreover, although the two fairing cones extend over the same length, because the radius of the small-radius fairing cone is less than one half of the maximum relative thickness of the vertical tail, the interaction between the boundary layers on the two sides in the junction region is not completely isolated. As a result, complex mutual interaction between the two boundary layers still exists in the junction region between the horizontal tail and the vertical tail. In particular, from the surface C_p, it can be seen that at the maximum-thickness location, i.e., the starting position of separation of the flow over the vertical-tail surface, there is a distinct high-pressure region (the location indicated by the red circle in Fig. 16b). This produces a strong adverse pressure-gradient region near the maximum-thickness location, thereby leading to flow separation in the corner region on the leeward side. In addition, the existence of the high-pressure region on the leeward side causes the overall force on the vertical tail to point toward the windward side, further resulting in directional static instability under small-sideslip-angle conditions. For the fairing-cone model with the larger radius, because its diameter exceeds the maximum relative thickness of the vertical tail, it basically isolates the direct mutual interference between the horizontal-tail and vertical-tail boundary layers. In both the original configuration and the small-radius fairing-cone configuration, the high-pressure region appearing at the maximum-thickness location on the windward side is completely suppressed, so that the flow remains attached without separation.

Fig. 16 C_p and streamlines of the small-radius fairing-cone model

Fig. 17 C_p and streamlines of the large-radius fairing-cone model

Figure 18 shows velocity contours at the station indicated by the dashed line in Fig. 17(b), for the original configuration and the large-radius fairing-cone configuration. It can be seen that, in the original configuration, separation begins at the $40\%C$ station and continues to the trailing edge. From the mean flow field, the separation region contains two vortex structures: a smaller vortex structure close to the wall, and a larger vortex located in the space on the leeward side.

Fig. 19

Figure 9: Fig. 19

Fig. 20

Figure 10: Fig. 20

Fig. 18 Contour of velocity at the same place of original and large fairing-cone models

In contrast, the large-radius fairing-cone configuration shows no such phenomenon as in the original configuration.

⋯the vortex structures exhibited in the model. This indicates that the large-radius fairing can effectively suppress flow separation on the leeward side. The flow fields of the two cases are basically similar on the windward side; thus it can be stated that the flow separation on the leeward side is the cause of the directional static instability.

Figure 19 shows the surface C_p distribution curves at the same station in the junction region for the two fairing models. It can be seen that, owing to the existence of separation on the leeward side, the smaller fairing model produces unloading over a relatively large area on the leeward side and loading on the windward side, thereby leading to directional static instability of the vertical tail. For the larger fairing model, although the C_p distributions on the windward and leeward sides are similar after 40% of the wing chord, the unloading on the windward side and loading on the leeward side in the leading-edge region are maintained; therefore, the characteristics of directional static stability at the small sideslip angle of the validation aircraft are preserved.

图 19　2 种整流模型交界处 C_p 曲线

Fig. 19　C_p of two fairing model at junction region

Figure 20 shows the drag characteristics of the two fairing models and the original model of the vertical tail at a 0° angle of attack and under different sideslip-angle conditions. The blue bar chart gives the drag reduction of the large-fairing model relative to the original model as a percentage of the drag of the original model.

图 20　不同构型阻力特性曲线

Fig. 20　Coefficients of drag calculated with different model

It can be seen that the drag characteristics of the smaller fairing model are basically consistent with those of the original model. This may be because the fairing increases part of the friction drag and pressure drag, while in the small fairing model the suppression effect on separation in the junction region is not obvious. By contrast, compared with the original model, the large fairing model produces a significant reduction in drag: at a sideslip angle of 0°, the drag is reduced by nearly 9%, and under the sideslip-angle conditions of ±4°, the maximum drag reduction is approximately 17%. Although the large fairing model brings about greater pressure drag and friction drag, because the separated-flow region in the junction region of the original model is relatively large, the interference drag it brings is also relatively large. The use of the larger fairing model can completely suppress the separated flow in the junction region, thereby reducing the drag of the horizontal and vertical tails as a whole.

4 Conclusions

In this study, the standard k-ω SST model with QCR correction was used to carry out computational validation of the Rood wing-section junction-zone benchmark and of a validation aircraft. At the same time, the directional static instability observed for the validation aircraft at small angles of attack and small sideslip angles was analyzed, and aerodynamic optimization was performed. The main conclusions are as follows.

1) The standard k-ω SST method with QCR correction can capture relatively well the location of the corner separated flow in the junction region and the typical flow features in the junction region. However, owing to the inherent deficiencies of the k-ω SST method, the captured false saddle points cannot be improved by the QCR correction and need to be corrected and improved by using other turbulence-model correction methods.

2) Under small-angle-of-attack and small-sideslip-angle conditions, corner-flow separation in the horizontal-tail/vertical-tail junction region not only increases drag, but may even lead to aircraft directional static instability and rudder-surface ineffectiveness. The use of a relatively large fairing that can completely suppress the separated flow in the junction region can not only eliminate the aircraft directional static instability, but also reduce the drag caused by the separated flow.

3) Corner separation on the leeward side of the vertical tail is the main reason for the generation of a reverse yawing moment by the vertical tail. By using a fairing that can completely enclose the horizontal-tail/vertical-tail junction region, the interaction between the boundary layers on the two sides can be effectively isolated, thereby suppressing the corner separated flow on both sides of the vertical tail.

In this study, fairings with two different radii were used to control flow separation in the junction region, and the ability of the fairings to improve directional static stability was qualitatively analyzed. However, the control effectiveness of the fairings on directional instability and flow separation under the influence of parameters such as different Mach numbers and Reynolds numbers requires further analysis and verification. At the same time, because isolating the boundary layers on both sides of the junction region requires a relatively large fairing and brings additional drag, whether fairing only in the region downstream of the maximum relative thickness can achieve a good flow-control effect requires further investigation.

References:

[1] SIMPSON R L. Junction flows [J] . Annual review of fluid mechanics, 2001, 33: 415-443.

[2] GAND F, DECK S, BRUNET V, et al. Flow dynamics past a simplified wing body junction[J]. Physics of Fluids, 2010, 22(11): 115111.

[3] PACIORRI R, BONFIGLIOLI A, DI MASCIO A, et al. RANS simulations of a junction flow[J]. International Journal of Computational Fluid Dynamics, 2005, 19(2): 179–189.

[4] SIMPSON R L. Aspects of turbulent boundary-layer separation[J]. Progress in Aerospace Sciences, 1996, 32(5): 457–521.

[5] FLEMING J L, SIMPSON R L, COWLING J E, et al. An experimental study of a turbulent wing-body junction and wake flow[J]. Experiments in Fluids, 1993, 14(5): 366–378.

[6] LIEN F S, KALITZIN G, DURBIN P A. RANS modeling for compressible and transitional flows[C]//Proceedings of the Summer Program. Stanford, California, USA: Stanford University Press, 1998, 1: 1998.

[7] ÖLÇMEN M S, SIMPSON R L. Theoretical and experimental pressure-strain comparison in a pressure-driven three-dimensional turbulent boundary layer[C]//Theoretical Fluid Mechanics Conference. Reston, VA, USA: AIAA, 1996: AIAA 1996-2141.

[8] ÖZCAN O, ÖLÇMEN M S. Measurements of turbulent flow behind a wing-body junction[J]. AIAA Journal, 1988, 26(4): 494–496.

[9] ÖLÇMEN S M, SIMPSON R L. Influence of wing shapes on surface pressure fluctuations at wing-body junctions[J]. AIAA Journal, 1994, 32(1): 6–15.

[10] ANDERSON S D, EATON J K. Experimental study of a pressure-driven, three-dimensional, turbulent boundary layer[J]. AIAA Journal, 1987, 25(8): 1086–1092.

[11] APSLEY D D, LESCHZINER M A. Investigation of advanced turbulence models for the flow in a generic wing-body junction[J]. Flow, Turbulence and Combustion, 2001, 67(1): 25–55.

[12] PARNEIX S, DURBIN P A, BEHNIA M. Computation of 3-D turbulent boundary layers using the V2F model[J]. Flow, Turbulence and Combustion, 1998, 60(1): 19–46.

[13] KEGERISE M A, NEUHART D H, HANNON J A, et al. An experimental investigation of a wing-fuselage junction model in the NASA Langley 14-by 22-

foot subsonic wind tunnel[C]//AIAA Scitech 2019 Forum. Reston, VA, USA: AIAA, 2019: AIAA 2019-0077.

[14] RUMSEY C L, AHMAD N N, CARLSON J R, et al. CFD comparisons with updated NASA juncture flow data[C]//AIAA Scitech 2021 Forum. Reston, VA, USA: AIAA, 2021: AIAA 2021-1427.

[15] EISFELD B, RUMSEY C L, TOGITI V, et al. Reynolds-stress model computations of NASA juncture flow experiment[J]. AIAA Journal, 2022, 60(3): 1643–1662.

[16] RUMSEY C L. The NASA juncture flow test as a model for effective CFD/experimental collaboration[C]//2018 Applied Aerodynamics Conference. Reston, VA, USA: AIAA, 2018: AIAA 2018-3319.

[17] RUMSEY C L, NEUHART D H, KEGERISE M A. The NASA juncture flow experiment: goals, progress, and preliminary testing (invited)[C]//54th AIAA Aerospace Sciences Meeting. Reston, VA, USA: AIAA, 2016: AIAA 2016-1557.

[18] ZHANG L, LI J, MOU Y F, et al. Numerical investigation of flow around a multi-element airfoil with hybrid RANS-LES approaches based on SST model[J]. Journal of Mechanics, 2018, 34(2): 123–134.

[19] MENTER F R. Two-equation eddy-viscosity turbulence models for engineering applications[J]. AIAA Journal, 1994, 32(8): 1598–1605.

[20] SPALART P R. Strategies for turbulence modelling and simulations[J]. International Journal of Heat and Fluid Flow, 2000, 21(3): 252–263.

[21] DEVENPORT W J, SIMPSON R L. Time-dependent and time-averaged turbulence structure near the nose of a wing-body junction[J]. Journal of Fluid Mechanics, 1990, 210: 23–55.

[22] LEE J P, CHEN J H, HSIN C Y. Study of junction flow structures with different turbulence models[J]. Journal of Marine Science and Technology, 2017, 25(2): 6.

[23] PAIK J, ESCAURIAZA C, SOTIROPOULOS F. On the bimodal dynamics of the turbulent horseshoe vortex system in a wing-body junction[J]. Physics of Fluids, 2007, 19(4): 045107.

[24] SRIKUMAR R, KUMAR S. LES of a novel wing/body junction: anti-fairing[D]. Delft: Delft University of Technology, 2019.

[25] RYU S, EMORY M, IACCARINO G, et al. Large-eddy simulation of a wing-body junction flow[J]. AIAA Journal, 2016, 54(3): 793–804.

[26] ALBERTS J. Wall-resolved large eddy simulation of a wing-body junction: high-fidelity data generation for data-driven turbulence modelling[D]. Delft: Delft University of Technology, 2021.

[27] DANDOIS J. Improvement of corner flow prediction using the quadratic constitutive relation[J]. AIAA Journal, 2014, 52(12): 2795–2806.

Figure 3

Figure 11: Figure 3

Figure 5

Figure 12: Figure 5

[28] GESSNER F B. The origin of secondary flow in turbulent flow along a corner[J]. Journal of Fluid Mechanics, 1973, 58(1): 1-25.

[29] BARBER T J. An investigation of strut-wall intersection losses[J]. Journal of Aircraft, 1978, 15(10): 676-681.

(Editor: Li Zhuyao)

Figures

Figure 6

Figure 13: Figure 6

Figure 7

Figure 14: Figure 7

Figure 12

Figure 15: Figure 12

Figure 15

Figure 16: Figure 15

Figure 16

Figure 17: Figure 16

Figure 17

Figure 18: Figure 17