High-Fidelity CFD Workshop 2021:
RANS SA-[neg]-QCR2000 Juncture Flow Model Case

Stephen Wood stephen.l.wood@nasa.gov, Chris Rumsey, Nash'at Ahmad

July 27, 2020

1 Introduction

This "challenge" test case is focused on the flow phenomenon within the junction region (between a wing and fuselage) on the upper surface. The recent experimental investigation of a Wing-Fuselage Junction Model in the NASA Langley 14- by 22-Foot Subsonic Tunnel NASA Langley Research Center (2019) is the basis for this case. Separated flow occurs near the trailing edge of the wing near the wing-root junction. Although other data were also taken, the main focus of the experiment was to document velocities and Reynolds stresses in the flow field near and upstream of the region of interest. An on-board laser doppler velocimetry (LDV) system was used, which measured through windows on the port side of the fuselage. The ultimate goal of the experiment was to provide information for validation and improvement of turbulence models, for predicting corner flows. The over-aching goal of this test case is the establishment of computational benchmarks for the prediction of corner flows. Comparison with experiment is beyond the focus of this workshop.

Participants are expected to have verified their solver and turbulence model implementations prior to conducting the simulations for this case. Participants are required to use the provided grids. Participants may additionally use their own grids if they make those available to all participants.

2 Governing Equations and Models

The compressible Reynolds Averaged Navier-Stokes equations should be used, with air as the working medium. The freestream Mach number is 0.189, a Reynolds number of 4307.4515 per mm (2.4 million based on crank chord of 557.174 mm), an angle of attack of 5 degrees, the heat capacity ratio is $\gamma = Cp/Cv = 1.4$, and the temperature is 519.92 Rankine. The dynamic viscosity is modeled using Sutherland's law. The Prandtl number is fixed to $Pr = 0.72$, and a turbulent Prandtl number of $Pr_t = 0.9$. Participants should use a freestream value $\nu_t/\nu = 3$ for the SA turbulence model.

For force calculations, the reference area is 965543.2302 mm^2 (semi-span model).

Participants must use the SA-QCR2000 turbulence model of Spalart (2000) or the the equivalent "negative-SA"[1] by Allmaras *et al.* (2012) with the QCR2000 terms, e.g. SA-neg-QCR2000. Because this will have a significant effect on the "truth" drag coefficient value, the turbulence model must be carefully documented. We strongly recommend the use of test cases from the NASA Turbulence Modeling Resource (`http://turbmodels.larc.nasa.gov/`) web site to verify correct

---

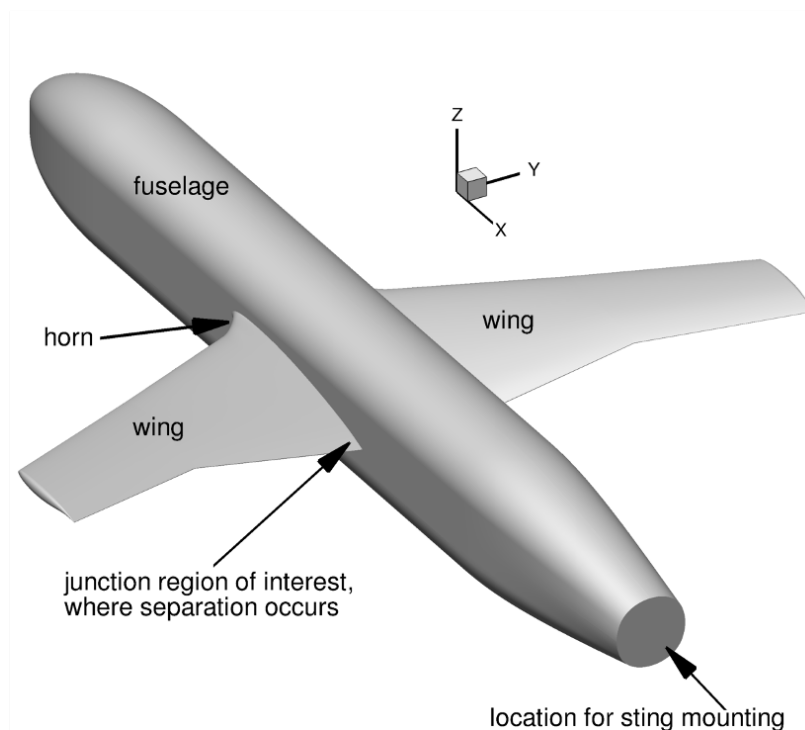[1]`http://www.iccfd.org/iccfd7/assets/pdf/papers/ICCFD7-1902_paper.pdf`

Figure 1: Juncture flow model geometry.

implementation of the turbulence model, where details of the mathematical formulation for the SA-[neg]-QCR2000[2] [3] model are also available

3 Boundary Conditions

The far field boundary can be imposed with a Riemann invariant or characteristic boundary condition. The wing and fuselage surfaces are imposed as a no-slip adiabatic wall. The full geometry of the model is shown in Figure 1. A half model will be used in CFD simulations for this test case to reduce computational cost. The x-z symmetry plane located on the centerline of the fuselage is imposed as symmetric boundary condition.

4 Common Inconsistencies

The following is a list of common inconsistencies that can influence solutions in ways that make comparisons with other CFD results unclear.

1. Using a different Prandtl number than 0.72.

2. Using constant viscosity rather than Sutherland's law.

3. Using isothermal wall rather than adiabatic wall.

4. Using modified SA turbulence models will have a significant effect.

---

[2]https://turbmodels.larc.nasa.gov/spalart.html#sa
[3]https://turbmodels.larc.nasa.gov/spalart.html#qcr2000

5. Using a freestream value other than $\nu_t/\nu = 3$.

5   Mandatory Campaign

5.1   Solutions

The over-aching goal of this test case is the establishment of computational benchmarks for the prediction of corner flows. The provided meshes must be used for all calculations. To represent their solution on each grid, participants should provide:

1. Lift and drag coefficients

2. Wing pressures at three locations, y=-254mm, y=-290.83mm, and y=-1663.7mm

3. Separation bubble apex x-location

4. Flow profile measurements of velocity, normal stresses, and shear stresses at four locations

Details of each of these four groups of deliverables are discussed in the following sections.

5.1.1   Lift and drag coefficients

The reference area used in lift, drag, and pressure moment calculations is 965543.2302 mm^2. Lift force is the component of net force acting on the wing and fuselage in the y-direction. The lift coefficient, C_L, is the dimensionless ratio of the lift force to the product of the free-stream dynamic pressure and the reference area.

$$C_L = \frac{2L}{\rho_\infty u_\infty^2 A_{ref}}$$

Similarly, the drag coefficient, C_D, is the dimensionless ratio of the component of the net force acting on the wing and fuselage in the x-direction to the product of the free-stream dynamic pressure and the reference area.

$$C_D = \frac{2D}{\rho_\infty u_\infty^2 A_{ref}}$$

Preliminary CFD simulations on a refined grid indicate $C_L \approx 0.85$ and $C_D \approx 0.07$.

5.1.2   Wing pressures

The pressure coefficient, C_P, is the dimensionless ratio that describes the relative pressure throughout a flow field.

$$C_P = \frac{2\,(p - p_\infty)}{\rho_\infty u_\infty^2}$$

The wing pressures will be reported using pressure coefficients on three x-z planes located at y=-254mm, y=-290.83mm, and y=-1663.7mm. These three planes coincide with experimental measurement locations included in Figure 2.

5.1.3   Separation bubble apex location

On this configuration, a separation occurs near the wing upper surface trailing edge, at the wing-root juncture. Participants should provide the x-coordinate of the apex of this separation. The apex is defined as the point on the wing-fuselage intersection where a surface streamline deviates in the spanwise direction along the foremost edge of the separated region. At this location, $c_{f,x}$ changes sign from positive to negative. A typical result might be $x \approx 2810$mm.

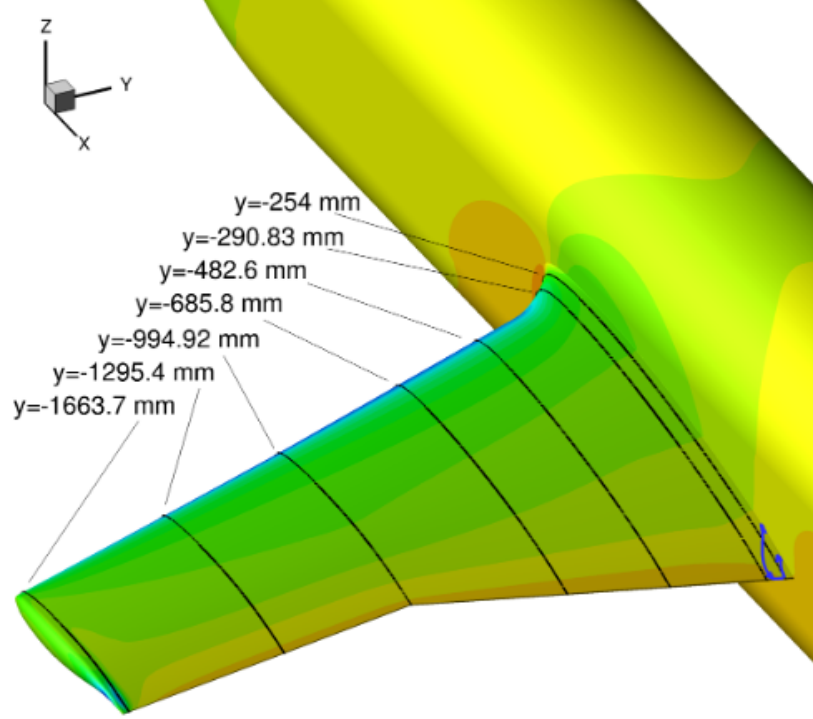Figure 2: Locations of wing pressure measurements. Participants only need to report C_p at three locations (y=-254mm, y=-290.83mm, and y=-1663.7mm).

5.1.4 Flow profile measurements

Participants should provide flow profile measurements of velocity components, turbulent normal stresses, and turbulent shear stresses along four lines detailed in Table 1 and depicted in Figures 3-6. The three components of velocity should be nondimensionalized by U_∞: u_i/U_∞. The six components of turbulent Reynolds stresses should be nondimensionalized by U_∞^2: $\overline{u_i' u_j'}/(U_\infty^2)$.

Table 1: Flow profile locations

Profile	Point	x	y	z
1	Begin	1168.4	surface (≈ -236.1)	0.0
	End	1168.4	-300.0	0.0
2	Begin	2747.6	-237.1	surface (≈ 16.75)
	End	2747.6	-237.1	130.0
3	Begin	2852.6	-237.1	surface (≈ -8.46)
	End	2852.6	-237.1	130.0
4	Begin	2852.6	-266.1	surface (≈ -3.68)
	End	2852.6	-266.1	130.0
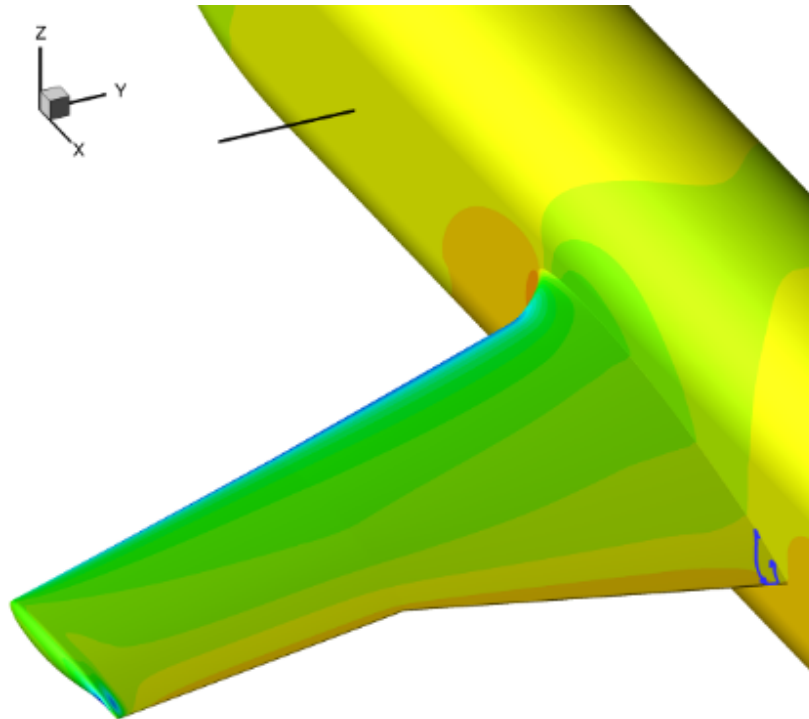
Figure 3:   Flow profile 1 from {1168.4, surface, 0.0} to {1168.4, -300.0, 0.0}.

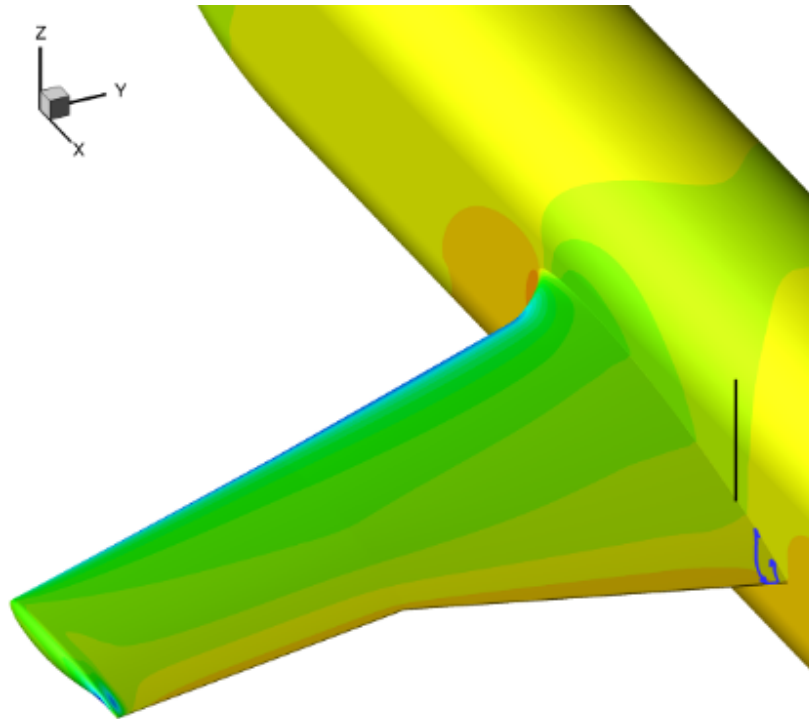Figure 4: Flow profile 2 from {2747.6, -237.1, surface} to {2747.6, -237.1, 130.0}.
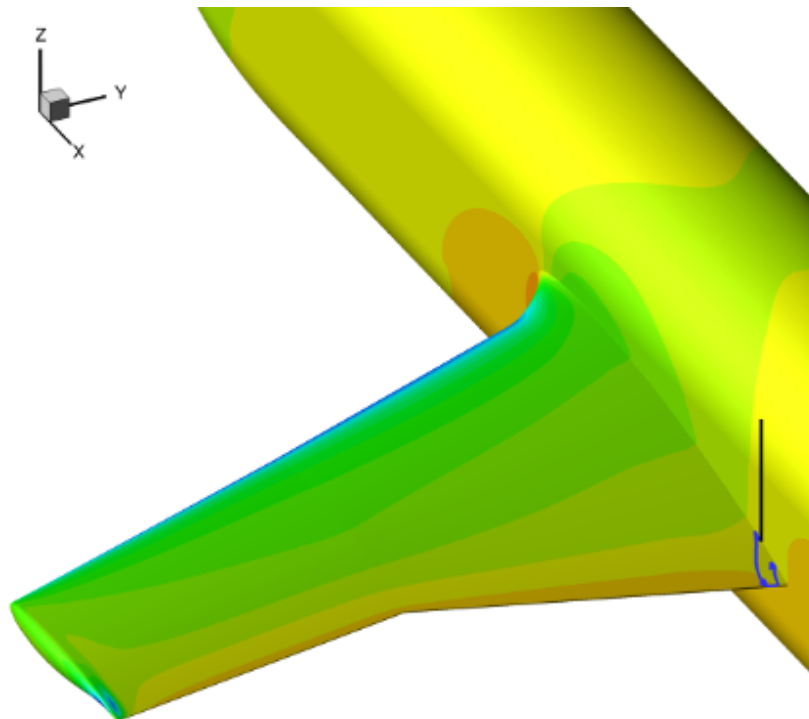

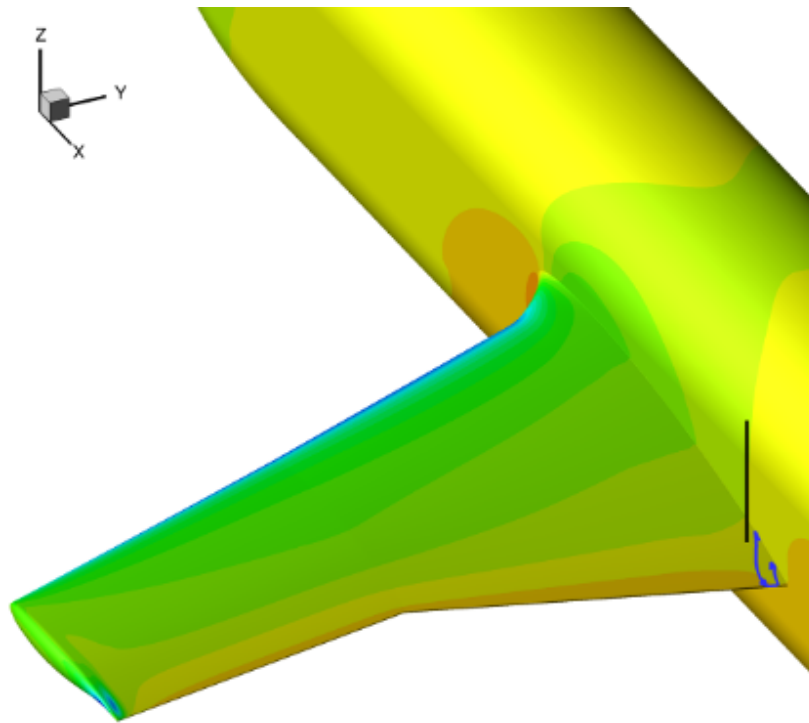Figure 5: Flow profile 3 from {2852.6, -237.1, surface} to {2852.6, -237.1, 130.0}.

Figure 6: Flow profile 4 from {2852.6, -266.1, surface} to {2852.6, -266.1, 130.0}.

5.2 Methods

5.2.1 Compute resources

Participants should provide a comprehensive a description of the computational resources used to produce the results submitted to the workshop. For example listing the CPU, GPU, and interconnect models and number of sockets as shown in Table 2.

Table 2: Compute resources

Grid	CPU		GPU		Interconnect	Wall time [hr]
	model	quantity	model	quantity		
fine	skylake	10			4X EDR Infiniband	10.4

5.2.2 Partitioning strategy

Participants should provide a comprehensive description of the partitioning applied to each grid and how compute resources are allocated to grid partitions.

5.2.3 Residual histories

Participants should provide the nonlinear residual histories for each grid. Linear residual histories are optional, but are encouraged.

References

Allmaras, Steven R., Johnson, Forrester T. & Spalart, Philippe R. 2012 Modifications and clarifications for the implementation of the spalart-allmaras turbulence model. Seventh International Conference on Computational Fluid Dynamics. Big Island, Hawaii.

NASA Langley Research Center 2019 Nasa juncture flow. `https://turbmodels.larc.nasa.gov/Other_exp_Data/junctureflow_exp.html`.

Spalart, P. R. 2000 Strategies for turbulence modelling and simulation. *International Journal of Heat and Fluid Flow* **21**, 252–263.