RANS Joukowski Airfoil

Marshall Galbraith & Carl Ollivier-Gooch
galbramc@mit.edu, cfog@mech.ubc.ca

Overview

This test case is designed as a verification case of the turbulence model of the RANS equations. Participants are required to use the provided grids, as they have been demonstrated to be able to provide the optimal convergence rate in drag. A Reynolds number of 1,000,000 is employed. For an adjoint consistent discretization, the optimal convergence rate is 2P. Otherwise, the convergence rate can be expected to be P. The Joukowski airfoil is used for this test as the cusped trailing edge removes the inviscid singularity at the trailing edge. However, there is still a singularity in skin friction. The provided grids are design to cluster nodes at both the trailing edge singularity and the stagnation point in order to capture the expected order of accuracy. Hence, all participants must use the provided grids. Please do not hesitate contact Marshall Galbraith as soon as possible if you are having difficulties using the provided grids.

Simulations

Governing Equations and models

The compressible Navier-Stokes equations should be used, with air as working medium. The freestream Mach number is 0.5, a Reynolds number of 1,000,000 based on chord, an angle of attack of 0 degrees, and the heat capacity ratio is $\gamma=Cp/Cv=1.4$. The dynamic viscosity is constant. The Prandtl number is fixed to $Pr=0.72$, and a turbulent Prandtl number of $Pr_t = 0.9$.

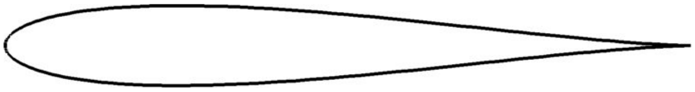
Participants may use their own choice of turbulence models. Because this will have a significant effect on the "truth" drag value, the turbulence model must be carefully documented. We strongly recommend the use of test cases from the NASA Turbulence Modeling Resource (http://turbmodels.larc.nasa.gov/) web site to verify correct implementation of the turbulence model.

Those using the SA turbulence model are strongly encouraged to use the "negative-SA" model outlined in this paper: http://www.iccfd.org/iccfd7/assets/pdf/papers/ICCFD7-1902_paper.pdf. Participants should use a freestream value $v_t/v = 3$.

Boundary Conditions

The far field boundary can be imposed with a Riemann invariant or characteristic boundary condition. Note that the far field boundary is not far enough away to be non-influential, in particular for high-order calculations. Hence the boundary condition used must be documented, and will likely lead to slight difference in the “truth” drag on the finest mesh. The airfoil surface is imposed as a no-slip adiabatic wall.

Geometry and grids



The geometry of the Joukowski airfoil is shown in the adjacent figure. Structured, unstructured quadrilateral, and unstructured triangular grids are provided. The provided grids must be used by all participants, through custom and adapted grids are also welcomed in addition to the provided grids. A python script (RANS.py) is provided that generates the grids. This allows participants to generate custom file formats if desired. Please contact Marshall Galbraith if you have any problems generating the grids.

Mandatory campaign

The main objective is to demonstrate grid convergence of drag on a sequence of successively refined meshes. The provided meshes must be used for all calculations. Participants should provide a non-dimensional drag error for each of the grids. Due to variations in boundary condition implementations, participants should compute a reference drag on the finest grid with the highest order of accuracy available in their software. The drag error for each grid is computed relative to this reference drag. Participants should also verify that machine zero lift is computed on all grids.

1. Start the simulation from a uniform free stream with M = 0.5 everywhere, and monitor the L_2 norm of the density residual. Compute the work units required to achieve a steady state where the density residual has dropped at least 10 orders of magnitude.
2. Perform this exercise for at least three different meshes and with different orders of accuracy to assess the performance of high-order schemes of various accuracy.
2. Plot the drag error vs. work units to evaluate efficiency, and drag error vs. length scale to assess the numerical order of accuracy.
3. Submit two sets of data to the workshop contact for this case
    a. Drag (not error) vs work units for different *h* and *p*
    b. Drag (not error) vs $h = 1/\sqrt{nDOFs}$ for different *h* and *p*

Common Inconsistencies

The following is a list of common inconsistencies that can lead to computing a different "truth" drag value.

1. Using a different Prandtl number than 0.72.
2. Using Sutherland's law rather than constant viscosity.
3. Using isothermal wall rather than adiabatic wall.
4. Using different turbulence models will have a significant effect.
5. Using something different from a freestream value of $v_t/v = 3$.