

# Stanford University Unstructured (SU<sup>2</sup>): Open-source Analysis and Design Technology for Turbulent Flows

Francisco Palacios\*, Thomas D. Economou†

Aniket C. Aranake‡, Sean R. Copeland‡, Amrita K. Lonkar‡, Trent W. Lukaczyk‡,  
 David E. Manosalvas‡, Kedar R. Naik‡, A. Santiago Padrón‡, Brendan Tracey‡, Anil Variyar‡  
 and Juan J. Alonso§

*Stanford University, Stanford, CA, 94305, U.S.A.*

This paper presents a comprehensive set of test cases for the verification and validation (V & V) of the Stanford University Unstructured (SU<sup>2</sup>) software suite within the context of compressible, turbulent flows described by the Reynolds-averaged Navier-Stokes (RANS) equations. SU<sup>2</sup> is an open-source (Lesser General Public License, version 2.1), integrated analysis and design tool for solving multi-disciplinary problems governed by partial differential equations (PDEs) on general, unstructured meshes. As such, SU<sup>2</sup> is able to handle arbitrarily complex geometries, mesh adaptation, and a variety of physical problems. At its core, the software suite is a collection of C++ modules embedded within a Python framework that are built specifically for both PDE analysis and PDE-constrained optimization, including surface gradient computations using the continuous adjoint technique.

V & V studies of two- and three-dimensional problems are presented for turbulent flows across a wide range of Mach numbers (from subsonic flat plate studies to a complex, transonic aircraft configuration). The presentation of this comprehensive V & V of SU<sup>2</sup> is intended to be the main contribution of this paper: the results generated with SU<sup>2</sup> in a variety of standard test cases compare well with experimental data and established flow solvers that have undergone similar V & V efforts. For completeness, the adjoint-based shape design capability within SU<sup>2</sup> is also illustrated.

## I. Introduction

The Stanford University Unstructured (SU<sup>2</sup>) software suite<sup>1</sup> has been recently developed for the specific task of solving PDE analyses and PDE-constrained optimization problems on general, unstructured meshes. While the framework is general and meant to be extensible to arbitrary sets of governing equations for solving multi-physics problems, the core of the suite is a Reynolds-averaged Navier-Stokes (RANS) solver capable of simulating the compressible, turbulent flows that are characteristic of typical problems in aerospace engineering. Furthermore, SU<sup>2</sup> was constructed with aerodynamic shape optimization problems in mind, and, therefore, adjoint-based sensitivity analysis is a key feature that is built directly into the RANS solver.

In order to increase our confidence in the results of any aerodynamic shape optimization problem, it is critical to assess the accuracy of the baseline RANS solver to guarantee that the improvements obtained can actually be realized. Moreover, since the code has been developed in an open-source environment, it is important to establish the accuracy of the results that it produces through a detailed validation study. The intent is to provide a clear expectation regarding what SU<sup>2</sup> can and cannot do. In this paper, we tackle this task by applying SU<sup>2</sup> across the flow regimes of interest in the aerospace sciences. In particular, the following 12 test cases have been chosen as a representative set of the broader range of Computational Fluid Dynamics (CFD) applications for which SU<sup>2</sup> could be used:

\*Engineering Research Associate, Department of Aeronautics & Astronautics, AIAA Senior Member.

†Ph.D. Candidate, Department of Aeronautics & Astronautics, AIAA Student Member.

‡Ph.D. Candidates (authors in alphabetical order), Department of Aeronautics & Astronautics, AIAA Student Members.

§Associate Professor, Department of Aeronautics & Astronautics, AIAA Associate Fellow.

- Unit/Test problems: a zero pressure gradient flat plate, a bump in a channel, and unsteady flow around a square cylinder.
- Subsonic airfoil geometries: the NACA 0012 with attached flow, the NACA 4412 with a recirculation bubble, and the McDonnell-Douglas 30P30N three-element high-lift configuration.
- Subsonic wing and rotorcraft configurations: a delta wing at a high angle of attack and the Caradonna and Tung rotor.
- Transonic airfoil geometries: the NACA 0012 with attached flow after a shock and the RAE 2822 with flow separation.
- Transonic wing and full aircraft configurations: the ONERA M6 wing and the DLR F6 aircraft model.

For each of the V & V cases listed above, we will present the results computed by SU<sup>2</sup> and compare them against experimental data and/or results from other simulation codes, when available. The flow / boundary conditions, the details of the computational mesh, the numerical discretization methods of choice, and a discussion of results for the major quantities of interest will be clearly detailed for each case. This comprehensive V & V process is intended to be the main contribution of this paper: it will demonstrate the accuracy of the results generated by the open-source SU<sup>2</sup> RANS solver for a wide range of problems. These demonstrations are supposed to serve as a future reference for the technology built into the SU<sup>2</sup> suite for the solution of turbulent flows.

However, a verified and validated RANS solver is just one of the tools needed for performing aerodynamic shape design. The shape design process in SU<sup>2</sup> consists of separate C++ modules whose sequential execution is automated within a Python framework. Each C++ module addresses a compute-intensive task in the process, and, in order to maintain high computational efficiency, encourage code reuse, and ease the integration of new features, the modules share a common set of classes and data structures within an object-oriented code architecture.

More specifically, in addition to solving the RANS equations for a candidate design, the adjoint RANS equations must also be solved for the purpose of sensitivity analysis for various functions of interest (lift, drag, moments, etc.). The entire design problem also requires an infrastructure for shape parameterization, grid deformation, and numerical optimization. SU<sup>2</sup> contains the following modules for completing these tasks:

- SU2\_CFD: The RANS CFD solver, which includes a solver for the adjoint RANS equations.
- SU2\_GPC: The Gradient Projection Code that allows for the calculation of sensitivities for use in optimization given a particular shape parametrization and the surface sensitivities from an adjoint solution.
- SU2\_MDC: The Mesh Deformation Code that can be used to perturb an existing volume mesh to conform to new surface geometries dictated by shape optimization.

Although the main focus of this paper is the V & V of the RANS solver, the adjoint-based shape design capability is demonstrated for the DLR F6 configuration. The true power of the open-source SU<sup>2</sup> suite lies in combining the efficient solution of the RANS equations (with industry-standard numerical methods in the presence of complex geometries) and the surrounding infrastructure for automatic shape design.

SU<sup>2</sup> is under active development in the Aerospace Design Lab (ADL) of the Department of Aeronautics and Astronautics at Stanford University. It has been released under an open-source license and is freely available to the community, so that developers around the world can continue the V & V process, contribute to the source code, and further improve the accuracy and capabilities of the suite. As described above, the modern structure of SU<sup>2</sup> makes it an ideal vehicle for multi-physics simulations and aerodynamic shape optimization. To accomplish this, the SU<sup>2</sup> development team has included industry-standard solver technology for turbulent flows while also developing numerical solution algorithms that result in robust, high rates of convergence. Lastly, from the point of view of design optimization, SU<sup>2</sup> includes continuous adjoint solver implementations for efficiently computing shape design gradients that we hope can be further improved via contributions from the community.

The paper is organized as follows. Section II describes the set of RANS equations (including the Spalart-Allmaras and SST turbulence models) and the corresponding adjoint RANS equations used in our work. It

includes a brief description of the main numerical discretization techniques to be verified and validated (more details of the discretization alternatives in SU<sup>2</sup> can be found in our previous work<sup>1</sup>). Section III provides a case-by-case validation study for each of the 12 test cases described earlier. Our conclusions are summarized in section IV.

## II. Governing Equations & Discretization

### A. Reynolds-averaged Navier-Stokes Equations

We are concerned with time-accurate, viscous flow around aerodynamic bodies in arbitrary motion which is governed by the compressible, unsteady Navier-Stokes equations. Consider the equations in a domain,  $\Omega \subset \mathbb{R}^3$ , with a disconnected boundary that is divided into a far-field component,  $\Gamma_\infty$ , and an adiabatic wall boundary,  $S$ , as seen in Fig. 1. The surface  $S$  represents the outer mold line of an aerodynamic body, and it is considered continuously differentiable ( $C^1$ ). These conservation equations along with a generic source term,  $\mathcal{Q}$ , can be expressed in an arbitrary Lagrangian-Eulerian (ALE)<sup>2</sup> differential form as

$$\begin{cases} \mathcal{R}(U) = \frac{\partial U}{\partial t} + \nabla \cdot \vec{F}_{ale}^c - \nabla \cdot \vec{F}^v - \mathcal{Q} = 0 & \text{in } \Omega, \quad t > 0 \\ \vec{v} = \vec{u}_\Omega & \text{on } S, \\ \partial_n T = 0 & \text{on } S, \\ (W)_+ = W_\infty & \text{on } \Gamma_\infty, \end{cases} \quad (1)$$

where the conservative variables are given by  $U = \{\rho, \rho\vec{v}, \rho E\}^T$ , and the convective fluxes, viscous fluxes, and source term are

$$\vec{F}_{ale}^c = \begin{Bmatrix} \rho(\vec{v} - \vec{u}_\Omega) \\ \rho\vec{v} \otimes (\vec{v} - \vec{u}_\Omega) + \bar{I}p \\ \rho E(\vec{v} - \vec{u}_\Omega) + p\vec{v} \end{Bmatrix}, \quad \vec{F}^v = \begin{Bmatrix} \cdot \\ \bar{\tau} \\ \bar{\tau} \cdot \vec{v} + \mu_{tot}^* c_p \nabla T \end{Bmatrix}, \quad \mathcal{Q} = \begin{Bmatrix} q_\rho \\ \vec{q}_{\rho\vec{v}} \\ q_{\rho E} \end{Bmatrix}, \quad (2)$$

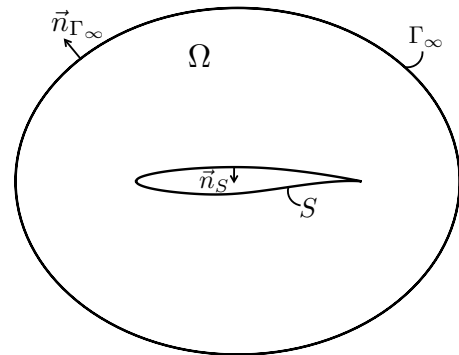
where  $\rho$  is the fluid density,  $\vec{v} = \{v_1, v_2, v_3\}^T \in \mathbb{R}^3$  is the flow speed in a Cartesian system of reference,  $\vec{u}_\Omega$  is the velocity of a moving domain (mesh velocity after discretization),  $E$  is the total energy per unit mass,  $p$  is the static pressure,  $c_p$  is the specific heat at constant pressure,  $T$  is the temperature, and the viscous stress tensor can be written in vector notation as

$$\bar{\tau} = \mu_{tot} \left( \nabla \vec{v} + \nabla \vec{v}^T - \frac{2}{3} \bar{I}(\nabla \cdot \vec{v}) \right). \quad (3)$$

The second line of Eqn. 1 represents the no-slip condition at a solid wall, the third line represents an adiabatic condition at the wall, and the final line represents a characteristic-based boundary condition at the far-field<sup>3</sup> with  $W$  representing the characteristic variables.

Including the boundary conditions given in Eqn. 1, the compressible RANS solver in SU<sup>2</sup> currently supports the following boundary condition types: Euler (flow tangency) and symmetry wall, no-slip wall (adiabatic and isothermal), far-field and near-field boundaries, characteristic-based inlet boundaries (stagnation, mass flow, or supersonic conditions prescribed), characteristic-based outlet boundaries (back pressure prescribed), periodic boundaries, nacelle inflow boundaries (fan face Mach number prescribed), and nacelle exhaust boundaries (total nozzle temp and total nozzle pressure prescribed).

The boundary conditions listed above make SU<sup>2</sup> suitable for computing both external and internal flows. Note that the boundary conditions also take into account any domain motion, but for problems on fixed grids ( $\vec{u}_\Omega = 0$ ), Eqn. 1 reduces to a purely Eulerian formulation. While all of the validation cases in this article are for turbulent flows on static



**Figure 1. Notional schematic of the flow domain,  $\Omega$ , the boundaries,  $\Gamma_\infty$  and  $S$ , as well as the definition of the surface normals.**

meshes, SU<sup>2</sup> is capable of solving unsteady flows on both rigidly transforming and dynamically deforming meshes. For unsteady problems, the temporal conditions will be problem dependent, and in this work, we will be interested in time-periodic flows where the initial and terminal conditions do not affect the time-averaged behavior over the time interval of interest,  $\mathbb{T} = t_f - t_o$ . For steady problems, we will use the free-stream fluid state as the initial condition for the mean flow, and this is a typical practice in external aerodynamics.

Assuming a perfect gas with a ratio of specific heats,  $\gamma$ , and gas constant,  $R$ , the pressure is determined from  $p = (\gamma - 1)\rho[E - 0.5(\vec{v} \cdot \vec{v})]$ , the temperature is given by  $T = p/(\rho R)$ , and  $c_p = \gamma R/(\gamma - 1)$ . In accord with the standard approach to turbulence modeling based upon the Boussinesq hypothesis,<sup>4</sup> which states that the effect of turbulence can be represented as an increased viscosity, the total viscosity is divided into a laminar,  $\mu_{dyn}$ , and a turbulent,  $\mu_{tur}$ , component. In order to close the system of equations, the dynamic viscosity,  $\mu_{dyn}$ , is assumed to satisfy Sutherland's law<sup>5</sup> (function of temperature alone), the turbulent viscosity  $\mu_{tur}$  is computed via a turbulence model, and

$$\mu_{tot} = \mu_{dyn} + \mu_{tur}, \quad \mu_{tot}^* = \frac{\mu_{dyn}}{Pr_d} + \frac{\mu_{tur}}{Pr_t}, \quad (4)$$

where  $Pr_d$  and  $Pr_t$  are the dynamic and turbulent Prandtl numbers, respectively.

The turbulent viscosity,  $\mu_{tur}$ , is obtained from a suitable turbulence model involving the flow state and a set of new variables. The Shear Stress Transport (SST) model of Menter and the Spalart-Allmaras (S-A) model are the two most common and widely used turbulence models for the analysis and design of engineering applications affected by turbulent flows. These two models will be used throughout the validation portion to the article, and brief descriptions of the two models are given below.

**SPALART-ALLMARAS (S-A) MODEL:** In the case of the one-equation Spalart-Allmaras<sup>6</sup> turbulence model, the turbulent viscosity is computed as

$$\mu_{tur} = \rho \hat{\nu} f_{v1}, \quad f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3}, \quad \chi = \frac{\hat{\nu}}{\nu}, \quad \nu = \frac{\mu_{dyn}}{\rho}. \quad (5)$$

The new variable  $\hat{\nu}$  is obtained by solving a transport equation where the convective, viscous, and source terms are given as follows:

$$\vec{F}^c = \vec{v}\hat{\nu}, \quad \vec{F}^v = -\frac{\nu + \hat{\nu}}{\sigma} \nabla \hat{\nu}, \quad Q = c_{b1} \hat{S} \hat{\nu} - c_{w1} f_w \left( \frac{\hat{\nu}}{d_S} \right)^2 + \frac{c_{b2}}{\sigma} |\nabla \hat{\nu}|^2, \quad (6)$$

where the production term  $\hat{S}$  is defined as  $\hat{S} = |\vec{\omega}| + \frac{\hat{\nu}}{\kappa^2 d_S^2} f_{v2}$ ,  $\vec{\omega} = \nabla \times \vec{v}$  is the fluid vorticity,  $d_S$  is the distance to the nearest wall, and  $f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}$ . The function  $f_w$  is computed as  $f_w = g \left[ \frac{1 + c_{w3}^6}{g^6 + c_{w3}^6} \right]^{1/6}$ , where  $g = r + c_{w2}(r^6 - r)$  and  $r = \frac{\hat{\nu}}{\hat{S} \kappa^2 d_S^2}$ . Finally, the set of closure constants for the model is given by

$$\sigma = 2/3, \quad c_{b1} = 0.1355, \quad c_{b2} = 0.622, \quad \kappa = 0.41, \quad c_{w1} = \frac{c_{b1}}{\kappa^2} + \frac{1 + c_{b2}}{\sigma}, \quad c_{w2} = 0.3, \quad c_{w3} = 2, \quad c_{v1} = 7.1. \quad (7)$$

The physical meaning of the far-field boundary condition for the turbulent viscosity is the imposition of some fraction of the laminar viscosity at the far-field. On viscous walls,  $\hat{\nu}$  is set to zero, corresponding to the absence of turbulent eddies very near to the wall.

**MENTER SHEAR STRESS TRANSPORT (SST) MODEL:** The Menter SST turbulence model<sup>7</sup> is a two equation model for the turbulent kinetic energy,  $k$ , and specific dissipation,  $\omega$ , that consists of the blending of the traditional  $k - \omega$  and  $k - \epsilon$  models. The definition of the eddy viscosity, which includes the shear stress limiter, can be expressed as

$$\mu_{tur} = \frac{\rho a_1 k}{\max(a_1 \omega, S F_2)}, \quad (8)$$

where  $S = \sqrt{2 S_{ij} S_{ij}}$  and  $F_2$  is the second blending function. The convective, viscous, and source terms for the turbulent kinetic energy are

$$\vec{F}^c = \rho k \vec{v}, \quad \vec{F}^v = -(\mu_{dyn} + \sigma_k \mu_{tur}) \nabla k, \quad Q = P - \beta^* \rho \omega k, \quad (9)$$

where  $P$  is the production of turbulent kinetic energy. The convective, viscous, and source terms for the specific dissipation are given by

$$\vec{F}^c = \rho\omega\vec{v}, \quad \vec{F}^v = -(\mu_{dyn} + \sigma_\omega\mu_{tur})\nabla\omega, \quad Q = \frac{\gamma}{\nu_t}P - \beta^*\rho\omega^2 + 2*(1 - F_1)\frac{\rho\sigma_\omega^2}{\omega}\nabla k\nabla\omega, \quad (10)$$

where  $F_1$  is the first blending function. The values for the constants and the forms for the blending functions and auxiliary relations are detailed in Rumsey<sup>8</sup> and Menter.<sup>7</sup>

## B. Continuous Adjoint Navier-Stokes Equations

A typical aerodynamic shape optimization problem seeks the minimization of a cost function,  $J(S)$  (lift, drag, moment, etc.), as chosen by the designer, with respect to changes in the shape of the boundary  $S$ . For the present description, we will focus on integrated forces and moments on the solid surface which depends on a scalar,  $j$ , evaluated at each point on  $S$ . Other objectives are possible, such as functions based on surface temperature or surface heat flux, for instance.

We note that any changes to the shape of  $S$  will result in perturbations in the fluid state,  $U$ , in the domain, and that these variations in the state are constrained to satisfy the RANS equations, i.e.,  $\mathcal{R}(U) = 0$  must be satisfied for any candidate shape of  $S$ . Therefore, the optimal shape design problem can be formulated as a PDE-constrained optimization problem:

$$\begin{aligned} \min_S J(S) &= \int_S j(\vec{f}, \vec{n}) ds \\ \text{subject to: } \mathcal{R}(U) &= 0 \end{aligned} \quad (11)$$

where  $\vec{f} = (f_1, f_2, f_3)$  is the time-dependent force on the surface (from fluid pressure and viscous stresses) and  $\vec{n}$  is the outward-pointing unit vector normal to the surface  $S$ . We will parameterize the shape by an infinitesimal deformation of size  $\delta S$  along the normal direction  $\vec{n}$  to the surface  $S$ . The new surface obtained after the deformation is then given by  $S' = \{\vec{x} + \delta S \vec{n}, \vec{x} \in S\}$ .

Using the continuous adjoint approach, the computation of the objective function gradient with respect to perturbations of the geometry will require the solution of the adjoint RANS equations given by

$$\begin{cases} -\frac{\partial \Psi^\top}{\partial t} - \nabla \Psi^\top \cdot (\vec{A}^c - \mu_{tot}^k \vec{A}^{vk}) - \nabla \cdot (\nabla \Psi^\top \cdot \mu_{tot}^k \vec{D}^{vk}) - \Psi^\top \frac{\partial Q}{\partial U} = 0 & \text{in } \Omega, \quad t > 0 \\ \vec{\varphi} = \vec{d} & \text{on } S, \\ \partial_n(\psi_{\rho E}) = 0 & \text{on } S, \end{cases} \quad (12)$$

where  $\Psi$  are the adjoint variables and we have introduced the following Jacobian matrices,<sup>9,10</sup>

$$\left. \begin{aligned} \vec{A}^c &= (A_x^c, A_y^c, A_z^c), & A_i^c &= \frac{\partial \vec{F}_i^c}{\partial U} \Big|_{U(x,y,z)} \\ \vec{A}^{vk} &= (A_x^{vk}, A_y^{vk}, A_z^{vk}), & A_i^{vk} &= \frac{\partial \vec{F}_i^{vk}}{\partial U} \Big|_{U(x,y,z)} \\ \vec{D}^{vk} &= \begin{pmatrix} D_{xx}^{vk} & D_{xy}^{vk} & D_{xz}^{vk} \\ D_{yx}^{vk} & D_{yy}^{vk} & D_{yz}^{vk} \\ D_{zx}^{vk} & D_{zy}^{vk} & D_{zz}^{vk} \end{pmatrix}, & D_{ij}^{vk} &= \frac{\partial \vec{F}_i^{vk}}{\partial (\partial_j U)} \Big|_{U(x,y,z)} \end{aligned} \right\} \quad i, j = 1 \dots 3, \quad k = 1, 2. \quad (13)$$

After satisfying the adjoint system, the final expression for the functional variation will become a surface integral that contains terms involving only the flow and adjoint variables multiplied by  $\delta S$ :

$$\delta J(S) = \int_S (\vec{n} \cdot \vec{\Sigma}^\varphi \cdot \partial_n \vec{v} - \mu_{tot}^2 C_p \nabla_S \psi_5 \cdot \nabla_S T) \delta S ds, \quad (14)$$

where  $\nabla_S$  represents the tangential gradient operator on  $S$ , and  $\vec{\Sigma}^\varphi = \mu_{tot}(\nabla \vec{\varphi} + \nabla \vec{\varphi}^\top - \frac{2}{3} \vec{I} \nabla \cdot \vec{\varphi})$ , which depends on the gradient of the adjoint variables. This computable formula is what we call the *surface sensitivity*, and it is the key result of the continuous adjoint derivation. The surface sensitivity provides a measure of the variation of the objective function with respect to infinitesimal variations of the surface shape in the direction of the local surface normal. This value is computed at every surface node of the numerical

grid with negligible computational cost. In this manner, the functional variation for an arbitrary number of shape perturbations will be computable at the fixed cost of solving the flow and adjoint PDE systems.

The ability to recover an analytic expression as a surface integral for the variation of the functional is commonly referred to as a surface formulation for computing gradients (with no dependence on volume mesh sensitivities). After early work in the area of continuous adjoints on unstructured meshes,<sup>11,12</sup> this type of surface formulation based on shape calculus was first demonstrated by Castro et al.<sup>13</sup> for inviscid and laminar flows and later extended to turbulent flows using the S-A turbulence model.<sup>9</sup> Extensions and advances of this formulation form much of the recent research activity within the SU<sup>2</sup> suite. In particular, the formulation has been extended to sonic boom minimization for supersonic aircraft,<sup>14</sup> aerodynamic design for unsteady problems on dynamic meshes,<sup>10,15,16</sup> mesh adaptation and design in nonequilibrium hypersonic flows,<sup>17</sup> and design for free-surface flows.<sup>18,19</sup>

## C. Numerical Implementation

A brief overview of the implementation details for the pertinent numerical methods is given below. Both the flow and adjoint problems are solved numerically on unstructured meshes with an edge-based data structure. Following the method of lines, the governing equations are discretized in space and time separately. This decoupling of space and time allows for the selection of different types of schemes for the spatial and temporal integration. Spatial integration is performed using the finite volume method (FVM), while integration in time is achieved through several available explicit and implicit methods. For time-accurate calculations, a dual time-stepping approach is used.

### 1. Spatial Integration via the Finite Volume Method

Partial Differential Equations (PDEs) in SU<sup>2</sup> are discretized using a finite volume method<sup>3,20–27</sup> with a standard edge-based structure on a dual grid with control volumes constructed using a median-dual, vertex-based scheme. Median-dual control volumes are formed by connecting the centroids, face, and edge-midpoints of all cells sharing the particular node. After integrating the governing equations over a control volume and applying the divergence theorem, the semi-discretized, integral form of a typical PDE (such as the RANS equations above) is given by,

$$\int_{\Omega_i} \frac{\partial U}{\partial t} d\Omega + \sum_{j \in \mathcal{N}(i)} (\tilde{F}_{c_{ij}} + \tilde{F}_{v_{ij}}) \Delta S_{ij} - Q |\Omega_i| = \int_{\Omega_i} \frac{\partial U}{\partial t} d\Omega + R_i(U) = 0, \quad (15)$$

where  $U$  is the vector of state variables, and  $R_i(U)$  is the numerical residual representing the integration of the spatial terms.  $\tilde{F}_{c_{ij}}$  and  $\tilde{F}_{v_{ij}}$  are the projected numerical approximations of the convective and viscous fluxes, respectively, and  $Q$  is a source term.  $\Delta S_{ij}$  is the area of the face associated with the edge  $ij$ ,  $\Omega_i$  is the volume of the control volume, and  $\mathcal{N}(i)$  is the set of neighboring nodes to node  $i$ .

The convective and viscous fluxes are evaluated at the midpoint of an edge. The numerical solver loops through all of the edges in the primal mesh in order to calculate these fluxes and then integrates them to evaluate the residual at every node in the numerical grid. The convective fluxes can be discretized using centered or upwind schemes in SU<sup>2</sup>. Several numerical schemes have been implemented (JST,<sup>28</sup> Roe,<sup>29</sup> AUSM,<sup>30</sup> HLLC,<sup>27</sup> Roe-Turkel,<sup>31</sup> to name a few), and the code architecture allows for the rapid implementation of new schemes. Limiters are available for use with higher-order reconstructions for the upwind convective schemes. In order to evaluate the viscous fluxes using a finite volume method, flow quantities and their first derivatives are required at the faces of the control volumes. The gradients of the flow variables are calculated using either a Green-Gauss or weighted least-squares method at all grid nodes and then averaged to obtain the gradients at the cell faces. Source terms are approximated using piecewise constant reconstruction within each of the finite volume cells.

### 2. Time Integration

Eqn. 15 must be valid over the entire time interval, so one has to make the choice of evaluating  $R_i(U)$  either at time  $t^n$  (explicit methods) or  $t^{n+1}$  (implicit methods). Focusing on the implicit integration (SU<sup>2</sup> also has an Euler Explicit and a Runge-Kutta explicit method), the following linear system should be solved to find

the solution update ( $\Delta U_i^n$ ),

$$\left( \frac{|\Omega_i|}{\Delta t_i^n} \delta_{ij} + \frac{\partial R_i(U^n)}{\partial U_j} \right) \cdot \Delta U_j^n = -R_i(U^n), \quad (16)$$

where  $\Delta U_i^n = U_i^{n+1} - U_i^n$  and if a flux  $\tilde{F}_{ij}$  has a stencil of points  $\{i, j\}$ , then contributions are made to the Jacobian at four points:

$$\frac{\partial R}{\partial U} := \frac{\partial R}{\partial U} + \begin{bmatrix} \ddots & & & & \\ & \frac{\partial \tilde{F}_{ij}}{\partial U_i} & \dots & \frac{\partial \tilde{F}_{ij}}{\partial U_j} & \\ & \vdots & \ddots & \vdots & \\ & -\frac{\partial \tilde{F}_{ij}}{\partial U_i} & \dots & -\frac{\partial \tilde{F}_{ij}}{\partial U_j} & \\ & & & & \ddots \end{bmatrix}. \quad (17)$$

The SU<sup>2</sup> framework includes the implementation of several linear solvers for solving Eq. 16. Currently, the following methods are available:

- The Generalized Minimal Residual (GMRES) method,<sup>32</sup> which approximates the solution by the vector in a Krylov subspace with minimal residual. The Arnoldi iteration is used to find this vector.
- The Biconjugate Gradient Stabilized (BiCGSTAB) method,<sup>33</sup> also a Krylov subspace method. It is a variant of the biconjugate gradient method (BiCG) and has faster and smoother convergence properties than the original BiCG.

For unsteady flows, a dual time-stepping strategy<sup>34,35</sup> has been implemented to achieve high-order accuracy in time. In this method, the unsteady problem is transformed into a steady problem at each physical time step which can then be solved using all of the well-known convergence acceleration techniques for steady problems. The current implementation of the dual-time stepping approach solves the following problem

$$\frac{\partial U}{\partial \tau} + R^*(U) = 0, \quad (18)$$

where

$$R^*(U) = \frac{3}{2\Delta t} U + \frac{1}{|\Omega|^{n+1}} \left( R(U) - \frac{2}{\Delta t} U^n |\Omega|^n + \frac{1}{2\Delta t} U^{n-1} |\Omega|^{n-1} \right), \quad (19)$$

where  $\Delta t$  is the physical time step,  $\tau$  is a fictitious time used to converge the steady state problem,  $R(U)$  denotes the residual of the governing equations, and  $U = U^{n+1}$  once the steady problem is satisfied.

### 3. Convergence Acceleration

Due to the nature of most iterative methods/relaxation schemes, high-frequency errors are usually well damped, but low-frequency errors (global error spanning the larger solution domain) are less damped by the action of iterative methods that have a stencil with a local area of influence. To combat this, SU<sup>2</sup> contains an agglomeration multigrid implementation that generates effective convergence at all length scales of a problem by employing a sequence of grids of varying resolution (SU<sup>2</sup> can automatically generate the coarse grids from the provided fine grid at runtime). Simply stated, the main idea is to accelerate the convergence of the numerical solution of a set of equations by computing corrections to the fine-grid solutions on coarser grids and applying this idea recursively.<sup>36–40</sup>

Preconditioning is the application of a transformation to the original system that makes it more suitable for numerical solution.<sup>41</sup> In particular, a linelet preconditioner has been implemented to improve the convergence rate of the Krylov subspace linear solvers.<sup>37,42</sup> A Roe-Turkel<sup>31</sup> preconditioning for low Mach number flows is available, and a Lower-Upper Symmetric-Gauss-Seidel (LU-SGS) method<sup>43–45</sup> is used to increase the convergence speed of the code.

## III. Validation

### A. Unit/Test Problems

#### 1. Turbulent Flat Plate

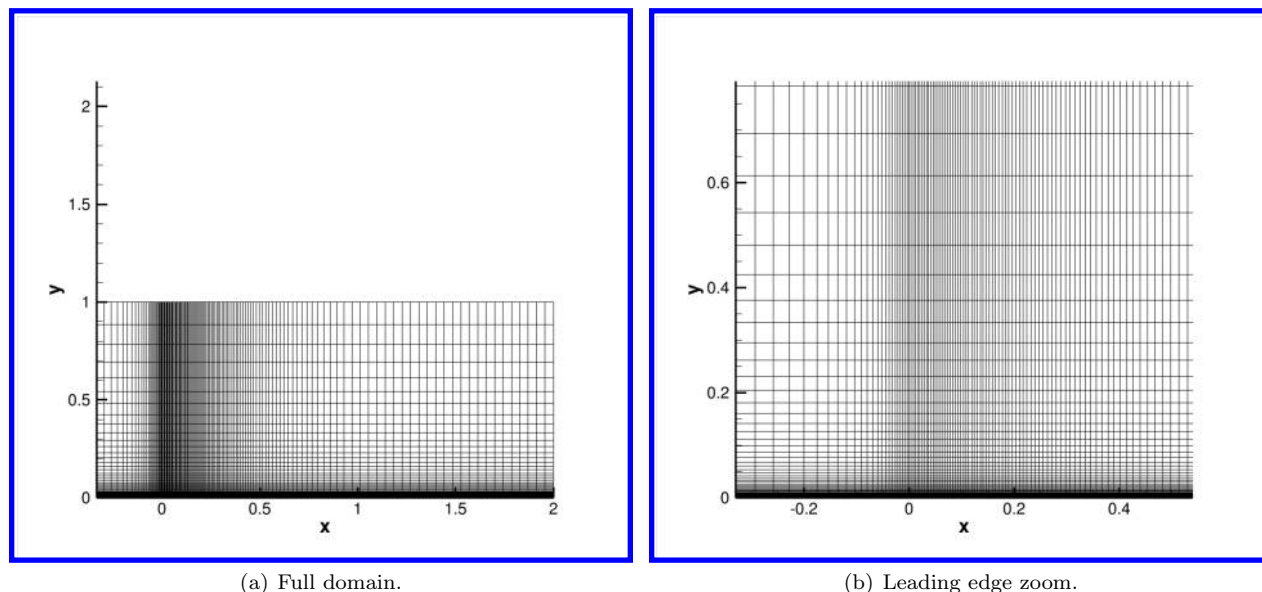
**CASE DESCRIPTION** Flow over a flat plate with zero pressure gradient is a useful validation case for any implementation of a turbulence model. Comparison data, computed using the code CFL3D, is available through the NASA Langley Research Center.<sup>46</sup> The flow conditions used for this case are summarized in Table 1. The implementations of both the SST and S-A models in SU<sup>2</sup> are considered, and both are found to agree with the available solution data.

$M_\infty$	0.2
$Re_L$	$5e6$
$T_\infty$	300K
$\alpha$	$0^\circ$
$\mathcal{R}_\nu(U)$	SA & SST

**COMPUTATIONAL DOMAIN** A rectangular mesh made up of rectangular elements is considered. The grid is shown in Fig. 2. The grid has  $137 \times 97$  cells in the stream-wise and normal directions, respectively.

**Table 1. Flat plate free-stream conditions.**

Total temperature and pressure are prescribed at the inlet. At the upper and outlet boundaries, only the back pressure is prescribed. The remaining quantities are computed using Riemann invariants. A symmetry condition is applied upstream of the flat plate on the lower boundary, and at the wall, a no-slip condition is enforced.



**Figure 2. Flat plate computational grids.**

**NUMERICAL METHODS** The mean flow is computed using the Roe scheme with second-order reconstruction and the Venkatakrishnan limiter. For baseline verification of the turbulence models, the S-A and SST equations are solved using a fully-upwind scheme with both first- and second-order reconstruction. A fixed CFL of 10.0 is used for all cases presented in this section.

**RESULTS** Fig. 3(a) shows the skin friction coefficient,  $C_f$ , plotted against the momentum thickness Reynolds number,  $Re_\theta$ . The quantity  $Re_\theta$  is determined by a trapezoidal integration of the flow solution. Both first- and second-order convective discretizations are shown. In Fig. 3(b), the typical velocity profile for a turbulent boundary layer is plotted. Very good agreement is observed between codes and turbulence models in the sublayer and log layer.

## 2. Turbulent Bump in a Channel

**CASE DESCRIPTION** Flow over a bump in a channel is another useful validation case for turbulence models, as it introduces non-zero pressure gradients on the wall. Comparison data, computed using the CFL3D and FUN3D codes, is available through the NASA Langley Research Center.<sup>47</sup> The flow conditions used for this case are summarized in Table 2. The implementation of the SST model in SU<sup>2</sup> is considered.



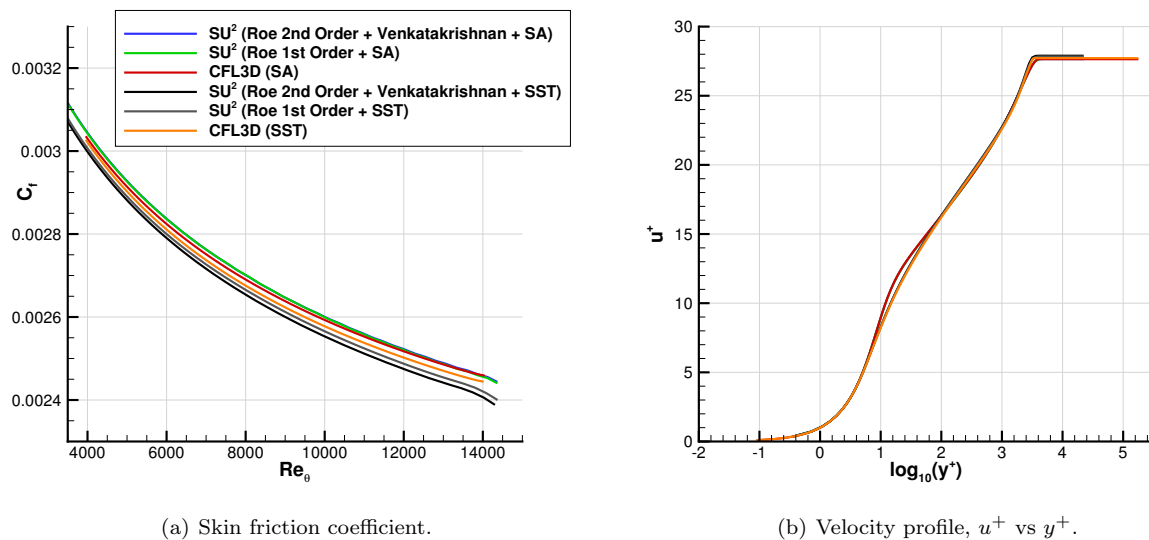


Figure 3. Verification of the zero pressure gradient flat plate.

**COMPUTATIONAL DOMAIN** A mesh made up of rectangular elements is considered. The bump is an adiabatic solid wall, and it extends between  $x = 0$  and  $x = 1.5$ . The overall flow domain and boundary conditions are shown in Fig. 4(a). The grid has  $352 \times 160$  cells in the stream-wise and normal directions, respectively. The grid near the bump is shown in Fig. 4(b).

Total temperature and pressure are prescribed at the inlet. At the outlet, only the back pressure is prescribed. The remaining quantities are computed using Riemann invariants. Symmetry conditions are applied up-stream and down-stream of the bump section as well as along the upper wall. A no-slip condition is enforced on the bump surface.

$M_\infty$	0.2
$Re_L$	$3e6$
$T_\infty$	$540R$
$\alpha$	$0^\circ$
$\mathcal{R}_\nu(U)$	SST

Table 2. Flow conditions for the bump in a channel case.

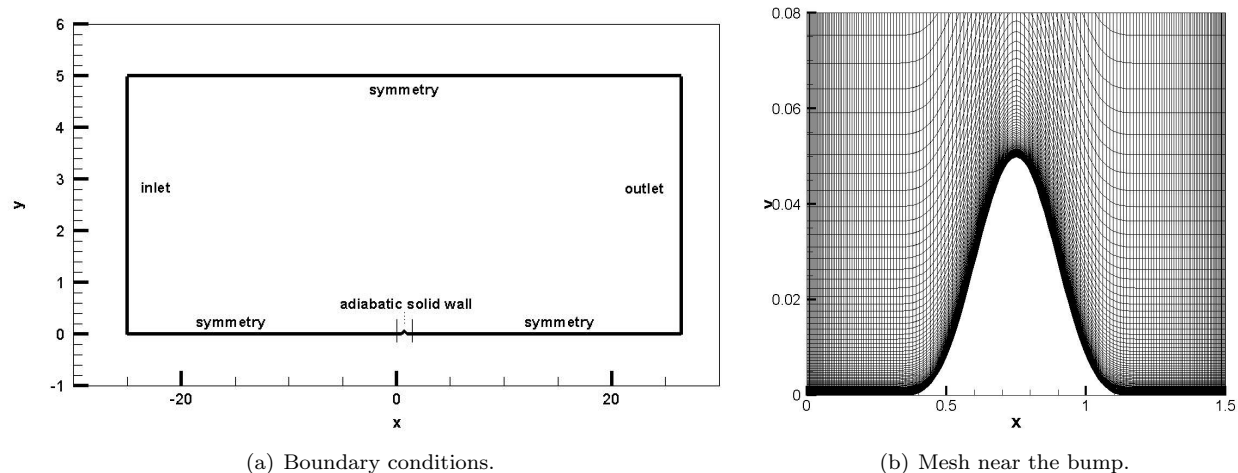
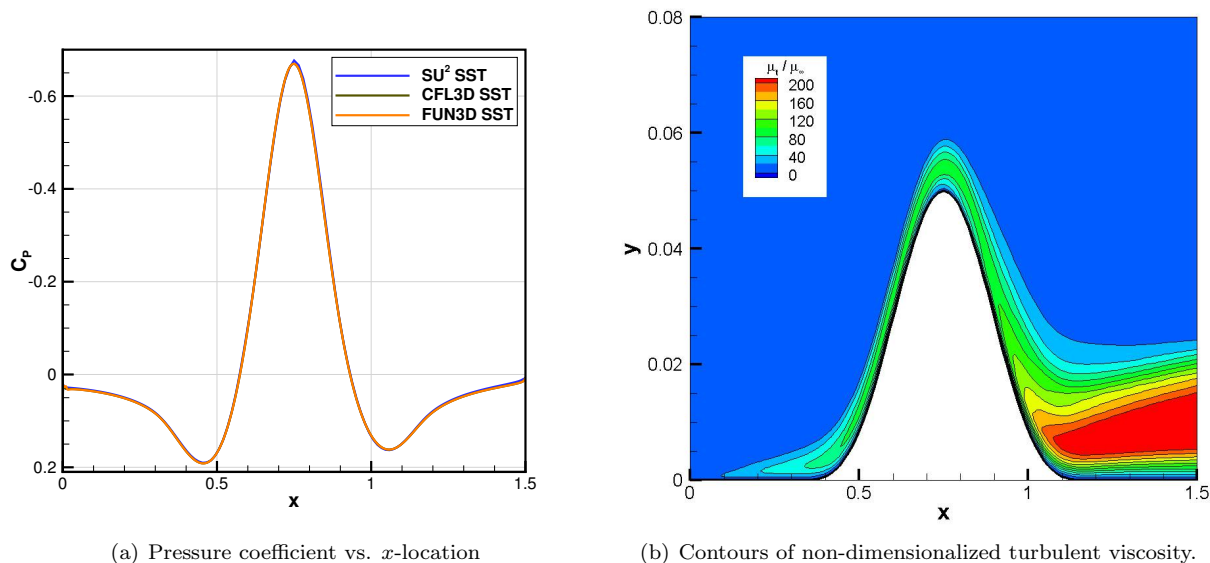


Figure 4. Bump in a channel problem setup and mesh.

**NUMERICAL METHODS** The mean flow is computed using the Roe scheme with second-order reconstruction and the Venkatakrishnan limiter. The SST equations are solved using a fully-upwind scheme with second-

order reconstruction and a fixed CFL of 5.0.

**RESULTS** The data were compared to results from FUN3D and CFL3D simulations which were run on a very fine grid ( $1408 \times 640$  cells).<sup>47</sup> Fig. 5(a) shows the pressure coefficient,  $C_p$ , plotted against the  $x$ -location. Excellent agreement is observed between the different codes. Fig. 5(b) shows contours of turbulent viscosity non-dimensionalized by the free-stream viscosity, and, while not shown here, results from SU<sup>2</sup> compare well to those of CFL3D.



**Figure 5. Verification of the bump in a channel case.**

### 3. Square Cylinder

**CASE DESCRIPTION** To demonstrate the use of the unsteady RANS equations in SU<sup>2</sup> for solving time-accurate problems, a square cylinder test case has been chosen. This case exhibits typical features encountered in flows past bluff bodies, such as separation, recirculation, and vortex shedding. The obtained simulation results are compared with experimental data from Lyn et al.,<sup>48</sup> Lee,<sup>49</sup> and Vickery,<sup>50</sup> as well as with computational results obtained from Rodi et al.<sup>51</sup> and Iaccarino et al.<sup>52</sup>

**COMPUTATIONAL DOMAIN** A two-dimensional, hybrid-element mesh with a rectangular far-field has been used with 22,500 total elements. The domain size is  $20h \times 14h$  in the stream-wise and span-wise directions, respectively, as recommended by Rodi et al.<sup>51</sup> and Iaccarino et al.<sup>52</sup> The value of  $h$  is one, and it is the characteristic length of the square cylinder. The square cylinder is located at  $x = 5h$ . The structured portion of the mesh is clustered close to the surface to better capture the boundary layer, and the spacing near the wall is  $8.1E - 4$  which is enough to guarantee  $y^+ < 1$ . The unstructured portion of the mesh starts at a distance of  $0.1h$  from the wall. This distance was chosen to guarantee that the boundary layer will remain in the structured portion of the grid.

Characteristic-based far-field conditions are applied at the outer domain boundaries. The square cylinder walls have no-slip and adiabatic boundary conditions enforced.

**NUMERICAL METHODS** For the purpose of this study, the convective flux for the mean flow equations will be calculated using the second-order JST scheme, and turbulence will be modeled using the S-A model. The convective flux in the SA model has been discretized with an upwind first-order method for the convective terms, and the viscous terms have been computed using a corrected average-gradient method. For time

$M_\infty$	0.2
$Re_L$	22,000
$T_\infty$	273.15K
$\mathcal{R}_\nu(U)$	SA

**Table 3. Square cylinder free-stream conditions.**

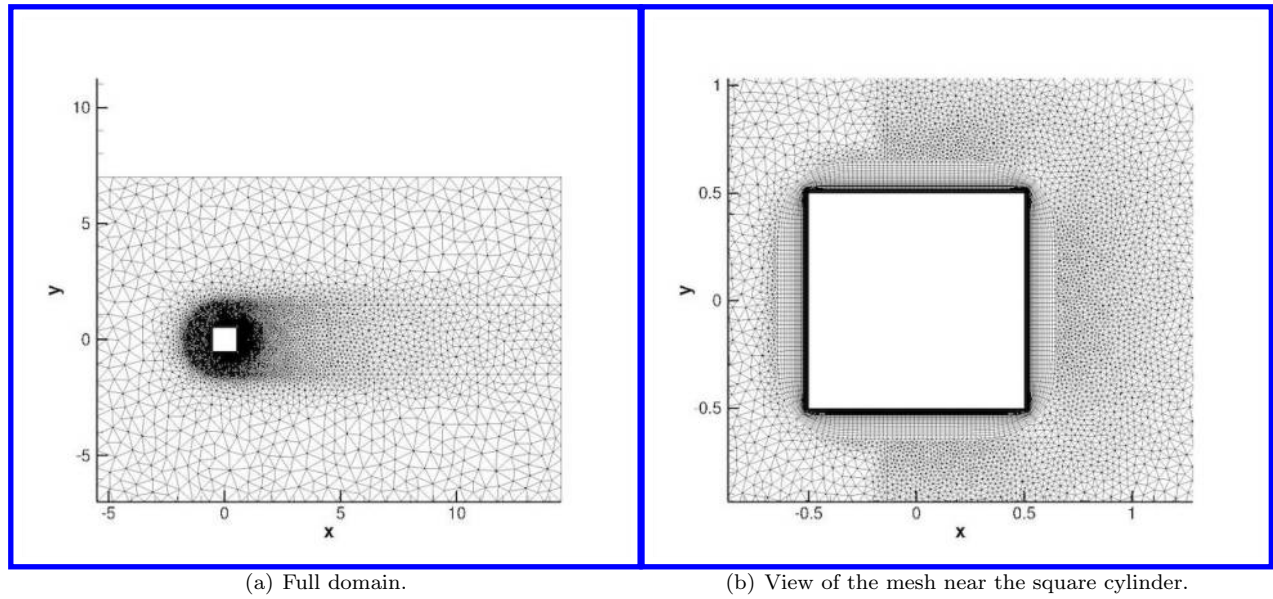


Figure 6. Square cylinder computational grids.

integration, a second-order accurate dual time stepping scheme was used with a physical time step of 0.0015  $s$  and a convergence criteria of 5 orders of magnitude for the relaxation in pseudo-time at each physical time step.

<i>Contribution</i>	<i>Model</i>	$x_r/h$	$\overline{C_d}$	$\tilde{c}_d$	$\tilde{c}_l$	$S_t$
<i>Lyn et al., 1995</i>	<i>Experiments</i>	1.38	2.1	—	—	0.132
<i>Lee et al., 1975</i>	<i>Experiments</i>	—	2.05	0.16 – 0.23	—	—
<i>Vickery, 1966</i>	<i>Experiments</i>	—	2.05	0.1 – 0.2	0.68 – 1.32	—
<i>Rodi et al., 1997</i>	<i>LES</i>	1.32	2.2	0.14	1.01	0.13
<i>Rodi et al., 1997</i>	<i>RANS <math>k - \omega</math></i>	1.25	2.004	—	—	0.143
<i>Iaccarino et al., 2003</i>	<i>RANS <math>v^2 - f</math></i>	1.45	2.22	0.056	1.83	0.141
<i>Present</i>	<i>RANS – SA</i>	1.45	2.428	0.0289	2.7743	0.1402

Table 4. Results for the square cylinder  $Re = 22,000$ .  $x_r/h$  is the recirculation length,  $\overline{C_d}$  is the time averaged drag coefficient,  $\tilde{c}_d$  are the drag-coefficient fluctuations,  $\tilde{c}_l$  are the lift coefficient fluctuations, and  $S_t$  is the Strouhal number

**RESULTS** As expected, the flow past a square cylinder exhibits a periodic vortex shedding in its wake. A summary of the present results from  $SU^2$  and several experimental and computational data sets is presented in Table 3. The frequency at which the cylinder's wake is oscillating is characterized by the Strouhal number, which is in good agreement with the experimental and computational results found in the literature. One of the most important features that has to be analyzed is the recirculation region just downstream of the cylinder. This region is formed due to separation, and the value measured is in good accordance with the computational and experimental results. The average drag coefficient and lift coefficient fluctuations are overestimated by this simulation, while the drag coefficient fluctuations are underestimated with respect to the experimental data. A plausible explanation for the over-prediction of the average drag coefficient and lift coefficient fluctuations is partially due to the lack of three-dimensional effects that are believed to have a great influence on the lift coefficient fluctuations.<sup>52</sup> On the other hand, the effect of different turbulence models on separation prediction should be carefully analyzed.

A qualitative picture of the fluid streamlines laid over a Mach number contour plot is presented in Fig. 7. These plots clearly show the periodic vortex shedding in the cylinder's wake. In addition, we can clearly see that vortices are shed alternating from each side of the cylinder, which are then convected downstream. The

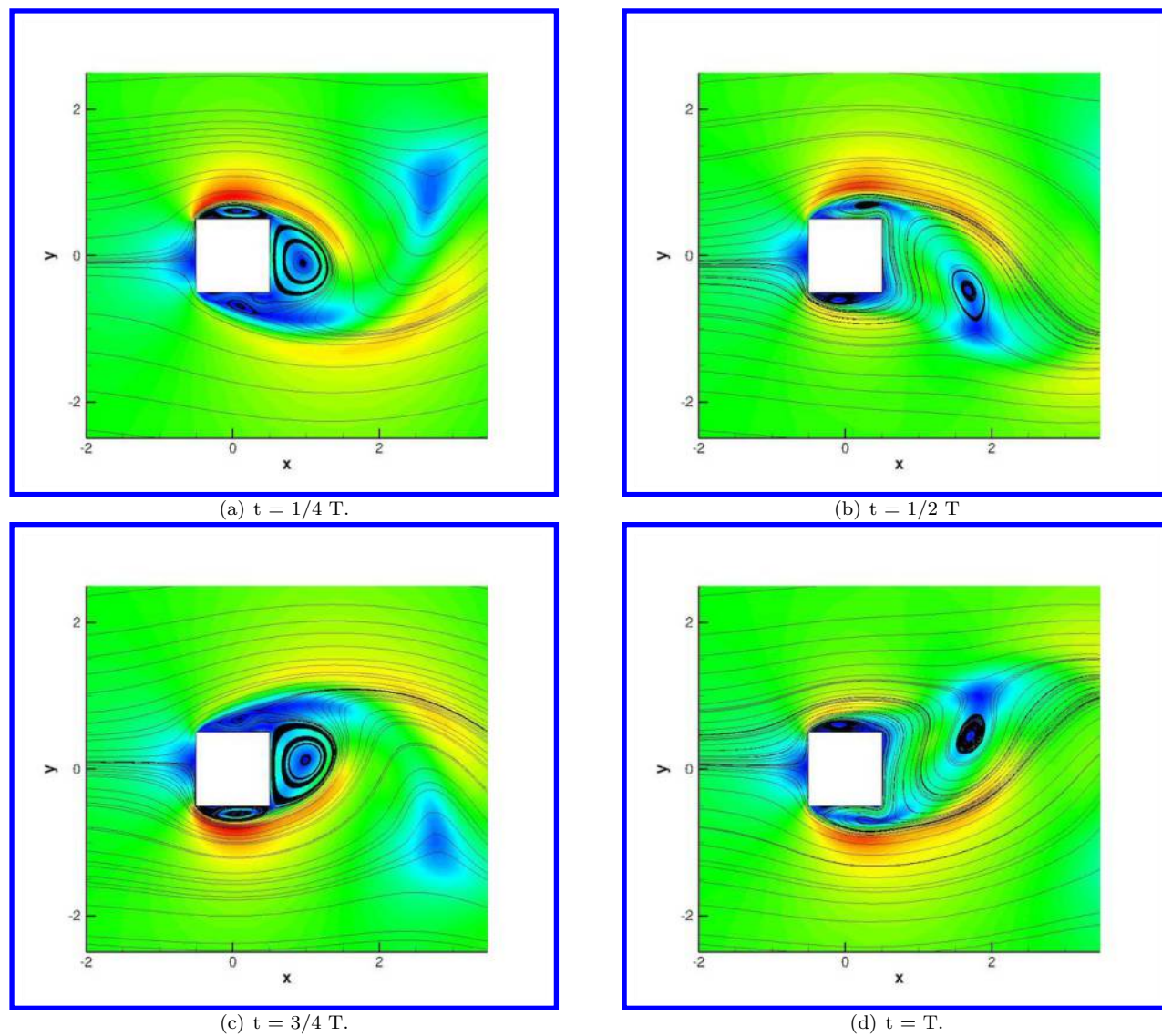


Figure 7. Time history of the streamlines past a square cylinder ( $T$  represents the period).



qualitative results shown are in agreement with the results presented by Iaccarino et al.<sup>52</sup>

## B. Subsonic Airfoil Geometries

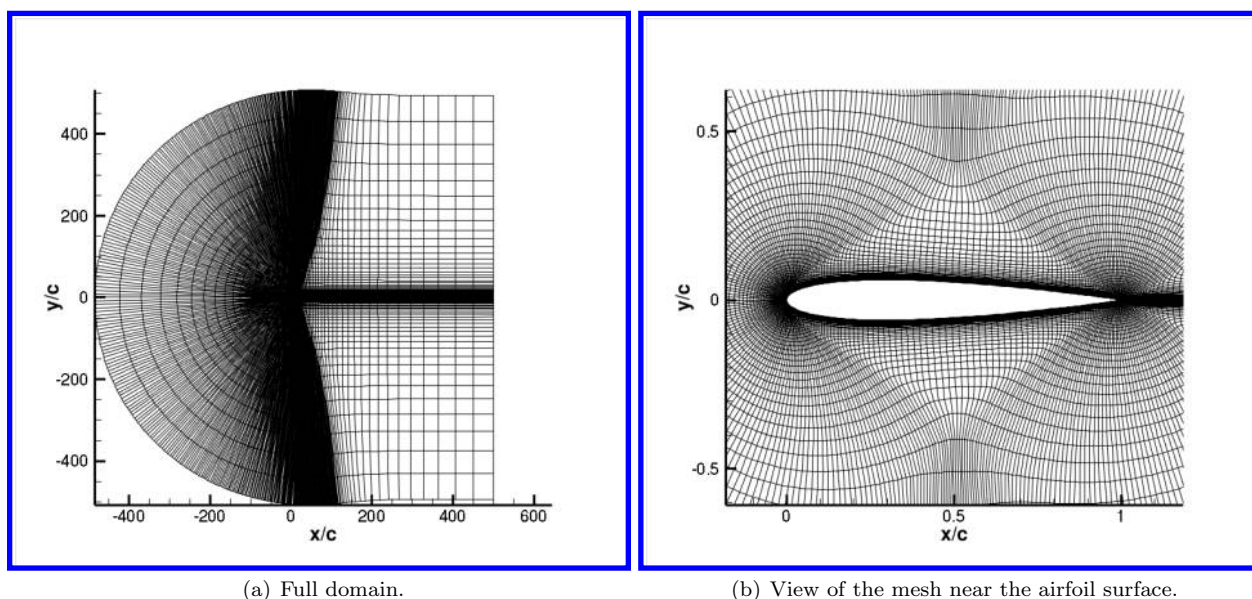
### 1. NACA 0012 Airfoil

**CASE DESCRIPTION** This test case simulates two-dimensional flow over a NACA 0012 airfoil under essentially incompressible free-stream conditions as specified by the AIAA Turbulence Model Benchmarking Working Group (TMBWG). SU<sup>2</sup> simulation outputs of pressure and skin friction coefficients are compared to published experimental data from Gregory,<sup>53</sup> and from NASA's CFL3D software at two angles of attack. The S-A and SST turbulence models are used for this test case, and the free-stream conditions are shown in Table 5.

**COMPUTATIONAL DOMAIN** The two-dimensional discretized volume consists of a C-mesh of quadrilateral elements conforming to the airfoil surface with 897 nodes in the airfoil-normal and 257 in the airfoil-tangent directions and is provided by the TMBWG (see Fig. 8). Domain boundaries for this case are placed 500 chord lengths from the airfoil surface to minimize the effect of the far-field boundaries on the surface solution. Characteristic-based far-field boundary conditions are applied to the outer domain boundaries, and an adiabatic wall boundary condition is enforced on the airfoil surface. Mesh spacing at the airfoil boundary is  $1.5E-6$  and is sufficient to ensure  $y^+ < 1$  over the airfoil surface.

$M_\infty$	0.15
$Re_c$	$6M$
$T_\infty$	$300K$
$\alpha$	$10^\circ, 15^\circ$
$\mathcal{R}_\nu(U)$	S-A & SST

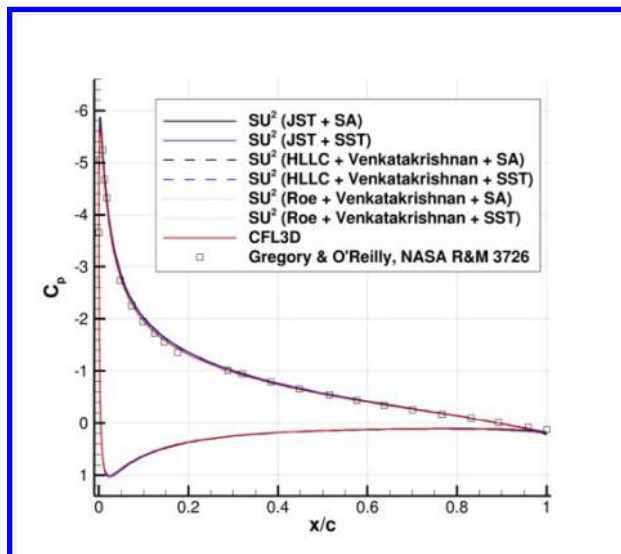
**Table 5. NACA 0012 free-stream conditions.**



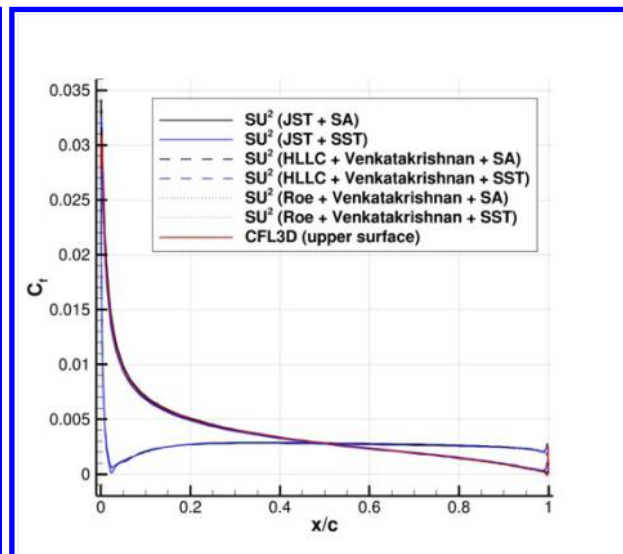
**Figure 8. NACA 0012 computational grids.**

**NUMERICAL METHODS** Three different second-order spatial discretization schemes are used to calculate the convective fluxes for this test case: JST, Roe, and HLLC. For the upwind methods, Venkatakrishnan's limiter is used on the primitive variables. Turbulent variables for the S-A and SST models are convected using a first-order scalar upwind method, and the viscous fluxes are calculated using a corrected average-gradient method. Implicit, local time-stepping is used to converge the problem to the steady-state solution, and the linear system is solved using the GMRES method with a maximum error tolerance of  $\mathcal{O}(10^{-6})$  for each nonlinear iteration of the flow solver.

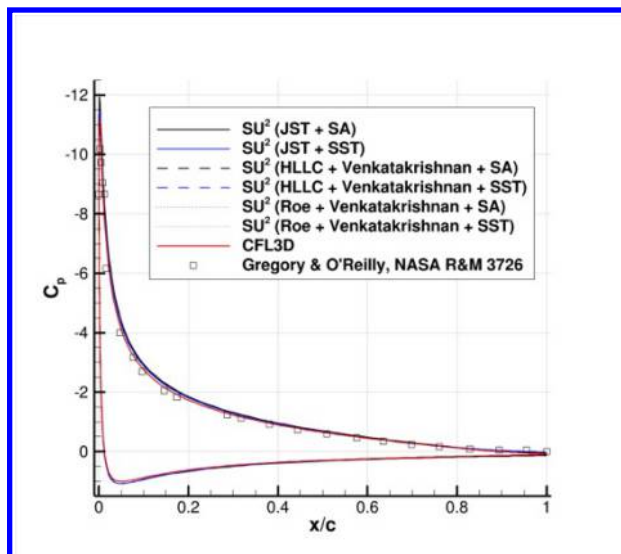
**RESULTS** Airfoil surface pressure and skin friction coefficients are shown in Fig. 9 for the two angle of attack conditions. The computed SU<sup>2</sup> solutions are in good agreement with the published data from Gregory. The



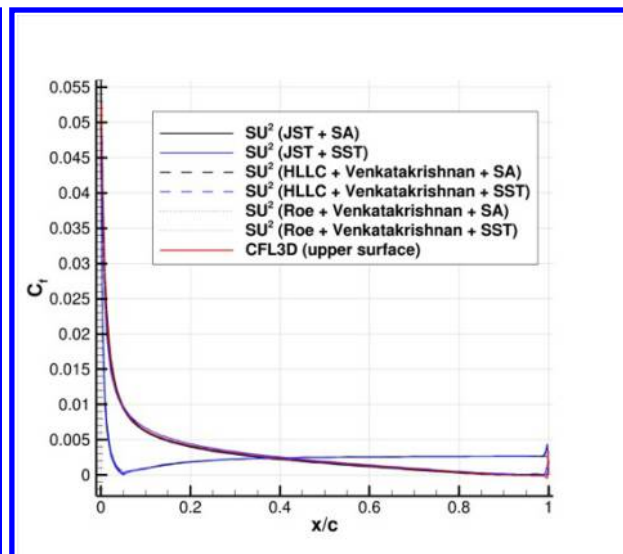
(a)  $C_p$  at  $\alpha = 10^\circ$ .



(b)  $C_f$  at  $\alpha = 10^\circ$ .



(c)  $C_p$  at  $\alpha = 15^\circ$ .



(d)  $C_f$  at  $\alpha = 15^\circ$ .

Figure 9. NACA 0012 surface pressure and skin friction coefficients.

computed values match the experimental pressure coefficient through the suction peak, the pressure recovery region, and back to the trailing edge for both angles of attack. When co-plotted, the effect of the convective scheme and turbulence model on the surface pressure is negligible at the specified run conditions. As there is no known reference for skin friction data for this test case, results are compared to outputs from CFL3D using the S-A turbulence model. Generally, there is good agreement with the NASA solver, though the selection of the convective numerical method and turbulence model does influence the predicted skin friction.

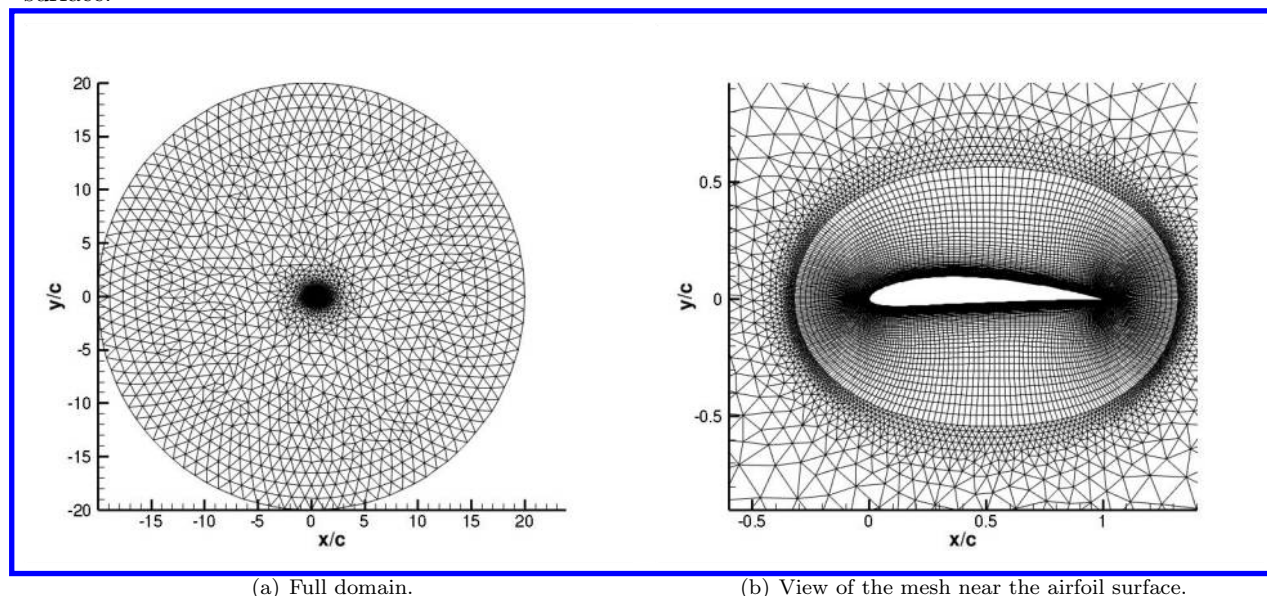
## 2. NACA 4412 Airfoil

**CASE DESCRIPTION** This test case simulates two-dimensional flow over a NACA 4412 airfoil under essentially incompressible free-stream conditions as specified by the NASA Langley Research Center.<sup>8</sup> Pressure coefficient values compute by SU<sup>2</sup> are compared to published experimental data from Coles and Wadcock<sup>54,55</sup> and results from NASA's CFL3D software. The S-A and SST turbulence models are used for this test case and the free-stream conditions are shown in Table 6.

**COMPUTATIONAL DOMAIN** The two-dimensional discretized volume consists of an unstructured, O-mesh conforming to the airfoil surface with 36,145 total elements with 325 edges making up the airfoil boundary and 100 edges on the far-field boundary. It is a hybrid-element mesh with quadrilaterals in the region adjacent to the airfoil surface and triangles in the remaining portion of the computational domain. The far-field boundary is located approximately 20 chord lengths away from the airfoil. A characteristic-based far-field condition is applied to the outer domain boundary, and an adiabatic, no-slip wall boundary condition is enforced on the airfoil surface. Mesh spacing at the airfoil boundary is 1E-5 and is sufficient to ensure  $y^+ < 1$  over the airfoil surface.

$M_\infty$	0.09
$Re_c$	$1.52M$
$T_\infty$	$300K$
$\alpha$	$13.87^\circ$
$\mathcal{R}_{\bar{\nu}}(U)$	S-A & SST

**Table 6. NACA 4412 free-stream conditions.**



**Figure 10. NACA 4412 computational grids.**

**NUMERICAL METHODS** Roe's second-order upwind scheme is used to calculate the convective fluxes for this test case. Venkatakrishnan's limiter is used on the primitive variables. Turbulent variables for the S-A and SST models are convected using a second-order scalar upwind method, and the viscous fluxes are calculated using the corrected average-gradient method. Implicit, local time-stepping is used to converge the problem to the steady-state solution, and the linear system is solved using the GMRES method with a maximum error tolerance of  $\mathcal{O}(10^{-6})$ .

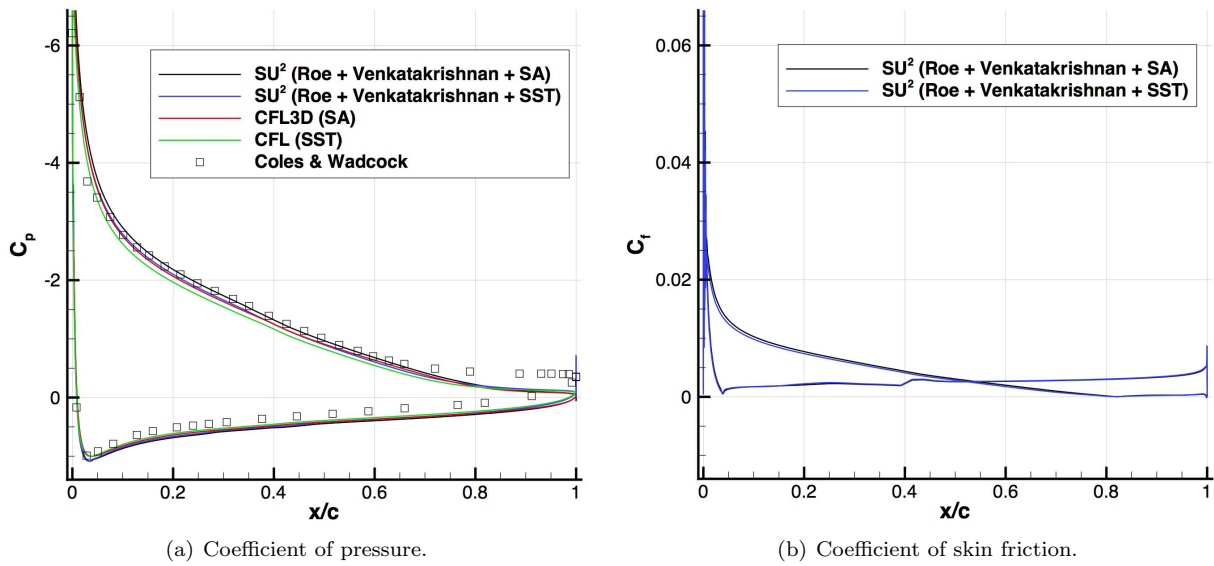


Figure 11. Comparison of the NACA 4412 results from  $SU^2$  with the experiments and numerical simulations.

**RESULTS** Airfoil surface pressure and skin friction coefficients are shown in Fig. 11(a) and 11(b), respectively. The computed  $SU^2$  solutions for the coefficient of pressure are in good agreement with the published data from Coles & Wadcock. The values computed by  $SU^2$  match the experimental pressure coefficient through the suction peak and the pressure recovery region. When co-plotted, the effect of the turbulence model on the surface pressure is negligible at the specified run conditions. The results are also compared to outputs from CFL3D using the S-A and SST turbulence model. There is a slight mismatch between the experimental data and the numerical results in the distribution of pressure near the trailing edge of this airfoil due to the presence of a large separation bubble at such high angles of attack. However, the numerical results from  $SU^2$  do match those from CFL3D. The solutions for the coefficient of skin friction also show the expected behavior with the two turbulence models predicting similar results, but experimental results are not available for comparison. It is worth noting that the simulation results from  $SU^2$  are for a blunt-trailing-edge airfoil (like the experiments), while the published results for CFL3D use a slightly modified airfoil with a sharp trailing edge.

### 3. 30P30N High-lift Configuration

**CASE DESCRIPTION** This test case simulates two-dimensional flow over a McDonnell-Douglas 30P30N high lift configuration. Pressure and skin friction coefficient results from  $SU^2$  are compared with published results from Rumsey<sup>56</sup> and experimental data from Chin et al.<sup>57</sup> at different angles of attack. The SST turbulence model is used for this test case, and the free-stream conditions are shown in Table 7.

**COMPUTATIONAL DOMAIN** The simulations are run on a coarse and a refined mesh for the same conditions (see Fig. 12). Both of the two-dimensional meshes contain quadrilateral cells in the near-wall region (for good boundary layer resolution) and in the wake region, and triangular cells are found everywhere else. Grid 1 contains  $\mathcal{O}(400,000)$  cells, and Grid 2 contains  $\mathcal{O}(700,000)$  cells. Domain boundaries for this case are placed 15 chord lengths from the geometry. Characteristic-based far-field conditions are applied to the outer domain boundaries, and an adiabatic wall boundary condition is enforced on the airfoil surface. Mesh spacing at the airfoil boundary is chosen to ensure  $y^+ < 1$  over the airfoil surface.

**NUMERICAL METHODS** Roe's second-order upwind scheme is used to calculate the convective fluxes for this test case. Venkatakrishnan's limiter is applied to the primitive variables. Turbulent variables for the

$M_\infty$	0.2
$Re_{c_{stowed}}$	$9M$
$T_\infty$	$300K$
$\alpha$	$8^\circ, 19^\circ$
$\mathcal{R}_\nu(U)$	SST

Table 7. 30P30N free-stream conditions.



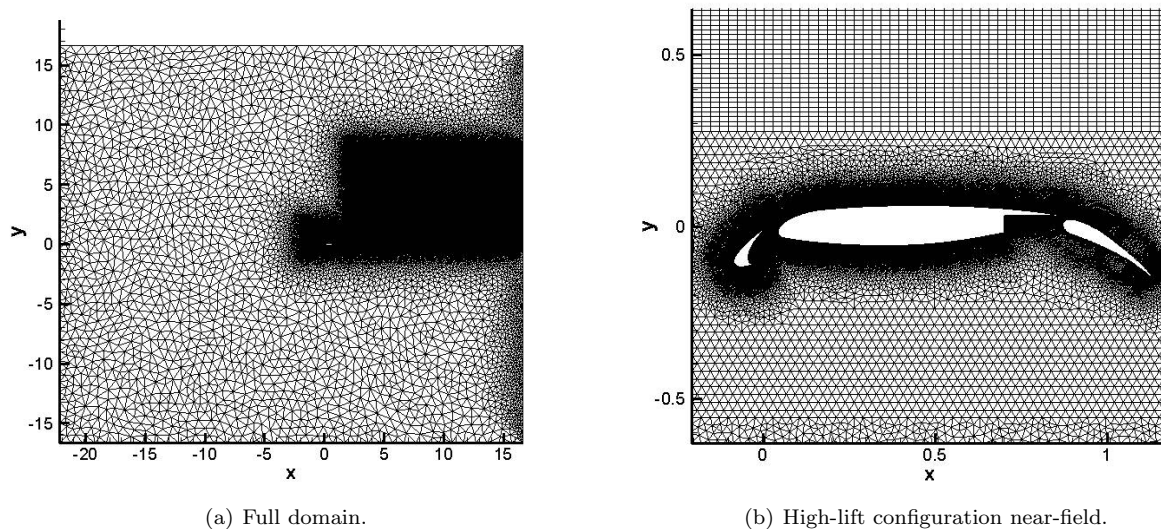


Figure 12. McDonnell-Douglas 30P30N high-lift configuration computational grids.

SST model are convected using a first-order scalar upwind method, and the viscous fluxes are calculated using the corrected average-gradient method. Implicit, local time-stepping is used to converge the problem to the steady-state solution, and the linear system is solved using the GMRES method with a maximum error tolerance of  $\mathcal{O}(10^{-6})$  for each nonlinear iteration of the flow solver.

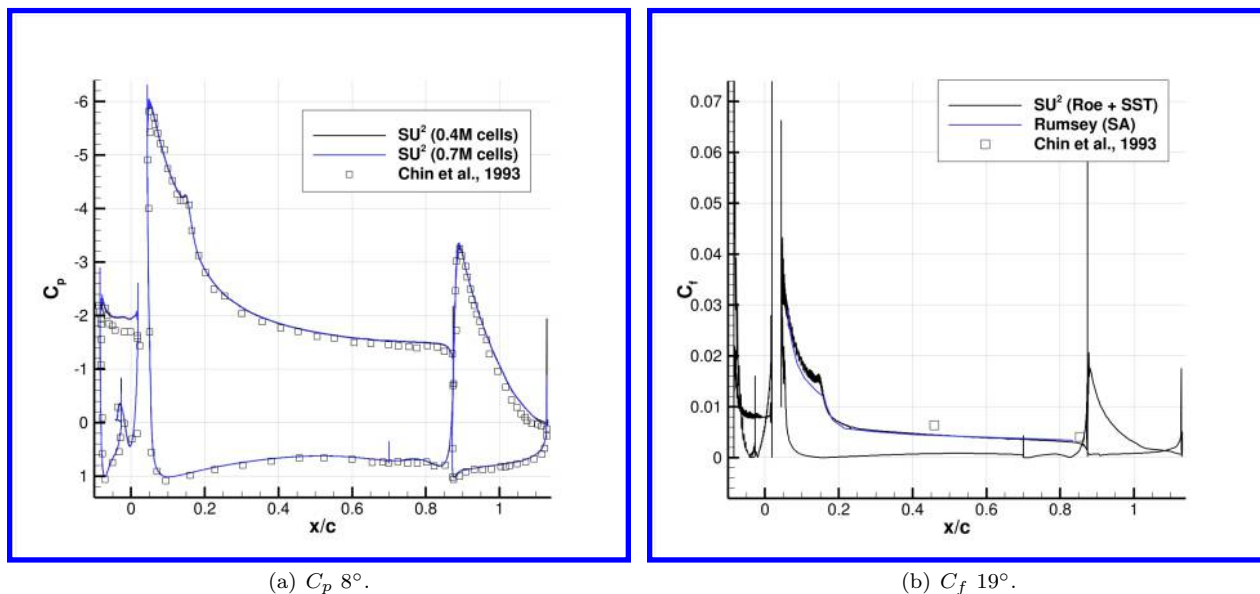


Figure 13. Pressure and skin friction coefficient for the 30P30N configuration.

**RESULTS** The  $C_p$  distributions for the 30P30N configuration from both grids are compared with experimental results from Chin et al.<sup>57</sup> The  $C_p$  distribution over the main element is captured well at both the leading and trailing edges, as seen in Fig. 13. The main discrepancy between the computed and experimental results is an over-prediction of the drop in  $C_p$  on the upper surface of the slat. The skin friction coefficient is also compared for an angle of attack of 19 degrees with S-A results from Rumsey<sup>56</sup> and experimental results.<sup>57</sup> The  $C_l$  for this configuration is computed at different angles of attack. These are shown below in the  $C_l$  vs. angle of attack plot in Fig. 14. The  $C_l$  values obtained using SU<sup>2</sup> are fairly close to the experimental

results<sup>57</sup> for the lower angles of attack. For the higher angle of attack cases where separation is significant, the  $C_l$  is under-predicted compared to the experimental results. This is probably because of the inability to model accurately the effect of separation behind the slat, which results in a reduced pressure peak for both the slat and the main element.

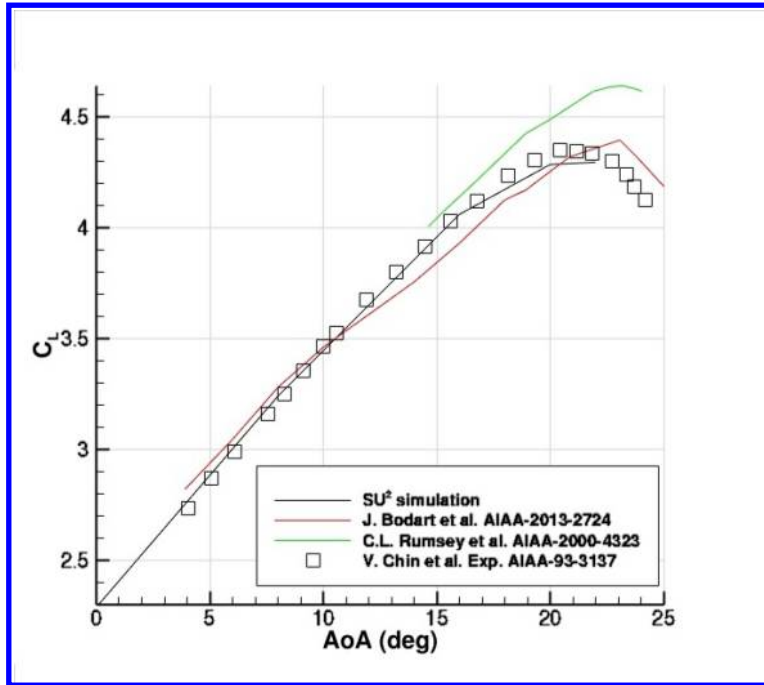


Figure 14.  $C_l$  vs. angle of attack for the 30P30N configuration.

## C. Subsonic Wing and Rotor Configurations

### 1. Delta Wing

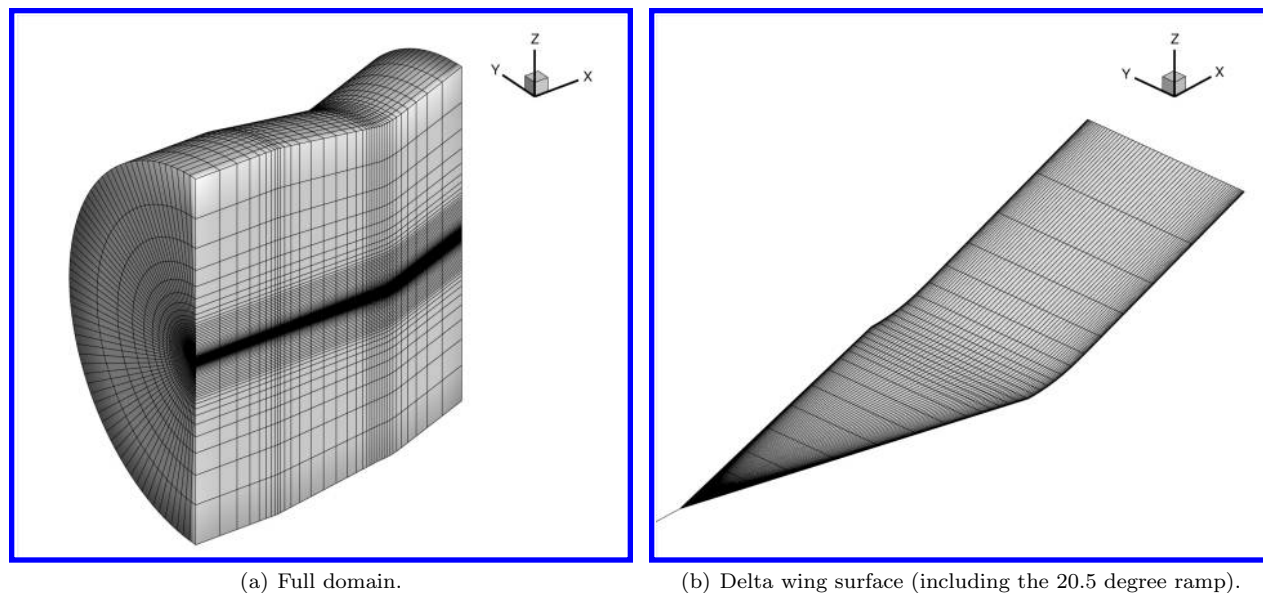
**CASE DESCRIPTION** This test simulates a three-dimensional flow field over a delta wing with 75 degrees of sweep at 20.5 degrees angle of attack (two stable leading edge vortices are present). The problem is modeled at a Reynolds number of 0.9 million and Mach number of 0.3. In this particular simulation, the pressure distribution computed with SU<sup>2</sup> will be compared with experimental data.<sup>58</sup> The free-stream conditions are shown in Table 8.

**COMPUTATIONAL DOMAIN** The three-dimensional discretized volume consists of a 65x65x33 (span-wise, radial, chord-wise) hexahedral numerical grid (see Fig. 15). Domain boundaries are placed 1.5 chord lengths from the delta wing surface. Characteristic-based far-field conditions are applied to the outer domain boundaries, and an adiabatic, no-slip wall boundary condition is enforced on the delta wing surface. Behind the delta wing there is a 20.5 degrees ramp created as a byproduct of the structured grid generation technique.

$M_\infty$	0.15
$Re_c$	0.9M
$T_\infty$	300K
$\alpha$	20.5°
$\mathcal{R}_\nu(U)$	Laminar flow, SA

**NUMERICAL METHODS** The JST convective scheme has been used with a very low value for the fourth-order artificial dissipation of  $\mathcal{O}(10^{-4})$ . The turbulent variable for the S-A model is convected using a first-order scalar upwind method, and the viscous fluxes are calculated using the corrected average-gradient method. Implicit, local time-stepping is used to converge the problem to the steady-state solution, and the linear system is solved using the GMRES method with a maximum error tolerance of  $\mathcal{O}(10^{-6})$  for each nonlinear iteration of the solver.

Table 8. Swept delta wing free-stream conditions.



**Figure 15. Delta wing computational grids.**

**RESULTS** The steady state solution to this problem is characterized by two stable, leading edge vortices. There is experimental data for velocity profiles, pressure on the surface, and the locations of the vortex cores. In particular, the wing surface pressure coefficient is shown in Fig. 16 at 4 different sections ( $x/L = 0.3, 0.5, 0.7, 0.9$ ). In general, the trend is well captured, and the agreement is acceptable (better near the symmetry plane). However, further studies are required to understand the pressure discrepancies close to the leading edge. The degenerate elements and the coarseness of the numerical grid appear to be the most plausible causes for the discrepancies.

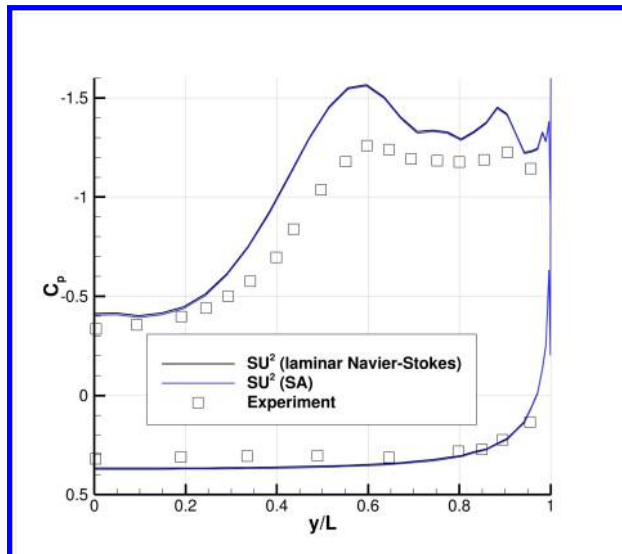
## 2. Caradonna and Tung Rotor

**CASE DESCRIPTION** The code's ability to simulate three-dimensional, viscous flows around rotor geometries is demonstrated by validating against the well-known experimental results of Caradonna and Tung.<sup>59</sup> The experiment comprises a two-bladed, rigid rotor in hover. The two blades are identical: each has a rectangular planform with an aspect ratio of 6, the rotor radius,  $R$ , is 1.143 m and the chord length,  $c$ , is 0.1905 m. The blades are untwisted, untapered, and maintain a constant NACA 0012 airfoil section along their entire span. The present study seeks to validate the specific test case wherein the blades are positioned at precone angle,  $\beta_0$ , of 0.5 degrees and pitched upwards at a collective angle,  $\theta_c$ , of 8 degrees. When viewed from above, the blades rotate in a clockwise direction at an angular velocity,  $\Omega$ , of 1250 RPM (i.e. 130.9 rad/s) - resulting in a tip Mach number,  $M_{tip}$ , of 0.439. Naturally, inasmuch as the rotor is in hover, the free-stream conditions are for still air. The general flow parameters are summarized in Table 9.

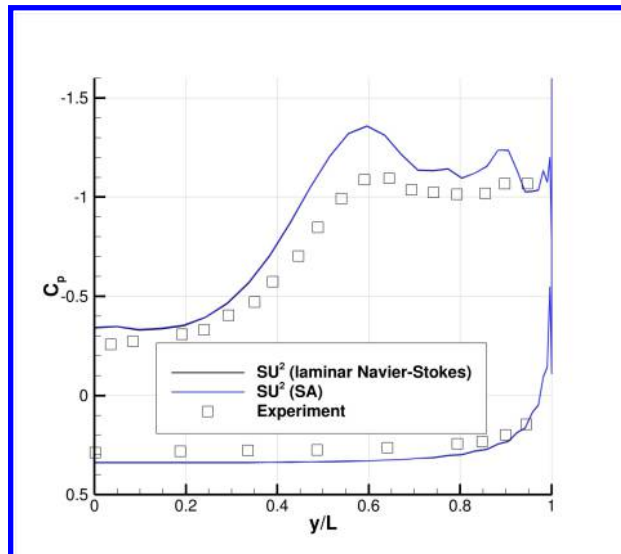
$M_{tip}$	0.439
$\Omega$	1250 RPM
$\theta_c$	8°
$\beta_0$	0.5°
$M_\infty$	0
$p_\infty$	95,680 Pa
$\rho_\infty$	1.2168 kg/m <sup>3</sup>
$\mathcal{R}_\nu(U)$	SA

**Table 9. Caradonna and Tung flow conditions.**

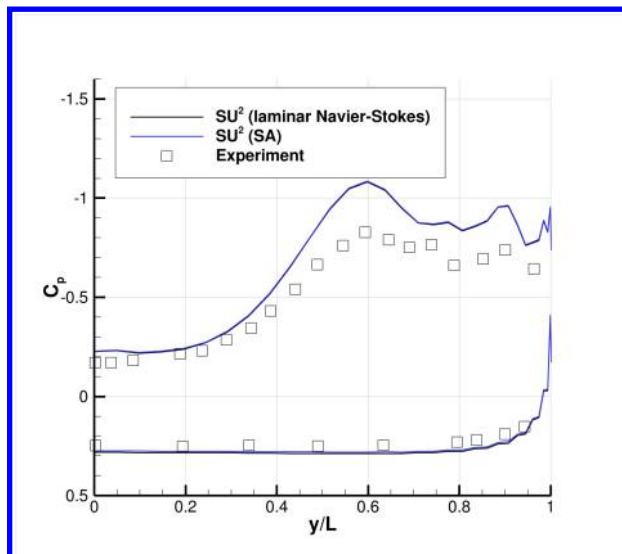
The numerical experiment uses a full, two-bladed representation of the rotor geometry, as seen in Fig. 17(a). Here, a time-accurate solution scheme is eschewed in favor of one in which the RANS equations are solved in a rotating reference frame, which offers significant savings in computational cost by recovering a steady solution procedure for this problem. Navier-Stokes closure is achieved by means of the S-A turbulence model. Validation of the numerical results will be done by comparing computed profiles of pressure coefficient,  $C_p$ , at certain blade cross sections with experimental values reported by Caradonna and Tung. It is well to mention that in the context of rotors, the coefficient of pressure is traditionally defined as  $C_p = (p - p_\infty)/(0.5\rho_\infty\Omega^2r^2)$ , where  $p_\infty$  is the ambient pressure,  $\rho_\infty$  is the ambient density, and  $r$  is the



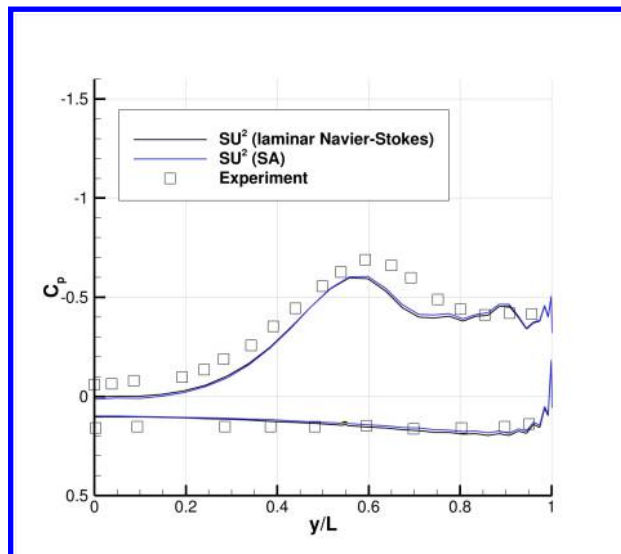
(a)  $C_p$  distribution at wing section  $x/L = 0.3$ .



(b)  $C_p$  distribution at wing section  $x/L = 0.5$ .



(c)  $C_p$  distribution at wing section  $x/L = 0.7$ .

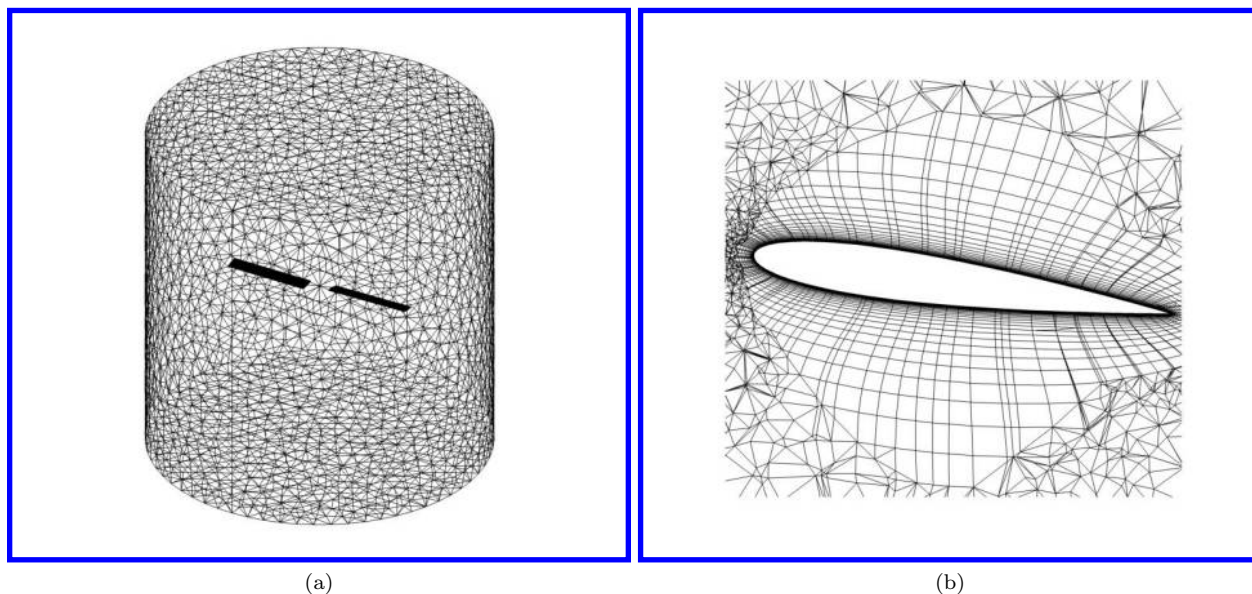


(d)  $C_p$  distribution at wing section  $x/L = 0.9$ .

Figure 16.  $C_p$  distribution at different delta wing sections.



radial distance at which the cross section of interest resides.



**Figure 17. Computational set-up.** (a) The physical geometry model of the two blades is seen embedded within the discretized flow domain. (b) A cross section through the geometry shows the refinement of the volume mesh near the blade surface.

**COMPUTATIONAL DOMAIN** The computational domain – seen in Fig. 17(a) – is an upright cylinder, which is large enough to encapsulate the entire plane of rotation. This flow domain is discretized by a hybrid-element mesh that uses 1.23 million tetrahedra, 3.06 million prisms, 26,478 pyramids, and 1.76 million nodes in total. The mesh refinement near the blade surfaces, shown in Fig. 17(b), has been verified to produce an acceptable dimensionless wall distance ( $y^+ < 1$ ).

Since the blade tips are not far from the upright walls of the cylindrical domain, the computational setup must be prepared to accommodate both the rotor wake and the induced velocities that simultaneously appear. This is accomplished by assigning characteristic-based inlet and outlet conditions to the outer boundaries of the domain. A no-slip, adiabatic condition is enforced along the blade surfaces.

**NUMERICAL METHODS** The convective fluxes for the mean flow equations are computed using the JST scheme. The turbulence working variable for the S-A model is convected using a first-order, scalar upwind method. As with many of the other test cases presented, all of the viscous fluxes have been computed using the average-of-gradients method, with a correction. The rotating-frame problem is driven to a steady state by means of an implicit, local time-stepping scheme. At each pseudo-time step, the GMRES method is employed to solve the linear system. The GMRES method is allowed to run for either 20 iterations or until an error tolerance of  $\mathcal{O}(10^{-6})$  is met.

**RESULTS** The computational problem described above was converged to the point where the density residual had decreased by three and a half orders of magnitude.

Because this case is simulating a rotor in hover, both blades see identical distributions of airloads along their spans. In Fig. 18,  $C_p$  profiles at four different radial stations have been plotted against the experimental data of Caradonna and Tung.

The  $SU^2$  RANS pressure distributions are in excellent agreement with the experimental data. Although the experimental value of the suction peak is not quite matched at the four sections investigated, the values of  $C_p$  throughout the pressure-recovery region, as well as along the trailing edge, are matched nearly exactly.

## D. Transonic Airfoil Geometries

### 1. RAE 2822 Transonic Airfoil

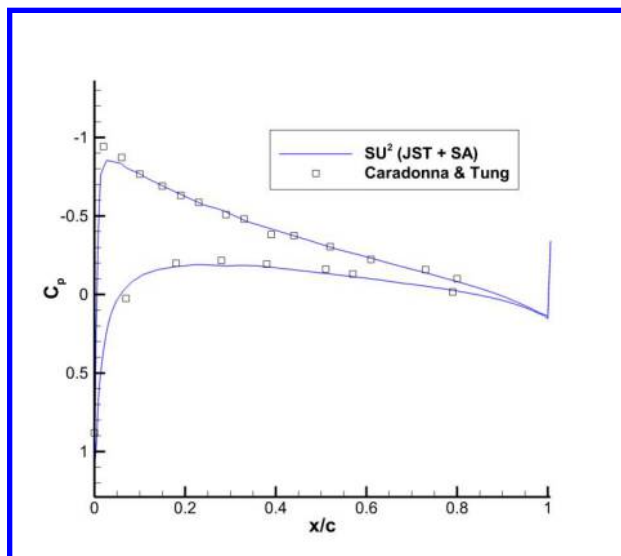
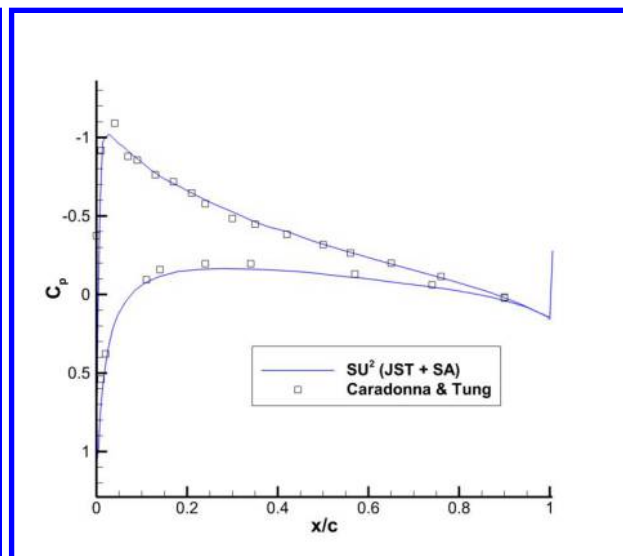
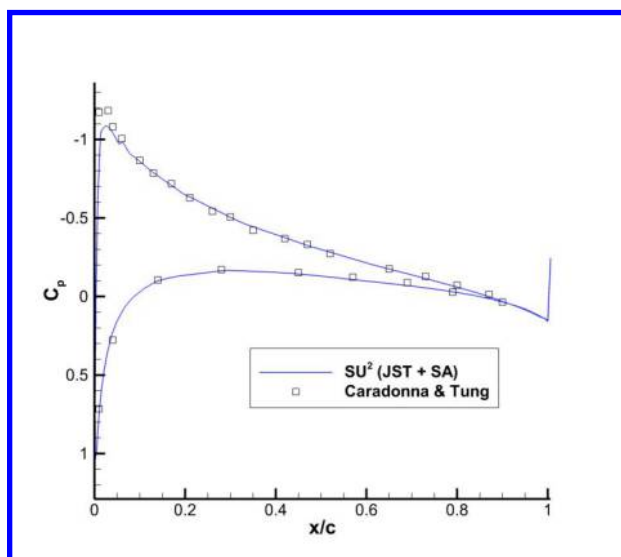
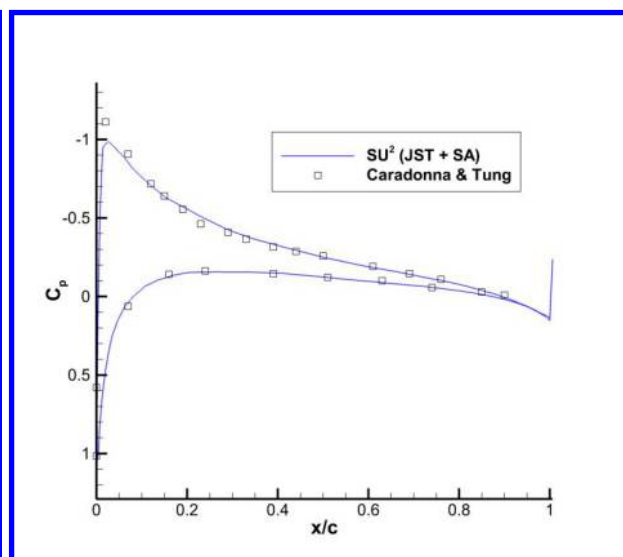
(a)  $r = 0.77724$  m(b)  $r = 0.91440$  m(c)  $r = 1.01727$  m(d)  $r = 1.09728$  m

Figure 18. Comparison of  $SU^2$  results (solid lines) with the experimental data of Caradonna and Tung (open squares).  $C_p$  distributions are plotted at various radial stations along the span of a given blade: (a)  $r/R = 0.68$ , (b)  $r/R = 0.80$ , (c)  $r/R = 0.89$ , and (d)  $r/R = 0.96$ .

**CASE DESCRIPTION** The RAE 2822 airfoil is a supercritical airfoil commonly used for validation of turbulence models. The experimental cases 6, 9, and 10 from AGARD<sup>60</sup> are considered. The flow conditions for these cases are detailed in Table 10. The flow conditions for case 6 are used by NASA's NPARC Alliance Verification and Validation Archive<sup>61</sup> and those of case 9 and 10 are from DLR.<sup>62</sup> These flow conditions are corrected from those of AGARD<sup>60</sup> to account for wind tunnel influences.

	Case 6	Case 9	Case 10
$M_\infty$	0.729	0.730	0.750
$Re_c$	6.5E6	6.5E6	6.2E6
$T_\infty$	273.15K	273.15K	273.15K
$\alpha$	2.31°	2.80°	2.80°
Scheme	JST & Roe	HLLC	HLLC
$\mathcal{R}_v(U)$	SST & SA	SA	SA

**COMPUTATIONAL DOMAIN** The mesh used is an unstructured, O-grid that wraps around the RAE 2822 airfoil. It has 22,842 elements in total with 192 edges making up the airfoil boundary and 40 edges along the far-field boundary. It is a hybrid-element mesh with quadrilaterals in the region adjacent to the airfoil surface and triangles in the remaining portion of the computational domain, as seen in Fig. 19. The first grid point off the airfoil surface is at a distance of  $1E-5$  chords, and the far-field boundary is located approximately one hundred chord lengths away from the airfoil. Characteristic-based far-field boundary conditions are enforced on the far-field boundary, and a no-slip, adiabatic boundary condition is enforced on the airfoil.

Table 10. RAE 2822 free-stream conditions.

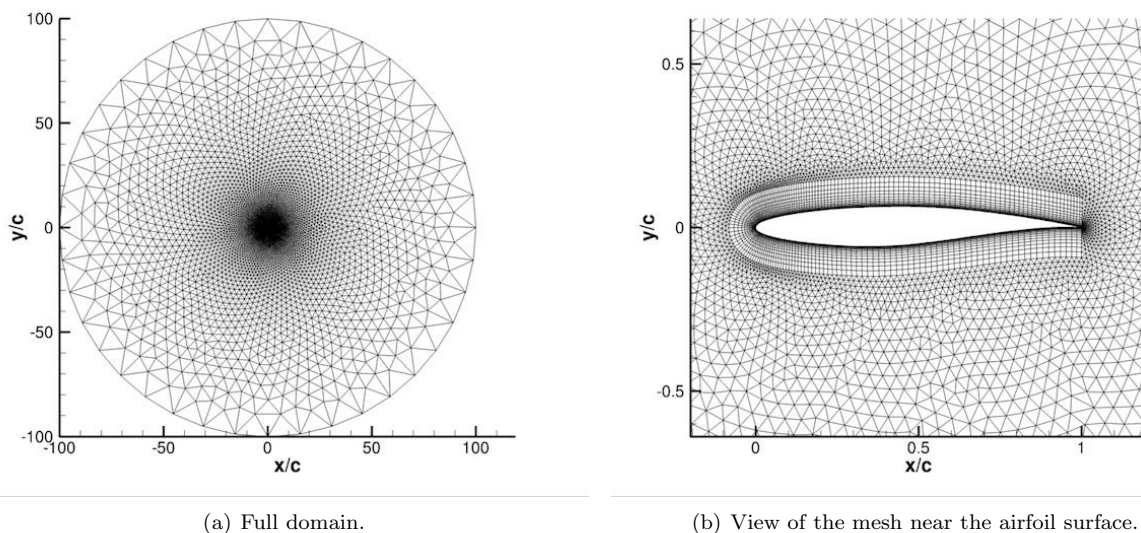


Figure 19. RAE 2822 computational grids.

**NUMERICAL METHODS** The mean flow convective fluxes are calculated using the schemes listed in Table 10. For the upwind methods, Venkatakrishnan's limiter is used on the primitive variables. Turbulent variables for the S-A and SST models are convected using a second-order scalar upwind method, and the viscous fluxes are calculated using the corrected average-gradient method. Implicit, local time-stepping is used to converge the problem to the steady-state solution, and the linear system is solved using the GMRES method with a maximum error tolerance of  $\mathcal{O}(10^{-6})$  for each nonlinear solver iteration. The effect of this error tolerance on total solution time (wall clock) is also explored for this case in the results.

**RESULTS** The coefficient of pressure obtained from the SU<sup>2</sup> simulation of case 6 compares favorably with the experimental data<sup>60</sup> with a small discrepancy in the shock wave position, as presented in Fig. 20. This discrepancy is also observed by the NASA study<sup>a</sup>. Changing the turbulence model (Fig. 20(a)) and the mean flow convective flux scheme (Fig. 20(b)) lead to small differences in the coefficient of pressure that are most apparent at the expansion region over the leading edge and at the shock wave location.

<sup>a</sup><http://www.grc.nasa.gov/WWW/wind/valid/raetaf/raetaf05/raetaf05.html>

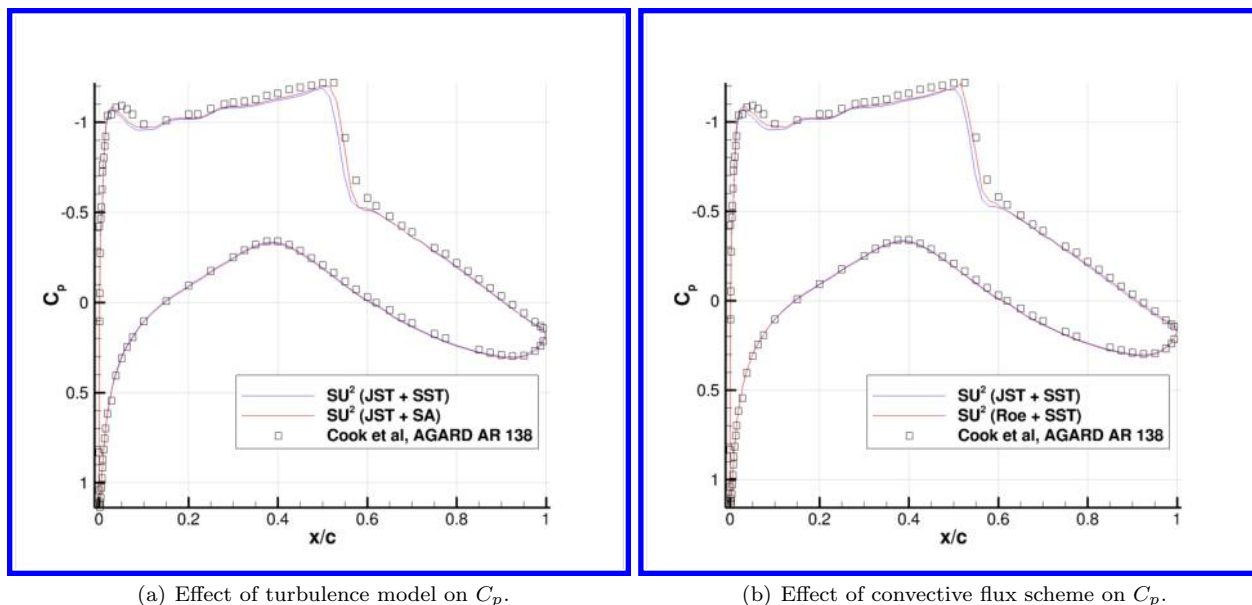


Figure 20. RAE 2822 surface pressure distributions for case 6.

For case 6 (using JST and SST), the effect of the linear solver level of convergence on the total solution time was investigated. Table 11 contains the results from this study. The flow solution was converged 6 orders of magnitude in the density residual while the level of convergence of the linear solver was varied from  $1\text{E}-1$  to  $1\text{E}-8$ , which is approximately equivalent to performing 1 to 10 linear solver iterations, respectively. A linear solver convergence criteria of  $1\text{E}-2$  (about 2 to 3 linear solver iterations) provided the fastest convergence time, and this time is used to normalize the other time entries in Table 11. Note that, for this problem, converging the linear solver more than  $1\text{E}-2$  is a waste of resources, as the number of nonlinear solver iterations needed to reach the desired tolerance remains fixed at 2770, while the number of linear iterations increases.

Convergence	Linear Iter	Time	Solver Iter
$1.0\text{E}-1$	1-2	1.017	2900
$1.0\text{E}-2$	2-3	1.0	2770
$1.0\text{E}-4$	5	1.054	2770
$1.0\text{E}-6$	7-8	1.099	2770
$1.0\text{E}-8$	10	1.150	2770

Table 11. Effect of linear solver convergence on total solution time.

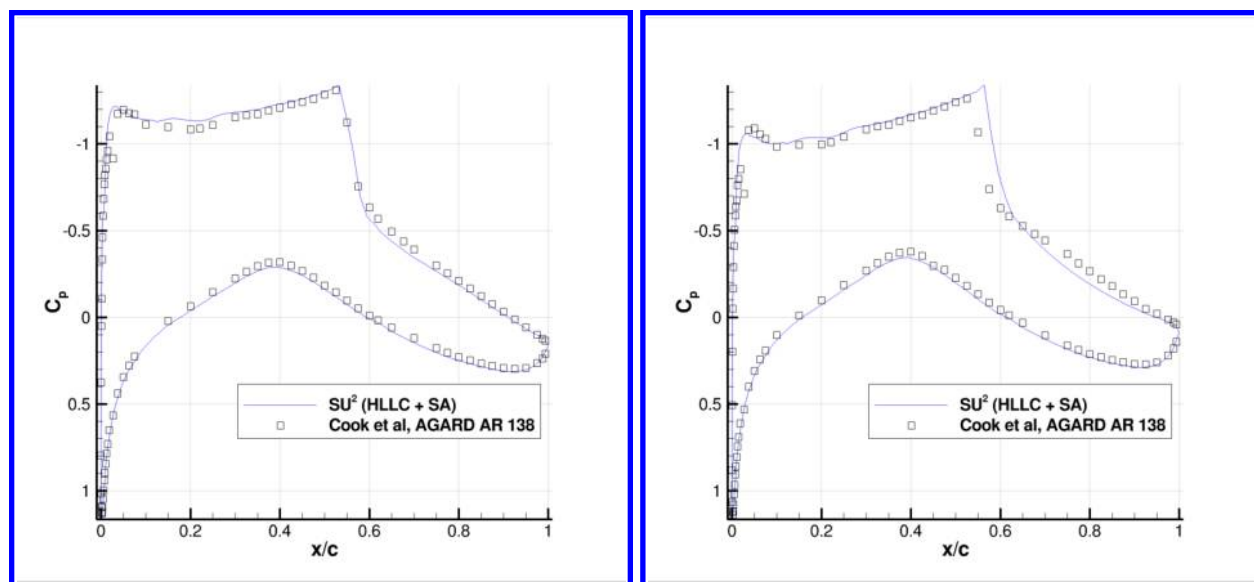
In addition to case 6, case 9 and 10 have also been simulated. The pressure distributions for both cases are compared with experimental results in Fig. 21(a) and Fig. 21(b). As before, a very good match between the simulations and experiments is observed. For the flow conditions of case 10, note that there is a massive detachment of the flow after the shock wave, which leads to the small discrepancies observed in Fig. 21(b).

## E. Transonic Wing and Full Aircraft Configurations

### 1. ONERA M6 Wing

**CASE DESCRIPTION** The ONERA M6 wing was designed in 1972 by the ONERA Aerodynamics Department as an experimental geometry for studying three-dimensional, high Reynolds number flows with some complex flow phenomena (transonic shocks, shock-boundary layer interaction, and separated flow, for instance). It has become a classic validation case for CFD codes due to the available geometric description, complicated flow physics, and the availability of experimental data. More specifically, it is a swept, semi-



(a) RAE 2822 surface  $C_p$  distribution for case 9.(b) RAE 2822 surface  $C_p$  distribution for case 10.**Figure 21. Evaluation of Case 9 and Case 10 for the RAE 2822.**

span wing with no twist that uses a symmetric airfoil (ONERA D sections). The aspect ratio is 3.8, and the leading edge angle is 30.0 degrees.

In this investigation, we will compare computational results from  $SU^2$  against experimental data from Schmitt and Charpin<sup>63</sup> for pressure coefficient distributions at several span-wise stations of the wing. The chosen flow conditions are from Test 2308:  $M_\infty = 0.8395$ , an angle of attack 3.06 degrees, and an angle of sideslip 0.0 degrees. These correspond to a Reynolds number of 11.72 million based on the mean aerodynamic chord of 0.64607 m. A summary of the conditions can be found in Table 1.

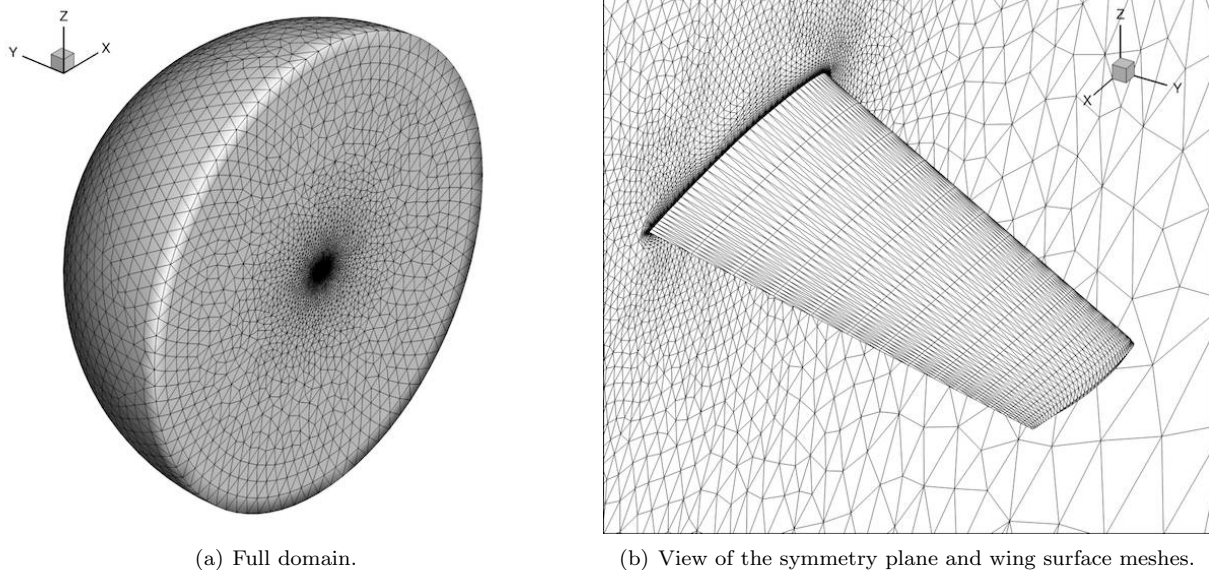
**COMPUTATIONAL DOMAIN** The unstructured, mixed-element mesh around the ONERA M6 wing consists of 220,145 tetrahedra, 107,477 prisms, and 1,431 pyramids (329,053 total interior elements) with a total of 96,252 nodes. A layer of prisms surrounds the wing surface for capturing the boundary layer, and the mesh spacing near the wall was set to achieve a  $y^+ < 1$  over the entire wing surface. A no-slip, adiabatic condition is satisfied on the wing surface, a symmetry plane is used to reflect the flow about the plane of the root airfoil section to mimic the effect of the full wing planform, and a characteristic-based condition is applied at a spherical far-field boundary. The full domain and surface meshes for the wing geometry and symmetry plane are shown in Fig. 22.

$M_\infty$	0.8395
$Re_{mac}$	11.72M
$T_\infty$	273.15K
$\alpha$	3.06°
$\mathcal{R}_{\bar{\nu}}(U)$	SA & SST

**Table 12. ONERA M6 free-stream conditions.**

**NUMERICAL METHODS** The mean flow convective fluxes for this test case are computed with the JST scheme. The turbulent variables for the S-A and SST models are convected using a first-order scalar upwind method, and all viscous fluxes are calculated using the corrected average-gradient method. Implicit, local time-stepping is used to relax the problem to a steady-state solution, and the linear system is solved at each pseudo-time step using the iterative GMRES method with a maximum error tolerance of  $\mathcal{O}(10^{-6})$ . For both the S-A and SST cases, the flow was converged five orders of magnitude in the density residual.

**RESULTS** Surface pressure coefficient distributions are shown at four different span-wise stations of the wing in Fig. 23. Overall, the computed  $SU^2$  distributions are in good agreement with the published data from Schmitt & Charpin. There is almost no noticeable difference between the computed results using the S-A and the SST models for this case (and with the other present numerical methods). Near the outboard region of the wing ( $y/b = 0.8, 0.95$ ), the computed values of pressure match experiment particularly well.



**Figure 22.** View of the computational grid for the ONERA M6 case.

The computed pressures along the upper surface for  $y/b = 0.2$  and  $y/b = 0.65$  show some discrepancies from the experimental values, but the pressure along the lower surface, leading edge, and the aft portion of the wing are in good agreement. The typical lambda shock pattern on the upper surface of the wing is captured, but the shocks could be captured more crisply with a finer mesh. However, the agreement between SU<sup>2</sup> and experiment is particularly impressive, as the mesh used for this case is relatively coarse (less than 100,000 nodes in total).

## 2. DLR-F6 Transonic Airplane

**CASE DESCRIPTION** Transonic flow over the DLR F6 aircraft (wing body configuration) is computed with SU<sup>2</sup> in this validation case. For the baseline geometry and case definition, we have chosen the DLR F6 configuration without a fairing that was used in the 3<sup>rd</sup> CFD Drag Prediction Workshop (Mach number 0.75 and Reynolds number 5E6). In order to match the experiment lift coefficient of 0.498, a zero angle of attack was required in the numerical settings. It is important to remark that the wind tunnel experiments were performed at a Reynolds number of 3E6 and an angle of attack of 0.49 degrees.

A detailed description of the geometry and experimental results can be found in the documentation produced by the 3<sup>rd</sup> CFD Drag Prediction Workshop<sup>b</sup>. The original reference for the baseline DLR F6 geometry is by Brodersen and Stürmer.<sup>64</sup> The S-A and SST turbulence models are used for this test case, and the free-stream conditions are shown in Table 2.

$M_\infty$	0.75
$Re_c$	3M, 5M
$T_\infty$	273K
$\alpha$	0°, 0.49°
$\mathcal{R}_\nu(U)$	SA & SST

**Table 13.** NACA 0012 free-stream conditions.

**COMPUTATIONAL DOMAIN** The mesh used in this study is a hybrid-element grid (see Fig. 24) composed of 8,773,810 total elements and 3,059,189 nodes (generated with the ANSYS ICEM CFD Mesh Generation Software). The mesh is composed of tetrahedra, prisms, and pyramids around a surface that has been discretized using triangles. The far-field boundary is located approximately 20 body lengths away from the aircraft with a suitable spacing in the boundary layer to allow for a  $y^+ \approx 1$ .

**NUMERICAL METHODS** A JST centered spatial discretization has been used to calculate convective fluxes. Turbulent variables for the S-A and SST models are convected using a first-order scalar upwind method, and

<sup>b</sup><http://aaac.larc.nasa.gov/tsab/cfdlarc/aiaa-dpw/Workshop3/>

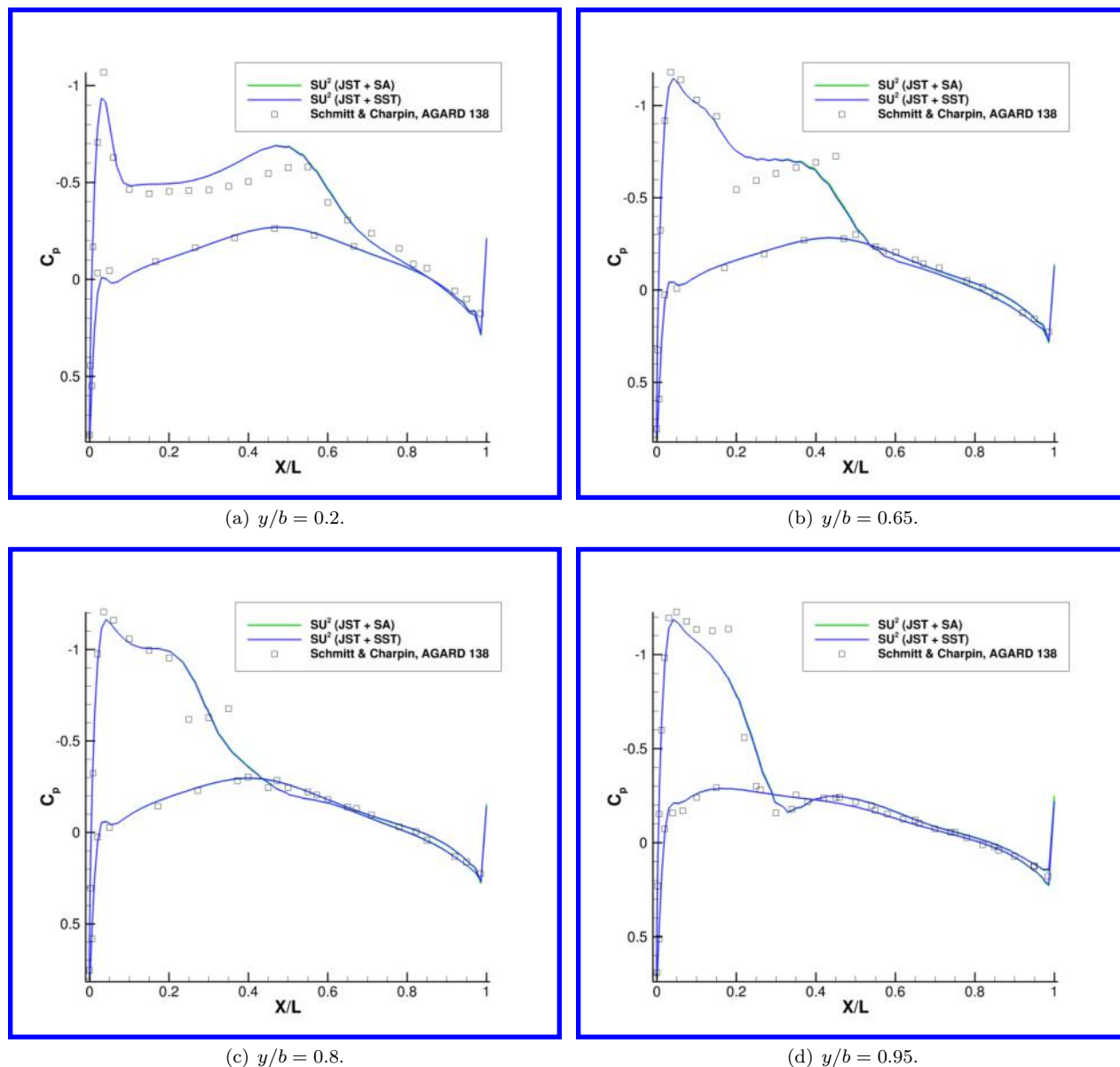
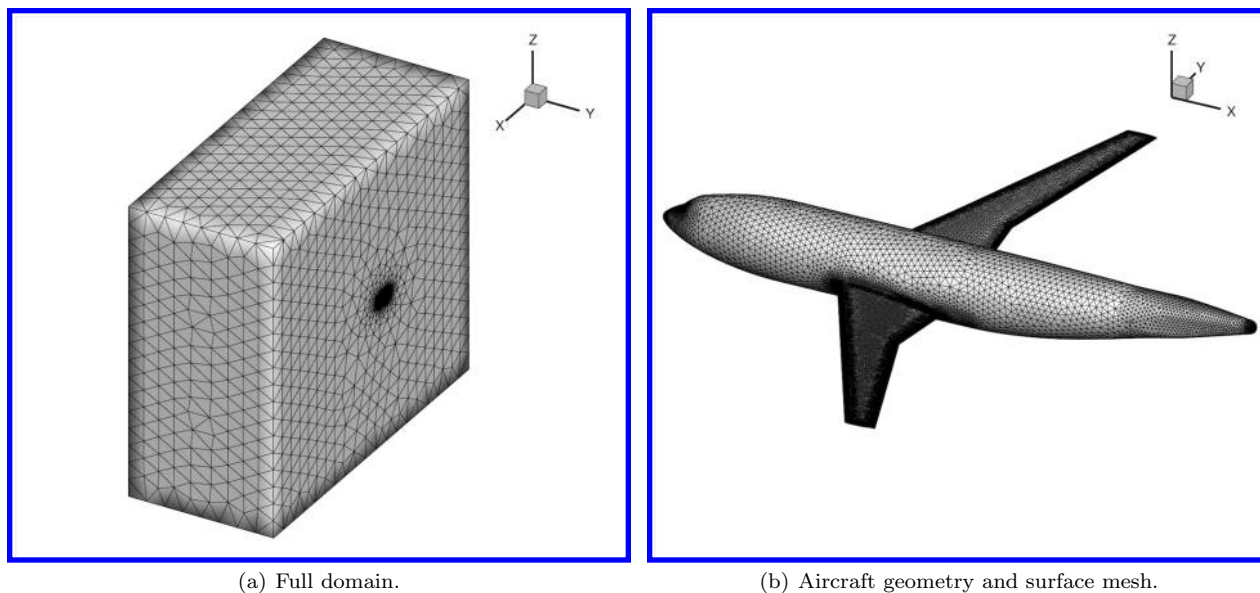


Figure 23.  $C_p$  distribution comparison between the experimental results of Schmitt and Charpin and SU<sup>2</sup> at different sections along the span of the wing.



**Figure 24. DLR F6 computational grids.**

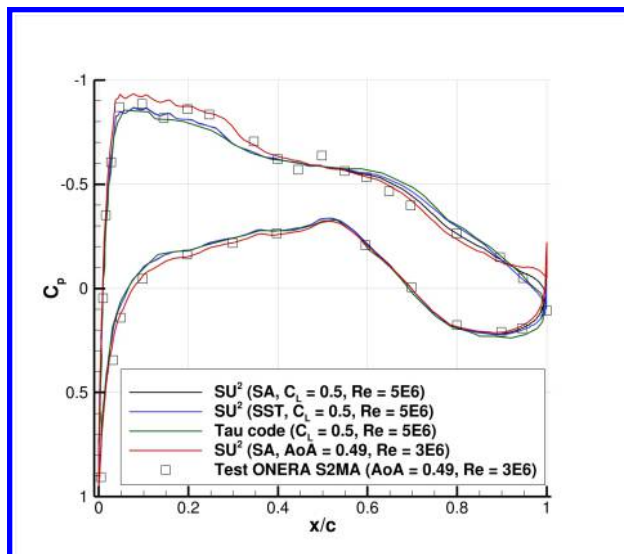
the viscous fluxes are calculated using the corrected average-gradient method. Implicit, local time-stepping is used to converge the problem to the steady-state solution, and the linear system is solved using the iterative GMRES method with a maximum error tolerance of  $\mathcal{O}(10^{-6})$ .

**RESULTS** A complete set of results has been obtained for this configuration (not all are shown in this paper). In this particular study, four representative sections of the wing ( $y/b = 0.150, 0.331, 0.409, 0.844$ ) are presented in Fig. 25. To obtain these results, two different sets of conditions have been used:

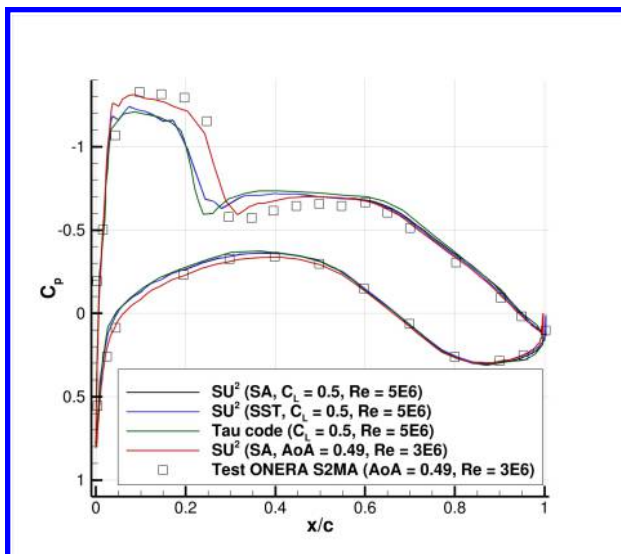
- First, the wind tunnel lift coefficient  $C_L = 0.5$  was matched at Reynolds number  $5E6$ . In this case, the results are compared with those obtained by the code Tau (DLR) with very good agreement. It is also important to highlight the small differences introduced by the turbulence models (more important in the inboard section close to a well-known recirculation region in the wing-fuselage intersection).
- Second, the conditions from the wind tunnel experiment were matched. In this case, the angle of attack is set to  $0.49$  degrees with a Reynolds number of  $3E6$ . With this particular setting, despite the fact that the lift coefficient is over-predicted ( $C_L = 0.53$ ), we obtain very good agreement with the experimental data, except near the most outboard section of the wing where there is a mismatch in the shock wave location (probably due to the low resolution of the numerical grid).

This complex, full aircraft configuration is a perfect example for demonstrating the adjoint RANS solver that is integrated in  $SU^2$  for obtaining the sensitivities needed for shape design. After solving the RANS equations, the direct flow solution and the same computational mesh can be immediately reused as inputs for solving the adjoint RANS equations in the solver (while taking advantage of similar numerical methods and the same code structure). While using less computational time and memory resources than in the direct problem with the present continuous adjoint formulation, it is possible to evaluate the surface sensitivity after solving the RANS adjoint equations for a particular objective function.

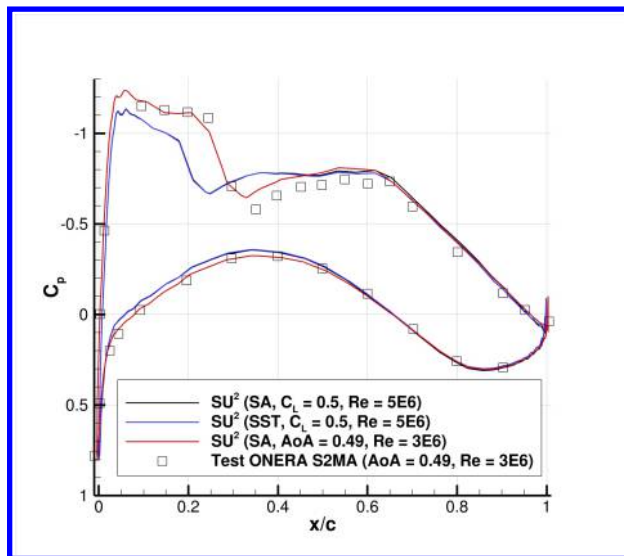
The pressure distribution on the upper and lower surfaces and the surface sensitivity (for the drag, lift, and pitching moment coefficients) are shown in Fig. 26. This sensitivity information reveals the impact of a particular geometrical change on the selected objective function and can be used for gradient-based shape optimization or directly by the designer to manually improve the shape of the aircraft. The DLR F6 case presented here is meant to serve as an example of how  $SU^2$  can be used both to analyze the performance of complex geometries using the RANS equations and to efficiently compute the sensitivities needed by a designer for a large-scale engineering problem.



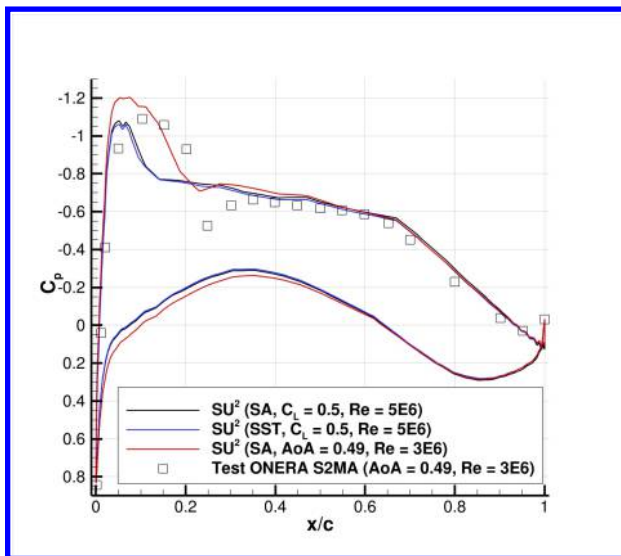
(a)  $C_p$  distribution at wing section  $y/b = 0.150$ .



(b)  $C_p$  distribution at wing section  $y/b = 0.331$ .



(c)  $C_p$  distribution at wing section  $y/b = 0.409$ .



(d)  $C_p$  distribution at wing section  $y/b = 0.844$ .

**Figure 25.**  $C_p$  distributions at  $C_L = 0.5$ ,  $R_N = 5E6$  (workshop), and  $AoA = 0.49$ ,  $R_N = 3E6$  (wind tunnel experiment).

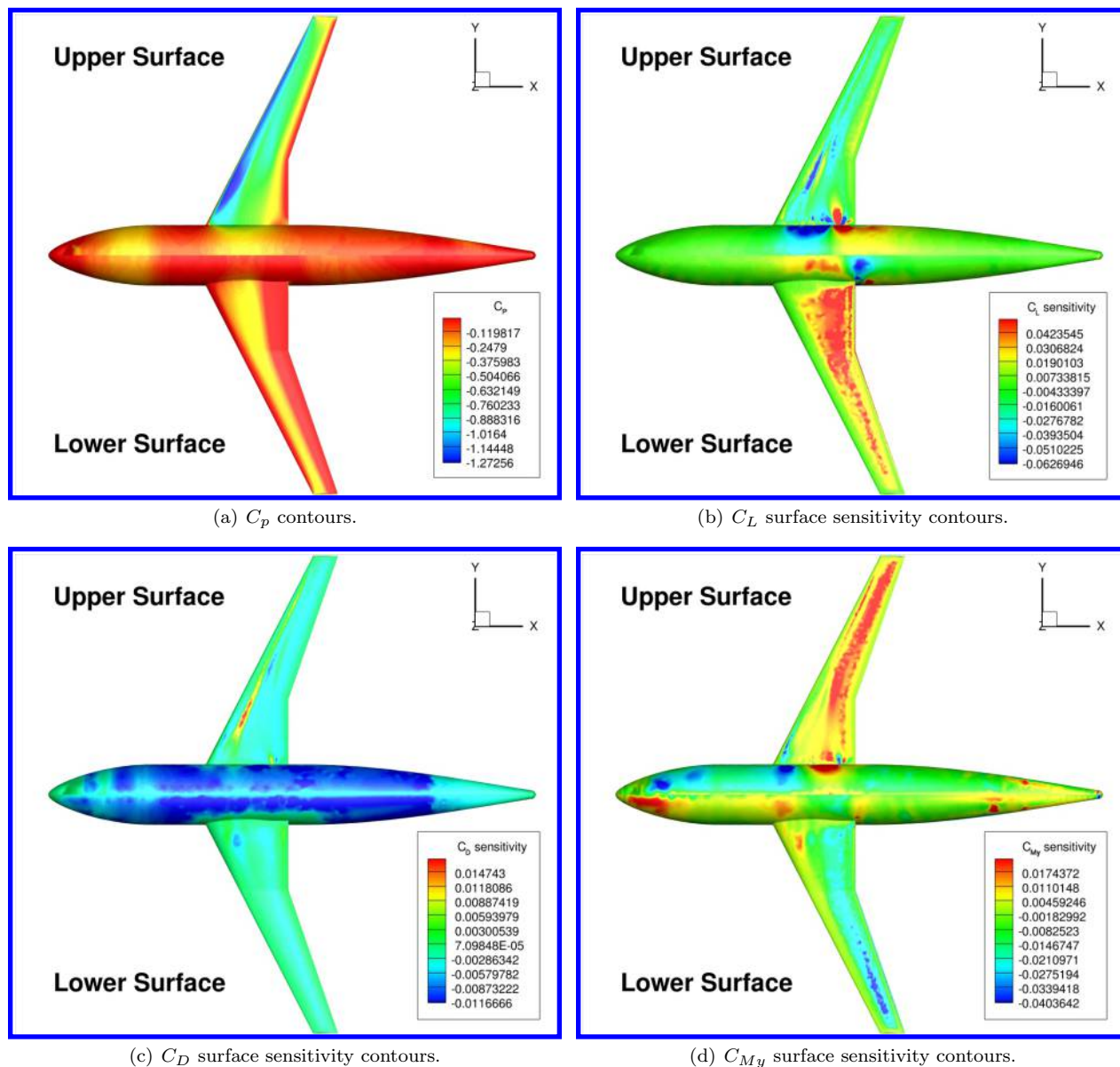


Figure 26. Pressure and surface sensitivity contours on the DLR F6 aircraft geometry (lower and upper surfaces).



## IV. Conclusions

In this work, we have completed a comprehensive V & V process for the RANS flow solver portion of the SU<sup>2</sup> software suite, using both the S-A and SST turbulence models. The validation test cases span a range of flow regimes pertinent to applications of broad interest in both aerospace and mechanical engineering fields. For the selected test cases, SU<sup>2</sup> solutions are shown to be in excellent agreement with both the available experimental data and numerical simulation results from other well-established computational tools developed at NASA. Based on our results, we have established that SU<sup>2</sup> provides accurate, high-fidelity computational simulations for the analysis of complex configurations on unstructured meshes.

Additionally, the flexibility of the class hierarchy in the SU<sup>2</sup> framework permits the straight-forward implementation of new turbulence models and numerical discretization methods. The code structure enables rapid implementations of new models and makes SU<sup>2</sup> an ideal test-bed for scientists in turbulence modeling research fields. Moreover, as the framework is tailor-made for coupled analyses, SU<sup>2</sup> dramatically lowers the implementation barrier for tackling aeroelastic, aeroacoustic, and aerothermodynamic problems of critical interest to the aerospace community.

Lastly, SU<sup>2</sup> is uniquely connected to a global community of researchers and developers in the field of scientific computing for engineering applications. The release of the software under the GNU LGPL (v2.1) has enabled engineers and scientists from around the world to work from a common code-base and provides worldwide access to industry-standard analysis tools. Advances in CFD, shape design, and numerical methods can be rapidly disseminated to a wide, knowledgeable user base in an established, online community.

## V. Acknowledgements

Francisco Palacios would like to acknowledge the support of the U.S. Department of Energy under the Predictive Science Academic Alliance Program (PSAAP). Aniket C. Aranake would like to acknowledge the support of the U.S. Department of Defense (DoD) through the National Defense Science & Engineering Graduate Fellowship (NDSEG) Program. The authors are grateful to Eran Arad, Gérald Carrier, Heather Kline, Shlomy Shitrit, and Saqib Mahmood for their help in the validation of the code.

## References

- <sup>1</sup>F. Palacios, M. R. Colonno, A. C. Aranake, A. Campos, S. R. Copeland, T. D. Economon, A. K. Lonkar, T. W. Lukaczyk, T. W. R. Taylor, and J. J. Alonso. Stanford University Unstructured (SU<sup>2</sup>): An open-source integrated computational environment for multi-physics simulation and design. *AIAA Paper 2013-0287*, 51st AIAA Aerospace Sciences Meeting and Exhibit, January 2013.
- <sup>2</sup>J. Donea, A. Huerta, J.-Ph. Ponthot, and A. Rodriguez-Ferran. *Arbitrary Lagrangian-Eulerian Methods in Encyclopedia of Computational Mechanics*. John Wiley and Sons, 2004.
- <sup>3</sup>C. Hirsch. *Numerical Computation of Internal and External Flows*. Wiley, New York, 1984.
- <sup>4</sup>D.C. Wilcox. *Turbulence Modeling for CFD*. 2nd Ed., DCW Industries, Inc., 1998.
- <sup>5</sup>F.M. White. *Viscous Fluid Flow*. McGraw Hill Inc., New York, 1974.
- <sup>6</sup>P. Spalart and S. Allmaras. A one-equation turbulence model for aerodynamic flows. *AIAA Paper 1992-0439*, 1992.
- <sup>7</sup>F.R. Menter. Zonal two equation  $k - \omega$ , turbulence models for aerodynamic flows. *AIAA Paper*, 93-2906, 1993.
- <sup>8</sup>C. Rumsey and et al. The menter shear stress turbulence model. World Wide Web electronic publication, 2012.
- <sup>9</sup>A. Bueno-Orovio, C. Castro, F. Palacios, and E. Zuazua. Continuous adjoint approach for the Spalart-Allmaras model in aerodynamic optimization. *AIAA Journal*, 50(3), 2012.
- <sup>10</sup>T. D. Economon, F. Palacios, and J. J. Alonso. A viscous continuous adjoint approach for the design of rotating engineering applications. *AIAA Paper 2013-2580*, June 2013.
- <sup>11</sup>W. K. Anderson and V. Venkatakrishnan. Aerodynamic design optimization on unstructured grids with a continuous adjoint formulation. *AIAA Paper 1997-0643*, 1997.
- <sup>12</sup>A. Jameson and S. Kim. Reduction of the adjoint gradient formula for aerodynamic shape optimization problems. *AIAA Journal*, 41(11):2114-2129, 2003.
- <sup>13</sup>C. Castro, C. Lozano, F. Palacios, and E. Zuazua. A systematic continuous adjoint approach to viscous aerodynamic design on unstructured grids. *AIAA Journal*, 45(9):2125-2139, 2007.
- <sup>14</sup>F. Palacios, J. J. Alonso, M. Colonno, J. Hicken, and T. Lukaczyk. Adjoint-based method for supersonic aircraft design using equivalent area distributions. *AIAA Paper 2012-0269*, 2012.
- <sup>15</sup>T. D. Economon, F. Palacios, and J. J. Alonso. Optimal shape design for open rotor blades. *AIAA Paper 2012-3018*, 2012.
- <sup>16</sup>T. D. Economon, F. Palacios, and J. J. Alonso. Unsteady aerodynamic design on unstructured meshes with sliding interfaces. *AIAA Paper 2013-0632*, 51st Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition, January 2013.

- <sup>17</sup>S. Copeland, A. K. Lonkar, F. Palacios, and J.J. Alonso. Adjoint-based goal-oriented mesh adaptation for nonequilibrium hypersonic flows. *AIAA Paper 2013-0552*, 2013.
- <sup>18</sup>F. Palacios, J.J. Alonso, and A. Jameson. Optimal design of free-surface interfaces using a level set methodology. *AIAA Paper 2012-3341*, 2012.
- <sup>19</sup>F. Palacios, J.J. Alonso, and A. Jameson. Design of free-surface interfaces using rans equations. *AIAA Paper 2013-3110*, 2013.
- <sup>20</sup>T. J. Barth. Aspect of unstructured grids and finite-volume solvers for the Euler and Navier-Stokes equations. In *Lecture Notes Presented at the VKI Lecture Series*, 1994 - 05, Rhode Saint Genese Begium, 2 1995. Von karman Institute for fluid dynamics.
- <sup>21</sup>A. Quarteroni and A. Valli. *Numerical approximation of partial differential equations*, volume 23 of *Springer series in computational mathematics*. Springer-Verlag Berlin Heidelberg New York, 1997.
- <sup>22</sup>A. Jameson. A perspective on computational algorithms for aerodynamic analysis and design. *Progress in Aerospace Sciences*, 37:197–243, 2001.
- <sup>23</sup>R. LeVeque. *Finite Volume Methods for Hyperbolic Problems*. Cambridge Univesity Press, 2002.
- <sup>24</sup>P. Wesseling. *Principles of Computational Fluid Dynamics*, volume 29 of *Springer Series in Computational Mathematics*. Springer-Verlag Berlin Heidelberg New York, 2000.
- <sup>25</sup>A. Jameson. Analysis and design of numerical schemes for gas dynamics 1 artificial diffusion, upwind biasing, limiters and their effect on accuracy and multigrid convergence. *RIACS Technical Report 94.15, International Journal of Computational Fluid Dynamics*, 4:171–218, 1995.
- <sup>26</sup>A. Jameson. Analysis and design of numerical schemes for gas dynamics 2 artificial diffusion and discrete shock structure. *RIACS Report No. 94.16, International Journal of Computational Fluid Dynamics*, 5:1–38, 1995.
- <sup>27</sup>E. F. Toro. *Riemann Solvers and Numerical Methods for Fluid Dynamics: a Practical Introduction*. Springer-Verlag, 1999.
- <sup>28</sup>A. Jameson, W. Schmidt, and E. Turkel. Numerical solution of the Euler equations by finite volume methods using Runge-Kutta time stepping schemes. *AIAA Paper 1981-1259*, 1981.
- <sup>29</sup>P. L. Roe. Approximate Riemann solvers, parameter vectors, and difference schemes. *Journal of Computational Physics*, 43:357–372, 1981.
- <sup>30</sup>M.-S. Liou and C. Steffen. A new flux splitting scheme. *J. Comput. Phys.*, 107:23–39, 1993.
- <sup>31</sup>E. Turkel, V. N. Vatsa, and R. Radespiel. Preconditioning methods for low-speed flows. *AIAA*, 1996.
- <sup>32</sup>Y. Saad and M. H. Schultz. GMRES: A generalized minimal residual algorithm for solving nonsymmetric linear systems. *SIAM J. Sci. Stat. Comput.*, 7:856–869, 1986.
- <sup>33</sup>H. A. Van der Vorst. Bi-CGSTAB: A fast and smoothly converging variant of Bi-CG for the solution of nonsymmetric linear systems. *SIAM J. Sci. and Stat. Comput.*, 13(2), 1992.
- <sup>34</sup>A. Jameson. Time dependent calculations using multigrid, with applications to unsteady flows past airfoils and wings. *AIAA Paper 1991-1596*, 1991.
- <sup>35</sup>A. Jameson and S. Schenectady. An assessment of dual-time stepping, time spectral and artificial compressibility based numerical algorithms for unsteady flow with applications to flapping wings. *AIAA Paper 2009-4273*, 2009.
- <sup>36</sup>A. Jameson, L. Martinelli, and F. Grasso. A multigrid method for the Navier-Stokes equations. *AIAA Paper*, 86-0208, 1986.
- <sup>37</sup>D. J. Mavriplis. On convergence acceleration techniques for unstructured meshes. Technical report, Institute for Computer Applications on Science and Engineering (ICASE), 1998.
- <sup>38</sup>D. J. Mavriplis. Multigrid techniques for unstructured meshes. Technical Report 95-27, ICASE, 1995.
- <sup>39</sup>A. Borzi. Introduction to multigrid methods. Technical report, Institut für Mathematik und Wissenschaftliches Rechnen (Karl-Franzens-Universität Graz), 2003.
- <sup>40</sup>F. Palacios and J. J. Alonso. New convergence acceleration techniques in the Joe code. Technical report, Center for Turbulence Research, Annual Research Brief 2011, 2011.
- <sup>41</sup>N.A. Pierce and M.B. Giles. Preconditioned multigrid methods for compressible flow calculations on stretched meshes. *J. Comput. Phys.*, 136:425–445, 1997.
- <sup>42</sup>O. Soto, R. Lohner, and F. Camelli. A linelet preconditioner for incompressible flow solvers. *Int. J. Numer. Meth. Heat Fluid Flow*, 13(1):133–147, 2003.
- <sup>43</sup>S. Yoon and A. Jameson. Lower-Upper Symmetric-Gauss-Seidel method for the Euler and Navier-Stokes equations. *AIAA Journal*, 26(9), 1988.
- <sup>44</sup>A. Jameson and E. Turkel. Implicit schemes and LU-decompositions. *Mathematics of Computation*, 37:385–397, 1981.
- <sup>45</sup>A. Jameson and Y. Seokkwan. Lower-Upper implicit schemes with multiple grids for the euler equations. *AIAA Journal*, 25(7):929–935, 1987.
- <sup>46</sup>C. Rumsey and et al. 2d zero pressure gradient flat plate verification case - intro page. World Wide Web electronic publication, 2013.
- <sup>47</sup>C. Rumsey and et al. 2d bump-in-channel verification case - intro page. World Wide Web electronic publication, 2013.
- <sup>48</sup>D.A Lyn. A laser-Doppler velocimetry study of ensemble-averaged characteristics of the turbulent near wake of a square cylinder. *Fluid Mechanics*, 304:205–332, 1995.
- <sup>49</sup>B. E. Lee. The effect of turbulence on the surface pressure field of a square prism. *Journal of Fluid Mechanics*, 69(02):263–289, March 2006.
- <sup>50</sup>J Vickery. Fluctuating lift and drag on a long cylinder of square n a turbulent stream cross-section in a smooth and i. *Fluid Mechanics*, 25:481–503, 1966.
- <sup>51</sup>W Rodi and M Breuer. Status of Large Eddy Simulation : Results of a Workshop. *Fluid Engineering*, 119(June 1997):248–262, 2007.



- <sup>52</sup>G. Iaccarino, A. Ooi, P.a. Durbin, and M. Behnia. Reynolds averaged simulation of unsteady separated flow. *International Journal of Heat and Fluid Flow*, 24(2):147–156, April 2003.
- <sup>53</sup>N. Gregory and C. L. O'Reilly. Low-speed aerodynamic characteristics of naca 0012 aerofoil sections, including the effects of upper-surface roughness simulation hoar frost. *NASA R & M 3726*, 1970.
- <sup>54</sup>D. Coles and A. J. Wadcock. Flying-hot-wire study of flow past an naca 4412 airfoil at maximum lift. *AIAA*, Vol. 17, No. 4, 1979.
- <sup>55</sup>A. J. Wadcock. Structure of the turbulent separated flow around a stalled airfoil. *NASA-CR-152263*, 1979.
- <sup>56</sup>Christopher L. Rumsey, Thomas B. Gatski, Susan X. Ying, and Arild Bertelrud. Prediction of high-lift flows using turbulent closure models. *AIAA Journal*, 36(5):765–774, 2013/12/11 1998.
- <sup>57</sup>V. Chin, D. Peters, F. Spaid, and R. McGhee. *Flowfield measurements about a multi-element airfoil at high Reynolds numbers*. American Institute of Aeronautics and Astronautics, 2013/12/11 1993.
- <sup>58</sup>S. Kjelgaard, W. Sellers, and R. Weston. Flowfield survey over a 75 deg swept delta wing at an angle of attack of 20.5 deg. *AIAA Paper 1986-1775*, 1986.
- <sup>59</sup>F.X. Caradonna and C. Tung. Experimental and Analytical Studies of a Model Helicopter Rotor in Hover. *NASA Technical Memorandum 81232*, September 1981.
- <sup>60</sup>PH Cook, MCP Firmin, and MA McDonald. Aerofoil RAE 2822 - Pressure Distributions, and Boundary Layer and Wake Measurements. *AGARD Advisory Report No. 138*, 1979.
- <sup>61</sup>JW Slater, JC Dudek, and KE Tatum. The NPARC Alliance Verification and Validation Archive. *NASA Technical Report 2000-209946*, 2000.
- <sup>62</sup>R. Rudnik. Untersuchung der Leistungsfähigkeit von Zweigleichungs-Turbulenzmodellen bei Profilumströmungen. *DLR Forschungsbericht 97-49*, 1997.
- <sup>63</sup>V. Schmitt and F. Charpin. Pressure distributions on the ONERA-M6-wing at transonic mach numbers. Technical Report AGARD, Report AR-138, 1979.
- <sup>64</sup>O. Brodersen and A. Stürmer. Drag prediction of engine-airframe interference effects using unstructured navier-stokes calculations. *AIAA Paper 2001-2414*, 2001.