

SIMULATING TURBULENT FLOWS IN COMPLEX GEOMETRIES

Krishnan Mahesh *

Department of Aerospace
 Engineering and Mechanics
 University of Minnesota
 Minneapolis, Minnesota 55455
 Email: mahesh@aem.umn.edu

George Constantinescu

Center for Integrated Turbulence Simulations
 Department of Mechanical Engineering
 Stanford University
 Stanford, California 94305-3030
 Email:gconstan@ctr.stanford.edu

Parviz Moin

Center for Turbulence Research
 Stanford University
 Stanford, California 94305-3030
 Email:moin@ctr.stanford.edu

ABSTRACT

We discuss development of a numerical algorithm, and solver capable of performing large-eddy simulation (LES) in geometries as complex as the combustor of a gas-turbine engine. The algorithm is developed for unstructured grids, is non-dissipative, yet robust at high Reynolds numbers on highly skewed grids. Results from validation in simple geometries is shown along with simulation results in the exceedingly complex geometry of a Pratt & Whitney gas turbine combustor.

NOMENCLATURE

- A_f face area
- cv1, cv2 cells that have in common a particular face f; the outward normal n points always from cv1 to cv2
- k time level
- l_e length of edge
- n_i components of unit vector \vec{n} normal to the face
- NL convective term in Navier Stokes equations
- u_i velocity components defined at cell centers
- \hat{u}_i predicted velocity components defined at cell centers
- p pressure potential
- V cell volume
- VISC viscous term in Navier Stokes equations
- v_n normal velocity component defined at face centers
- v_t tangential velocity component along the edge in 2D
- \hat{v} predicted normal velocity component defined at face centers
- ϕ scalar defined at cell centers

ω vorticity

INTRODUCTION

The Navier-Stokes equations for incompressible flow are

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j}; \quad \frac{\partial u_i}{\partial x_i} = 0. \quad (1)$$

Large-eddy simulation (LES) is a computational approach where one filters the unsteady Navier-Stokes equations in space, and then numerically solves for the large-scales of motion, while modeling the effect of the filtered scales. Assuming that the spatial filter commutes with the spatial and temporal derivatives, the LES equations for the filtered velocity and pressure are

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j}; \quad \frac{\partial \bar{u}_i}{\partial x_i} = 0. \quad (2)$$

where $\tau_{ij} = \overline{u_i u_j} - \bar{u}_i \bar{u}_j$ is the subgrid stress, and is modeled.

There is considerable general incentive to develop LES on unstructured grids. However, LES on unstructured grids requires development of the appropriate numerical methods. Specifically, the numerical methods used to solve the RANS equations are not directly applicable to LES. RANS typically uses upwinded numerical methods; upwinding provides numerical dissipation, which makes the solution-procedure robust. However, when used for LES, upwinding severely compromises accuracy (e.g. [1]),

*Address all correspondence to this author.

since the numerical dissipation competes with, and often overwhelms the physical dissipation of the subgrid model.

By definition, the dissipative scales are not resolved by the grid in a LES. In practice this leads to numerical instability if straight-forward non-dissipative central-difference schemes are used. One solution to this problem is to develop non-dissipative numerical schemes that discretely conserve not only first order quantities such as momentum, but also second - order quantities such as kinetic energy [2–5]. Discrete energy conservation ensures that the flux of kinetic energy, $\sum_{cvs} u_i \partial(u_i u_j) / \partial x_j$ only has contributions from the boundary faces. This makes the solution robust without the use of numerical dissipation. Note that satisfying one constraint discretely, does not ensure the other - both constraints have to be simultaneously enforced when deriving the algorithm. The Harlow-Welch algorithm [6] possesses this property on *structured* grids, and has therefore been widely used for LES on structured grids in simple geometries. We have developed a numerical method suitable for LES on unstructured grids that has been implemented on massively parallel computers [7–9]. The development was evolutionary and is summarized below. First, the Harlow-Welch formulation was generalized to unstructured grids using a rotational form of the convection terms. While elegant, this formulation was found lacking when extended to three-dimensions. In particular its restriction to tetrahedral elements, and lack of robustness on skewed grids were serious limitations. An alternative formulation was therefore derived which can be applied to arbitrary elements. Robust yet accurate solutions are now obtained at high Reynolds numbers in very complex geometries on highly skewed grids. Details follow.

STAGGERED ROTATIONAL FORMULATION

We describe below the spatial discretization of the two-dimensional incompressible Navier-Stokes equations on an unstructured grid of triangles. The formulation is a conceptual extension of the popular staggered formulation on structured grids [6]. Figure 1 shows a single triangular element. Note that the pressure is stored inside the element (at the circumcenter), while the velocities normal to the edges of the triangle are stored at the edge-centers.

Denoting the edge-normal velocity by v_n , we have:

$$\frac{\partial v_n}{\partial t} - (\vec{u} \times \vec{\omega}) \cdot \vec{n} + \frac{\partial}{\partial n} \left(\frac{\vec{u} \cdot \vec{u}}{2} \right) = -\frac{1}{\rho} \frac{\partial p}{\partial n} + \nu \left(\nabla^2 \vec{u} \right) \cdot \vec{n} \quad (3)$$

We have $\nabla^2 \vec{u} = -\nabla \times \vec{\omega} + \nabla (\nabla \cdot \vec{u})$. Also in two dimensions, $(\vec{u} \times \vec{\omega}) \cdot \vec{n} = \omega v_t$.

If the velocity field is divergence-free, this implies that $(\nabla^2 \vec{u}) \cdot \vec{n} = -\frac{\partial \omega}{\partial s_t}$, where $\frac{\partial}{\partial s_t}$ denotes the tangential derivative. To time-advance the momentum equation, we therefore need the

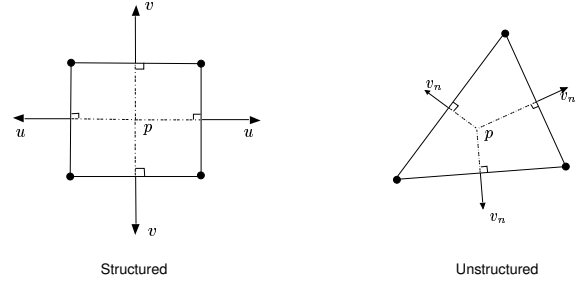


Figure 1. THE STAGGERED POSITIONING OF VARIABLES ON AN UNSTRUCTURED MESH OF TRIANGLES IS CONTRASTED WITH THAT ON A STRUCTURED MESH.

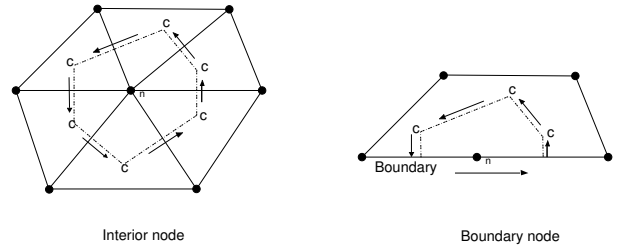


Figure 2. ILLUSTRATION OF THE DUAL MESH USED TO COMPUTE THE VORTICITY AT THE NODES.

vorticity at the nodes and the edge-centers, tangential velocity at edge centers, total kinetic energy and pressure at the cell-centers.

The vorticity at the nodes is computed using Green's theorem i.e. $\int_A \omega dA = \int_C \vec{u} \cdot d\vec{l}$. The area over which the integration is performed is shown in Fig. 2, and is obtained by joining the circumcenters of the triangles that surround the node. It is easily seen that the velocity components parallel to the edges of this area are normal to the edges of the triangles that make up the area. The vorticity at the node 'n' is therefore computed as $\omega = \frac{1}{A_{dual}} \sum_{edges} v_n l_{e-dual}$, where the length of the edges of the dual mesh is the distance between the circumcenters of the triangles that constitute the dual mesh.

Once the nodal vorticities are known, the vorticity at the edge-centers may be computed as the average of the vorticity at the corresponding nodes. Also the tangential derivative of the vorticity at the edge-center may be computed as the difference of the nodal vorticities divided by the edge length. These approximations are second-order accurate in the edge length. The tangential velocities at the edges are obtained by interpolating from neighboring edges. A fractional step approach for the pressure is derived as follows. Denote the nonlinear terms by NL and the viscous terms by V , and use the Adams-Bashforth method for both. Ignoring the pressure p in the first fractional step, we get

$$\frac{\hat{v}_n - v_n^k}{\Delta t} = \frac{1}{2} \left[3(NL + VISC)^k - (NL + VISC)^{k-1} \right] \quad (4)$$

$$\frac{v_n^{k+1} - \hat{v}}{\Delta t} = -\frac{\partial p^{k+1}}{\partial n} \quad (5)$$

We require that the divergence of the velocity field at t_{k+1} be zero; i.e. $A_v \nabla \cdot \vec{u} = \sum_e v_{ne} l_e = 0$. Equation (5) therefore implies that

$$\frac{1}{\Delta t} \sum_e v_n^{k+1} l_e - \frac{1}{\Delta t} \sum_e \hat{v} l_e = -\sum_e \frac{\partial p^{k+1}}{\partial n} l_e \quad (6)$$

which along with the continuity constraint requires that the pressure p (more exactly the pressure potential) satisfy

$$\sum_e \frac{\partial p^{k+1}}{\partial n} l_e = \frac{1}{\Delta t} \sum_e \hat{v} l_e \quad (7)$$

Since p is stored at the circumcenter of the volume, its gradient normal to the edge is easily computed. This yields a set of discrete equations for p that are then solved. These discrete equations are of the form,

$$c(i)\phi(i) + \sum_{j=1}^3 c(i,j)\phi(j) = \text{rhs}(i) \quad (8)$$

where the sum over j is a sum over the three volumes (faces in 2D) that are neighbors of volume, 'i'. Equation (8) is easily written for the interior volumes. However, when it is applied to a boundary volume, it appears that the gradient of p at the boundary is required. However this is not the case [10]; this requirement can be circumvented as follows. The divergence-free condition requires that

$$\sum_{\text{edges of CV}} v_{ne} l_e = 0; \text{ i.e. } \sum_{\text{interior edges}} v_{ne} l_e = -v_{nb} l_{eb}. \quad (9)$$

where the subscript 'b' refers to the boundary faces. This implies that the pressure equation for the boundary elements may be obtained by summing Eqn. (8) over the interior edges alone, and using Eqn. (9) to relate the interior sum of the velocity to the normal velocity at the boundary; i.e.

$$\sum_{\text{interior edges}} \frac{\partial p^{k+1}}{\partial n} l_e = \frac{1}{\Delta t} \sum_{\text{interior edges}} \hat{v} l_e + \frac{1}{\Delta t} v_{nb} l_{eb}. \quad (10)$$

This eliminates the need for boundary conditions on p .

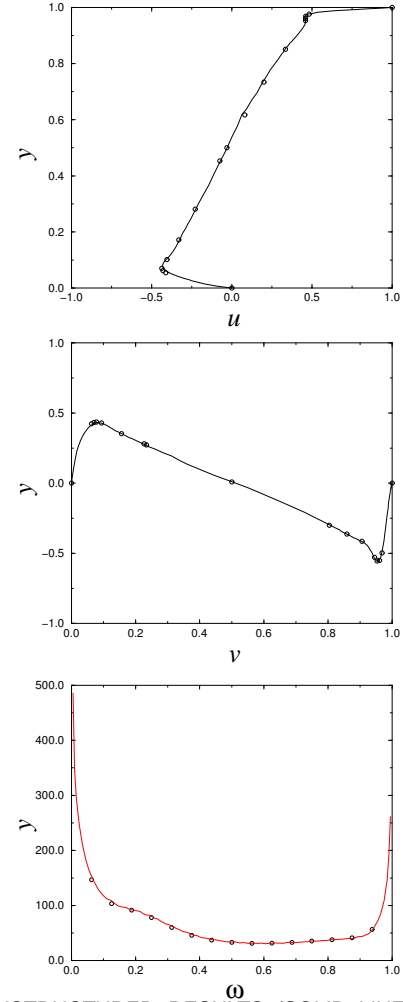


Figure 3. UNSTRUCTURED RESULTS (SOLID LINES) ARE COMPARED TO RESULTS FROM GHIA ET AL. (symbols). THE REYNOLDS NUMBER IS 5,000. (a) VERTICAL PROFILE OF STREAMWISE VELOCITY AT $X/D = 0.5$ (b) STREAMWISE PROFILE OF VERTICAL VELOCITY COMPONENT AT $Y/D = 0.5$ (c) VORTICITY AT THE TOP BOUNDARY.

Validation

We consider the steady laminar flow in a two-dimensional driven cavity at a Reynolds number of 5,000. Results from [11] are used for validation in Fig. 3. The quantities validated include velocity profiles and the vorticity at the center of the primary vortex. The unstructured grid is seen to allow fewer points to be used; at a Reynolds number of 5,000, comparable results were obtained using 10576 nodes (30,925 triangles) as compared to 66,049 nodes (65,536 Cartesian elements) in Ghia et al's computations. Very good agreement with Ghia's results is observed. No attempt was made to optimize the unstructured grids.

Extension to three-dimensions

The algorithm is extended to tetrahedral elements as follows. The pressure and any scalars are stored at the circumcenter of the tetrahedron. The velocity component normal to each face, v_n is stored at the circumcenter of each face. The convection term is computed in rotational form as $(\vec{u} \times \vec{\omega}) \cdot \vec{n} = v_t \cdot (\omega_t \times \vec{n})$. The tangential velocity component v_t is obtained by interpolating from the neighboring faces, and there are two steps to obtaining the vorticity components in the plane of each face. First, the circulation theorem is invoked to obtain the vorticity along each edge of the face. This vorticity is then projected along the tangential basis vectors on the face, and averaged to obtain ω_t at the face circumcenter. The circulation theorem is applied on a closed circuit around each edge. This circuit is obtained by joining the circumcenters of the tetrahedra to which this edge belongs. This is made possible by the fact that the property of the circumcenter that all such segments will lie in the same plane. The circulation theorem $A_{\text{dual}} \omega = \sum_{\text{dual edges}} v_n l_e$ yields the vorticity component along the edge. The edge-vorticities are interpolated to obtain the tangential vorticity components at the face circumcenter. The viscous term is obtained from the edge-vorticities using the identity, $A_f (\vec{\nabla} \times \vec{\omega}) \cdot \vec{n} = \sum_e \omega_e l_e$, where ω is the vorticity component along the edges previously computed, and l_e denotes the length of the edges of the face.

Problems with formulation

While elegant, the above formulation has some limitations. It is restrictive in that pressure (and scalars if any) is stored at the circumcenter of the triangular elements. This restricts the grid to elements whose circumcenter lies within them. For example, in two-dimensions, consider right-triangles. Their circumcenter lies on the hypotenuse, making it impossible to determine the pressure gradient normal to the hypotenuse. Highly skewed elements are another source of problem since the circumcenter will now lie outside the element. Although projection of the velocity field is still possible in this situation, the inaccurate computation of the pressure gradient is cause for concern.

Another more fundamental limitation concerns the restriction of the algorithm essentially to tetrahedra. While well-suited to grid very complex geometries, experience shows that hexahedral elements are better suited for unsteady computations since (i) hexahedral elements are more easily aligned with flow gradients such as boundary layers, and (ii) it takes fewer hexahedral elements to fill space than comparable tetrahedra. For example, a three-dimensional grid generated for a Pratt and Whitney gas-turbine combustor required a tetrahedral grid with approximately 600,000 nodes and 6.35 million faces (normal velocities to be solved) while a hexahedral grid with only 1.5 million nodes and faces yielded significantly higher resolution near the walls.

NON-STAGGERED FORMULATION

An alternative approach was therefore derived. The basic idea is that the robustness at high Reynolds numbers is determined essentially by the convection term, while robustness on skewed grids is determined by both convection and the pressure gradient term. A formulation is derived that emphasizes energy conservation for the convection term on arbitrary grids. Accordingly, the cell velocities u_i and the face-normal velocities v_n are treated as essentially independent variables. This storage is similar to that used in [12] to suppress odd-even decoupling in a colocated formulation. Then, one can construct a formulation in which the convective term is discretely energy conserving. The concept can be illustrated for a scalar ϕ defined at the cell centers. The velocity components u_i will play the role of the scalar ϕ . It is readily shown that if the passive scalar ϕ satisfies the equation

$$\frac{\partial \phi}{\partial t} + \frac{\partial}{\partial x_i} \phi u_i = 0 \quad (11)$$

then ϕ^2 satisfies the same equation

$$\frac{\partial \phi^2}{\partial t} + \frac{\partial}{\partial x_i} \phi^2 u_i = 0. \quad (12)$$

if the velocity field is divergence-free. We would like to have the same property for the discrete case. Integrating Eqn. (12), we get $V \frac{d\phi^2}{dt} + \sum_{\text{faces}} \phi_f v_n A_f = 0$ where V denotes the cell volume. This discrete equation has the same conservative property for ϕ^2 if $\phi_f = (\phi_{cv1} + \phi_{cv2}/2)$ and $\sum_{\text{faces of cv}} v_n A_f = 0$. Here cv1 and cv2 are the two cells that shear a particular face. This observation is extended to the Navier-Stokes equations by computing the convection term in a similar manner. Note that the symmetry of the interpolation is retained even on non-uniform grids. This makes the solution robust on skewed grids since the discretization does not 'see' the underlying rough grid. As the results will show, this robustness results in no visible degradation of accuracy.

The convection term computed in the above manner yields a predicted value for the cell velocities \hat{u}_i . These predicted velocities are interpolated to obtain a prediction for the face-normal velocities \hat{v} . The face-normal velocities are projected using $v_n - \hat{v} = -\frac{\partial p}{\partial n}$ and the pressure (p is in fact the pressure multiplied by the time step) is obtained from the discrete Poisson equation that results. The gradient of p at the cell-centers is then used to update the cell velocities; i.e. $u_i = \hat{u}_i - \frac{\partial p}{\partial x_i}$.

It turns out that the details of how $\frac{\partial p}{\partial x_i}$ are computed, affect the robustness of the solution on highly skewed grids. An obvious approach to computing the gradient at cell-centers is to use the gradient theorem; $\frac{\partial p}{\partial x_i} = \frac{1}{V} \sum_{\text{faces}} p_f A_f n_i$. When applied to flows such as homogeneous turbulence, turbulent channel flow

and even a coaxial combustor for which the grids are very regular, very accurate results are obtained. However when applied to highly skewed grids such as those in the Pratt & Whitney combustor, unstable solutions are obtained. This behavior is found at both high and low Reynolds numbers, pointing to the pressure-gradient as the source of the problem. It is readily seen that for a staggered formulation the pressure term in the discrete kinetic energy equation, in which we performed summation over all the cells of the computational domain, can be written as:

$$-\sum_{CV} \sum_{faces\ of\ cv1} v_n A_f (p_{cv1} - p_{face}) = \sum_{CV} \sum_{faces\ of\ cv1} v_n A_f p_f - \sum_{CV} p_{cv1} \sum_{faces\ of\ cv1} v_n A_f$$

The second term in the last expression is zero due to the continuity equation that is satisfied in discrete form, while in the first term we are left only with contributions from the boundary faces, as for the interior faces each face is counted twice, once as a face of cv1 and once as a face of cv2, v_n changes sign and p_f is just a scalar. So the pressure term contains only contributions from the boundary faces:

$$\sum_{boundary\ faces} v_n A_f p_{face} \quad (13)$$

However in the collocated formulation this is not necessarily true,

$$u_i - \hat{u}_i = -\frac{\partial p}{\partial x_i} \Rightarrow \sum_{cv} u_i \sum_{faces\ of\ cv1} \left(\frac{p_{cv1} + p_{cv2}}{2} \right) n_i A_f = \sum_{cv} \frac{u_i}{2} \sum_{faces\ of\ cv} p_{cv2} A_f n_i$$

as the last term does not reduce to summation over the boundary cells. This is because pressure is not obtained by taking the discrete divergence of the above equation; doing so will result in odd-even decoupling. As a result, $\frac{1}{2} \left(\frac{\partial p_{cv1}}{\partial x_i} + \frac{\partial p_{cv2}}{\partial x_i} \right) n_i \neq \frac{\partial p}{\partial n}$ at the faces. The pressure gradient term is therefore no longer discretely energy conserving. We therefore developed the following solution. Once p is known, $\frac{\partial p}{\partial x_i}$ is computed such that for each cell, $\frac{\partial p}{\partial x_i} n_i$ at the faces equals $\frac{\partial p}{\partial n}$ in a least squares sense. The resulting algorithm is found to be both robust on highly skewed grids as well as accurate.

Results

Flow over a circular cylinder The flow over a circular cylinder is chosen as an example of external flow. DNS was per-

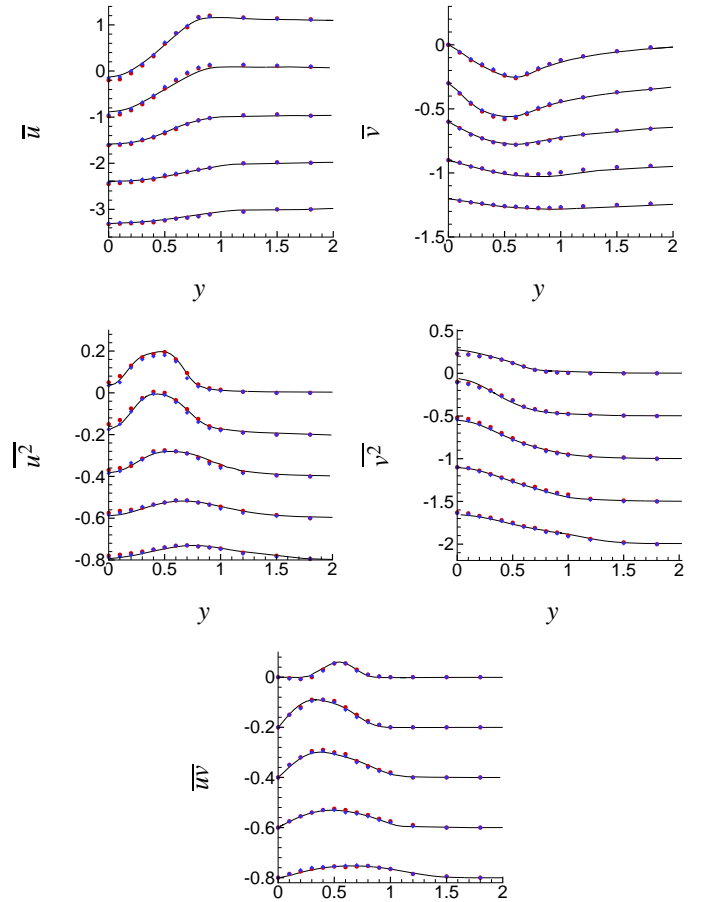


Figure 4. VERTICAL PROFILES AT STREAMWISE STATIONS BEHIND A CYLINDER AT $Re = 300$. THE UNSTRUCTURED SOLUTIONS (—) ARE COMPARED TO THE B-SPLINE-FOURIER SIMULATIONS OF KRAVCHENKO AND MOIN (\circ) AND SPECTRAL SIMULATIONS OF MITTAL AND BALACHANDAR (\diamond).

formed at a cylinder Reynolds number of 300, and LES was performed at a Reynolds number of 3,900. Only the $Re = 300$ DNS is shown here in the interest of brevity. An unstructured grid of 1.2 million hexahedral control volumes was generated. The spanwise domain was πD in extent, and was spanned by 32 volumes. The unstructured capability was used to cluster points in the boundary layer and the wake. The simulation results (mean velocity and components of the Reynolds stress tensor at different sections downstream of the cylinder) are compared in Fig. 4 to the B-spline-Fourier computations by Kravchenko and Moin [14], and the spectral computations of Mittal and Balachandar [13] and good agreement is observed.

Pratt & Whitney gas turbine combustor Simulations were performed in the exceedingly complex geometry of a combustor corresponding to a Pratt & Whitney gas-turbine engine. The geometry was provided in IGES format. Experimental data for mass-splits and pressure drop across the injector was used for validation. The Reynolds number in the pre-diffuser based on the bulk velocity and cross-section was 500,000; values in the main (core) swirler channel were 150,000. Turbulent fluctuations from a separate calculations in a pipe sector of identical shape as the pre-diffuser inlet section were fed at the inlet. Figure 5 shows the very complex flow pattern inside the main combustion chamber due to the interactions among the swirling jets exiting the injector and the jets entering the combustion chamber through the inner and outer dilution holes. Table 1 compares LES predictions of the flow splits to experiment; good agreement is observed.

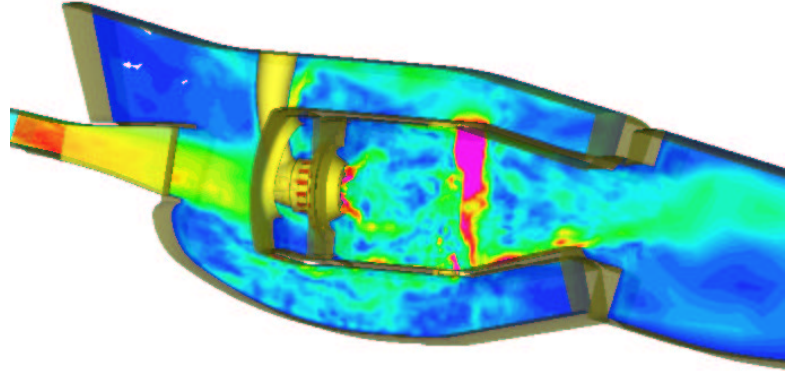


Figure 5. INSTANTANEOUS VELOCITY MAGNITUDE CONTOURS IN THE COMBUSTOR GEOMETRY.

Location	LES Error % wrt expt.	LES Error % wrt inlet
OD dilution hole	3.1	0.8
ID dilution hole	3.5	0.5
Core (main swirler)	10.3	0.14
Second (OD) swirler	7.5	0.35
Third (Guide) swirler	0.4	0.02

Table 1. COMPARISON TO EXPERIMENT OF MASS FLOW SPLITS IN THE PRATT & WHITNEY COMBUSTOR GEOMETRY.

Summary

We have discussed development of a numerical algorithm, and solver capable of performing large-eddy simulation (LES) in geometries as complex as the combustor of a gas-turbine engine. The algorithm is developed for unstructured grids, has non-dissipative convection, and yet is robust at high Reynolds numbers on highly skewed grids. Results from validation in simple geometries is shown along with simulation results in the exceedingly complex geometry of a Pratt & Whitney gas turbine combustor.

ACKNOWLEDGMENT

We are immensely grateful to Mr. Gianluca Iaccarino for his assistance in generating the grid used in the combustor simulations and to Dr. Sourab Apte for useful discussions. This work was supported by the DOE under the Accelerated Strategic Computing Initiative at Stanford University.

REFERENCES

- [1] Mittal, R., Moin, P., 1997. "Suitability of upwind biased schemes for large-eddy simulation". *AIAA J.*, **30** (8), pp. 1415–1417.
- [2] Arakawa, A., 1966. "Computational design for long term numerical integration of the equations of fluid motion: two-dimensional incompressible flow, Part I". *J. Comp. Phys.*, **1**, pp. 119–143.
- [3] Fromm, J.E., Harlow, F.H., 1963. "Numerical solution of the problem of vortex street development". *Phys. Fluids*, **6**, pp. 175–182.
- [4] Lilly, D.K., 1965. "On the computational stability of numerical solutions of time-dependent non-linear geophysical fluid dynamics problems". *Monthly Weather Review*, **93**, pp. 11–26.
- [5] Mansour, N.N., Moin, P., Reynolds, W.C., Ferziger, J.H., 1979. "Improved methods for large-eddy simulation for turbulence". In *Proceedings of Turbulent Shear Flows I*, F. Durst, B.E. Launder, F.W. Schmidt and J.H. Whitelaw, Eds., Springer, Berlin, pp. 386–401.
- [6] Harlow, F.H., Welch, J.E., 1965. "Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface". *Phys. Fluids*, **8**, pp. 2182–2189.
- [7] Mahesh, K., Ruetsch, G.R., Moin, P., 1999. "Towards large-eddy simulation in complex geometries". *Annual Research Briefs - 1999*, Center for Turbulence Research, Stan-

- ford University and NASA-Ames, pp. 379–387.
- [8] Mahesh, K., Constantinescu, G., Moin, P., 2000. “Large-eddy simulation of gas-turbine combustors”. Annual Research Briefs - 1999, Center for Turbulence Research, Stanford University and NASA-Ames, pp. 219–228.
 - [9] Mahesh, K., Constantinescu, G., Apte, S., Iaccarino, G., Moin, P., 2001. “Large-eddy simulation of gas-turbine combustors”. Annual Research Briefs - 1999, Center for Turbulence Research, Stanford University and NASA-Ames, pp. 3–17.
 - [10] Moin, P., Kim, J., 1980. “On the numerical solution of time-dependent viscous incompressible flows involving solid boundaries”. *J. Comp. Phys.*, **35**, pp. 381–392.
 - [11] Ghia, U, Ghia, K.N., Shin, C.T., 1982. “High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method”. *J. Comp. Phys.*, **48**, pp. 387-411.
 - [12] Rhie, C.M., Chow, W.L., 1983. “A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation”. *AIAA J.*, **21**, pp. 1525–1532.
 - [13] Mittal, R., Balachandar, S., 1997. “On the inclusion of three-dimensional effects in simulation of two-dimensional bluff body wake flows”. In Proceedings of ASME Fluids Engineering Division Summer Meeting, Vancouver, British Columbia, Canada, June 22-26, 1997.
 - [14] Kravchenko, A., Moin, P., 1998. “B-spline method and zonal grids for numerical simulation of turbulent flows”. Report TF - 73, Mechanical Engineering Dept., Stanford University, Stanford, California.