Computation of Turbulence with Stabilized Methods

Kenneth Jansen [1]

1 Dept. of Mech. Engr., Aero. Engr. & Mech.
Scientific Computation Research Center (SCOREC)
Rensselaer Polytechnic Institute, Troy, New York, USA
jansen@rpi.edu

1. INTRODUCTION

Large-eddy simulation (LES) has emerged as a useful tool in the computation of turbulence. Important building blocks have recently been put into place making possible a careful study of LES on unstructured grids. The first key step was an efficient mesh generator which allowed the placement of points according to smooth variation of physical length scales. This length scale variation is in all three directions independently, which allows a large reduction in points when compared to structured-grid methods, which can only vary length scales in one direction at a time. The second step was the development of a dynamic model appropriate for unstructured grids. The principle obstacle was the development of an unstructured-grid filtering operator. New filtering operators were developed in Jansen [1]. These filters have been implemented into a highly parallelized finite element code (see Jansen *et al.* [2] and Johan *et al.* [3]).

Within this paper we will briefly describe the finite element formulation and the salient features leading to efficient parallel implementation. We also give a description of the extension of the dynamic sub-grid model to unstructured-grid simulations. After a cursory description of mesh generation techniques, results of past and current simulations will be provided to illustrate the efficiency and the accuracy of the approach. Results will be presented both from simple validation problems and from the very challenging simulation of flow over an airfoil at maximum lift.

There are many finite element and finite volume formulations available for application with unstructured grids. However, few have had their accuracy and stability as carefully scrutinized to the degree of what have become known as stabilized finite element methods. Two important members of this family, Galerkin/Least-Squares (GLS) and Streamline Upwind Petrov Galerkin (SUPG), have been proven stable and higher order accurate (converging at the optimal rate for a given function space) for the full range of flows, inviscid to viscous dominated, for multi-dimensional linear advective-diffusive systems (see Hughes *et al.* [4] and Hughes *et al.* [5]) and the frozen coefficient Navier-Stokes equations (see Franca and Hughes [6]).

A semi-discrete formulation is used requiring the introduction of a time integrator to relate the time derivative to the solution variable, yielding a non-linear matrix problem to be solved at each time step. An implicit, second order accurate family of time integrators has been developed. The family is characterized by one free parameter, the amplification factor as the time step tends to infinity (ρ_∞), which varies from 0 (Gears Method) to 1 (Midpoint Rule). The non-linear matrix problem is solved in a predictor-corrector format yielding successive linear problems. Each linear problem is solved using the Matrix-Free Generalized Minimal RESidual (MF-GMRES) solution technique. Convergence of the non-linear problem is confirmed before moving to the next time step.

2. ACCURACY VERIFICATION

Finite element methods are relatively new to the computation of turbulence. Most applications thus far have been to steady, laminar and Reynolds averaged flows. Until recently their application to unsteady turbulent flows was deemed too expensive and not sufficiently accurate. This assessment is puzzling given that the methods can be proven higher order accurate and that the unstructured grid can lead to point reductions which more than offset the additional work associated with the method. In fact the inclusion of a consistent mass matrix leads to a super-convergent phase accuracy in regions where the mesh is uniform or slowly varying. This super-convergence is the result of the rational approximation (Padé like) and can be illustrated by studying the growth of a small disturbance with the shape of the most unstable eigenmode of parallel channel flow. Here we repeat the flow conditions studied by Malik *et al.* [7] ($Re = 7500, \alpha_r = 1$). The growth of the disturbance energy within the Navier-Stokes code (no subgrid-scale model) can be compared to linear stability theory to test the accuracy of the numerical method.

In [8] the growth rates of our formulation were compared to that obtained by central differences (CD) on a staggered grid (the workhorse of the LES community). There, the FEM was shown to converge almost two refinement levels ahead of the FDM. This result clearly challenges the notion that kinetic energy conservation (a property of CD-FDM not enjoyed by stabilized FEM) is paramount since, at least in this problem, accurate interaction of the disturbance with the mean flow is more critical than the very small additional dissipation generated by the stabilization operator.

3. PARALLEL IMPLEMENTATION

The original implementation of this approach was on a Thinking Machines CM5 following the so-called data parallel approach. On the CM5 the gather/scatter operations are performed using CMSSL library procedures requiring no preprocessing of the data. This made application to this platform quite easy. Unfortunately, other parallel platforms do not have libraries of this type. For this reason a second version of the code was developed, employing the Message Passing Interface (MPI) to do the gather and scatter operations. This version of the code requires that the problem be pre-processed to break the computational domain into n pieces where n is the number of processors, on which, the problem will be run. This preprocessing step is non-trivial for unstructured grids since load balance (equal distribution of the elements onto each processor) and minimal communication are necessary. Recently we have accomplished this preprocessing procedure and have begun testing our MPI version of the code on a variety of platforms (IBM-SP2, SGI-Origin, SUN-Sparc and Cray J90). The algorithm has shown perfect scalability on moderate sized problems (typically the bigger the better) due to the low ratio of communication to calculation that is inherant in finite element methods.

4. DYNAMIC MODEL DEVELOPMENT

To close the filtered Navier-Stokes equations an expression for the subgrid-scale viscosity μ_T must be given. In the early nineties a dynamic procedure for determining this viscosity was developed by Germano *et al.* [9]. The model is based on a comparison of results of applying a subgrid-scale model (most applications use Smagorinsky) at two different grid resolutions. Usually, one grid resolution is chosen to be that of the calculation and the other is obtained by filtering the solutions obtained from the calculation. The filtering process is carried out through the convolution of the function with a filtering kernel. Applications of this model flourished within the spectral methods community and the finite difference community where it gained wide acceptance. Use of this model on unstructured grids required an extension of the filtering operator. A variety of filtering operators were developed in [1]. To date the most successful one in applications has been the simplest one, the generalized tophat filter. In this approach the filter function for each nodal value is taken to be one on the domain formed by elements touching that node.

5. MODEL VALIDATION

To verify that both the numerical method and the subgrid-scale model are appropriate for large eddy simulation two canonical flows have been simulated; decay of isotropic turbulence and turbulent channel flow [8]. Here, we only have space to briefly discuss the isotropic turbulence studies. We have intentionally used very course grids to get an indication of the models performance with the resolution expected to be attainable in the more complex flows for which these methods are directed. It is worth pointing out that these flows do not have adequate length scale variation to give unstructured grids the point reduction advantage that we see in more complex flows.

5.1 Decay of Isotropic Turbulence

A common validation test for large-eddy simulation codes is the isotropic homogeneous turbulence decay. The simulation was run on a 32^3 periodic grid and is designed to replicate the results of the experiment conducted by Comte-Bellot & Corrsin [10]. To accurately match the experiment, the initial velocity field of the simulation must match the *filtered* spectrum given at the first experimental station, corresponding to $t = 42$ with the conventions of the reference. Then, the actual decay is simulated and results are compared to the experimental data given at $t = 98$ and 171. During this computation, the time step is constant with 45 time steps per eddy turnover time at the first experimental station ($t = 42$).

Figure 1 shows the decay of the kinetic turbulent energy obtained with the dynamic model, with the Smagorinsky coefficient fixed at $C_s = 0.17$, or when no subgrid-scale model is used. The variations observed are important since they show that the numerical dissipation does not dominate the subgrid-scale model. Though the fixed-coefficient Smagorinsky model seems to provide even better results, the velocity spectra, presented in on the right in Figure 1, show that this would be a an errant conclusion since the dynamic model represents the shape of the spectrum much better while the fixed coefficient simply benefits from error cancellation. Finally the spectrum obtained with no model is flatter, which indicates that the

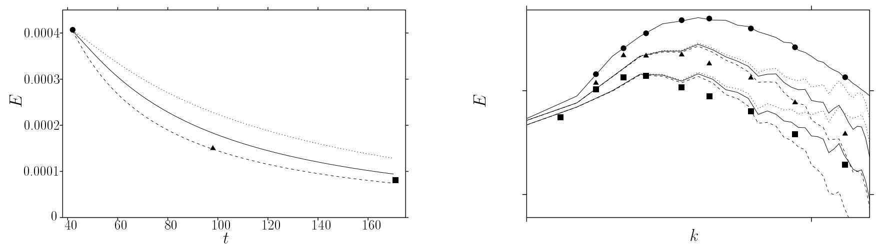

Figure 1. Decay of isotropic turbulence on a 32^3 grid compared to the experimental data of [10]. On the left is the decay of resolved kinetic turbulent energy; on the right the filtered velocity spectra (energy vs. wavenumber) —— Present dynamic model; $----$ fixed coefficient Smagorinsky model ($C_s = 0.17$); $\cdots\cdots$ no model. $\bullet$, $\blacktriangle$, $\blacksquare$ Experiment at $t = 42$, 98 and 171 respectively.

numerical dissipation inherent to the scheme is not dominant.

5.2 Airfoil Simulation

As mentioned earlier, the real promise of this method is its application to more complex flows where there is a large variation in length scales requiring resolution. Here, unstructured grids can refine or coarsen to match the physical length scale requirements of a particular region rather than letting the length scale of one region determine the grid requirements for the entire problem. While the same can be done, to some extent by overset or Chimera grids, for many flows there is a gradual and continuous change in length scales making this approach impractical.

Though the geometric complexity remains mild ("2D" prismatic body with the only complexity arising from a sawtooth transition strip and wind tunnel walls), the length scales requiring resolution vary from one region of the flow to another by over two orders of magnitude. This variation is due to the growth of an extremely thin boundary layer, present near the leading edge, to a massively separated flow near the trailing edge. By adapting the grid size to the local resolution requirements (through user interaction), the unstructured grid achieves a 27-fold point reduction over structured grids attempting to achieve the same resolution. The adaptivity is carried out both when moving from the leading edge to the tail and in layers moving away from the airfoil surface as can be seen in Figure 2. Here, the grid on four planes parallel to the surface is shown. Note the smooth gradation in element size with increasing distance from the surface. The contours of the streamwise velocity are shown on the same planes. The grid has been made to match the size of the structure and it changes smoothly in successive layers as can be seen in the back cross-sectional plane. Note also that the unstructured grid has been used to represent the serrated trip strip used in the experiment (first two layers pass through the trip strip). Structured grid simulations of the same flow have proven too costly due to the 27-fold point increase [11].

6. CONCLUSIONS AND FUTURE DIRECTIONS

The simulations performed to date have served as a vehicle for development of more efficient techniques. To this end we have begun development of new, higher-

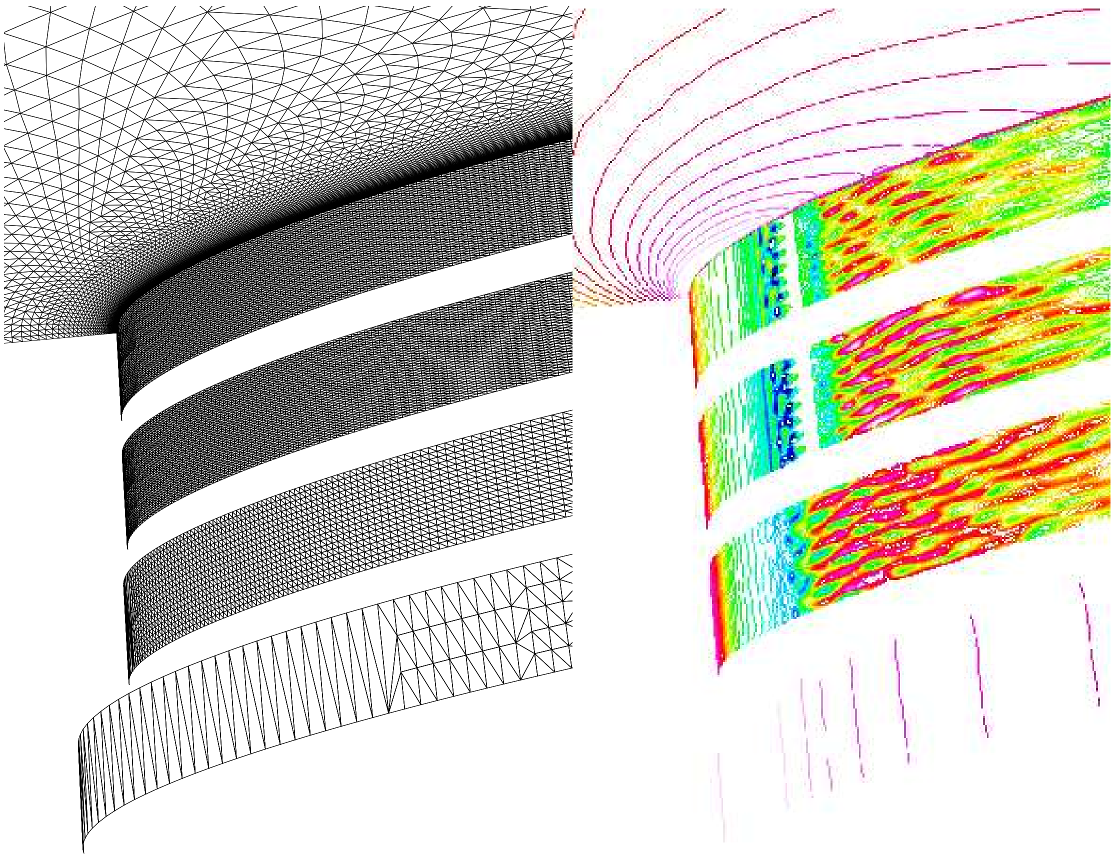

Figure 2. On the left, an example of a "hand-crafted" mesh which meets the smoothness requirements in the turbulent regions is shown. On the right, the contours of the streamwise velocity are shown on the same planes. Here, light contours denote high speed while dark contours denote low speed.

order methods which employ hierarchical basis functions to add p adaptivity to the already present h adaptivity [12]. Their implementation into the Navier-Stokes code is complete and they are currently being applied to turbulent flows.

Other simulations are also being carried out with this code though the space remaining here allows only a brief mention of them. Bastin [13] is using the code to compute the noise from a supersonic jet ($M = 1.4$, $Re_D = 1.5 \times 10^6$). We have also recently started a simulation of turbulent flow over a cavity ($Re_H = 27000$, $Re_\theta = 5100$) with the goal of predicting/altering the acoustic resonance which sometimes occurs in this geometry.

7. ACKNOWLEDGMENTS

The author wishes to acknowledge the collaboration of Francois Bastin, S. Scott Collis, Juin Kim, Christian Whiting, Parviz Moin, Thomas Lund, Hans Kaltenbach, Rajat Mittal, and Massimiliano Fatica and others at the Center for Turbulence Research (CTR). This work has been supported by the AFOSR and by CTR (a joint NASA-Stanford University program). It has also enjoyed computer resources from the Army High Performance Computing Research Center (AHPCRC) and the

Naval Research Lab (NRL).

REFERENCES

[1] K.E. Jansen, "Unstructured grid large eddy simulation of flow over an airfoil", in *Annual Research Briefs*, Center for Turbulence Research, NASA Ames / Stanford University, (1994) 161–173.

[2] K.E. Jansen, Z. Johan, and T. J. R. Hughes, "Implementation of a one–equation turbulence model within a stabilized finite element formulation of a symmetric advective–diffusive system", *Computer Methods in Applied Mechanics and Engineering*, **105** (1993) 405.

[3] Z. Johan, T. J. R. Hughes, K. K. Mathur, and S. L. Johnsson, "A data parallel finite element method for computational fluid dynamics on the Connection Machine system", *Computer Methods in Applied Mechanics and Engineering*, **99** (1992) 113.

[4] T. J. R. Hughes, L. P. Franca, and M. Mallet, "A new finite element formulation for fluid dynamics: VI. Convergence analysis of the generalized SUPG formulation for linear time–dependent multidimensional advective–diffusive systems", *Computer Methods in Applied Mechanics and Engineering*, **63** (1987) 97–112.

[5] T. J. R. Hughes, L. P. Franca, and G. M. Hulbert, "A new finite element formulation for fluid dynamics: VIII. the Galerkin / least–squares method for advective–diffusive equations", *Computer Methods in Applied Mechanics and Engineering*, **73** (1989) 173–189.

[6] L. Franca and T. J. R. Hughes, "Convergence analyses of Galerkin least–squares methods for symmetric advective-diffusive forms of the Stokes and incompressible Navier-Stokes equations", *Computer Methods in Applied Mechanics and Engineering*, **105** (1993) 285–298.

[7] M.R. Malik, T. A. Zang, and M. Y. Hussaini, "A spectral collocation method for the Navier-Stokes equations", *Journal of Computational Physics*, **61** (1985) 64–88.

[8] K.E. Jansen, "Large-eddy simulation using unstructured grids", in *Advances in DNS/LES*, Greyden Press, Columbus, (1997) 117–128.

[9] M. Germano, U. Piomelli, P. Moin, and W. H. Cabot, "A dynamic subgrid–scale eddy viscosity model", *Physics of Fluids*, **3** (1991) 1760.

[10] G. Comte-Bellot and S. Corrsin, "Simple Eulerian time-correlation of full and narrow-band velocity signals in grid-generated 'isotropic' turbulence", *Journal of Fluid Mechanics*, **48** (1971) 273–337.

[11] H.J. Kaltenbach and H. Choi, "Large-eddy simulation of flow around an airfoil on a structured mesh", in *Annual Research Briefs*, Center for Turbulence Research, NASA Ames / Stanford University, (1995) 51–60.

[12] S. Dey, *Geometry-based three-dimensional hp-finite element modelling and computations*, Ph.D. thesis, Rensselaer Polytechnic Institute, 1997.

[13] F. Bastin, "Jet noise using large eddy simulation", in *Annual Research Briefs*, Center for Turbulence Research, NASA Ames / Stanford University, (1996) 115–132.