# Direct Numerical Simulation of Turbulent Flows Using Spectral Methods

K. Sengupta,<sup>\*</sup> and F. Mashayek<sup>†</sup> University of Illinois at Chicago

G. B. Jacobs,<sup>‡</sup>

San Diego State University

Direct numerical simulation (DNS) is the most accurate method of solving turbulence in fluids. In DNS the Navier-Stokes equations are solved on a fine mesh to resolve all the spatial and temporal scales present in the flow. In order to ensure high accuracy of the discrete solution, schemes with low numerical errors are necessary. Spectral methods with their low dissipation and dispersion errors are very attractive in this regard. Since their inception in early 1970's, spectral methods have been routinely applied towards direct simulation of turbulent flows. Some of the earliest applications were in isotropic turbulence using Fourier series based methods. As the community turned its attention to solving practical flows, the need for schemes that are efficient in handling complex geometries arose. This led to the development of spectral element methods.

In this paper, we describe the application of a semi-structured spectral element method for direct simulation of compressible, wall bounded turbulent flows. A brief outline of the method is first presented. Then we review some of the results for turbulent channel flow and flow over a backward-facing step. The accuracy of the technique is established by comparing our simulation results with experiment and a previous DNS study.

# I. Introduction

Turbulence, the chaotic and apparently unpredictable state of fluid flow, is one of the most challenging problems in fluid physics. Since almost all large scale engineering and atmospheric flows are turbulent, fundamental understanding of the physics of turbulence is of great practical importance. Currently, the different approaches to the study of turbulent flows can be broadly classified into three categories: theory, physical experiments and numerical simulation. The explosive growth of computational power in the last three decades, has generated huge interest in the area of numerical simulation of turbulence. Computational studies have also gained momentum with the advent of efficient and accurate numerical schemes for solving coupled and highly nonlinear partial differential equations that are used to describe turbulent flows. Simulation methodologies can again be classified, based on the level of representation of the physics and accuracy, into direct numerical simulation (DNS), large-eddy simulation (LES) and Reynolds-averaged Navier-Stokes (RANS) approaches.

The most accurate is the direct numerical simulation, where all the spatial and temporal scales of the flow are resolved. This entails solution of time-dependent Navier-Stokes equations without any modeling. Numerical schemes play a crucial role in determining the fidelity of direct-numerical simulations. In order to resolve the smallest of scales accurately, methods with low dissipation errors must be sought. It was shown in Ref.<sup>1</sup> that for low-order methods the high-wavenumber part of the energy spectrum is heavily distorted

<sup>\*</sup>Graduate student, Department of Mechanical and Industrial Engineering, 842 W. Taylor Street, Chicago, IL 60607.

<sup>&</sup>lt;sup>†</sup>Professor, Department of Mechanical and Industrial Engineering, 842 W. Taylor Street, Chicago, IL 60607, AIAA Associate Fellow.

<sup>&</sup>lt;sup>‡</sup>Assistant Professor, Department of Aerospace Engineering, 5500 Campanile Drive, San Diego, CA 92182

by truncation errors. Therefore, to resolve the small scales of turbulence with standard low-order schemes, unrealistically large number of grid points are necessary. In addition, these schemes have large dispersion error, which introduces phase-speed errors of wave in compressible flows.

High-order methods on the other hand, are very efficient in resolving the small scales of turbulence. Currently, high-order methods used in direct simulation of turbulence are compact finite difference, spectral, and spectral element methods. In this article we will focus our attention on spectral element methods. Since its inception in early 1970's, spectral methods have been extensively applied to turbulent flows. Orszag<sup>2</sup> developed a fourier series based method for solution of isotropic turbulence, which he termed as pseudo-spectral method. Since then many variants have been developed. Standard pseudo-spectral methods have some severe restrictions. The Fourier series based method imposes restriction on boundary conditions, and as such cannot be applied to practical flows that require non-periodic, physical boundary conditions. The standard Chebyshev spectral method does not impose periodicity requirement at the domain boundaries. However, they require that the computational domain be simple enough to be mapped to a unit cube. Spatial resolution can be increased by increasing the order of the polynomial. The derivative approximations are usually performed (especially for high-order polynomials) using fast fourier transform methods, since matrix multiplication technique becomes prohibitively expensive. Moreover, for pseudo-spectral methods, the time step decreases with the square of the polynomial order. This imposes severe time step restrictions for large polynomial degrees.

The above drawbacks of the standard spectral schemes led to the development of the concept of domain decomposition. The basic idea is to subdivide the computational domain into multiple sub-domains or elements, and use Chebyshev or Legendre polynomials for approximating the solution in each sub-domain/element. The use of lower-order polynomials within each element implies that efficient matrix vector multiplication techniques can be used for computing the spatial derivatives. The spatial resolution can be conveniently altered either by increasing the number of elements (h-refinement) or by increasing the polynomial order within the elements (p-refinement). In smooth solution spaces, the method provides asymptotically exponential rate of spatial convergence with p-refinement. Patera<sup>3</sup> was first to introduce the method for solution of incompressible Navier-Stokes equations. Multi-domain methods for compressible flows were later developed by Hesthaven<sup>4</sup> and Kopriva.<sup>5</sup> Most of the early versions of spectral element methods were based on quadrilateral (2D) and hexahedral (3D) elements. For such d-cube type elements, multidimensional approximations are obtained through tensor product of one-dimensional polynomial expansion bases. Although very efficient for moderately complex flow configurations, their limitations show up for arbitrarily complex geometries. The issue of complex geometries were addressed by the development of spectral element methods on triangles and tetrahedrons (see Refs.<sup>6,7</sup>), commonly referred to as unstructured spectral elements. The success of these methods hinges on the identification of multivariate Lagrange polynomials within triangles/tetrahedron along with optimal collocation points.

In this paper, we describe the application of a spectral spectral element method to direct numerical simulation of compressible turbulent flows. The remaining part of the paper is organized as follows. First governing equations for compressible flows are enumerated. Then, we outline the spectral element methodology. The methodology is applied to simulation of turbulent channel flow and flow over a backward-facing step.

# II. Governing equations

The governing equations for the compressible and viscous fluid flow are the conservation statements for mass, momentum and energy. They are presented in non-dimensional, conservative form with Cartesian tensor notation,

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0, \tag{1}$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j + p\delta_{ij})}{\partial x_i} = \frac{\partial\sigma_{ij}}{\partial x_i},\tag{2}$$

$$\frac{\partial(\rho e)}{\partial t} + \frac{\partial[(\rho e + p)u_j]}{\partial x_j} = -\frac{\partial q_j}{\partial x_j} + \frac{\partial(\sigma_{ij}u_i)}{\partial x_j}.$$
(3)

The total energy, viscous stress tensor, and heat flux vector are, respectively, given as

$$\rho e = \frac{p}{\gamma - 1} + \frac{1}{2}\rho u_k u_k,\tag{4}$$

$$\sigma_{ij} = \frac{\mu}{Re} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right),\tag{5}$$

$$q_j = -\frac{\mu}{(\gamma - 1)RePrM_f^2} \frac{\partial T}{\partial x_j}.$$
(6)

The Reynolds number Re is based on the reference density  $\rho_f^*$ , velocity  $U_f^*$ , length  $L_f^*$ , and molecular viscosity  $\mu_f^*$  and is given by  $Re = \rho_f^* U_f^* L_f^* / \mu_f^*$ .  $Pr = \mu_f^* C_p / k^*$  is the Prandtl number. The non-dimensional viscosity is taken as  $\mu = \mu(T)/\mu_f^*$ , where  $\mu(T)$  is the molecular viscosity at temperature T. The astrict denotes dimensional quantities. The above equation set is closed by the equation of state,

$$p = \frac{\rho T}{\gamma M_f^2},\tag{7}$$

where  $M_f$  is the reference Mach number, taken to be 1 in this work.

The conservation equations can be cast in the matrix form

$$\frac{\partial \boldsymbol{Q}}{\partial t} + \frac{\partial \boldsymbol{F}_i^a}{\partial x_i} - \frac{\partial \boldsymbol{F}_i^v}{\partial x_i} = 0, \tag{8}$$

where

$$\boldsymbol{Q} = \begin{pmatrix} \rho \\ \rho u_1 \\ \rho u_2 \\ \rho u_3 \\ \rho e \end{pmatrix}, \tag{9}$$

$$\boldsymbol{F}_{i}^{a} = \begin{pmatrix} \rho u_{i} \\ \rho u_{1} u_{i} + p \delta_{i1} \\ \rho u_{2} u_{i} + p \delta_{i2} \\ \rho u_{3} u_{i} + p \delta_{i3} \\ (\rho e + p) u_{i} \end{pmatrix},$$
(10)

$$\boldsymbol{F}_{i}^{\upsilon} = \begin{pmatrix} 0 \\ \sigma_{i1} \\ \sigma_{i2} \\ \sigma_{i3} \\ -q_{i} + u_{k}\sigma_{ik} \end{pmatrix}.$$
(11)

Here Q is the vector of the conserved variables and will also be referred to as the state vector in this work.  $F_i^a$  and  $F_i^v$  are the advective and viscous flux vectors respectively, in the  $x_i$  direction.

# III. Staggered grid Chebyshev multi-domain method

This section briefly describes the staggered-grid Chebyshev multidomain method. For a more complete description of the method see Ref.<sup>5</sup> In this method, the computational domain,  $\Omega$ , is divided into non-overlapping hexahedral sub-domains,  $D_k$ , such that

$$\Omega = \sum D_k.$$
(12)

Chebyshev polynomials are naturally defined on the interval [-1, 1] in one space dimension (tensor product expansion for multiple dimensions). However, in this method the sub-domains are mapped onto a unit



Figure 1. Staggered arrangement of solution variable and fluxes.

hexahedron  $D_k \leftrightarrow [0,1] \times [0,1] \times [0,1]$  by an isoparametric mapping. The use of isoparametric mapping ensures that the spectral accuracy of the scheme is not affected by the domain boundary approximation. The staggered grid method uses two sets of grids, one for the solution (Chebyshev-Gauss grid) and one for computation of the fluxes (Chebyshev-Gauss-Lobatto grids). In one space dimension the Gauss and Gauss-Lobatto quadrature points are defined by,

$$X_{j+1/2} = \frac{1}{2} \left\{ 1 - \cos\left[\frac{\pi 2j + 1}{2N}\right] \right\}, j = 0, \dots N - 1,$$
(13)

and

$$X_j = \frac{1}{2} \left\{ 1 - \cos\left[\frac{\pi j}{N}\right] \right\}, j = 0, \dots N,$$
(14)

respectively, on the unit interval [0, 1]. The gauss grid in three-dimension, henceforth called the ggg grid, is the tensor product of the one-dimension grid defined in (13). The solution values  $\tilde{Q}$  are approximated on the ggg grid. Its polynomial interpolant is defined as,

$$\tilde{\boldsymbol{Q}}^{ggg}(X,Y,Z) = \sum_{i=0}^{N-1} \sum_{j=0}^{N-1} \sum_{k=0}^{N-1} \tilde{\boldsymbol{Q}}^{ggg}_{i+1/2,j+1/2,k+1/2} h_{i+1/2}(X) h_{j+1/2}(Y) h_{k+1/2}(Z),$$
(15)

where N is the approximation order. Here,  $h_{i+1/2,j+1/2} \in \mathbf{P}_{N-1}$  is the Lagrange interpolating polynomial defined on the Gauss grid,

$$h_{i+1/2}(\xi) = \prod_{m=0, m \neq i}^{N-1} \left( \frac{\xi - X_{m+1/2}}{X_{i+1/2} - X_{m+1/2}} \right).$$
(16)

The flux,  $\mathbf{F}_i$ , in each direction is defined on the Gauss-Lobatto set of grids, shown in figure 1. The *x*-direction flux  $(\mathbf{F}_1)$  is evaluated at the Lobatto-Gauss-Gauss grid (lgg), denoted by open squares in the figure,  $(X_i, Y_{j+1/2}, Z_{k+1/2}), i = 0, 1, \dots N, j, k = 0, 1, \dots N - 1$ , the *y*-direction flux  $(\mathbf{F}_2)$  at the Gauss-Lobatto-Gauss grid (gg), denoted by open circles,  $(X_{i+1/2}, Y_j, Z_{k+1/2}), j = 0, 1, \dots N, i, k = 0, 1, \dots N - 1$ , and the *z*-direction flux  $(\mathbf{F}_3)$  at the Gauss-Lobatto grid (ggl), denoted by closed squares,  $(X_{i+1/2}, Y_{j+1/2}, Z_k), k = 0, 1, \dots N, i, j = 0, 1, \dots N - 1$ . The flux vectors are computed by reconstructing the solution at the Lobatto points through interpolations using polynomials of the type (15). This leads to the following

$$\boldsymbol{Q}^{lgg}(X_i, Y_{j+1/2}, Z_{k+1/2}) = \sum_{m=0}^{N-1} \sum_{n=0}^{N-1} \sum_{p=0}^{N-1} \frac{\tilde{\boldsymbol{Q}}_{m+1/2, n+1/2, p+1/2}^{ggg}}{J_{m+1/2, n+1/2, p+1/2}} h_{m+1/2}(X_i) h_{n+1/2}(Y_{j+1/2}) h_{p+1/2}(Z_{k+1/2}),$$
(17)

in the x-direction, which reduces to a one-dimensional operation,

$$\boldsymbol{Q}^{lgg}(X_i, Y_{j+1/2}, Z_{k+1/2}) = \sum_{m=0}^{N-1} \frac{\tilde{\boldsymbol{Q}}_{m+1/2, j+1/2, k+1/2}^{ggg}}{J_{m+1/2, j+1/2, k+1/2}} h_{m+1/2}(X_i),$$
(18)

due to the cardinal property of the Lagrange interpolating polynomial. Similarly for the y and z directions the interpolants are given by,

$$\boldsymbol{Q}^{glg}(X_{i+1/2}, Y_j, Z_{k+1/2}) = \sum_{n=0}^{N-1} \frac{\tilde{\boldsymbol{Q}}_{i+1/2, n+1/2, k+1/2}^{ggg}}{J_{i+1/2, n+1/2, k+1/2}} h_{n+1/2}(Y_j), \tag{19}$$

 $4~{\rm of}~10$ 

American Institute of Aeronautics and Astronautics

$$\boldsymbol{Q}^{ggl}(X_{i+1/2}, Y_{j+1/2}, Z_k) = \sum_{p=0}^{N-1} \frac{\tilde{\boldsymbol{Q}}_{i+1/2, j+1/2, p+1/2}^{ggg}}{J_{i+1/2, j+1/2, p+1/2}} h_{p+1/2}(Z_k).$$
(20)

Once the solution values are interpolated to the Lobatto grid the advective fluxes are computed. The interface points will have different flux values due to discontinuity of solution values at the sub-domain boundaries. The patching of the advective fluxes are described in the following paragraph. The viscous fluxes are computed in two steps. The solution interpolant at the Lobatto grids must be continuous for a unique first derivative at the subdomain interfaces. This is ensured by a Dirichlet type patching. After the Lobatto interpolants for the solution values are patched, their derivatives are computed at the Gauss points. The gradients are then interpolated back to Lobatto points. The viscous fluxes are computed using the functional relations (5) and (6). After application of the interface and Neumann boundary condition, the mapped viscous fluxes are computed. The total mapped flux is obtained by adding the inviscid and viscous parts.

Interpolation of the solution by equations (18)-(20) leads to different solution values at the sub-domain interface points, one from each of the contributing sub-domains. These two values coincide only in the limit of infinite resolution. The coupling between sub-domains is enforced by computing a single flux at the interface points, separately of the inviscid and viscous fluxes. The inviscid fluxes are computed using an approximate Riemann solver. Formally, given the two solution states  $\mathbf{Q}_N^{k-1}$  and  $\mathbf{Q}_0^k$ , the flux in each spatial direction, with the assumption that waves are normal to the interface, can be written as,

$$\Gamma^{a}(\boldsymbol{Q}_{N}^{k-1},\boldsymbol{Q}_{0}^{k}) = \frac{1}{2}(\boldsymbol{F}^{a}(\boldsymbol{Q}_{N}^{k-1}) + \boldsymbol{F}^{a}(\boldsymbol{Q}_{0}^{k})) - \frac{1}{2}\boldsymbol{R}|\lambda|\boldsymbol{R}^{-1}(\boldsymbol{Q}_{0}^{k} - \boldsymbol{Q}_{N}^{k-1})$$
(21)

where  $\mathbf{F}^{a}$  is the vector of advective fluxes. **R** is the matrix of the right eigenvectors of the Jacobian of  $\mathbf{F}^{a}$ , computed using Roe-average of  $\mathbf{Q}_{N}^{k-1}$  and  $\mathbf{Q}_{0}^{k}$ . For imposing Euler boundary conditions, the physical boundary can be viewed as an interface between external flow and the computational domain. The Riemann solver can then be applied between the external and internal state of the solution variables. The way of specifying the external state depends on the specific physical boundary and flow.

For patching of the viscous fluxes, first a single solution value at an interface point is computed, by averaging the solutions from the two neighboring domains or by specifying a value if the interface is at the boundary. This Dirichlet patching ensures a unique first derivative at the interface point. Once the viscous fluxes are computed by the method outlined in the previous sub-section, they are averaged at the interface points,

$$\Gamma_0^{v,k} = \Gamma_N^{v,k-1} = \frac{1}{2} (\boldsymbol{F}_0^{v,k} + \boldsymbol{F}_N^{v,k-1}).$$
(22)

The Neumann boundary conditions are imposed at the boundary points at this stage. Once the total fluxes are computed at the lgg, glg and ggl points, the flux interpolants are constructed as,

$$\tilde{F}_{1}(X,Y,Z) = \sum_{m=0}^{N} \sum_{n=0}^{N-1} \sum_{p=0}^{N-1} \tilde{F}_{1m,n+1/2,p+1/2} l_{m}(X) h_{n+1/2}(Y) h_{p+1/2}(Z),$$
(23)

$$\tilde{F}_{2}(X,Y,Z) = \sum_{m=0}^{N-1} \sum_{n=0}^{N} \sum_{p=0}^{N-1} \tilde{F}_{2m+1/2,n,p+1/2} h_{m+1/2}(X) l_n(Y) h_{p+1/2}(Z),$$
(24)

$$\tilde{F}_{3}(X,Y,Z) = \sum_{m=0}^{N-1} \sum_{n=0}^{N-1} \sum_{p=0}^{N} \tilde{F}_{3m+1/2,n+1/2,p} h_{m+1/2}(X) h_{n+1/2}(Y) l_p(Z).$$
(25)

These polynomials are differentiated and evaluated at the Gauss-Gauss-Gauss grid, to give pointwise derivatives,

$$\frac{\partial \tilde{F}_{1}(X_{i+1/2}, Y_{j+1/2}, Z_{k+1/2})}{\partial X} = \sum_{m=0}^{N} \tilde{F}_{1}(X_{m}, Y_{j+1/2}, Z_{k+1/2}) \frac{\partial l_{m}(X_{i+1/2})}{\partial X}$$
(26)

$$\frac{\partial \tilde{F}_{2}(X_{i+1/2}, Y_{j+1/2}, Z_{k+1/2})}{\partial Y} = \sum_{n=0}^{N} \tilde{F}_{2}(X_{i+1/2}, Y_{n}, Z_{k+1/2}) \frac{\partial l_{n}(Y_{j+1/2})}{\partial Y}$$
(27)

#### $5~{\rm of}~10$

American Institute of Aeronautics and Astronautics



Figure 2. Mean normalized streamwise velocity plotted in, global coordinate (top) and wall coordinate (bottom), in channel flow.

$$\frac{\partial \tilde{F}_{3}(X_{i+1/2}, Y_{j+1/2}, Z_{k+1/2})}{\partial Z} = \sum_{p=0}^{N} \tilde{F}_{3}(X_{i+1/2}, Y_{j+1/2}, Z_p) \frac{\partial l_p(Z_{k+1/2})}{\partial Z}.$$
(28)

Finally the semi-discrete equation for the solution unknowns at the Gauss-Gauss-Gauss grid is given by,

$$\left[\frac{d\tilde{\boldsymbol{Q}}}{dt}\right]_{i+1/2,j+1/2,k+1/2} + \left[\frac{\partial\tilde{\boldsymbol{F}}_i}{\partial X_i}\right]_{i+1/2,j+1/2,k+1/2} = 0.$$
(29)

# IV. Results and discussion

#### A. Turbulent channel flow

In this section, simulation results from a compressible, plane, parallel, 3D channel flow using the Chebyshev multi-domain spectral (CMSM) code are presented. This configuration provides a case study of wall bounded inhomogeneous turbulent flows. The flow is simulated at bulk Reynolds number of  $Re_f = 3000$  based on channel half width and bulk velocity. The Mach number is taken to be Ma = 0.4 based on bulk velocity and wall temperature ( $T_{wall}$ ). The computational model is similar to the one used in Ref.<sup>9</sup> The domain extents are  $L_x=6$ ,  $L_y=2$ , and  $L_z=2$  in the streamwise, spanwise, and wall normal directions, respectively. Periodic boundary conditions are employed in the streamwise and spanwise directions, while the conservative wall boundary condition of Jacobs et al.<sup>10</sup> is used for the bottom and top wall.

Following Lenormand et al.,<sup>8</sup> a time dependent forcing term is included in the streamwise momentum equation in order to drive the flow. The velocity field is initialized with the laminar parabolic profile for u with a small random disturbance superimposed on it. Similarly, temperature is initially set to laminar Poiseuille profile. The flow was simulated with 10 sub-domains in both streamwise and spanwise directions, and 16 sub-domains in the wall normal direction. The sub-domains in the wall direction are stretched out towards center of the channel with a cosine distribution. Previous study<sup>11</sup> has shown that at Ma=0.4, plane parallel channel flow is pseudo-incompressible. Therefore, simulation results are compared with incompressible DNS study of Moser et al.<sup>12</sup> and experimental measurements of Niedershulte et al.<sup>13</sup>

The Favre averaged streamwise velocity normalized with the skin friction velocity,  $u_{\tau}$ , is plotted against global, z, and wall coordinates,  $z^+$ , in figure 2. CMSM results compare well with benchmark data. In wall coordinates, our simulation is able to reproduce the linear law in the viscous sub-layer ( $z^+ < 5$ ) and the log law in the outer layer. Figure 3 compares the Favre-fluctuating streamwise turbulent stresses from CMSM simulation with the benchmark cases. The results are normalized with  $u_{\tau}^2$  and are plotted in wall coordinates. The peak value is accurately captured by the CMSM simulation and overall agreement is also quite good.



Figure 3. Streamwise normal stresses in channel flow.

The wall and spanwise normal stresses, shown in figure 4 also compare well. The shear stress profile has larger discrepancies. Under-prediction near the wall is compensated by over-prediction at the center of the channel.

#### B. Flow over backward-facing step

Next, we apply the CMSM methodology to a problem with increased complexity, by simulating the flow over a backward-facing step. A schematic of the configuration studied is presented in figure 5. The boundary layer upstream of the step separates at the step corner and subsequently reattaches to the bottom wall. After reattaching, the flow recovers to a boundary layer profile. The separated shear layer bounds a reverse flow region near the step, usually known as the recirculation bubble. The geometry often serves as a building block model for dump-combustors and flame anchors in ramjet engines. The selected configuration was studied by Wengle et al.<sup>14</sup> in their experimental and DNS investigation using finite volume method. The Reynolds number of the flow is 3000, based on the step height and the free stream velocity at the step. The inflow profile is a laminar boundary layer with  $\delta_{0.99}/h = 0.21$ , where h is the step height. Under these conditions the separated shear layer becomes transitional in nature.

At the outflow, mean profile from the experiment is used for the velocities, while the thermodynamic quantities are set to their free stream values. The bottom-wall is isothermal and at the top boundary free stream conditions are specified. As a final note, it should be pointed out that both experiment and simulation of Wengle et al. was for incompressible flow, while our simulations here are for compressible flow. Results from three different cases are reported here. At this point only averaged quantities are evaluated for comparison with Ref.<sup>14</sup> DNS was performed with polynomial order p = 10 and p = 6 for Mach number 0.4, results of which are compared with the experiment in figure 6. Favre averaged streamwise and wall normal velocity profiles are plotted along the z-direction for different axial locations. It can be seen that the results compare reasonably well at x = 8 and at x = 12, while there are significant differences at x = 9 and x = 11. The differences are amplified for the wall normal velocity (w), due to their smaller range as compared to streamwise velocity (u). The shear layer behaves laminar till about x = 8, after which disturbances from below are amplified leading to vortex roll-ups and onset of turbulence. The laminar portion of the separated shear layer bounds a dead water region of little turbulent activity. However, as the shear layer goes into transition a recirculation region of strong turbulent activity is created. The mean recirculation lengths for the p = 10 and p = 6 cases were 5.36 and 5.45, respectively, which amounts to an under-prediction of 16.25% and 14.7% over the experimental value. The poor agreement between our simulation and experiment is most likely due to compressibility effects within the shear-layer at Ma = 0.4. In order to verify this argument we simulate the flow for Ma = 0.2 (at p = 6), where compressibility effects are expected to be significantly less.



Figure 4. Normal stresses in the spanwise and wall-normal direction (top) and shear stress (bottom), in channel flow.



Figure 5. Schematic of the backward-facing step (not to scale).

Figure 7 indicates that results are indeed better for Ma = 0.2. We conjecture that large pressure dilatation dissipation in the recirculation region at Ma = 0.4 leads to a shorter recirculation region and consequent disagreement in the averaged velocity profiles. However, this needs to be verified with the calculation of the dissipation terms appearing in the turbulent kinetic energy equation.

# V. Conclusions

Numerical schemes with low dissipation errors are required for direct numerical simulation of turbulent flows, where all the scales of motion are resolved. Moreover, for long time-accurate solutions, methods must also be minimally dispersive. Spectral methods have proven to be very effective in meeting these requirements. The different types of spectral methods in use, can be broadly divided into standard spectral schemes and spectral element methods. Standard spectral methods impose restrictions on boundary conditions and geometry. Spectral element methods on the other hand allow for high-order discretization of complex geometries.

In this paper we have reviewed the application of a multi-domain spectral method to direct numerical simulation of turbulent flows. The computational domain is decomposed into multiple sub-domains, which are mapped onto a standard hexahedron. The solution values and fluxes are approximated on two different grids using high-order Chebyshev polynomials. The staggered arrangement makes the method fully conservative. The robustness and accuracy of the method is demonstrated with simulation of turbulent channel flow and



Figure 6. Mean streamwise velocity (top) and mean wall normal velocity (bottom) at different axial locations for the backward facing step.

flow over a backward facing step. Excellent agreement with experimental data and previous DNS results is observed for both cases.

### Acknowledgments

The support for this work was provided by the U.S. Office of Naval Research with Dr. G.D. Roy as the Program Officer. The computational time was provided by the National Center for Supercomputing Applications (NCSA).

# References

<sup>1</sup>Kravchenko, A. G. and Moin, P., On the effect of numerical errors in large eddy simulation of turbulent flows, *Journal of Comput. Physics*, 131, 310-322, 1997.

<sup>2</sup>Orszag, S. A., A comparison of pseudospectral and spectral approximations, *Stud. Appl. Math.*, 51, 253-259, 1972.

<sup>3</sup>Patera, A. T., A spectral element method for fluid dynamics: laminar flow in channel expansion, *Journal of Comput. Physics*, 54, 468-488, 1984.

<sup>4</sup>Hesthaven, J. S., A stable penalty method for compressible Navier-Stokes equations. III. Multidimensional domain decomposition schemes, *SIAM J. Sci. Comput.*, 20, 62-93, 1998.

<sup>5</sup>Kopriva, D. A., A staggared-grid multidomain spectral method for compressible Navier-Stokes equations, *Journal of Comput. Physics*, 143, 125-158, 1998.

<sup>6</sup>Hesthaven, J. S. and Gottlieb, D., Stable spectral methods for conservation laws on triangles with unstructured grids, *Comput. Methods Appl. Engrg.*, 175, 361-381, 1999.

<sup>7</sup>Hesthaven, J. S. and Teng, C. H., Stable spectral methods on tetrahedral elements, *Comput. Methods Appl. Engrg.*, 21(6), 2352-2380, 2000.

<sup>8</sup>Lenormand, E., Sagaut, P. and Ta Phuoc, T., Large eddy simulation of subsonic and supersonic channel flow at moderate Reynolds number, *Int. J. Numer. Meth. Fluids*, 32, 369-406, 2000.

<sup>9</sup>Jacobs, G. B., Kopriva, D. A. and Mashayek, F., Validation study of a multidomain spectral code for simulation of turbulent flows, *AIAA Journal*, 43(6), 1256-1264, 2005.

<sup>10</sup>Jacobs, G. B., Kopriva, D. A. and Mashayek, F., A conservative isothermal wall boundary condition for the compressible Navier-Stokes equation, *Journal of Scientific Computing*, 30(2), 177-192, 2007.

<sup>11</sup>Ridder, J. B. and Beddini, R., Large eddy simulation of compressible channel flow, *NASA Tech. Report*, NGT-50363, 1993.

<sup>12</sup>Moser, R., Kim, J. and Mansour, N. N., Direct numerical simulation of turbulent channel flow up to  $Re_{\tau} = 590$ , Phys. of Fluids, 11(4), 943-945, 1999.

<sup>13</sup>Niederschulte, M. A., Adrian, R. J. and Hanratty, T. J., Measurements of turbulent flow in a channel at low Reynolds numbers, *Exp. Fluids*, 9, 222–230, 1990.



Figure 7. Mean streamwise velocity (top) and mean wall normal velocity (bottom) at different axial locations for the backward facing step.

<sup>14</sup>Wengle, H., Huppertz, A., Barwolff, G. and Janke, G., The manipulated transitional backward-facing step flow: an experimental and direct numerical investigation, *Eur. J. Mech. B-Fluids*, 20, 25-46, 2001.