Assignment - Part II

Ph.D. Course 2009:
An Introduction to DG-FEM for solving partial differential equations

This the second part of the mandatory part of the course and completion of it is required to pass the course with full credit.

This part should, in combination with the results for Assignment part I, form a report which must be submitted electronically no later than

Friday, October 2, 2009

We shall consider the solution of the two-dimensional Navier-Stokes equations for a compressible gas. The Navier-Stokes equations are given as

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0, \tag{1}$$

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2 + p)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} = \frac{1}{\mathrm{Re}}\left(\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y}\right), \tag{2}$$

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2 + p)}{\partial y} = \frac{1}{\mathrm{Re}}\left(\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y}\right), \tag{3}$$

$$\begin{aligned}&\frac{\partial E}{\partial t} + \frac{\partial(E+p)u}{\partial x} + \frac{\partial(E+p)v}{\partial y} \\ &= \frac{1}{\mathrm{Re}}\left(\frac{\partial(u\tau_{xx} + v\tau_{xy} - q_x)}{\partial x} + \frac{\partial(u\tau_{xy} + v\tau_{yy} - q_y)}{\partial y}\right),\end{aligned} \tag{4}$$

These equations reflect conservation of density, ρ, momentum, $(\rho u, \rho v)$, and total energy, E. The pressure, p, is expressed through the connection between the energy and momentum as

$$E = \rho\left(T + \frac{1}{2}\left(u^2 + v^2\right)\right),$$

and the ideal gas law

$$p = (\gamma - 1)\rho T.$$

Here $\gamma = 1.4 = c_p/c_v$ is the ratio of specific heats for atmospheric air.

Scientific Computing Section, DTU Informatics, Kgs.-Lyngby, Denmark.

The temperature relates to the heat flux, ∇q, as

$$\nabla q = -\gamma \frac{1}{Pr} \nabla T,$$

where $\mathrm{Pr} = 0.72$ is the Prandtl number.

All variables are normalized with respect to the free-steam quantities, ρ_∞, U_∞, μ_∞, and a reference length, D, which yields a free-stream Reynolds number, Re, defined as

$$\mathrm{Re} = \frac{\rho_\infty U_\infty D}{\mu_\infty}.$$

This is the Reynolds number we will use for comparisons later in this work. In many problems one accounts for the temperature dependence of the dynamic viscosity, μ_∞. However, in this work we assume that it is constant.

The viscous tensors, τ, is given as

$$\tau_{xx} = \frac{4}{3}\frac{\partial u}{\partial x} - \frac{2}{3}\frac{\partial v}{\partial y}, \quad \tau_{yy} = \frac{4}{3}\frac{\partial v}{\partial y} - \frac{2}{3}\frac{\partial u}{\partial x},$$

$$\tau_{xy} = \frac{\partial u}{\partial y} - \frac{\partial v}{\partial x}.$$

We shall consider solving this problem for a problem of flow around a cylinder of radius D. The boundary conditions on the boundary of the cylinder are, $u_{cyl} = v_{cyl} = 0$ and $T_{cyl} = T_0$ where T_0 is the normalized temperature of the cylinder, i.e., if $T_0 = T_\infty$ the cylinder is isothermal. No boundary condition is needed on the density, ρ_{cyl}, which can be taken to be the internal value determined by the solver, i.e., the boundary condition of the density is a Neumann condition along the wall.

At the outer boundary we assume a uniform mean flow, $(\rho, \rho u, \rho v, E) = (1, 1, 0, E_\infty)$. The energy is computed from the free-steam Mach number, M_∞, as

$$M_\infty = \frac{U_\infty}{c_\infty}, c_\infty = \sqrt{\gamma(\gamma-1)T_\infty},$$

which implies that

$$T_\infty = \frac{1}{\gamma(\gamma-1)M_\infty^2},$$

assuming that $U_\infty = 1$.

At inflow parts of the outer boundaries, you can impose the uniform mean flow conditions. At the outflow part of the out boundary, you can impose the pressure at infinite where the pressure at infinity can be determined from the temperature, the density and the ideal gas law.

For the second order terms you can take all derivates to be determined by the code itself, i.e., only Dirichlet conditions are needed at the boundaries.

1. Implement a two-dimensional DG-FEM solver for the compressible Navier-Stokes solver. You can chose the external boundary to have any shape you wish but the diameter of the cylinder should be one, i.e., $D = 1$.

2. Discuss how you decide on the time-step.

3. The initial conditions of the solver can be a uniform flow throughout the domain. To account for initial conditions satisfying the boundary conditions at the cylinder surface, it is recommend that the solver is running the first 100-200 timesteps with a very strong filter which is then gradually removed. It is also recommended that the time-step initially is very small and then gradually increased to the stable time-step.

4. Run the code at $\mathrm{Re} = 100$ and $M_\infty = 0.4$ – after a while you should observe the famous von Karman vortex streak after the cylinder – the reason a flag wiggles in the wind !

To validate the accuracy of the code we will consider the Stroudal number – the frequency associated with the von Karman streak. You can measure this by taking a time series of the pressure some where behind the cylinder and then determine the frequecy from the data. For $Re = 100$, the measured value is 0.164.

1. Validate to see if you can reproduce this value in your code.

2. Determine whether the size or shape of the computational domain, i.e., the distance from the cylinder to the outer boundary, has any substantial effect on the accuracy of the Stroudal number prediction.

3. For $M_\infty = 0.2$ and $60 \leq \mathrm{Re} \leq 180$, an empirical relation between the Stroudal number and Re is given as

$$S = \frac{-3.3265}{\mathrm{Re}} + 0.1816 + 0.00016\mathrm{Re}.$$

   Validate the code for 5 different Reynolds numbers using this scaling formula.

4. Try to run the code for different values of M_∞, e.g., 0.1 and 2.0 – what do you observe for the timestep and the general behavior of the solution in the two cases ? – can you explain your observations ?

5. Plot a couple of snap-shots of the computed solution

Now that we have implemented and tested the code, pick a different geometry - a square cylinder, a plate or something entirely different - and run the code for this.

Scientific Computing Section, DTU Informatics, Kgs.-Lyngby, Denmark.

1. Generate different grids to enable a convergence study and test that the computed solution(s) are converged.
2. Run the code for different values of the Re and discuss your observations.
3. Plot a couple of snap-shots of the computed solution

Enjoy !