GPU Parallelisation of basic CFD solvers

Dr Y Sudhakar
School of Mechanical Sciences, Indian Institute of Technology Goa

This project focuses on developing basic Computational fluid dynamic (CFD) parallel programs on GPUs. The aim is to write codes with detailed comments and necessary documentation. These will be published in open source repositories to serve as a starting point for students/researchers to learn GPU programming in the context of CFD.

The student will be writing codes for solving the following equations on GPUs using OpenACC and CUDA.

- 1D diffusion equation

$$\frac{\partial u}{\partial t} - \alpha \frac{\partial^2 u}{\partial x^2} = f(x)$$

  As a first step, an explicit time integration will be used to solve this equation. This is the simplest possible CFD program, and is the best way to start learning. After the completion of the above, we will use an implicit time integration whose solution requires solving a tridiagonal matrix system. The student will be implementing the well-known Thomas algorithm for the same.

- 2D diffusion equation

$$\frac{\partial u}{\partial t} - \alpha \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) = f(x, y)$$

  Similar to the first task, both explicit and implicit time integration will be used. The linear algebraic system resulting from the discretisation will be solved using Jacobi, Gauss-Seidel, and SOR algorithms.

For the above tasks, the performance on OpenACC and CUDA will be compared. After completing these tasks, the student can also work on the GPU parallelisation of the Navier-Stokes solver developed at IIT Goa to simulate turbulent fluid flows.