Meeting Notes:

* Two abstracts for conference papers
  + IMECE (Deadline: March 6)
  + InterPACK (Deadline: Feb 1)
  + IEEE Cluster 2017 (Deadline: Apr 24)
  + Sc17 (Deadline: Mar 20 / Apr 3)
* Get TDMA solutions done as quickly as possible.
  + Review Mukherjee’s code to make sure I am implementing it in the same way.

Chapters:

* Abstract
* Introduction
  + Background
  + Motivation
* Objective / Scope
* Mathematics Overview
  + Computational Fluid Dynamics
    - General
    - Diffusion Problems
    - Microchannel Flow Boiling
  + Linear Algebra (Solvers)
    - Serial Solutions
    - Parallel Solutions
  + Sparse Matrix Formats
* Numerical Model Setup
  + Diffusion
  + Two-phase Flow Boiling
* Results
* Conclusions

Title Idea:

Performance Optimization in Computational Fluid Dynamics with Parallelism for Linear System Solvers

Introduction

Background (where it’s used, how it got there, lead into short comings in academia)

(aerospace)

Since the 1960s, Computational Fluid Dynamics (CFD) has been a tool implemented by scientists and engineers to solve a variety of complex problems related to fluid flow, heat and mass transfer, chemical reactions, and related phenomena [Versteeg 2007]. The aerospace industry was an early adopter of CFD, primarily using the technology to study the effects of drag and lift on aerodynamic bodies. Aircraft manufacturers like Airbus, Boeing, and Lockheed Martin, typically spend millions of dollars creating prototype aircraft to study the effects of fluid flow around the designed aircraft body [need a ref here]. By introducing a tool like CFD, aircraft manufacturers are able to filter out potential design flaws prior to actually manufacturing any form of aerodynamic body. As CFD has matured as a standard technology, it (CFD) has even replaced the need to physically test the aerodynamic effect of new aircraft designs [reference Boeing 787 design].

(performance racing)

In performance racing, the use of CFD has become so integrated in the design process that every F1 car manufacturer / performance team has a dedicated CFD department. Furthermore, the use of CFD has expanded past simply modeling aerodynamic effects it now encompasses energy efficiency predictions in the combustion engines of.

(microchannel flow boiling)

(cfd history)

[what is CFD]

[early implementations of CFD]

[development of SIMPLE algorithm]

[development of other algorithms]

[entering the digital age]

[hitting the gpu age]

(state of the art / difficulties with legacy CFD code)

Object / Scope

The scope of this study is to provide a framework for improving the computational performance of legacy CFD code, perform an optimization study of the current solutions to CFD problems, and determine the costs associated with implementing new technologies in legacy CFD code. The framework for improving the computational performance of legacy CFD code considers two classes of examples. The first class of examples, considers simple diffusion problems solved using the finite volume method and the centralized differencing scheme. This class of example is primarily used due to its simplicity as implementing parallelism on a simple problem provide a higher adherence of understanding when compared to a problem with increased complexity. The second class of examples focused on a two-phase flow boiling problem that incorporates a solution to the full, incompressible Navier-Stokes equations. This example is used to demonstrate the effectiveness of implementing parallelism in a complex, CFD problem.

The optimization study considers the rate of convergence for the solutions to sparse linear systems, the computational resource requirement for different solutions to sparse linear systems, and the duration of computation for given solutions on a variety of hardware platforms. The study also considers different means of implementation for each solution algorithm and how that implementation affects the performance metrics of the optimization study. The optimization study considers the same two classes of CFD problems as mentioned previously and considers cases for a 1-dimensional, 2-dimensional, and 3-dimensional environment.

Finally, the determination of cost associated with implementing each particular CFD solution is examined for both the simple diffusion problem and the two-phase boiling problem. The performance indications of cost are considered for the 1-dimensional, 2-dimensional, and 3-dimensional environments and assess the required revision and restructuring of legacy code necessary to implement new solution types.