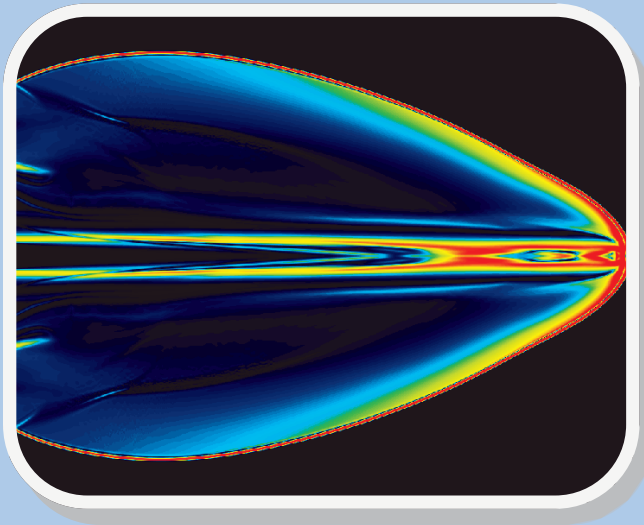


Dartmouth SIAM Student Chapter Presents

Computational Fluid Dynamics: History, Methods, and Modern Applications.



Abstract: Computational Fluid Dynamics (CFD) can be defined as the science of solving the equations which govern fluid flow, heat and mass transfer, and related physical phenomena using numerical techniques. Sophisticated CFD software packages have been developed over many years to provide efficient, robust, and accurate numerical solutions for a wide range of fluid flow problems. Applications are numerous and ubiquitous in science and engineering, and include numerical weather prediction, biomedical flows, the prediction of lift and drag on automobiles and aircraft, and simulation of industrial equipment performance, from combustors and turbines in jet engines to internal flow systems in chemical processing and manufacturing. In all cases, the principal goal of CFD is to gain a better understanding the flow physics which, in many cases, can not be fully determined using experimental observations and measurement techniques.

By

Franklyn J. Kelecy,

Ph.D., ANSYS Inc., Lebanon, NH.

May 21st 2012

Haldeman 041

6:00-7:30

Pizza will be served

The talk will provide a brief history of CFD, discuss the equations that are solved, and survey the numerical techniques that are used to obtain solutions. The methods primarily used in commercial CFD software packages will be emphasized. Following the talk, a brief demonstration of the ANSYS Inc. CFD software package will be given, wherein a simple fluid flow problem is created, meshed, and solved.

