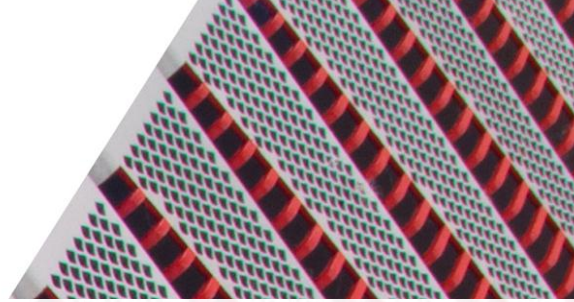


TRACC

TRANSPORTATION RESEARCH AND ANALYSIS COMPUTING CENTER



Training Course in Hydraulic and Aerodynamic CFD Analysis Using STAR-CCM+

March 27-28, 2013

Wednesday-Thursday

This course is designed for hydraulic and wind engineers and other analysts with knowledge of fluid mechanics but limited experience in using CFD software to analyze fluid flow problems. The course covers both CFD theory as well as practical applications of STAR-CCM+. Participants will be introduced to CFD principles, governing equations, physics models, data requirements, capabilities of the software, problem setup, post processing to graph and visualize results, and procedures for running large jobs in parallel on the TRACC clusters.

Hands on training is being planned in the form of tutorials that cover the steps needed to set up problems, run the analysis, and visualize the results. Temporary licenses for STAR-CCM+ will be provided to attendees at TRACC and to remote participants.

Remote Location Participation

The training sessions will also be broadcast over the Internet. The detailed information on remote participation via the internet will be provided to registered participants. Temporary licenses for the CFD software and tutorial files will also be provided to allow remote participants to work through the tutorials during the class. Remote participants will be able to ask questions.

Registration

Participation in the training course is free. Travel, lodgings, and other expenses are the responsibility of the participant. Please contact TRACC at the number or Email address shown below if you would like to participate in the training sessions either by internet or in person.

Training Site: Argonne National Laboratory

Building 222 | Second Floor
Rooms A253/C253

9700 South Cass Avenue
Argonne, IL 60439-4815

Contact information:

630-252-5290 | CFD_TRACC@anl.gov | www.tracc.anl.gov

STAR-CCM+ Training Sessions

Wednesday-Thursday

March 27-28, 2013

9:30 AM-4:30 PM (CST)

General Topics

Introductions and Agenda

Basics of Computational Fluid
Dynamics

STAR-CCM+ CFD Software Basics

STAR-CCM+ features

Graphical User Interface

Workflow

Meshing with STAR-CCM+

Post-Processing Basics

Becoming a TRACC User

How to use STAR-CCM+ on the
TRACC Cluster

Hydraulics and Aerodynamics Tutorials

Temporary licenses and STAR-CCM+
software will be provided to participants
to carry out a number of tutorials under
the guidance of instructors. Tutorials
will include:

- Free Surface Channel Flow
- Forces on a Flooded Bridge Deck
- Wind Load on a Sign over Highway
- Flow Through a Lab Scale Culvert
- River Bed Erosion / Mesh Morphing
- Propeller with Free Surface Model
- Truck Tire Spray under Bridge
- Dynamic Fluid Body Interaction with Bridge Cable Section
- DEM Particle-Particle Interaction
- Flow through the Street Drain Grates

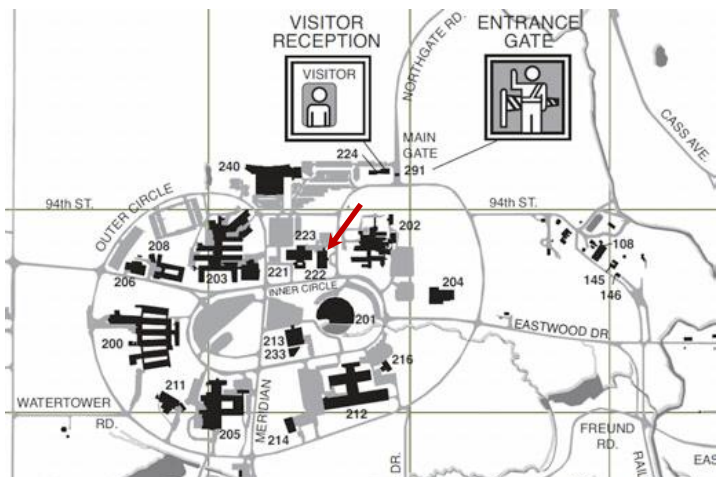


Daily Schedule

Sessions will start at 9:30 AM (CST)
and end at 4:30 PM (CST).

Lunch will be from 12 PM till 1:15 PM.
There will be a 10 minute break in
each half day session.

Training Location Map



[Map of Argonne campus](#)

Contact: 630-252-5290 | CFD_TRACC@anl.gov | www.tracc.anl.gov