



Free ANSYS Training & Consultation: April 16– 17, 2014

Finite-Element Analysis (FEA) & Computational Fluid Dynamics (CFD)

ANSYS, Inc. is pleased to offer free training and consultation sessions at Cornell University. The tools covered will be ANSYS Mechanical for FEA and ANSYS FLUENT for CFD. The training sessions will be a combination of presentations, demonstrations and hands-on workshops. Food will be provided at all sessions.

Location: 318 Philips Computer Lab

Date: April 16, 2014

Non-Linear Finite-Element Analysis: Best Practices

Time: 10:00 a.m. to 12:00 p.m.

Topics: Geometric, material & contact non-linearities, solution algorithms
Refreshments provided

Structural Mechanics: Industrial Applications

Time: 12:15 p.m. to 2:15 p.m.

Topics: Wind turbine blades, composite wings, bolts, gears, assemblies
Lunch provided

CAD Import /Clean Up and Structural Mechanics 3D Meshing

Time: 5:30 p.m. to 7:30 p.m.

Topics: CAD for FEA & CFD, 3D meshing techniques, complex assemblies
Dinner provided

Date: April 17, 2014

CFD: Turbulence Modeling in ANSYS FLUENT

Time: 10:00 a.m. to 12:00 p.m.

Refreshments provided

CFD: Industrial Applications

Time: 12:15 p.m. to 2:15 p.m.

Topics: Wind turbines, external aerodynamics, internal flows
Lunch provided

CFD: 3D Meshing, Post-Processing and Visualization for Complex Geometries

Time: 5:30 p.m. to 7:30 p.m.

Dinner provided

Open Office Hours for Consultations with ANSYS Engineers: April 16th and 17th, 3:00 to 4:30 p.m. (Held in the Swanson lab, 163 Rhodes)

Contact: Rajesh Bhaskaran (bhaskaran@cornell.edu)



<http://goo.gl/TX7LCo>

Registration is required