

ANSYS Seminar Series: University of Michigan

ANSYS is pleased to host a complimentary Seminar and Workshop Series at the University of Michigan.

This two-day series will allow University of Michigan faculty and students to engage one-on-one with ANSYS experts and learn the many ways in which ANSYS simulation tools are being used both in academia and the industry to solve complex, challenging engineering problems.

Presentation sessions of the event will involve ANSYS experts presenting real world examples and case studies.

Workshop sessions of the event will include hands-on, self paced workshops, allowing attendees to get experience with the software and to interact one-on-one with ANSYS engineers.

Information on how to obtain the newly released \$25 student version of ANSYS will also be made available.

Seminar Topics, Dates and Locations:

Computational Fluid Dynamics Seminar

Date: March 26, 2013

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

Location: 1180 Duderstat Center

Workshop Time: 1:00 p.m. to 4:00 p.m.

Location: Duderstat Center Windows Training Rooms 1&2

Structural Mechanics Seminar

Date: March 27, 2013

Presentation Time: 9:00 a.m. to 10:15 a.m.

Workshop Time: 10:15 a.m. to 12:00 p.m. and 12:45 p.m. to 2:00 p.m.

Location: Duderstat Center Windows Training Room 2

Location: University of Michigan

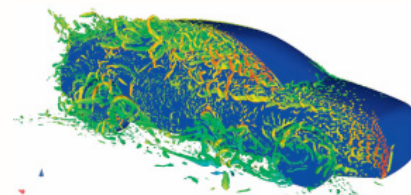
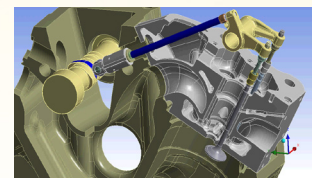
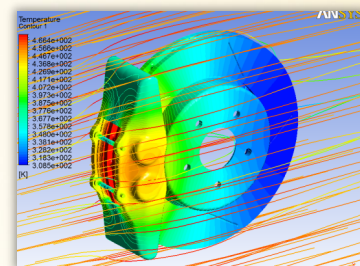
Dates: March 26 and 27, 2013

Cost: Free with Lunch Provided

Contact: Amadi Nwankpa
amadi@umich.edu

Sean Harvey
sean.harvey@ansys.com

Registration: www.ansys.com/umichigan



Seminar Session Agendas

March 26: Computation Fluid Dynamics Seminar

| | |
|-------------------------|---|
| 10:00 a.m. – 10:15 a.m. | Introduction |
| 10:15 a.m. – 12:00 p.m. | <p>CFD Presentation</p> <p>This seminar introduces the basic concepts of CFD (Computational Fluid Dynamics) through industrial examples and success stories from a variety of different industries. It is intended to provide a high level overview of CFD. Typical applications of CFD are engineering problems involving fluid flows and heat transfer combined with phenomena such as turbulence, chemical reactions, particulates, multiple-phase flows with or without phase change, as well as fluid-structure interactions (FSI). CFD usage for internal combustion engines, moving-boundaries problems, bio-fluid, electronics cooling, as well as external aerodynamics will be among the many application areas discussed.</p> |
| 12:00 p.m. – 1:00 p.m. | Lunch (1180 Duderstat Center Foyer) |
| 1:00 p.m. – 4:00 p.m. | CFD Hands-on Workshops |

March 27: Structural Mechanics Seminar

| | |
|-------------------------|--|
| 9:00 a.m. – 9:15 a.m. | Introduction |
| 9:15 a.m. – 10:15 a.m. | <p>Structural Mechanics Presentation</p> <p>This seminar will discuss the ANSYS Mechanical software capabilities, highlight the Industry usage of ANSYS Mechanical, and provided overview examples. Capabilities such as meshing, linear and non-linear solutions, multi-body kinematics, contact, working with large assemblies, among others will be discussed. Multiphysics capabilities, fluid-structure interaction (FSI) and pre and post processing of complex composite models will also be illustrated through examples.</p> |
| 10:15 a.m. – 12:00 p.m. | Structural Mechanics Hands-on Workshops |
| 12:00 p.m. – 12:45 p.m. | Lunch (1180 Duderstat Center Foyer) |
| 12:45 p.m. – 2:00 p.m. | Structural Mechanics Hands-on Workshops (continued.) |