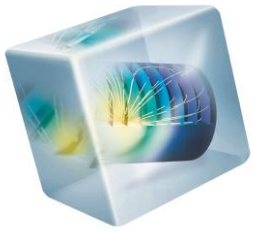
Finite Element Modeling Using COMSOL Multiphysics Workshop



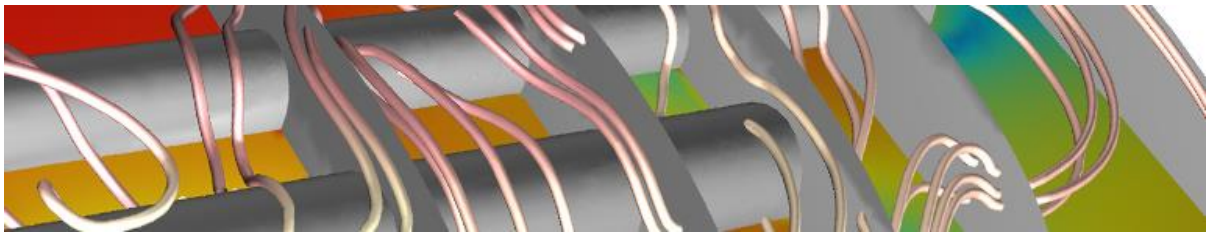
Date: Tuesday, March 17, 2015

Time: 9:00 AM to 3:00 PM

Place: King Khaled University

Event Page: .....

Join us for this unique opportunity to advance your skills in multiphysics simulation. This workshop day begins with a walk-through of the fundamental modeling steps in COMSOL Multiphysics. Attendees will then have the chance to set up and solve a simulation through a hands-on exercise, guided by a COMSOL expert. You will leave with new skills to work on your own applications using your free, two-week COMSOL trial.



The COMSOL Multiphysics simulation software environment facilitates all steps in the modeling process – defining your geometry, material, specifying your physics, meshing, solving, and then visualizing your results.

Model set-up is quick, thanks to a number of predefined physics interfaces for applications ranging from fluid flow and heat transfer to structural mechanics. Material properties, source terms and boundary conditions can all be arbitrary functions of the dependent variables.

Predefined multiphysics-application templates solve many common problem types. You also have the option of choosing different physics and defining the interdependencies yourself. Or you can specify your own partial differential equations (PDEs) and link them with other equations and physics.

COMSOL Multiphysics operates as the primary tool for all your future modeling needs. Its versatility, flexibility and usability can easily be extended with its add-on modules unlike any other FEA Software solution.

Discover the capabilities and features of COMSOL Multiphysics and get a quick overview of the add-on products. Learn the natural workflow of the COMSOL Desktop user interface through which all physical phenomena are set up. See how to efficiently create and modify your models, and optimize your designs, step-by-step. Experience the speed and ease of modeling in the COMSOL environment, shown through a hands-on multiphysics simulation example.

You don't need to be an expert in modeling in order to appreciate the seminar.

Workshop Agenda
Finite Element Modeling Using COMSOL Multiphysics
17th of March, 2015

08:45 AM to 09:00 AM	**Registration**
09:00 AM to 10:30 AM	**Intro to COMSOL Multiphysics**
10:30 AM to 11:00 AM	**Tutorial session: Getting Started with COMSOL** • Parallel Plate Capacitor • Stresses and Strains in a Wrench
11:00 AM to 12:00 PM	**Tutorial session: Getting Familiar with COMSOL** • Thermal Cube • Heat Sink
12:00 PM to 01:00 PM	**Coffee Break & Prayer**
01:00 PM to 03:00 PM	**Tutorial session: Application in Multiphysics Modeling** • Microresistor Beam • Shell & Tube Heat Exchanger • Sea Bed Logging

According to the length of the workshop and the interests of the participants, some of the above models will be considered.

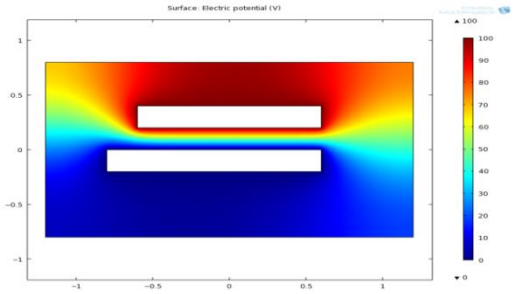
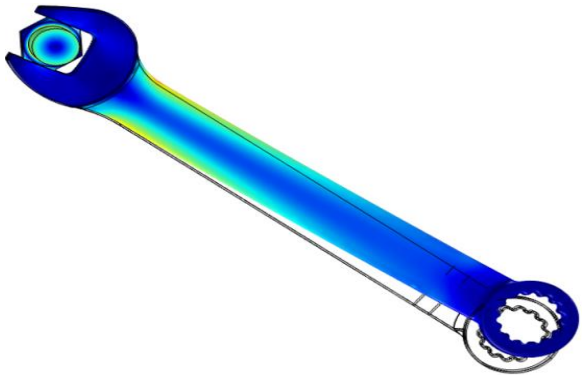
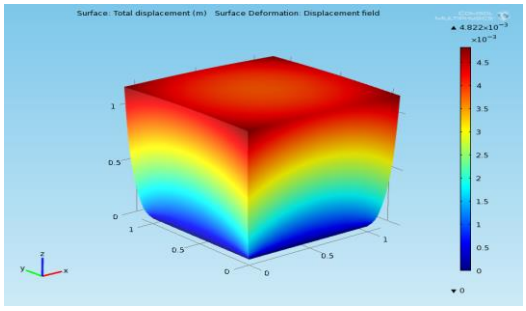
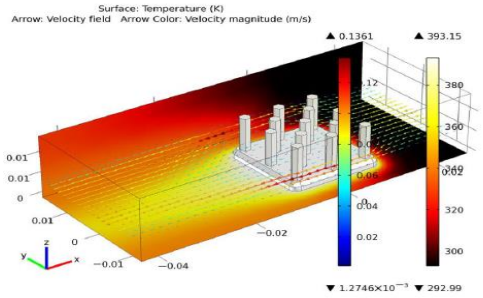
Speaker

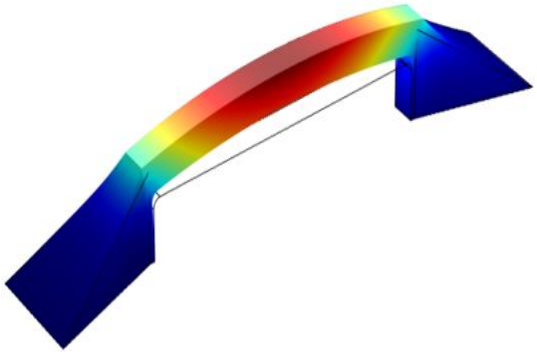
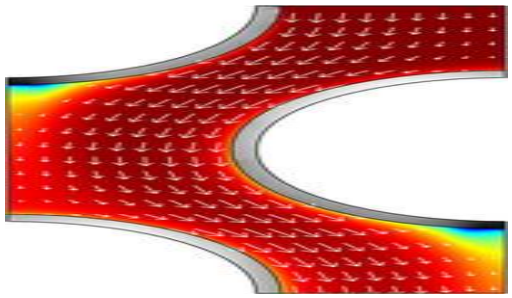
Eng. Ahmed Gamal

Technical Engineer Graduated from the Faculty of Engineering, Cairo University working as COMSOL technical support and application engineer at Elnady Engineering & Agencies.

Elnady Engineering & Agencies • 2 Elgabal Elakhdar Buildings • 11471 Cairo, Egypt
Phone: +20-2-2342 57 63 • Fax: +20-2-2342 17 93
Email: info@elnadycompany.com •Web: www.elnadycompany.com

Model Definition

• . Parallel Plate Capacitor This model main objective is to get familiar with COMSOL interface, Sketching geometries and How to have a domain “not active” in the analysis.	
• Stresses and Strains in a Wrench This tutorial demonstrates how to set up a simple static structural analysis. The analysis is exemplified on a combination wrench during the application of torque on a bolt. Despite its simplicity, and the fact that very few engineers would run a structural analysis before trying to turn a bolt, the example provides an excellent example of structural analysis in COMSOL Multiphysics.	
• Thermal Cube This model main objective is to demonstrate Multiphysics capabilities in COMSOL by modeling coupled Heat Transfer in solids with convective cooling and Structure mechanics to show the effect of thermal expansion.	
• Heat Sink This model is intended as a first introduction to simulations of fluid flow and conjugate heat transfer. It shows you how to: Draw an air box around a device in order to model convective cooling in this box. Set a total heat flux on a boundary using automatic area computation, and display results in an efficient way using selections in data sets.	

• Microresistor Beam This example illustrates the ability to couple thermal, electrical, and structural analysis in one model. This particular application moves a beam by passing a current through it; the current generates heat, and the temperature increase leads to displacement through thermal expansion. Although the model involves a rather simple 3D geometry and straightforward physics, it provides a good example of multiphysics modeling.	
• Shell and Tube Heat Exchanger In this model, you will study a part of a shell-and-tube heat exchanger, where hot water enters from above. The cooling medium flows through the tubes that, in this model, impose a constant temperature at the walls. Furthermore, the tubes are assumed to be made of stainless steel and the heat flux is also modeled through them.	
• Sea bed Logging The Controlled Source Electromagnetic (CSEM) method for oil prospecting has emerged as a promising technique during recent years. The method uses a mobile horizontal electric dipole transmitter and an array of seafloor electric receivers. The seafloor receivers measure the low frequency electrical field generated by the source. Some of the transmitted energy is reflected or guided by the resistive reservoir and results in a higher received signal. The model here demonstrates this method. It uses a mobile, horizontal 1 Hz electric dipole antenna that is towed 150 m above the sea floor. Arrays of sea floor receivers measure the electric field at various distances from the antenna.