Computational Fluid Dynamics

Author: Rajesh Bhaskaran, Cornell University

This module is from our free online simulations course at edX.org (sign up here). The edX interface provides a better user experience, so we recommend going through the module there.

1. Introduction

What is Computational Fluid Dynamics or CFD?

CFD Software We'll Use

What's Under the CFD Blackbox?

Commercial CFD software such as ANSYS Fluent is a blackbox for the CFD user. It is important for the user to know the major elements of what's under the blackbox in order to use the tool effectively and avoid "garbage in, garbage out". Earlier, we discussed a framework that lays out the major elements of what's under the blackbox and used that in the FEA case studies. Please review that section since we'll be using that framework for the CFD case studies also.

Some Cool CFD Animations

Go to Step 2. Finite Volume Method

Go to all (FLUENT) Learning Modules