FLUID DYNAMICS SIMULATIONS USING ANSYS CERTIFICATE

Program Description

From aerospace to manufacturing, mechanical engineers often have a need to model computational fluid dynamics (CFD) through simulation.

This certificate program empowers you to create reliable and validated simulations without the need to focus on all the underlying mathematics. Using a proven methodology, these courses will help you approach CFD problems like an expert. Throughout the courses, you will simulate a variety of 2D and 3D flows, such as flows over a car body, cooling fan, and airplane body, using Ansys, the leading simulation platform for industrial applications. You will then apply the insights and experience gained in the coursework to solve a new problem of your own. After practice with problem-based learning methodology using various forms of computer models, you will be able to apply this approach to create your own simulations for a wide array of situations.

You will be most successful in this course if you have a strong foundation in high school-level calculus, physics, and algebra. For the best experience in this program, it is strongly recommended to take these courses in the order that they appear.

For the best experience in this program, we strongly recommend you use a desktop version of Ansys. If you do not already have access to Ansys, be sure your machine meets the following hardware requirements in order to download and use the free student desktop version of Ansys:

For Desktop Application:

- Supported Platforms and Operating Systems
- Microsoft Windows 10, 64-bit
- Minimum Hardware Requirements
- Processor(s): Workstation class
- 4 GB RAM
- 25 GB hard drive space
- Computer must have a physical C:/" drive present
- Graphics card and driver: Professional workstation class 3-D
- OpenGL-capable

If your machine does not meet these requirements, you may also use an online version of Ansys. Be sure you meet the following bandwidth requirements:

For Web Application:

- Bandwidth Requirements
- 5Mbps download speed
- 100 ms maximum roundtrip latency

Key Takeaways

- Create simulations using Ansys software for a range of practical flow problems
- Explain the mathematical model underlying each simulation, including governing equations, boundary conditions, physical principles, and assumptions
- Predict expected results using hand calculations
- Defend simulation results by undertaking a "verification and validation" procedure

What You'll Earn

- Fluid Dynamics Simulations Using Ansys Certificate from Cornell Engineering, Sibley School of Mechanical and Aerospace Engineering
- 70 Professional Development Hours (7 CEUs)

Who Should Enroll

- Engineers who work with computer-aided design (CAD) software
- Engineering analysts
- Simulation engineers
- Mechanical engineers
- Naval engineers
- Aerospace engineers
- Physicists

Total Investment

- 2.5 months to complete all the courses

How To Enroll

For more information on how to enroll, please visit Fluid Dynamics Simulations Using Ansys (https://ecornell.cornell.edu/certificates/engineering/fluid-dynamics-simulations-using-ansys/).

Courses

Code	Title	Hours
eCornell MAE111	Foundations of CFD	0
eCornell MAE112	2D Laminar Flows	0
eCornell MAE113	3D Turbulent Flows	0
eCornell MAE114	Rotating Machinery Flows	0
eCornell MAE115	Compressible Flows	0