

Training at PDSVISION

Training is one of the best investments a company can make; it is not a huge expense, it creates a better quality of work, it creates efficiency and it is fun! PDSVISION is proud to be a Certified Platinum Training Provider of PTC courses.

Whether you are a first-time or an experienced user we offer suitable training for your needs. Greater knowledge contributes to increased efficiency and higher quality of work.

Ansys Fluent - Getting Started - Part 1

Course Length - 1 Day

Course Description:

This course teaches the essential skills you will need to start performing basic computational fluid dynamics (CFD) simulations on your own using Ansys Fluent software. Instructor demos and hands-on workshops cover every aspect of the end-to-end CFD workflow, including working with CAD models in Ansys SpaceClaim, creating quality CFD meshes with Ansys Meshing, and all aspects of performing CFD simulations in ANSYS Fluent. Course modules are organized into geometry (Ansys SpaceClaim), meshing (Ansys Meshing) and solver (Ansys Fluent) topics, and each of the three course modules takes students through a complete end to end workflow of a different CFD problem.

This course is part of a two course sequence. The second course in the sequence, Ansys Fluent Getting Started – Part 2, builds on skills learned in Part 1 to enable students to become proficient in performing basic CFD simulations. Part 2 is strongly recommended for all students upon completion of Part 1.

Prerequisites:

A technical education and background in fluid mechanics and heat transfer is recommended but not mandatory. An engineering degree is not required.

Target Audience:

Engineers and Designers

Teaching Method:

Lectures and computer practical sessions to validate acquired knowledge.