Rand SIM
Innovation Through Insight

Ansys part of Synopsys — ELITE CHANNEL PARTNER

ANSYS FLUENT FUNDAMENTALS

Whether you're exploring CFD for the first time or adding simulation to your design process, this session is for you. We'll cover the essentials of flow, turbulence, and heat transfer to help you feel confident using Fluent.

SIMULATION WORKFLOWS

Intro to Discovery Modeler, CAD interaction, geometry cleanup, volume extraction, named selections, and meshing basics (size, accuracy, boundary layers)

FLUENT MODELING

Fluent GUI, physics and setup options, materials (custom & Granta), boundary conditions, monitors, data collection, and post-processing. Intro to user-defined functions (UDFs), expressions, porous media, and basic multiphase flow.

FLUENT BEST PRACTICES

Industry best practices, meshing & sanity checks, and model inspection (qualitative and quantitative)

HANDS-ON APPLICATION

Full CFD simulation exercise from start to finish, team collaboration, and open Q&A.

Available for quarterly open-enrollment or for company exclusive sessions.

**Content may vary depending on time required for basic topics to be completed*

www.randsim.com

Connect with a Rand SIM Rep Today!

888-483-0674