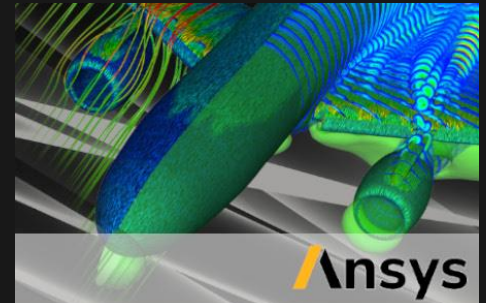


Ansys CFD

6 November 2020



Agenda

09:00 – 09:30	Welcome
09:30 - 12:30	<p>Ansys CFD Overview</p> <p>Preprocessing for CFD simulations</p> <p>Workshop 01 – Global controls for mesh sizing and inflation</p> <p>Workshop 02 – Multiple methods and local controls combined to create a conformal hybrid mesh</p> <p>CFD workflow</p> <p>Workshop 03 – Fluid flow and heat transfer in a mixing tee</p>
12:30 – 13:00	Quick Lunch
13:00 – 17:00	<p>Heat transfer</p> <p>Workshop 04 – Electronics cooling with natural convection and radiation</p> <p>Coupled field analysis</p> <p>Workshop 05 – One way fluid structure interaction for a flow probe</p> <p>Multi-species flow and postprocessing</p> <p>Workshop 06 – The release of heat and combustion gases from an engine fire in a ventilated garage</p>