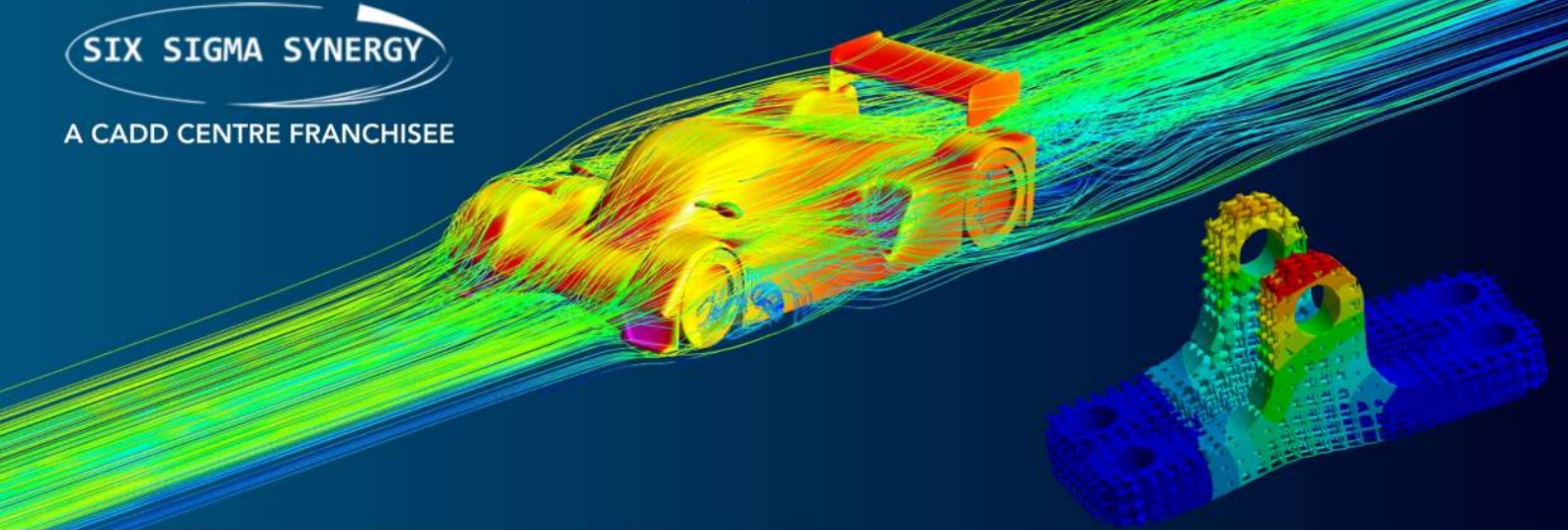# ANSYS FLUENT
Duration: 3 Days

SIX SIGMA SYNERGY

A CADD CENTRE FRANCHISEE

ANSYS FLUENT is a powerful computational fluid dynamics (CFD) software for optimizing product's performance. Its broad physical modeling ability help model flow, turbulence, and heat transfer. This knowledge of input values and result analysis will enhance productivity and performance.

COURSE CURRICULUM

DAY 1	GUI, Model Setup With Meshing, Creating Enclosures/Fluid Domain
DAY 2	Assigning Fluid Properties, External & Internal Flow System, Natural & Force Convection With Boundary Conditions (Thermal Flow Case)
DAY 3	Finding Out Drag & Life With Monitors, Plot Results With Contour & Vector Plot, Post Processing