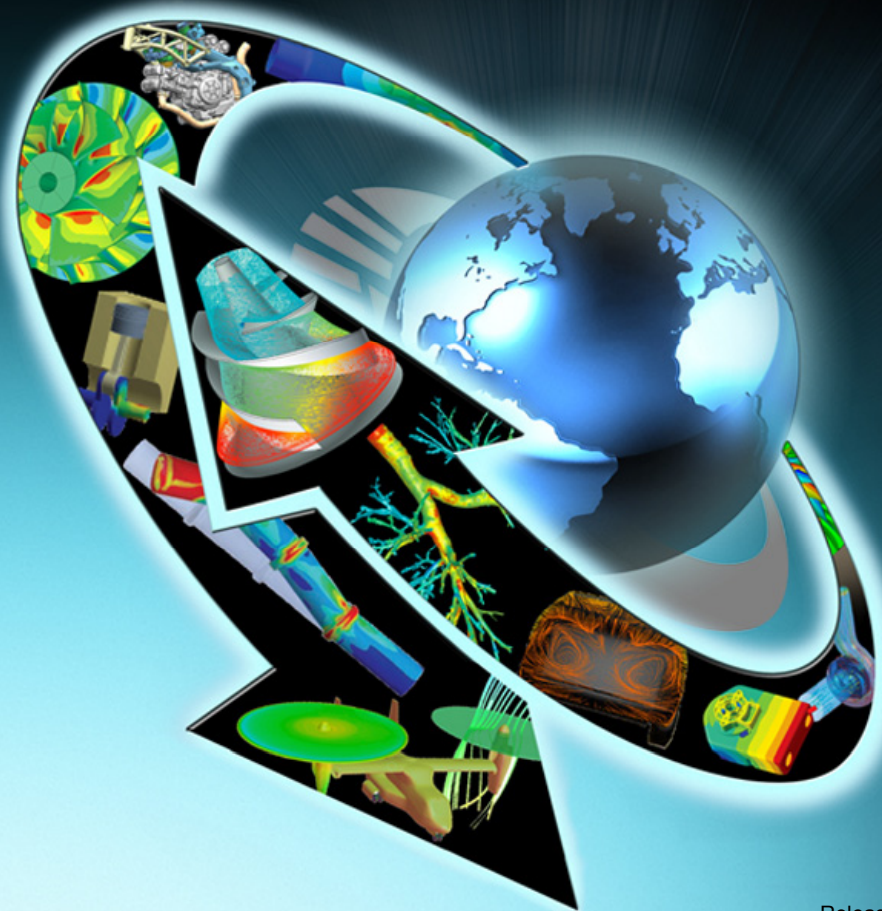




Customer Training Material

Table of Contents

Introduction to ANSYS FLUENT



1.	Welcome and Introduction to ANSYS	1-1
2.	Introduction to CFD	2-1
3.	Overview of the FLUENT Interface	3-1
4.	Boundary Conditions and Cell Zones	4-1
5.	Solver Settings	5-1
6.	Turbulence Modeling	6-1
7.	Heat Transfer Modeling	7-1
8.	User-Defined Functions	8-1
9.	Advanced Physics	9-1
10.	Transient Flow Modeling	10-1
11.	Postprocessing with FLUENT and CFD-Post	11-1