



	ANSYS® CFX® 11	CFX-Flo 11	FLUENT® 6.3	FloWizard® 3.0	FLUENT® for CATIA® V5 3.1	POLYFLOW® 3.11		
<b>Usability</b>								
Easy wizard interface with built-in expert guidance for all functions				•				
Wizard interface for basic setup	•	•						
ANSYS® Workbench™ integration	•	•						
Custom applications (service option)				•				
Fully embedded in CATIA V5					•			
<b>Geometry</b>								
Imports industry-standard geometry formats	<b>For geometry functions, compatible with other ANSYS Workbench integrated tools such as ANSYS® DesignModeler™ and CFX-Mesh</b>		<b>Comes with GAMBIT®</b>		<b>CATIA V5 based</b>		<b>Comes with GAMBIT</b>	
Full-featured geometry creation								
CAD geometry manipulation								
Manually controlled flow volume extraction								
Automatic flow volume extraction								
Manually controlled geometry cleanup								
Semi-automatic geometry cleanup								
Automatic geometry cleanup mode								
<b>CAD Connections</b>								
CAD connectivity to SolidWorks®, Inventor®, NX®, Pro/ENGINEER®	<b>Through ANSYS Workbench and ANSYS DesignModeler</b>		<b>Comes with GAMBIT</b>				<b>Comes with GAMBIT</b>	
CAD associativity to NX and Pro/ENGINEER								
CAD associativity with GAMBIT								
<b>Mesh</b>								
Fully featured manually controlled mesher	<b>Multiple Options</b>	<b>CFX-Mesh</b>	<b>Comes with GAMBIT</b>	<b>Optional (GAMBIT)</b>	<b>CATIA V5 based</b>	<b>Comes with GAMBIT</b>		
Built-in automated mesher				•				
Import industry standard mesh formats	•	•	•	•		•		
Handles fully unstructured meshes	•	•	•	•	•	•		
Solution-based mesh adaption	•	•	•	•		•		
Polyhedral mesh conversion to increase solution speed	NA	NA	•	•	•	NA		
<b>Physics and Boundary Conditions</b>								
Variety of inlet and outlet BC	•	•	•	•	•	•		
Steady state flow	•	•	•	•	•	•		
Transient flow	•	•	•	•	•	•		
2-D flow (dedicated solver option)			•			•		
2-D flow (using thin 3-D segment)	•	•	•	•	•	•		
3-D flow	•	•	•	•	•	•		
Time-dependent boundary conditions	•	•	•	•	•	•		
Incompressible flow	•	•	•	•	•	•		
Compressible flow	•	•	•	•	•	•		
Natural convection	•	•	•	•	•	•		
Fan model	•	•	•	•	•			
Periodic domains	•	•	•			•		
Porous media	•	•	•	•	•	•		
Heat transfer	•	•	•	•	•	•		
Conjugate heat transfer	•	•	•	•	•	•		
Non-Newtonian viscosity	•	•	•	•	•	•		
Visco-elastic flow option						•		
Turbulence (isotropic)	•	•	•	•	•			
Turbulence (anisotropic/strong swirl/RSM)	•	•	•	•	•			
Turbulence (LES)	•	•	•					

	ANSYS® CFX® 11	CFX-Flo 11	FLUENT® 6.3	FloWizard® 3.0	FLUENT® for CATIA® V5 3.1	POLYFLOW® 3.11
Rotating equipment (MRF/frozen-rotor)	•		•	•	•	
Rotating equipment (sliding mesh/stage or MST)	•		•			•
Rotating and intermeshing equipment	•		•			•
Moving–deforming mesh	•	•	•			•
Flow-driven solid motion	•		•			
Internal radiation–participating media	•	•	•	•		
Internal radiation–transparent media	•		•	•		
External radiation	•		•	•		•
Solar radiation & load	•		•			
Species modeling	•	•	•			•
Flow pathlines (massless)	•	•	•	•	•	•
Particle tracking (with mass)	•		•			
Coupled discrete phase modeling	•		•			
Acoustics	•		•			
Chemical reaction	•		•			•
Combustion	•		•			
Cavitation	•	•	•			
Multiphase (Eulerian)	•		•			
Multiphase (free surface)	•	•	•			•
Fluid structure interaction (FSI; implementation and capabilities vary)	•		•		•	•
Specialty extrusion models						•
Inverse extrusion die design						•
Specialty film casting (membrane) models						•
Specialty blow molding models						•
Specialty thermoforming models						•
Specialty mold filling models						•
Specialty fiber spinning models						•
<b>Post-Processing</b>						
Plane cuts	•	•	•	•	•	•
Iso-surfaces	•	•	•	•	•	•
Velocity vectors	•	•	•	•	•	•
Pathlines	•	•	•	•	•	•
Contour plots	•	•	•	•	•	•
Symmetry mirroring	•	•	•			•
Periodic repeats	•	•	•			•
X–Y charts	•	•	•			•
Automated generation of HTML reports with embedded images	•	•		•	•	
Animations	•	•	•	•	•	•
Basic quantitative post-processing	•	•	•	•	•	•
Advanced quantitative post-processing	•	•	•			•
Statistical particle track analysis						•
<b>Solver Options</b>						
Full control over numerics and solving	•	•	•			•
Fully automated numerics and solving				•	•	
Pressure-based coupled solver	•	•	•			
Density-based coupled solver			•			
Finite element method solver						•
Pressure-based segregated solver			•	•	•	
Remote solve (FLUENT RSolve) option			•	•		•
Parallel solving on local PC option	•	•	•	•	•	•
Parallel solving over network option	•	•	•		•	
Upward compatibility with ANSYS CFX	NA	•				
Upward compatibility with FLUENT			NA	•	•	



ANSYS, Inc.  
Southpointe  
275 Technology Drive  
Canonsburg, PA 15317  
U.S.A.  
724.746.3304  
ansysinfo@ansys.com

Toll Free U.S.A./Canada:  
1.866.267.9724  
Toll Free Mexico:  
001.866.267.9724  
Europe:  
44.870.010.4456  
eu.sales@ansys.com

ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICFM CFD is a trademark used under license. All other brand, product, service and feature names or trademarks are the property of their respective owners. Image Credits: Some images courtesy Aavid Thermalloy, ICT Prague and Silesian University of Technology—Institute of Thermal Technology.

© 2007 ANSYS, Inc. All Rights Reserved. Printed in U.S.A.

11-07