



	ANSYS® CFX® 11	CFX-Flo 11	FLUENT® 6.3	FloWizard® 3.1	FLUENT® for CATIA® V5 4.0	POLYFLOW® 3.12
Usability						
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					•	
Geometry						
Imports industry-standard geometry formats	For geometry functions, compatible with other ANSYS Workbench integrated tools such as ANSYS® DesignModeler™ and CFX-Mesh		Comes with GAMBIT® and TGrid™	•	CATIA V5 based	Comes with GAMBIT
Full-featured geometry creation				Optional (GAMBIT)		
CAD geometry manipulation				•		
Manually controlled flow volume extraction				Optional (GAMBIT)		
Automatic flow volume extraction				•		
Manually controlled geometry cleanup				•		
Semi-automatic geometry cleanup				•		
Automatic geometry cleanup mode				•		
CAD Connections						
CAD connectivity to SolidWorks®, Inventor®, NX®, Pro/ENGINEER®	Through ANSYS Workbench and ANSYS DesignModeler		Comes with GAMBIT and TGrid	•		Comes with GAMBIT
CAD associativity to NX and Pro/ENGINEER				•		
CAD associativity with GAMBIT				•		
Mesh						
Fully featured manually controlled mesher	Multiple Options	CFX-Mesh	Comes with GAMBIT and TGrid	Optional (GAMBIT)	CATIA V5 based	Comes with GAMBIT
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
Physics and Boundary Conditions						
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® 3.1	FLUENT® for CATIA® V5 4.0	POLYFLOW® 3.12
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						•
Post-Processing						
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						•
Solver Options						
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 R Solve) option			•	•		•
Parallel solving on local PC option	•	•	•	•	•	•
Parallel solving over network option	•	•	•		•	
Upward compatibility with ANSYS CFX	NA	•				
Upward compatibility with FLUENT			NA	•	•	



GSA Contract Holder

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.

© 2008 ANSYS, Inc. All Rights Reserved. Printed in U.S.A.

MKT0000271 9-08