



E06 - LOGICIEL Fluent

PRINCIPE :

FLUENT is the CFD solver of choice for complex flows ranging from incompressible (low subsonic) to mildly compressible (transonic) to highly compressible (supersonic and hypersonic) flows. Providing multiple choices of solver options, combined with a convergence-enhancing multigrid method, FLUENT delivers optimum solution efficiency and accuracy for a wide range of speed regimes. The wealth of physical models in FLUENT allows you to accurately predict laminar and turbulent flows, various modes of heat transfer, chemical reactions, multiphase flows, and other phenomena with complete mesh flexibility and solution-based mesh adaption.

TYPE :

Version 6.3.26

CARACTERISTIQUES :

- 2D planar, 2D axi-symmetriC, 2D axi-symmetric with swirl, and 3D flows
 - Unstructured mesh (triangle and quadrilateral elements for 2D; tetrahedral, hexahedral, prism and pyramid elements for 3D)
 - Steady-state or transient flows
-

