Computational fluid dynamics

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

A landmark study by Lin et al. addressed the problem of fluid characteristics in ventricular catheters using a two-dimensional simulation program of computational fluid dynamics (CFD).

Most commercially available ventricular catheters have an abnormally increase flow distribution pattern. New catheter designs with variable hole diameters along the catheter tip will allow the fluid to enter the catheter more uniformly along its length, thereby reducing the probability of its becoming occluded [1].

see Computational fluid dynamics for intracranial aneurysms

1)
Galarza M, Giménez A, Valero J, Pellicer OP, Amigó JM. Computational fluid dynamics of ventricular catheters used for the treatment of hydrocephalus: a 3D analysis. Childs Nerv Syst. 2014 Jan;30(1):105-16. doi: 10.1007/s00381-013-2226-1. Epub 2013 Jul 24. PubMed PMID: 23881424.