

Computational Fluid Dynamics

Commonly used acronym: CFD

Created on: 09-04-2020 - Last modified on: 10-04-2020

Contact person

Mohammad Rahimi Gorji

Organisation

Name of the organisation Ghent University (UGent)

Department Faculty of Medicine and Health Sciences

Country Belgium

Geographical Area Flemish Region

SCOPE OF THE METHOD

| | |
|----------------------------------|--|
| The Method relates to | Animal health, Environment, Human health |
| The Method is situated in | Translational - Applied Research |
| Type of method | In silico |

DESCRIPTION

Method keywords

computational fluid dynamics

in silico

Mathematical modeling

Numerical method

Scientific area keywords

Biomedical Engineering

biomechanics

Medical physics

Bio-engineering

Method description

Computational Fluid Dynamics (CFD) is being applied to characterize the fluid flow in different applications. CFD has obtained significant interest in both the medical and engineering community because of its non-invasive character. It can predict the fluid flow characteristics when one or multiple input flow variables are changed. In addition, it permits investigation of different flow variables and fluid forces to a level of fine detail.

Method status

Still in development

Published in peer reviewed journal

PROS, CONS & FUTURE POTENTIAL

Advantages

- Time consuming ;
- Ethics aspects (for human and animals) ;
- Cost reduction ;
- Quick assessment of design variations ;
- Possibility of simulation for different conditions ;
- Comprehensive information.

Challenges

Can be developed more and more always.

REFERENCES, ASSOCIATED DOCUMENTS AND OTHER INFORMATION

Coordinated by



Financed by

