

# 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

<b>The Method relates to</b>	Animal health, Environment, Human health
<b>The Method is situated in</b>	Translational - Applied Research
<b>Type of method</b>	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

