

# FSI Project

Spin off

University of Rome Tor Vergata



---

**OBJECT:** Fluid Structure Interaction (FSI) Modeling in ANSYS Fluent

## PRODUCT FEATURES

A dedicated FEM solver has been developed through the implementation of User Defined Function (UDF) coding as plug-in for ANSYS Fluent.

Dedicated Graphical User Interface (GUI) and Text User Interface have been developed.

The product aim is to allow FSI problems simulation entirely within ANSYS Fluent; the present approach uses the CFD point of view, focusing on how the interaction with the structures influences the fluid behaviour; the CFD user is now allowed to set structure material properties and constraints in order to easily simulate its presence in a very family way, evaluating at same time both structural and fluid dynamics behaviours of the analyzed shapes, to quantify, visualize and optimize the performances in the design process.

## MAIN FUNCTIONALITY

- General robust FEM solver implemented
- Structural simulations in the range of small and large displacements, including non linear and contact problems
- 2D model for beams with implicit and explicit FEM approach
- 3D model for shells structure with implicit and explicit FEM approach
- Parallel implementation
- Static and dynamic transient analysis
- Multiphase and turbulence ANSYS Fluent model
- Post-processing with ANSYS Fluent

## PRODUCT STRENGTHS

- High efficiency
- Very light
  - ⇒ The result is a very light code that leaves almost unaffected the global CPU time
- General purpose
- CFD users oriented
- There are no interpolation errors
  - ⇒ Interpolation errors are avoided by the use of conformal meshes at the interface between fluid and solid
- No other software knowledge is required
- Highly customizable for client-oriented solutions, services and consultancy