UNIVERSITÀ DEGLI STUDI DI PADOVA

Centro di Ateneo di Studi e Attività Spaziali "Giuseppe Colombo"

PROPOSAL OF RESEARCH ACTIVITY

A NOVEL NUMERICAL METHOD FOR FLUID-STRUCTURE INTERACTION PROBLEMS

Scuola di Dottorato in Scienze Tecnologie e Misure Spaziali (STMS)
Curriculum: Sciences and Technologies for Aeronautics and Satellite Applications (STASA)

Ciclo XIII

PhD Student: Federico Dalla Barba
Matricola: 1173676

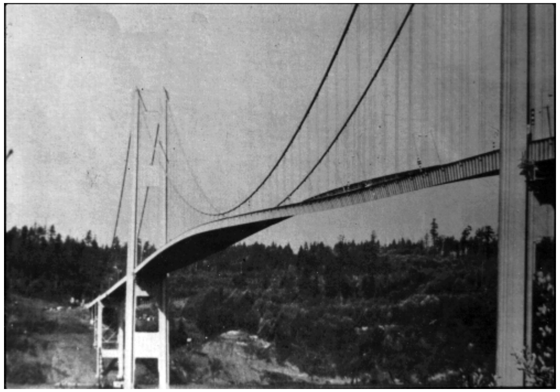
Fluid-Structure Interaction (FSI) in aeronautics: aeroelastic flutter

- **Aeroelastic flutter of lifting surfaces**: dynamic, **self-sustained instability** that affects flexible aerodynamic surfaces invested by a free stream. It is caused by the **competition between fluid forces and the elastic forces** generated as a response of the deflection of the body. **Can lead to catastrophic failure**.

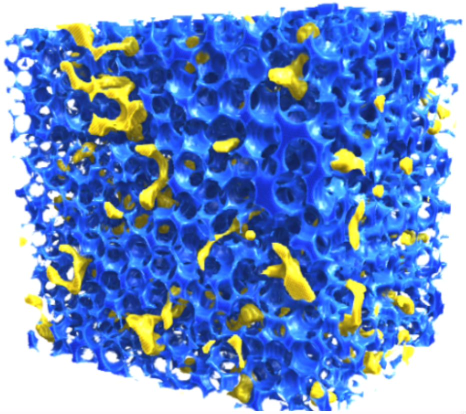
FSI problems in engineering and sciences

FSI: Interaction among a fluid flow and a rigid or deformable solid media.

- The interaction occurs through a **sharp solid-fluid interface**.
- The solid and fluid domains are governed by **different constitutive equations**.

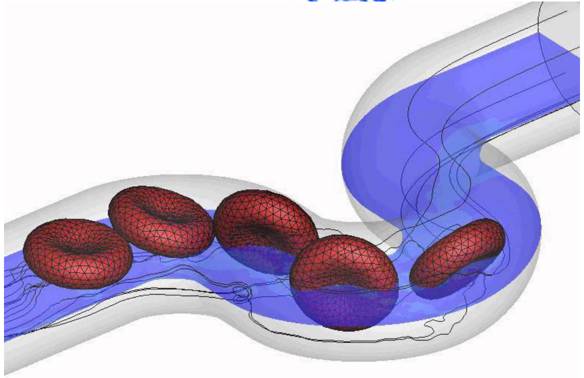


FSI problems are multidisciplinary, strongly non-linear problems. To date, a satisfactory comprehension of FSI problems remains a challenge and the capabilities of existing numerical models for applications are still limited.



Multidisciplinary applications:

- *Aeroelastic flutter of lifting surfaces.*
- *Aeroelasticity of structural elements in civil engineering.*
- *Erosion of immersed bodies.*
- *Flows in porous media.*
- *Cells in physiological flows.*



FSI problems: state of the art

Possible approaches to FSI problems:

❖ **Simplified models**

- **Analytical solutions:**
  - Applicable to simple cases.
  - Limited capabilities of reproducing complex dynamics.
- **Boundary Element Methods**:
  - Equations are not solved inside fluid domain.
  - Fluid stress and pressure are estimated on solid/fluid interface.
  - Limited capabilities of reproducing complex dynamics.

❖ **Complex models**

- **Finite Element Method and CFD solver:**
  - Capabilities of reproducing complex dynamics.
  - High computational cost.
  - Parallelization problems due to non-locality.
  - Usually only mean flow is solved (RANS).

❖ **Experimental test**

- Expensive.
- Scale models are often necessary.

The most part of models and numerical approaches do not resolve turbulent structures in the fluid domain but only mean flow.

Objectives of the proposed research activity

TASK1 - Development of an innovative numerical tool for FSI problems:

- *Incompressible* ***Navier-Stokes*** *equations solver equipped with turbulence closure model (Large Eddy Simulation).* ***Largest turbulent structures are resolved****.*
- *Peridynamics solver based on massively parallelized particle dynamics engine.*
- ***Immersed Boundary Method*** *(IBM) for coupling fluid/solid dynamics.*
- ***Massive parallelization*** *to optimize computational efficiency.*
- *Capabilities of working on computer clusters with* ***high scalability****.*

TASK 2 - Validation of the code and numerical investigation of aeroelastic flutter:

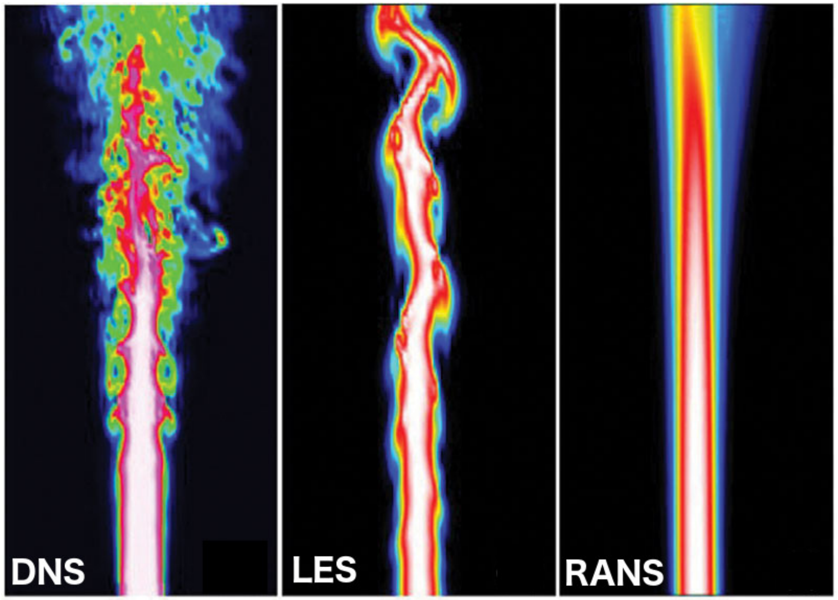
- *Validation tests.*
- *Numerical simulation of the aeroelastic flutter of a scale wing model.*

Large Eddy Simulation (LES)

Usually FSI simulations solve only the mean flow (RANS).

LES: largest turbulent structures of the fluid flow are **directly resolved** on the computational fluid grid.

Turbulent structures are non-stationary and affect the dynamics of solid structures driving oscillations over a certain frequency range.



DNS, LES and RANS of a burning jet.

PROS:

- **Largest turbulent structures are directly resolved.**
- Affordable computational cost.

CONTRAS:

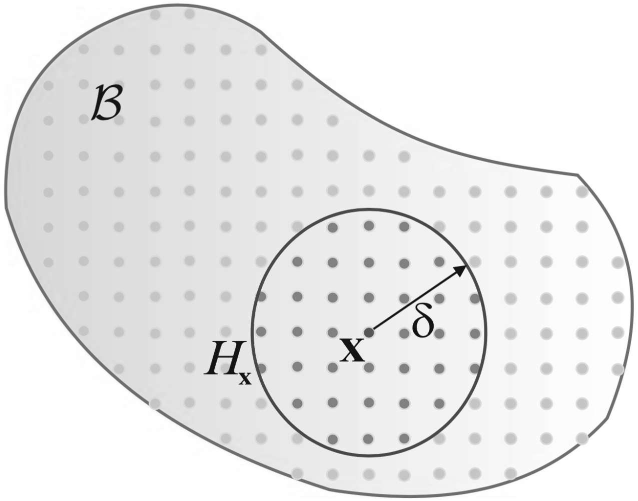
- Computational cost increases with the Reynolds number.
- The effect of smaller turbulent structures is accounted through **turbulence closure models**.

Peridynamics

The body is represented by a finite set of **particles interacting among each other trough micro-potentials** that generate bond forces.

Mutual interaction between particles extinguishes after a distance called horizon.

The body displacements and deformation fields are described by local, integral equations.



PROS:

- "Easy-to-implement" and **high efficiency parallelization** due to locality of equations.

- **Crack detection** and **crack propagation** can be easily accounted by the peridyanmics formulation: local micro-fracture occurs when the bond force exceeds a limit value.

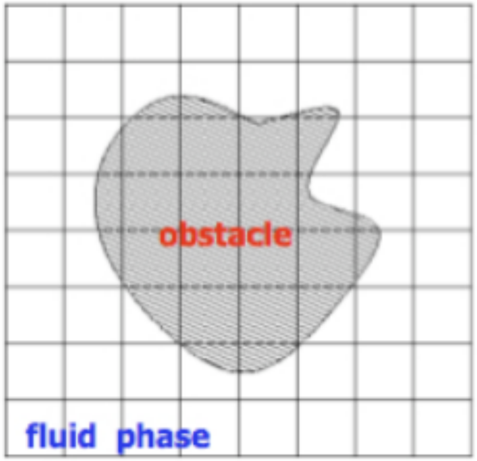
CONTRAS:

- Peridynamics is a novel numerical technique. Few archival literature and know-how are available.

Coupling: Immersed Boundary Method (IBM)

No-slip and **no-penetration** boundary conditions for fluid flow are not directly imposed. BCs are converted into **equivalent forcing terms** in the right hand side of Navier-Stokes equations:

$$\rho \left[\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u}\mathbf{u}) \right] = -\nabla p + \mu \nabla^2 \mathbf{u} + \rho \mathbf{f}$$



Fluid and solid grids do not have to conform to the interface.

IBM force that accounts for the presence of the solid obstacle.

PROS:

- Simple, regular Eulerian grid for fluid domain discretization. → Efficient solver.
- No re-meshing is needed for moving bodies. → Low computational cost.

CONTRAS:

- Possible effects on accuracy, stability and conservation of properties.

Parallelization

Parallel codes: **work on multiple cores** or even multiple computers in order to reduce computational time.

The principle of parallelization consists of the the **domain decomposition**: the simulation is performed by more processes each of them advance the solution in one single portion of the domain.

Fluid domain parallelization

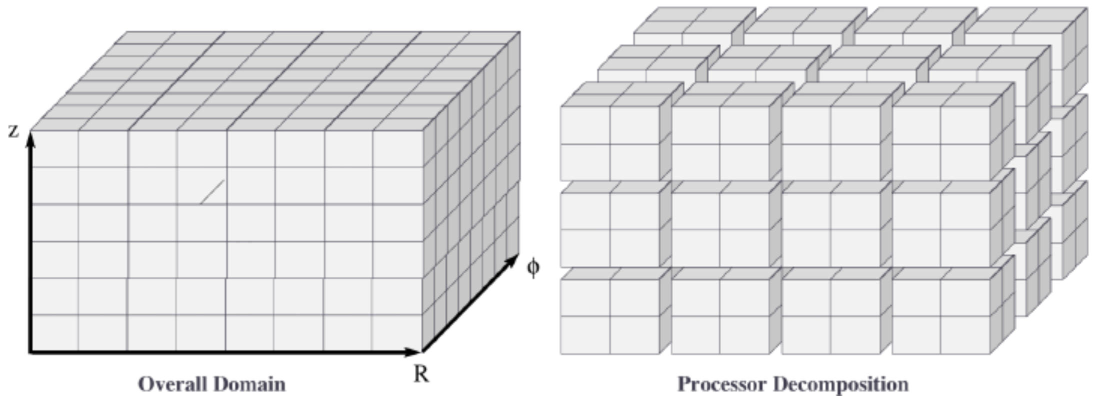
⬇

Subsets of grid nodes

Solid domain parallelization

⬇

Subsets of peridynamic particles



Main advances introduced by the proposed method

❖ **PERIDYNAMIC APPROACH TO SOLID MECHANICS**

- Locality of equation.
- "Easy" to parallelize.
- High scalability on large clusters.

- Crack prediction and propagation.
- Automatic tracking of fragments.

❖ **IMMERSED BOUNDARY METHOD**

- Eliminates the need for mesh update strongly reducing computational costs.
- Simple grid → Efficient solver.

❖ **NAVIER-STOKES LES SOLVER**

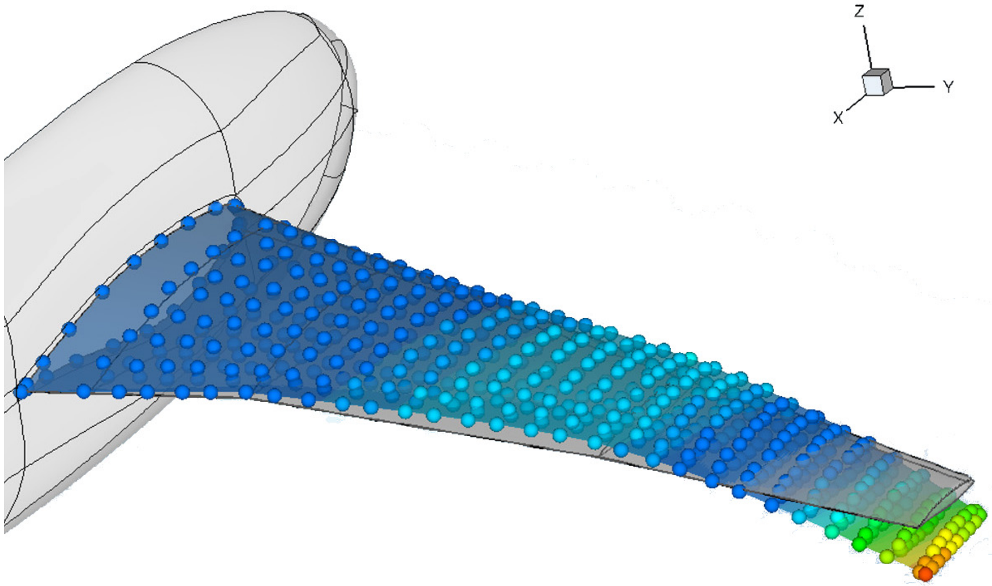
- Resolve time-evolving largest turbulent structures in the fluid domain.

- Accounts for the dynamic effects of turbulence on structures.

LES of a scale wing model fluttering

Large Eddy Simulation of a scale wing model:

- Up to **10^8 points** (fluid grid nodes and peridynamics particles).
- Reynolds number up to **Re = 500000.**
- Computational resources provided by **CINECA**.
- **Analysis of the dataset** to study aeroelastic flutter and test the numerical code capabilities.



Final remarks

During the proposed research activity:

- **A highly efficient numerical code able to reproduce general FSI problems working on high performance computer clusters will be developed.**
- **A novel numerical method for the description of FSI problem** will be tested.
- The **simulation of a scale wing model** will be performed.
- The phenomenon of **aeroelastic flutter** will be investigated.
- A **contribution to he know-how concerning the numerical reproduction of FSI problem** will be given.

Thank you for your attention.

GANTT bar chart of the proposed research activity