1D-3D coupling between GT-Power and OpenFOAM for cylinder and duct system domains

G. Montenegro°, A. Onorati°, M. Zanardi°, M. Awasthi+, J. Silvestri+

(°) Dipartimento di Energia - Politecnico di Milano
(+) Gamma Technologies Inc.
What is OpenFOAM?

- **OpenFOAM**® is an open source, freely available CFD Toolbox, licensed under the GNU General Public Licence, written in highly efficient C++ object-oriented programming. Currently it is the most advanced research CFD code. It can simulate almost any problem in computational continuum mechanics.

- It uses finite volume numerics to solve systems of partial differential equations ascribed on any 3D unstructured mesh of polyhedral cells.

- **Domain decomposition** parallelism is fundamental to the design of OpenFOAM® and integrated at a low level so that solvers can generally be developed without the need for any 'parallel-specific' coding.
The ENGINE library

- The Internal Combustion Engine Group of Politecnico di Milano has contributed to develop the engine library:
  - Moving mesh algorithms
  - Spray modeling
  - Combustion process modeling
  - 1D-3D coupling interface
  - Non-linear acoustics modeling
  - DPF modeling

GT-Power / OpenFOAM coupling

- Closed valve cylinder modeling (Diesel and S.I.)
- Intake and exhaust systems modeling

Gianluca Montenegro
Cylinder coupling

- Values are passed from GT-Power to OpenFOAM at the beginning of the 3D simulation:
  - Initial pressure
  - Initial temperature
  - Initial swirl and tumble coefficient
  - Initial turbulence level
  - EGR, fuel, products and oxidant mass fractions

- Other parameters, such as injection law, IVC, EVO and others, are set recurring to standard OpenFOAM dictionaries

- Instantaneous values of pressure, temperature, turbulence and mass fraction of EGR, products, oxidant and fuel are passed back to GT-Power at each time step
• **Fully Automatic Mesh Adaption (FAMA) can be used when the piston can be meshed by means of a layered grid**

• The mesh is moved and layers of cells are added or removed, during the compression and expansion stroke respectively, according to user defined tolerances
• **MUltiple MESH to MESH Interpolation (MUMMI)** can be used when the engine geometry is complex (bowl in GDI engines)

• The mesh is automatically remapped to a new mesh whenever the mesh quality deteriorates excessively

• The set of meshes must be provided at the beginning of the calculation. Meshes can be created exploiting the built-in mesh generator (blockMesh) or adopting OpenFOAM compatible mesh generators (Gambit, ProAM, ICEM, NetGen ...)

Gianluca Montenegro
• OpenFOAM can follow the time step given by the 1D model or it can perform subcycling according to the Courant number required by the 3D
Duct system coupling

- Values are passed from GT-Power to OpenFOAM at each time

- 1D values can be assigned uniformly at the interface

- A coupling procedure based on the solution of the Riemann problem allows to assign non uniform fields compatible with the 3D solution

- Transport of chemical species will be implemented to track different gas compositions such as EGR inside of intake systems
Thank you for your attention