A COUPLED FINITE VOLUME SOLVER FOR THE SOLUTION OF LAMINAR/TURBULENT INCOMPRESSIBLE AND COMPRESSIBLE FLOWS

L. Mangani

Maschinentechnik
CC Fluidmechanik und Hydromaschinen

Hochschule Luzern
Technik& Architektur
Technikumstrasse 21, CH-6048 Horw
T +41 41 349 33 11, F +41 41 349 39 60
e-mail: luca.mangani@hslu.ch

Lucerne University of
Applied Sciences and Arts
HOCHSCHULE
LUZERN

C. Bianchini

Dipartimento Energetica "S. Stecco"

Università degli Studi di Firenze
Via Santa Marta 3, IT-50139 Firenze
T +39 055 4796575
e-mail: cosimo.bianchini@htc.de.unifi.it

D•E

Fifth OpenFOAM Workshop, June 22-24 2010, Gothenburg, Sweden

Presentation outline

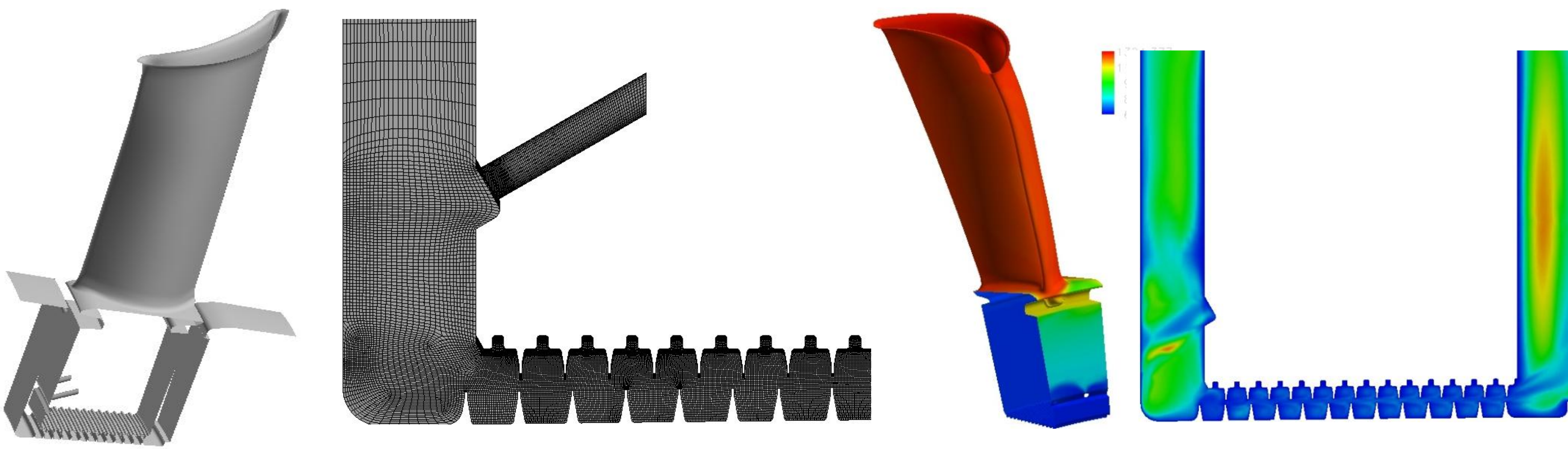
- Background-State of the Art
- Coupled Algorithms vs. Segregated Approach
- Pressure Velocity Coupling
- Developments and present contribution
- Results and discussion
- Conclusions

Background-State of the Art

- The “engine” of computational fluid dynamics (CFD) is the pressure-velocity coupling algorithm that drives the fluid flow
  - Segregated approach: predictor-corrector loops
  - Coupled approach: pressure and velocity treated as a single vectorial unknown
- In past years efforts to develop more robust and efficient velocity-pressure algorithms based on:
  - Choice of primitive variables density-based versus pressure-based
- For density but specially for pressure-based algorithms *the coupled versus segregated approach dichotomy has not been completely resolved yet!*
- Renewed interest in coupled solvers due to the increase in computers memory: commercial solvers

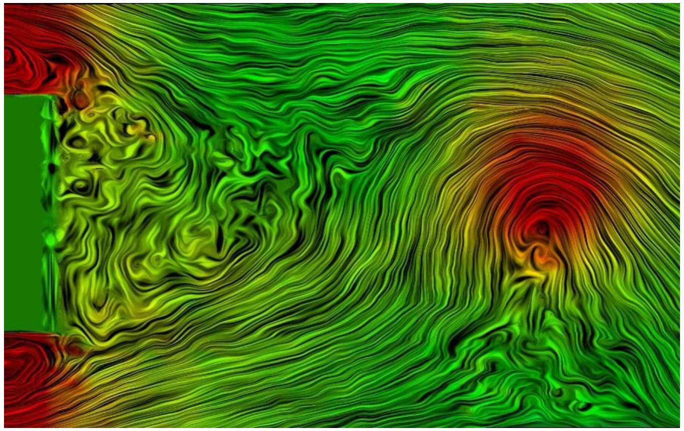
Coupled Algorithms vs. Segregated Approach

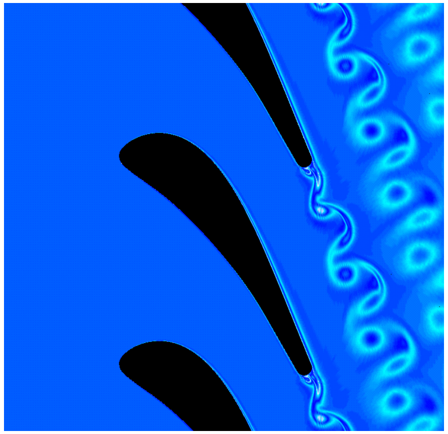
- Coupled Algorithm pros:
  - Fast Convergence: iterations only for momentum non linearity
  - Increase in efficiency for Steady State and time resolved arbitrary time step simulations
  - Less influence from the initial field, quasi initialization independence
  - Convergence speed grid independent
  - Fast convergence for simulations with extreme range of Mach:
    - ✓ Ex: Stator Rotor Cavity applications



Coupled Algorithms vs. Segregated Approach

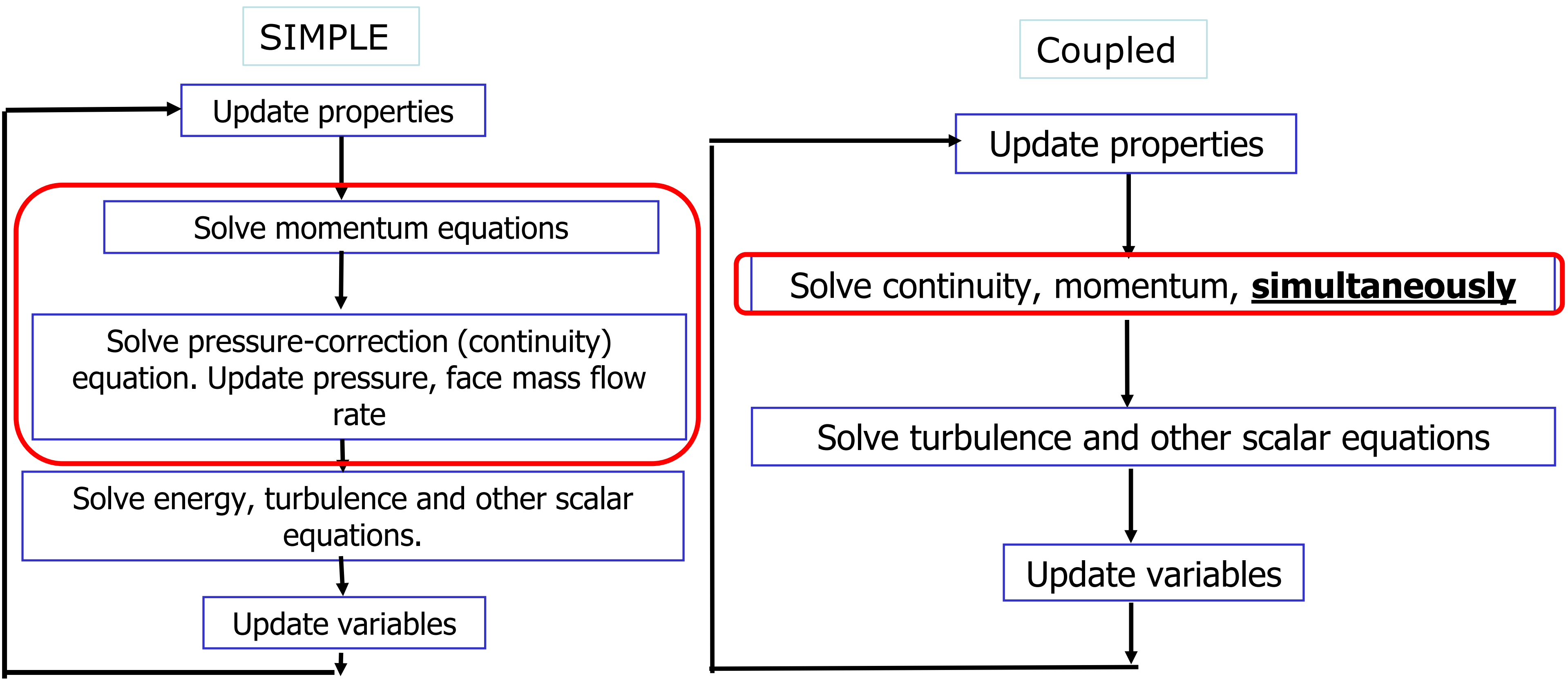
- Coupled Algorithm cons:
  - Huge memory allocation
  - Less flexible
  - Critical choice of the linear solver for inner iterations
  - Critical choice of the controls parameter of the linear solver
  - Bad scalability for huge parallel calculations
  - Inefficient for time accurate simulations Co < 1





Pressure Velocity Coupling

- SIMPLE (Semi-Implicit Method for Pressure-Linked Equations)
  - Segregated approach for the pressure velocity coupling
- Coupled Pressure based U-p coupling without energy



Pressure Velocity Coupling: Momentum Equation

- Momentum equation discretization
  - Pressure gradient becomes implicit
  - Pressure goes on the LHS

$$\sum_{f=nb(P)} (\rho \mathbf{v}\mathbf{v} - \mu \nabla \mathbf{v})_f \cdot \mathbf{S}_f + \boxed{\sum_{f=nb(P)} p_f \mathbf{S}_f} = \mathbf{b}_P \Omega_P$$

Gradient discretization

$$p_f = g_f p_P + (1 - g_f) p_F$$

$$\begin{cases} a_P^{uu} u_P + \underline{\underline{a_P^{uv} v_P}} + \underline{a_P^{up} p_P} + \sum\limits_{F=NB(P)} a_F^{uu} u_F + \underline{\underline{\sum\limits_{F=NB(P)} a_F^{uv} v_F}} + \underline{\sum\limits_{F=NB(P)} a_F^{up} p_F} = b_P^u \\ a_P^{vv} v_P + \underline{\underline{a_P^{vu} u_P}} + \underline{a_P^{vp} p_P} + \sum\limits_{F=NB(P)} a_F^{vv} v_F + \underline{\underline{\sum\limits_{F=NB(P)} a_F^{vu} u_F}} + \underline{\sum\limits_{F=NB} a_F^{vp} p_F} = b_P^v \end{cases}$$

  - Coupling coefficients for the momentum equations

$$a_F^{uu} = a_F^{vv} = \mu_f \frac{\mathbf{S}_f \cdot \mathbf{S}_f}{\mathbf{S}_f \cdot \mathbf{d}_{PF}} + \|\dot{m}_f, 0\|$$

$$a_P^{uu} = \sum_{F=NB(P)} a_F^{uu} \qquad a_P^{vv} = \sum_{F=NB(P)} a_F^{vv}$$

$$a_F^{up} = (1 - g_f) S_f^x \qquad a_F^{vp} = (1 - g_f) S_f^y$$

$$a_P^{up} = \sum_{f=nb(P)} g_f S_f^x \qquad a_P^{vp} = \sum_{f=nb(P)} g_f S_f^y$$

Coupled algorithm: Comments

- If pressure equation is **NOT** introduced and the momentum and continuity equations are used:
  - We have a Saddle Block Matrix problem

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla\bullet(\mathbf{u}\mathbf{u}) - \nabla\bullet(\nu\nabla\mathbf{u}) = -\nabla p$$

$$\nabla\bullet\mathbf{u} = 0$$

$$\boxed{\begin{bmatrix} [A_{\mathbf{u}}] & [\nabla(.)] \\ [\nabla\bullet(.)] & [0] \end{bmatrix} \begin{bmatrix} \mathbf{u} \\ p \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}}$$

- Since no pressure equation is derived, zeros are present in the main diagonal of the discretized continuity equation
- Leading to an ill conditioned system of equations

Pressure Velocity Coupling: Continuity Equation

- The use of the pressure equation solve the Saddle block matrix issue
  - Pressure equation derived from the continuity and momentum equation

$$\sum_{f=nb(P)} \rho_f \mathbf{v}_f \cdot \mathbf{S}_f = 0$$

  - Using Rhie-Chow interpolation

$$\sum_{f=nb(P)} \rho_f [\overline{\mathbf{v}_f} - \overline{\mathbf{D}_f}(\nabla p_f - \overline{\nabla p_f})] \cdot \mathbf{S}_f = 0$$

  - Discretized equation

$$\sum_{f=nb(P)} \rho_f (-\overline{\mathbf{D}_f} \nabla p_f) \cdot \mathbf{S}_f + \sum_{f=nb(P)} \rho_f \overline{\mathbf{v}_f} \cdot \mathbf{S}_f = \sum_{f=nb(P)} \rho_f (-\overline{\mathbf{D}_f \nabla p_f}) \cdot \mathbf{S}_f$$

  - Pressure-velocity coupling coefficients derived from mass fluxes imbalance

$$a_P^{pp} p_P + a_P^{pu} u_P + a_P^{pv} v_P + \sum_{F=NB(P)} a_F^{pp} p_F + \sum_{F=NB(P)} a_F^{pu} u_F + \sum_{F=NB(P)} a_F^{pv} v_F = b_P^p$$

$$a_F^{pu} = (1 - g_f) S_f^x \quad a_F^{pv} = (1 - g_f) S_f^y$$
$$a_P^{pu} = \sum_{f=nb(P)} g_f S_f^x \quad a_P^{pv} = \sum_{f=nb(P)} g_f S_f^y$$

Matrix Form

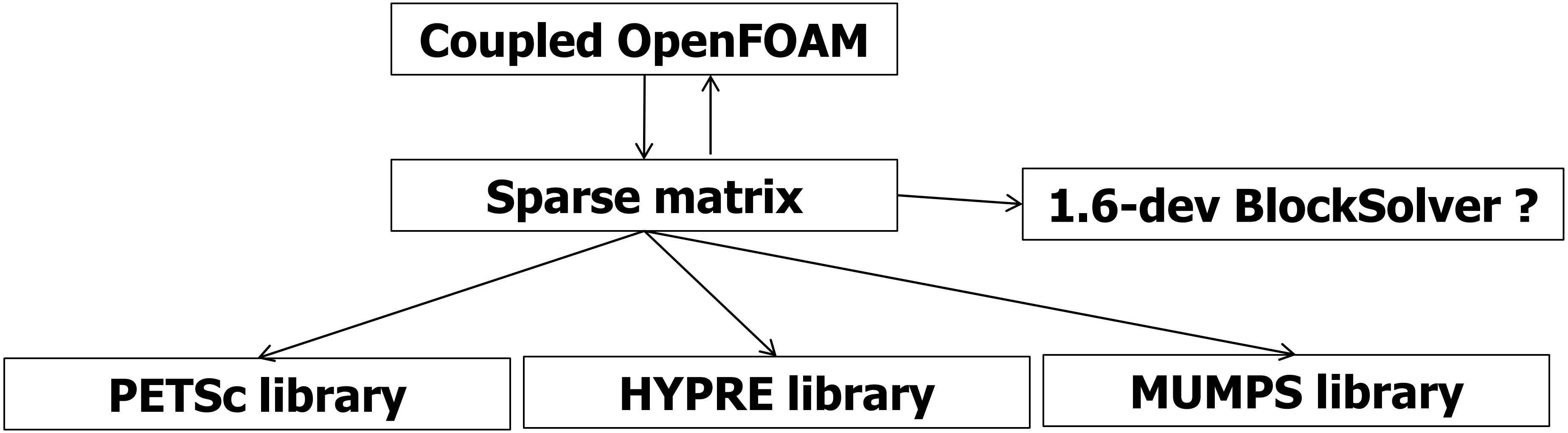
- Cell based variable storage
  - Global matrix is composed of a 4x4 local matrix for each cell (equivalent to fvMatrix.A() coeffs)
  - Matrix pattern expanded to include the influence of neighboring control volumes (equivalent to fvMatrix.H() coeffs)

- Local matrix structure:

$$\begin{pmatrix} UU & & & UP \\ & VV & & VP \\ & & WW & WP \\ PU & PV & PW & PP \end{pmatrix} \begin{pmatrix} U \\ V \\ W \\ P \end{pmatrix} = \begin{pmatrix} S_U \\ S_V \\ S_W \\ S_P \end{pmatrix}$$

OpenFOAM Development

- OpenFOAM current version cannot handle block matrix
  - lduMatrix addressing is referred to the mesh size
- Development of a generic matrix interface to handle external linear solvers



Compressibility and Turbulence

- If flow is in compressible regime, the change in fluid density should be taken into account

$$\boxed{\rho = \rho^* + \rho' = \rho^* + \frac{\partial \rho}{\partial p} p'}$$

  - The convection flux should also be modified in the pressure equation

- Turbulence model was added based on k-ω SST model with Low-Reynolds or automatic wall treatment
  - k and ω are solved also in a coupled way
  - A block sparse matrix for turbulence is solved too

Coupled U-p → **Energy Equation** → **Coupled k-ω**

Results

- Comparison is performed between coupled and segregated solver on reference test cases

  (coupled = solid line, segregated = dash line)

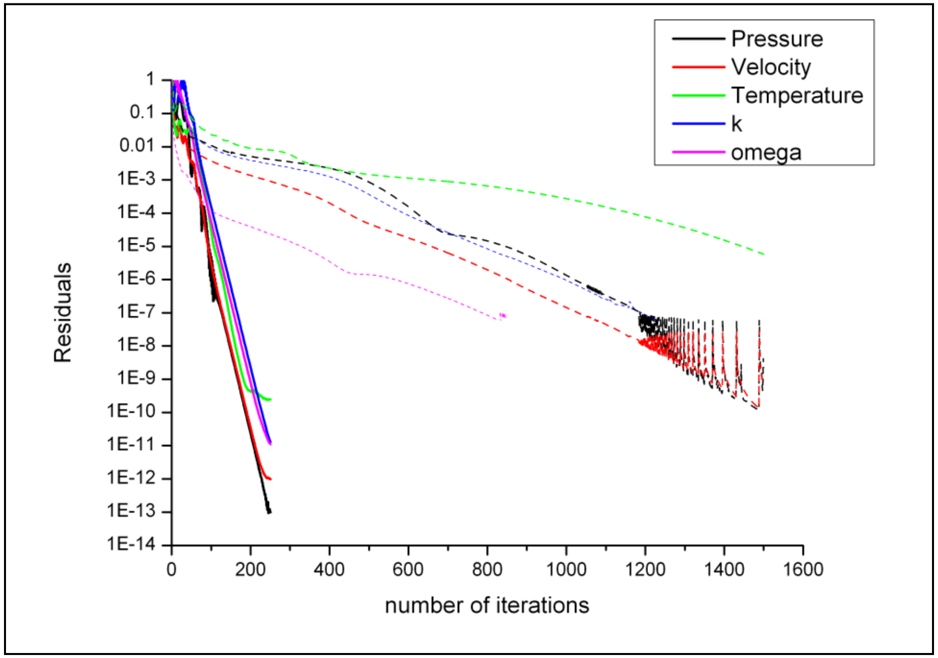
- Convergence speed is checked plotting normalized residuals

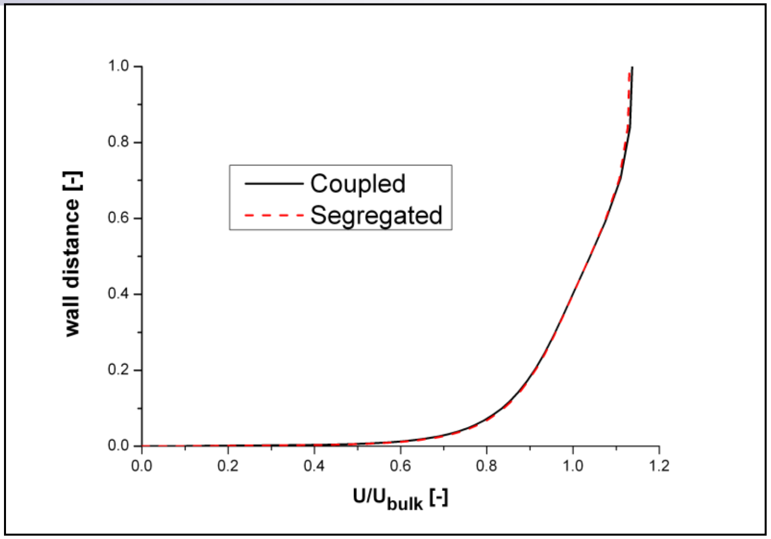
$$R_{\phi} = \frac{\sqrt{\sum \left(\phi_{old} - \phi_{new}\right)^2}}{\sqrt{\sum \left(\phi_{old} - \tilde{\phi}_{old}\right)^2}}$$

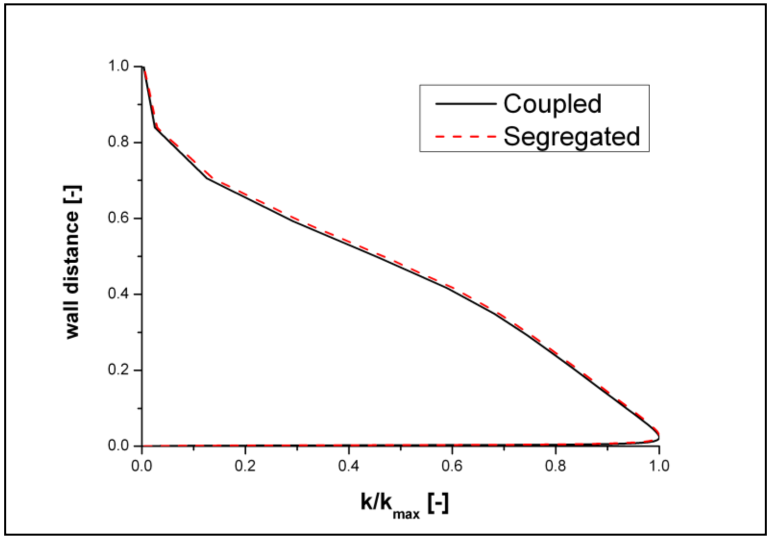
- Uniform initialization, energy and turbulence activated from beginning
- Incompressible and compressible formulation
- Inviscid, laminar and turbulent test cases
- Periodic boundaries

Turbulent Flat Plate

- Adiabatic 2D flat plate at Ma_{in}=0.2
- Turbulent boundary layer integrated up to the wall $y^+ \approx 0.1$
- Fixed localCo = 5000

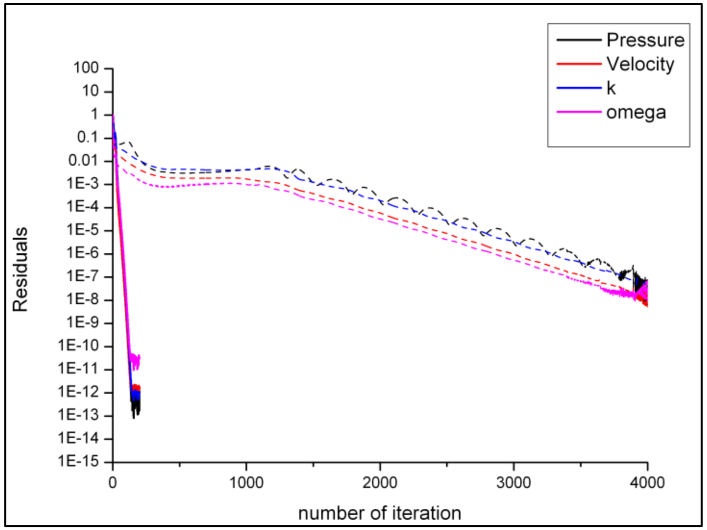


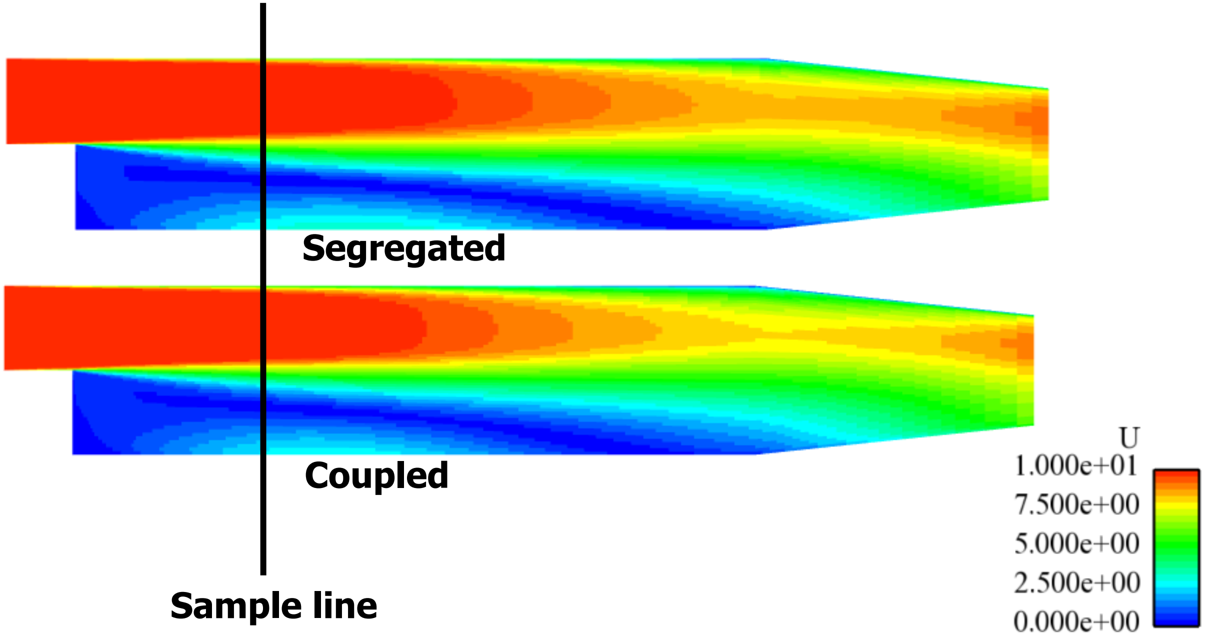


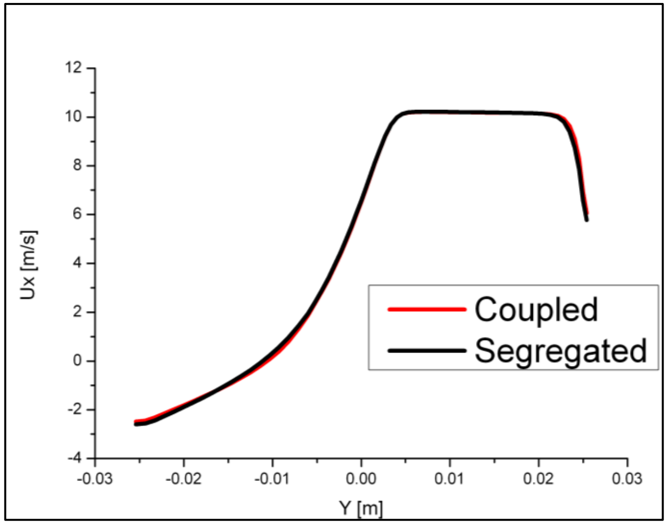


Backward Facing Step Test

- Incompressible isothermal turbulent flow
- Automatic wall treatment
- No time derivative (Co=∞)
- Explicit relaxation factors

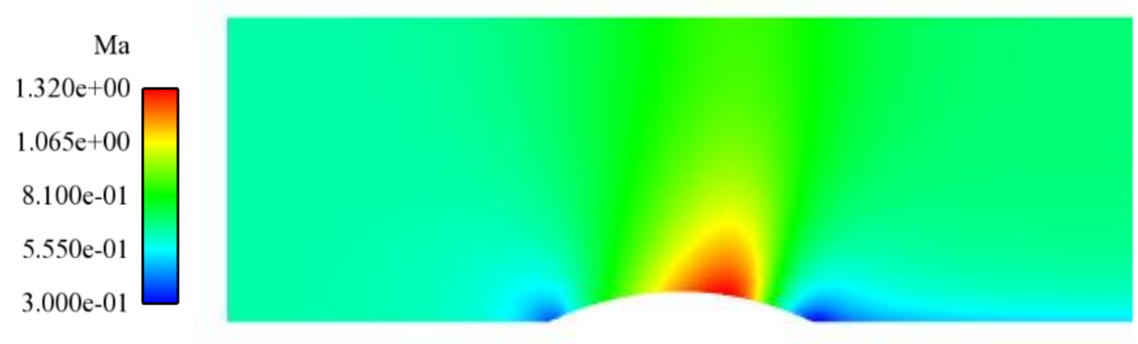


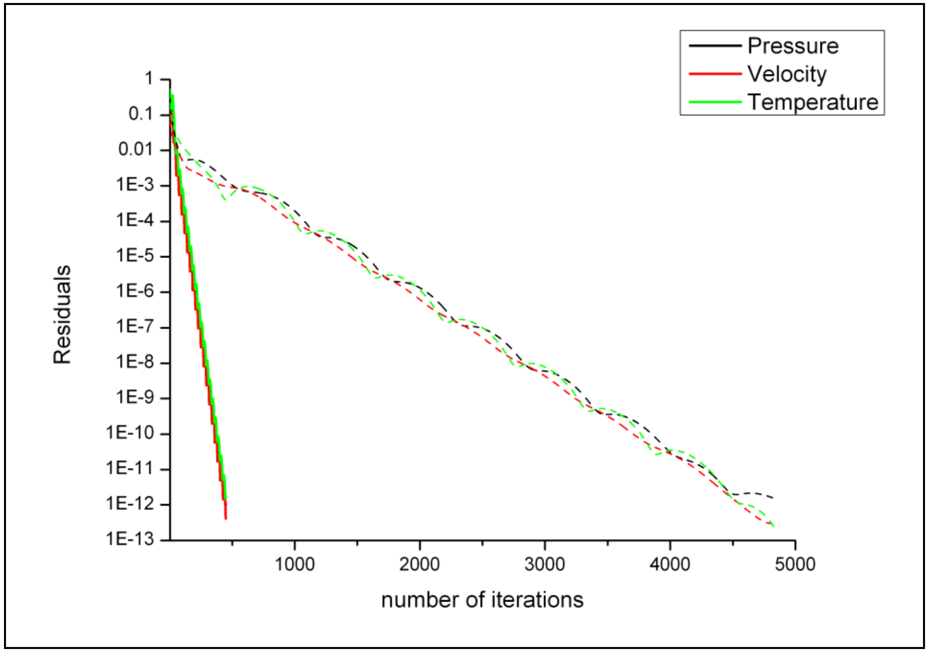


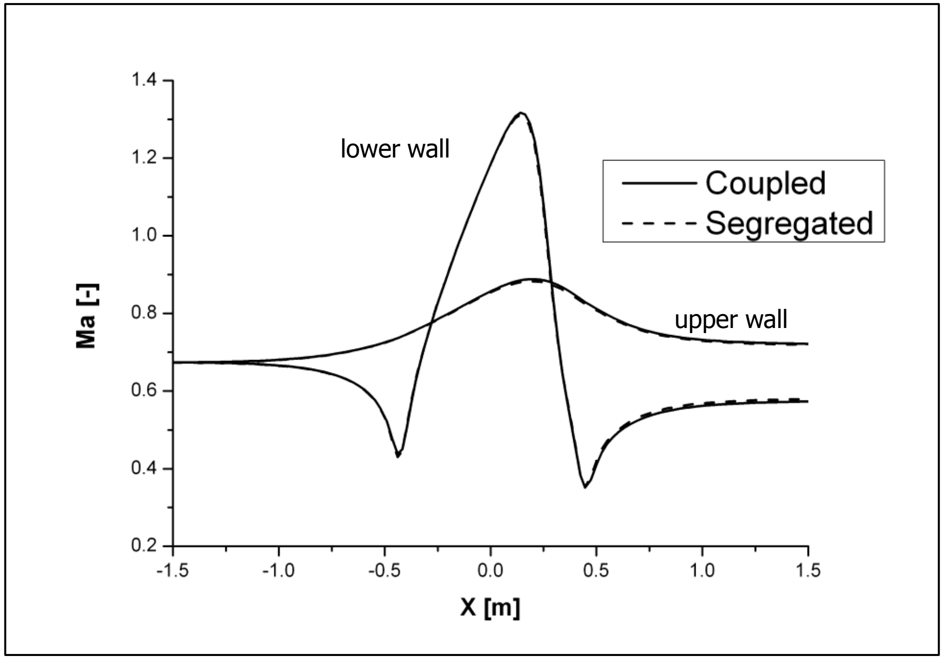


GAMM Test

- Inviscid transonic test case
- Confined circular bump at $Ma_\infty = 0.675$
- Fixed localCo = 600

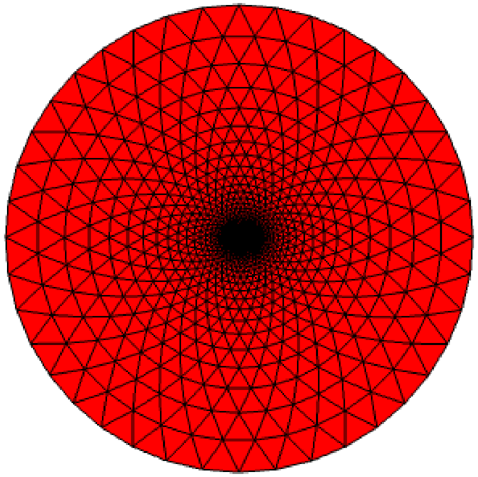




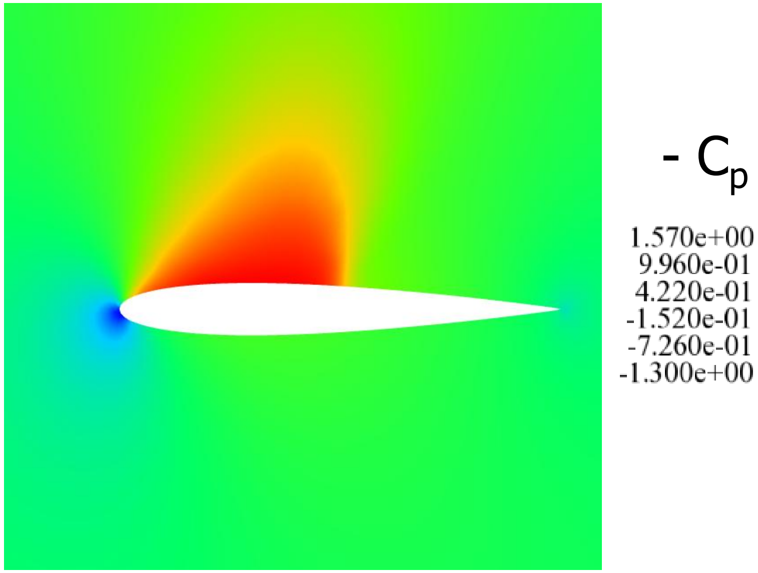


NACA 0012

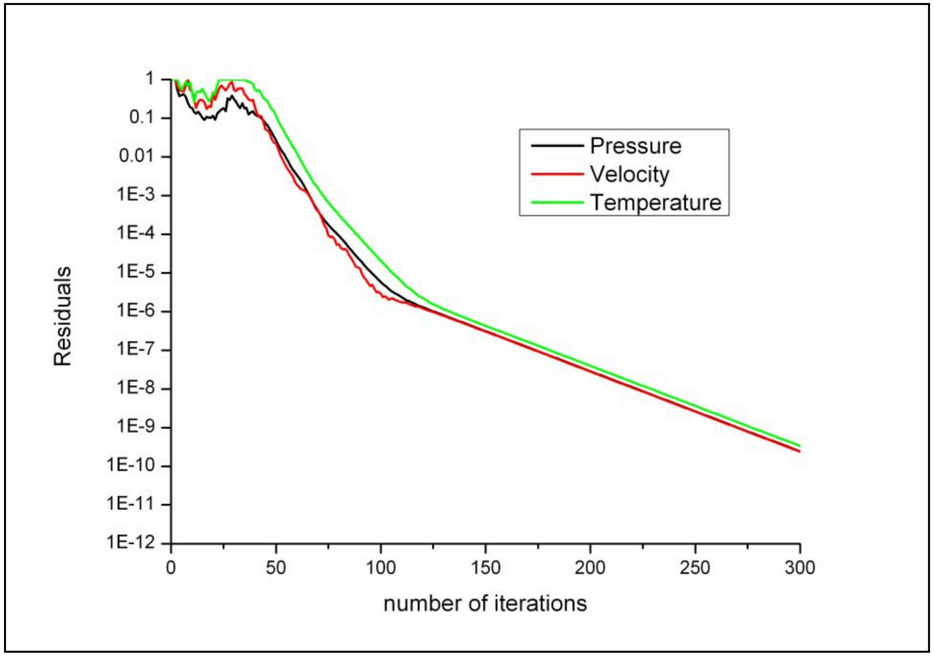
- Isolated 2D profile
- Inviscid test case
- Transonic test $Ma_\infty = 0.75, \alpha = 4°$
- Circular domain, inletOutlet BC
- Tetrahedral mesh
- Fixed localCo = 600
- Results in terms of dimensionless pressure

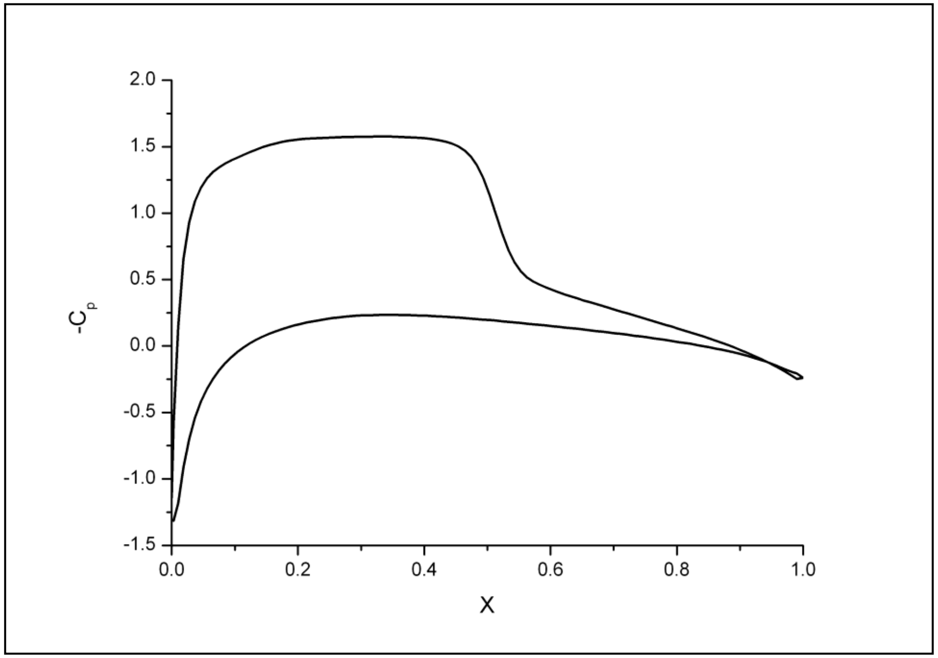


$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty U_\infty^2}$$



NACA 0012

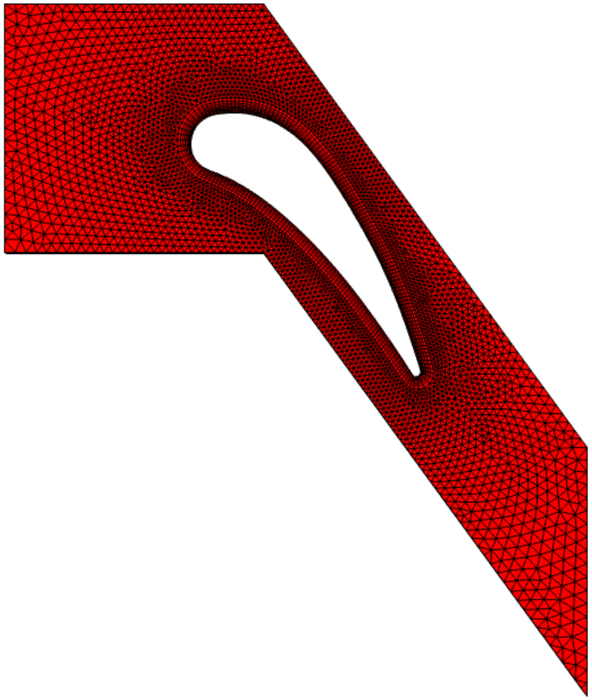


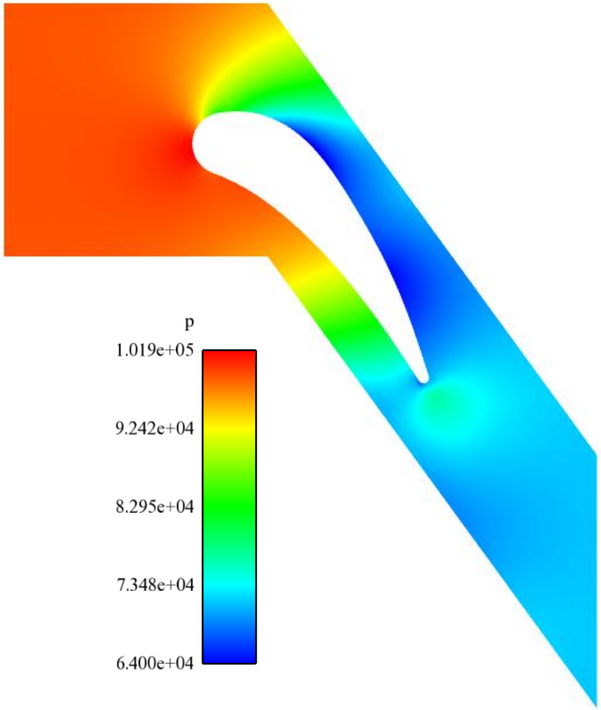


- Only 300 iterations to reach the convergence
- Even though more diffused the shock location is well predicted
- Profile load correctly reproduced

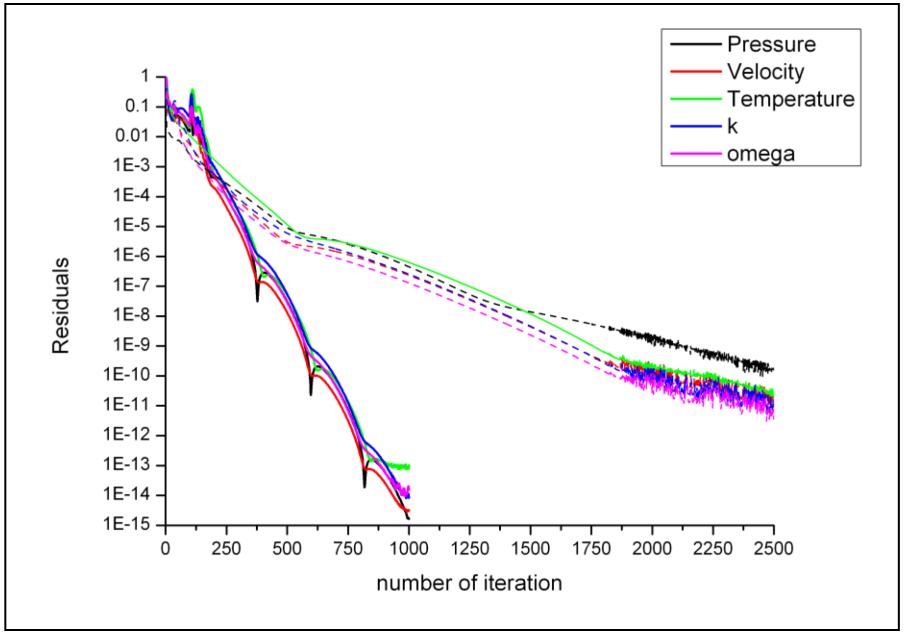
Goldman Test

- 2D linear cascade -> fully implicit coupled boundary
- Highly compressible turbulent test case $Ma_{in}=0.2$
- average y^+ = 50 automatic wall treatment
- Adiabatic surface, fixed velocity and static pressure
- Fixed localCo = 500

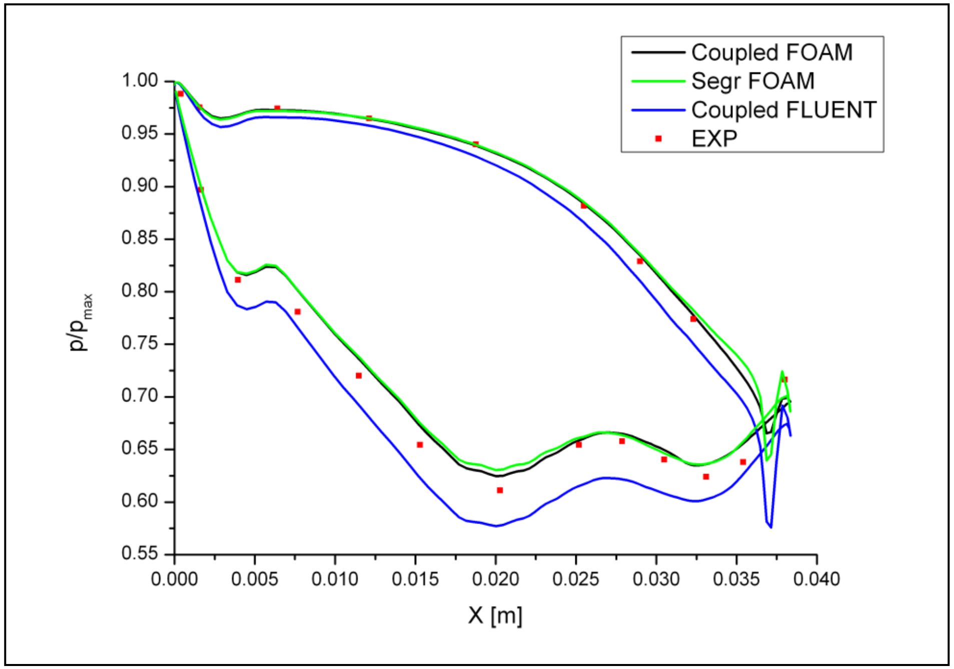




Goldman Test



- Faster convergence than segregated
- Lower level of residuals than segregated



- Pressure profile coincident with segregated
- Good agreement with experimental values
- Drift respect to other codes due to BC and turbulence model

Conclusions

- A turbulent compressible and incompressible coupled solver has been developed in OF framework
  - Consistent results compared to the segregated solver were reproduced among:
    - ✓ Inviscid/viscous/turbulent, compressible/incompressible, LowReynolds/WallFunction, periodic flows
- Improved convergence and stability respect to segregated solvers
- Main Drawbacks
  - Speed of linear solver can be improved
  - Great amount of memory allocation can be reduced with a more efficient implementation
  - Further generalization in the code to be achieved
- Applications to more complex cases to be achieved
  - Multi-phase, Combustion, FSI,…