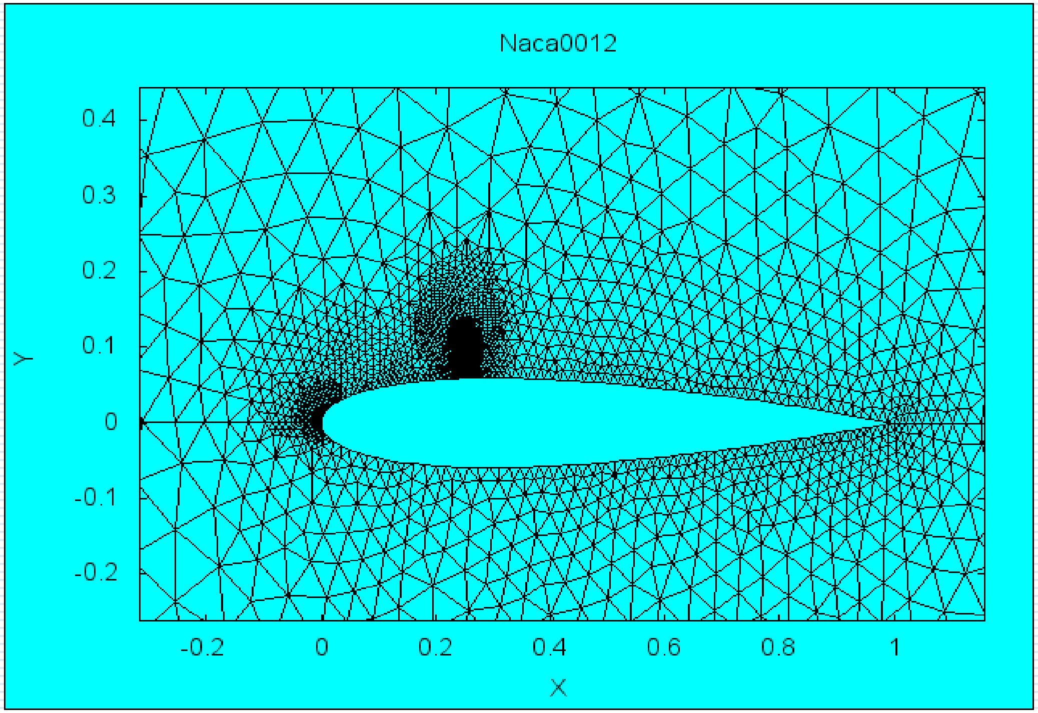
Development of mesh refinement methods at CFD codes for Computational Fluid Mechanics problems

- Description of problem
- Development
- Test cases
- Conclusion



Mesh Refinement

Reasons for choosing mesh refinement

- Increase local resolution at areas of interest – shock waves, boundary layers – whose position is not known a priori
- Efficient at time consuming hyperbolic systems of conservation laws. Large spatial scale
- Speed up calculations

Navier Stokes equations

Integration at volume Ω , with boundary $\partial\Omega$:

- $$\frac{\partial}{\partial t}\int_\Omega \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E \end{pmatrix} d\Omega + \oint_{\partial\Omega} \left(\begin{pmatrix} \rho V \\ \rho u V + n_x p \\ \rho v V + n_y p \\ \rho w V + n_z p \\ \rho\left(E + \frac{p}{\rho}\right) V \end{pmatrix} - \begin{pmatrix} 0 \\ n_x \tau_x x + n_y \tau_x y + n_z \tau_x z \\ n_x \tau_y x + n_y \tau_y y + n_z \tau_y z \\ n_x \tau_z x + n_y \tau_z y + n_z \tau_z z \\ n_x \Theta_x + n_y \Theta_x + n_z \Theta_z \end{pmatrix} \right) ds = \int_\Omega \vec{Q}\, d\Omega$$

With:

- $\Theta_i = u\tau_{ix} + v\tau_{iy} + w\tau_{iz} + k\frac{\partial T}{\partial i}$, i=x,y,z , viscous stress and heat conduction in fluid
- $p = (\gamma - 1)\rho[E - \frac{u^2+v^2+w^2}{2}]$, equation of state for ideal gases

Navier Stokes equations

- **Conservative variables with cell centered scheme**

  - Mass density
  - Momentum density
  - Energy density

$$\vec{U} = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E \end{pmatrix}$$

- **Discretization**

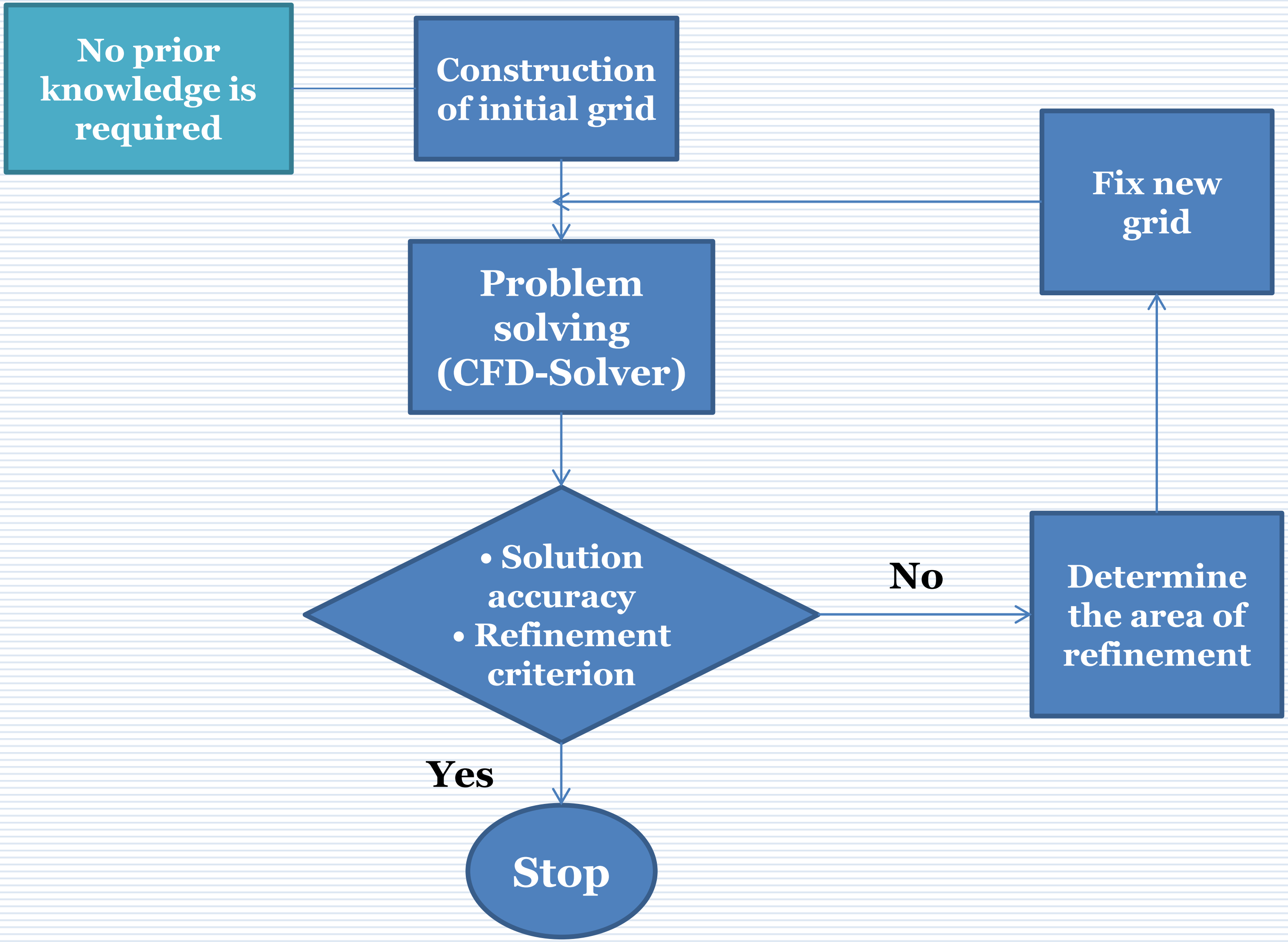
Finite Volume method

- Calculation of the variables at the center of each cell
- 2nd order of accuracy in time and space

Method of calculation

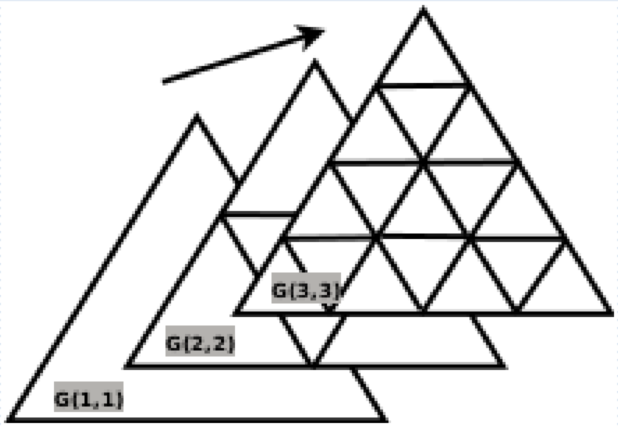
- Method of Characteristics – Solution of Riemann problem

Implementation of mesh refinement at CFD problems



Adaptive mesh

- **Grid changes**



- Finer grid replaces a coarse one
- Properly suited
- Spatial adaptation at current time step

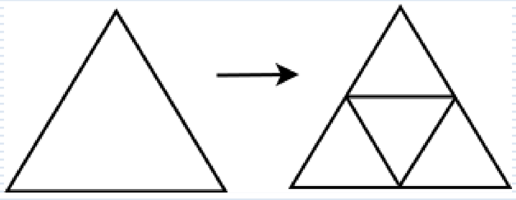
- **Computational steps**

- From coarser to finer mesh
- Fix new cells (1 or more sub-domains)
- Fix the boundaries between fine and coarse grid
- Interpolation of variables (coarse → fine mesh)

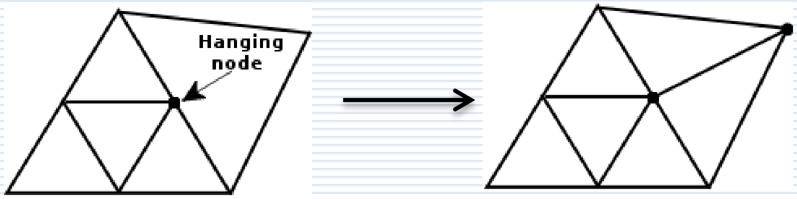
Geometry of Grid

- Geometric decomposition at new cells (children-cells)
- Refined regions and interface between coarse and fine grid

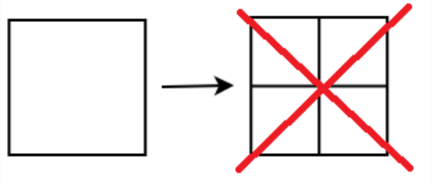
- **Unstructured grid (2D)**

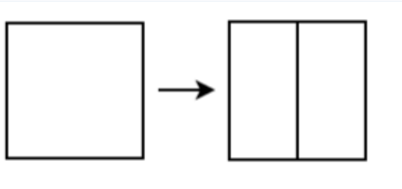


PROBLEM: Hanging nodes !



- **Structured grid (2D)**



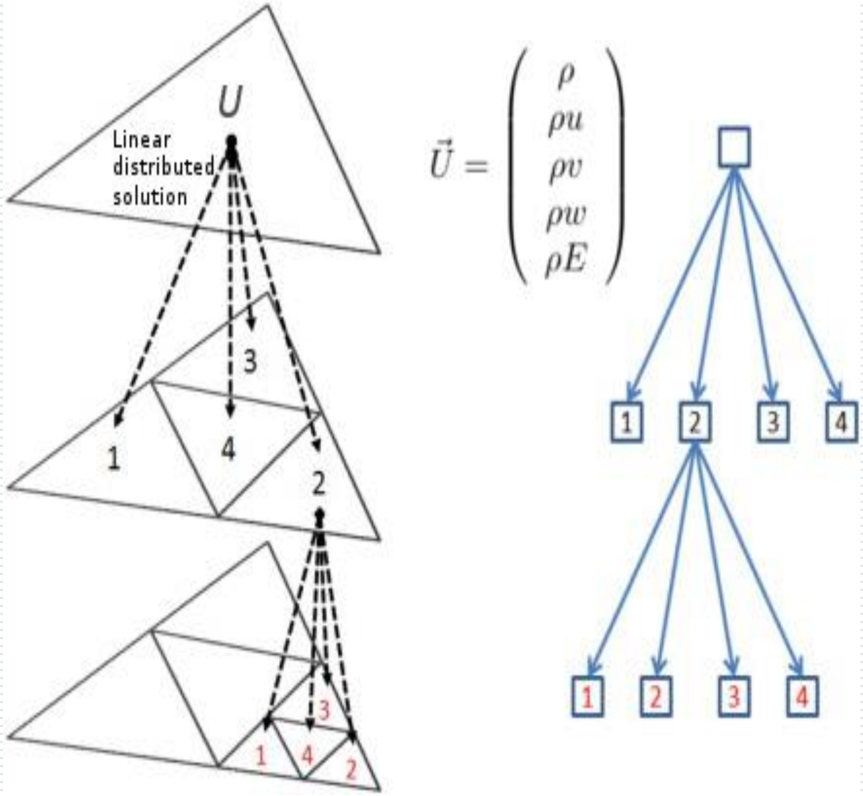


Interpolation of variables

- fine ← coarse grid

Update values at computational cells

- New cells at mesh
- Pass the physical information
- Suitable method for the update



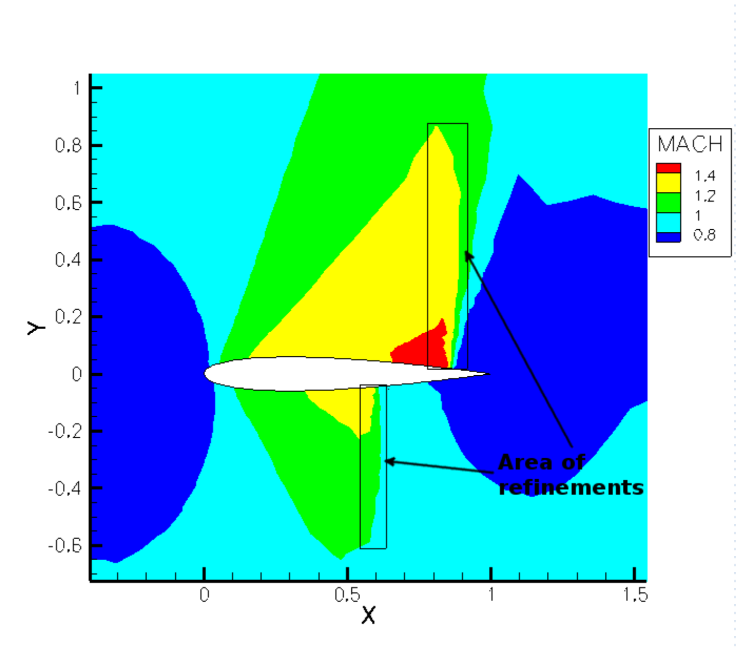
- Children-cells at (t+Δt) receive the same value with the parental cell (t)

- Conservative scheme

Determine the area to refine

Flow criteria

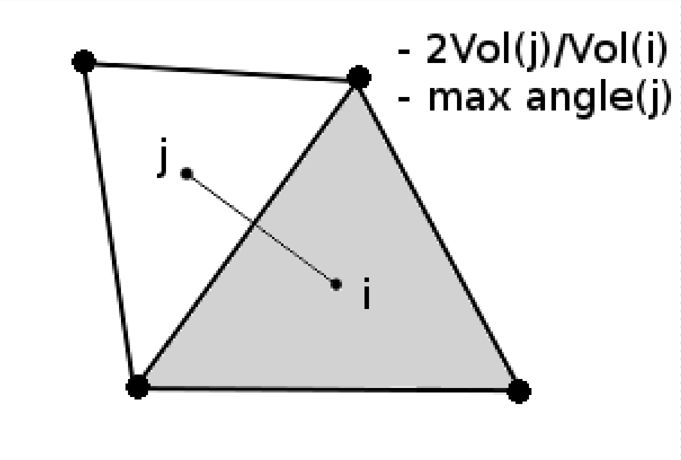
- They primary determine the area that needs high resolution
- Functions of the flow quantities (Mach, pressure etc)
- Error per cell (energy error) for a specific time step of solution
- Limit to the amount of selection



Determine the area to refine

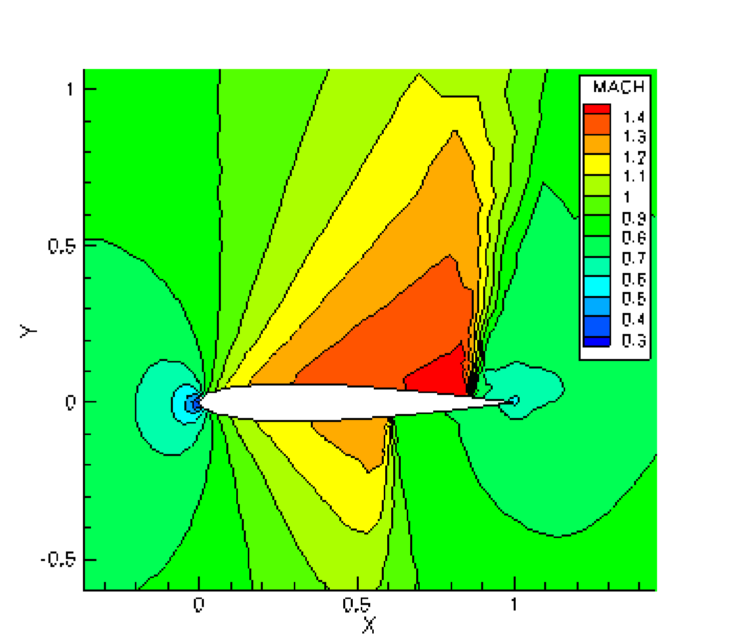
Geometric criteria

- Smooth transition from finer to coarse grid
- Areas that do not fulfill the fluid criteria
- Eliminate the generation of “bad” cells (eg overwhelmed nodes)
- Volume ratio with neighboring cells, new angles. Cells with desirable aspect ratio (fig)
- Limit to the amount of selection



$Ma_{\infty} = 0.85, angle\ of\ inflow = 1.25°$

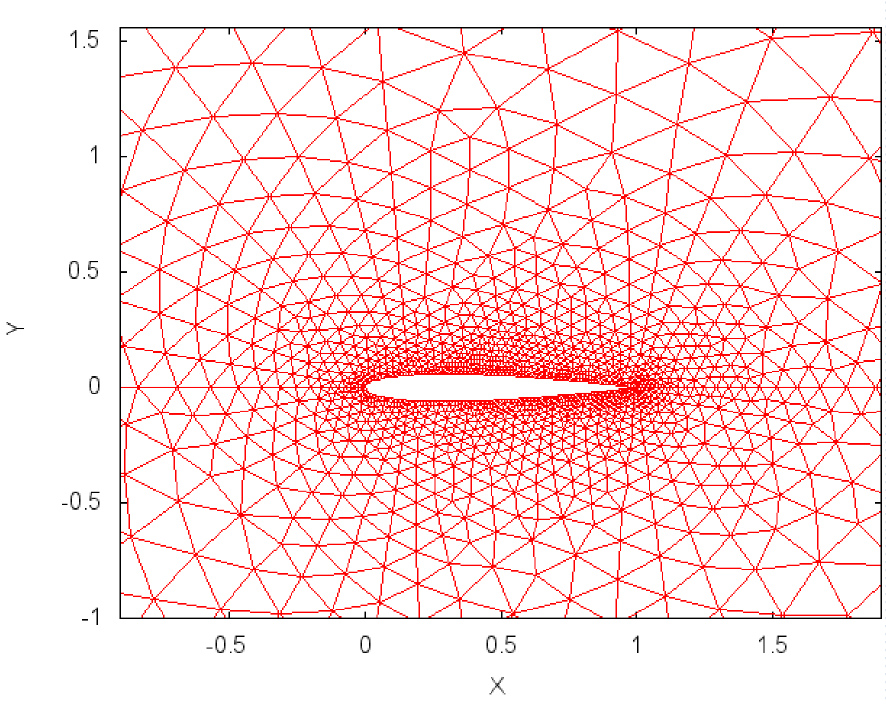
- Inviscid flow around Naca0012, $Ma_{\infty} = 0.85$, angle = 1.25°

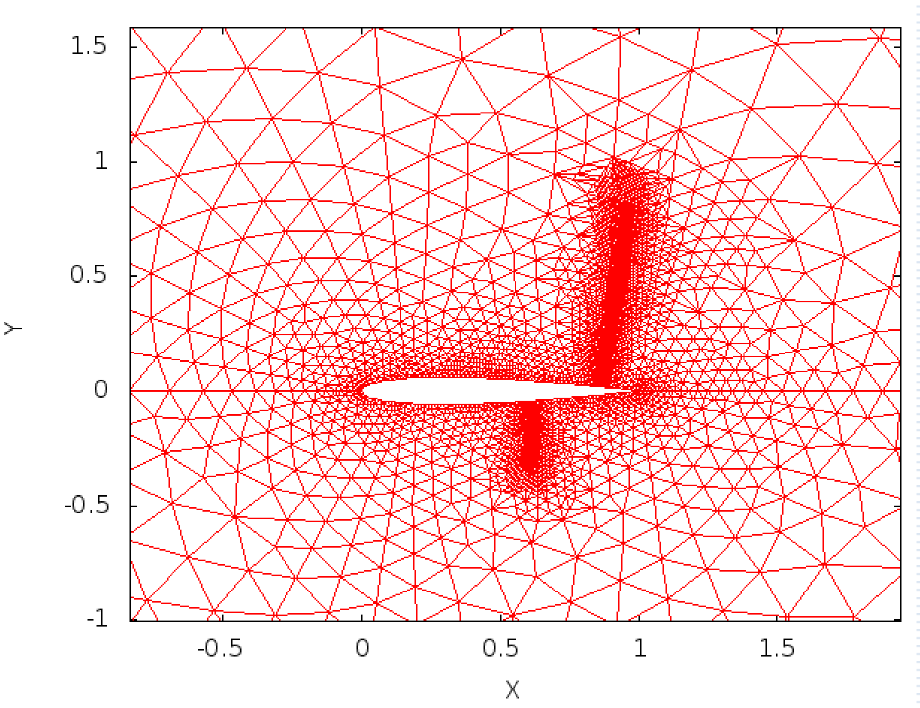


- 2 shock waves
- Flow condition for error energy per cell (1e-01). Condition for Mach number
- 5 levels of refinement
- Coarse and fine initial grids tested (area of refinement: 5% of the initial gird for fluid criteria)

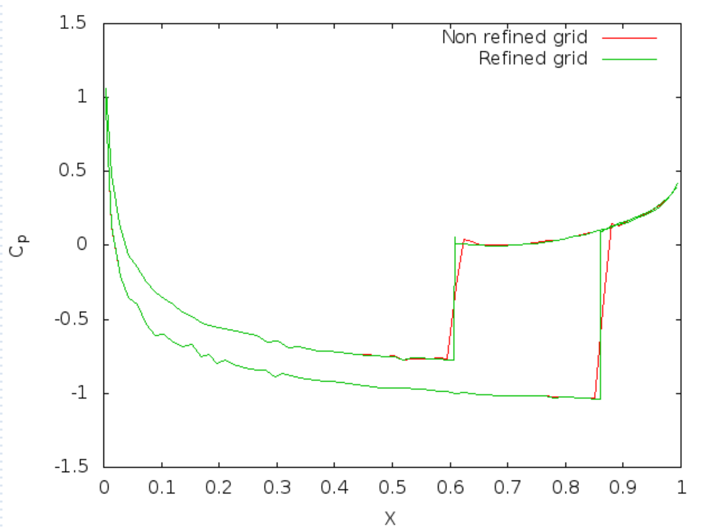
$Ma_{\infty} = 0.85, angle\ of\ inflow = 1.25°$

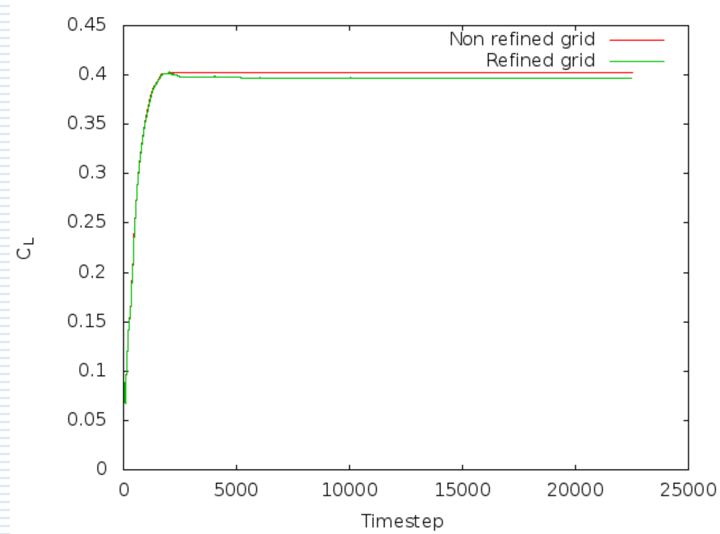
- Initial unstructured grid: 3658 triangle cells. Final grid: 27026 cells
  CFL=5

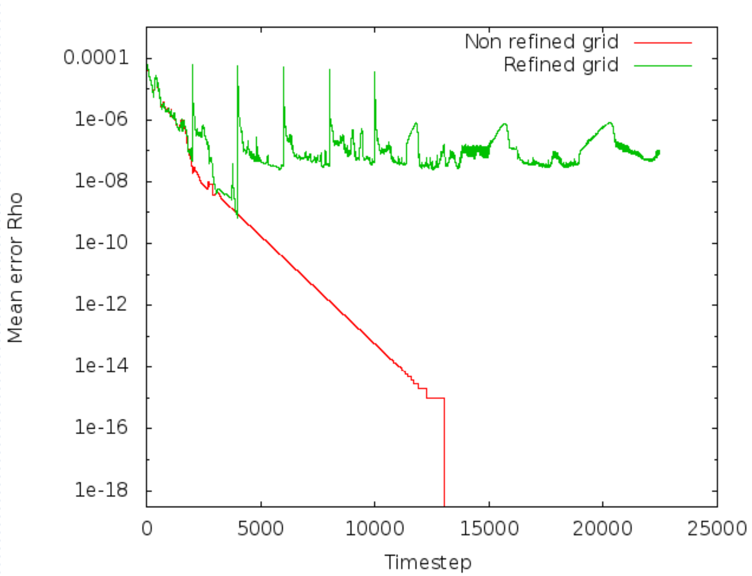


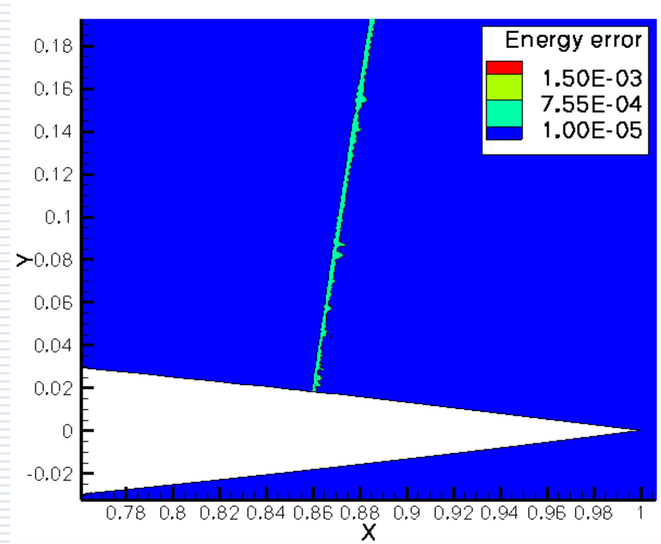


$$Ma_\infty = 0.85, angle\ of\ inflow = 1.25°$$



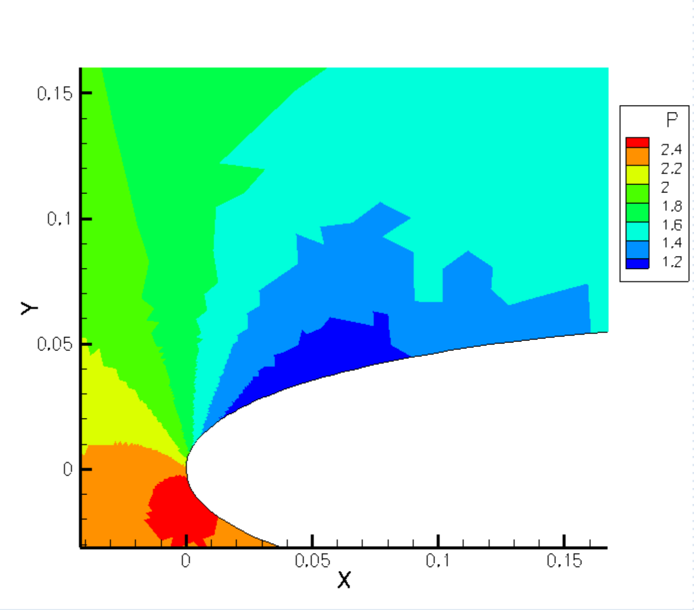






$Ma_{\infty} = 0.6, angle\ of\ inflow = 4.0°$

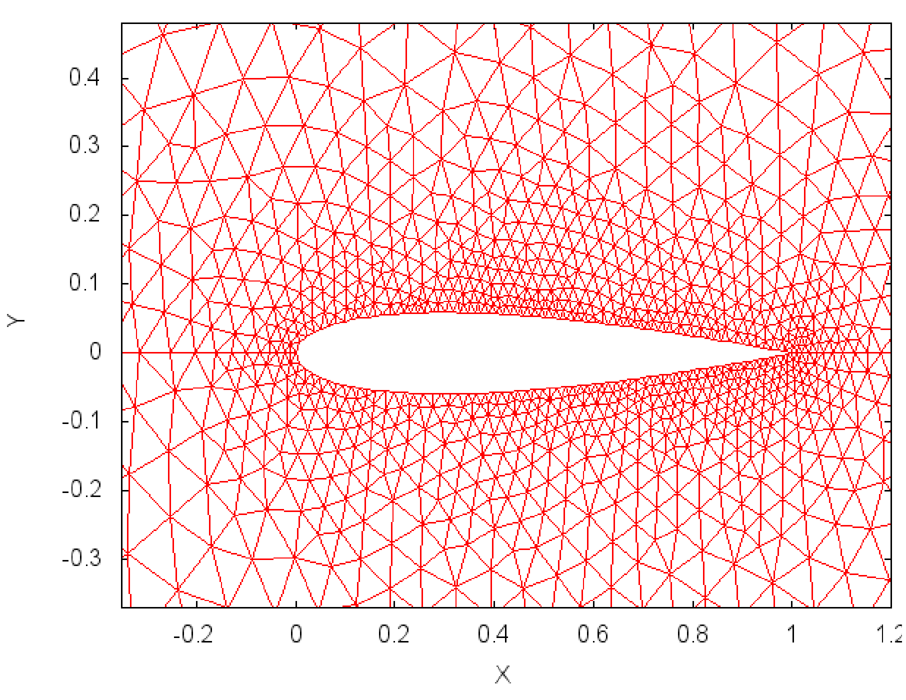
- Inviscid flow around Naca0012, $Ma_{\infty} = 0.6$, angle = 4.0°

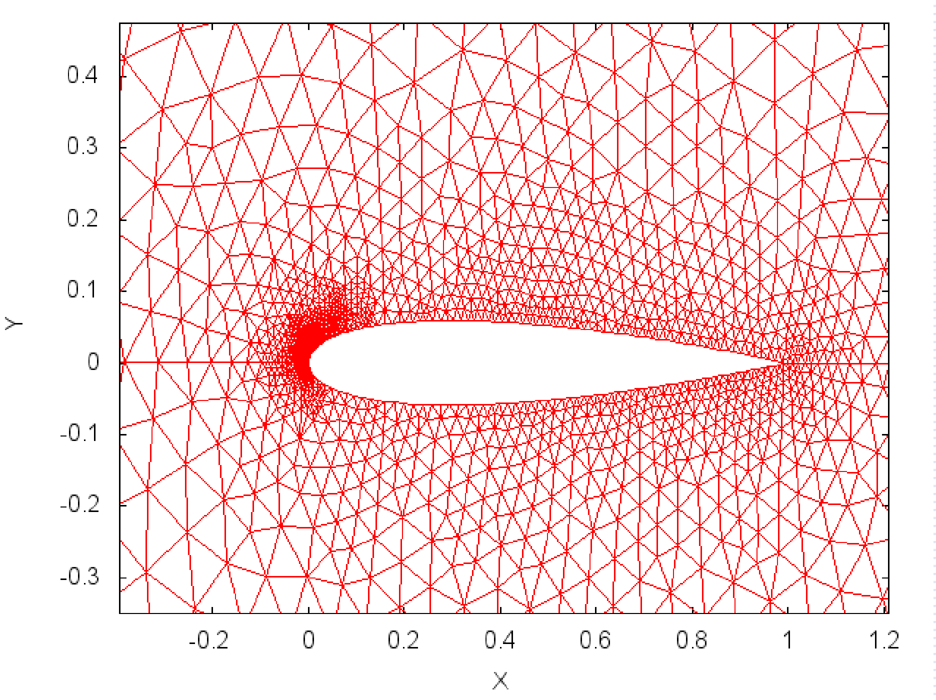


- Weak shock wave. Stagnation point region
- Pressure criterion and energy error per cell (10% of the initial grid)
- 4 levels of refinement

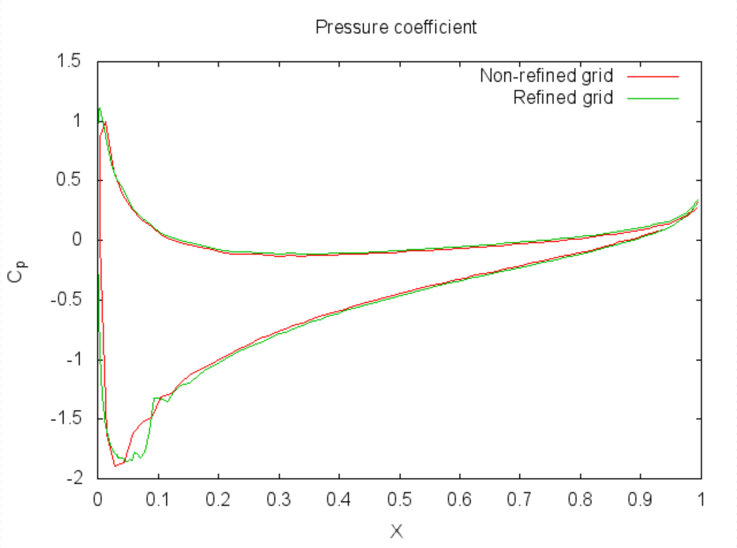
$Ma_\infty = 0.6, angle\ of\ inflow = 4.0°$

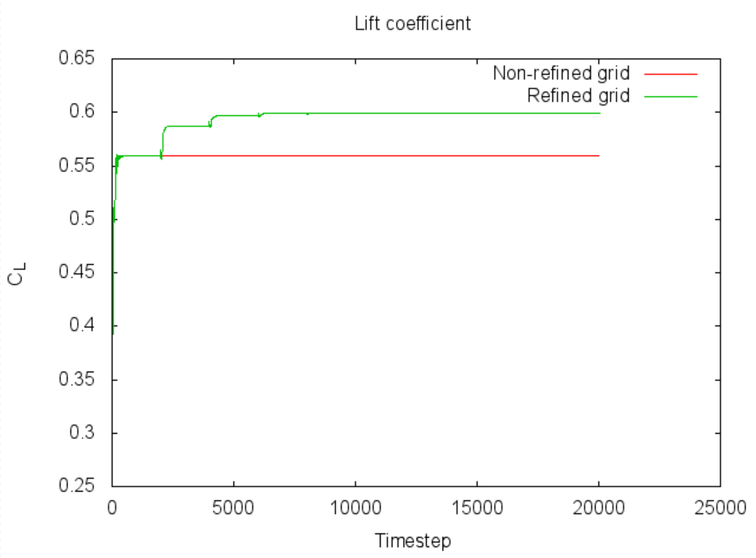
- Initial unstructured grid: 3658 triangles. Final grid: 8351 triangles
  CFL=20

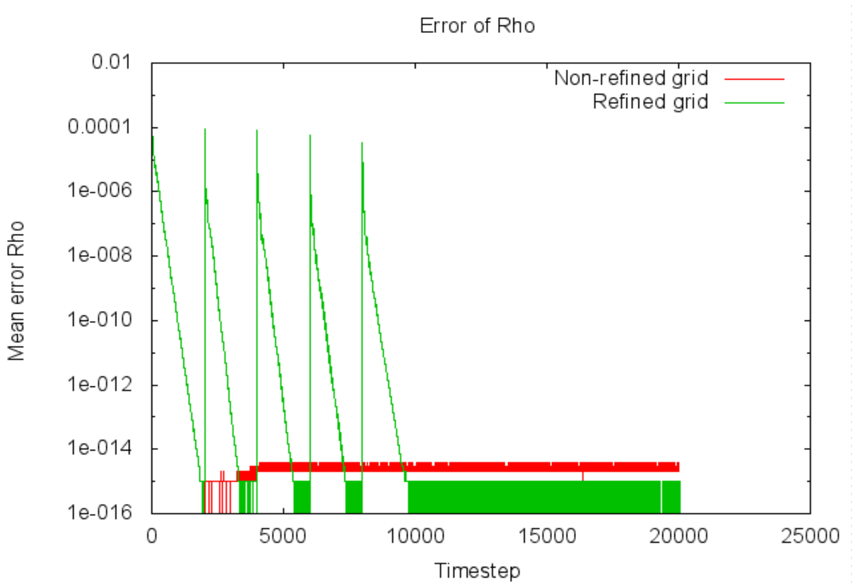




$Ma_{\infty} = 0.6, angle\ of\ inflow = 4.0°$

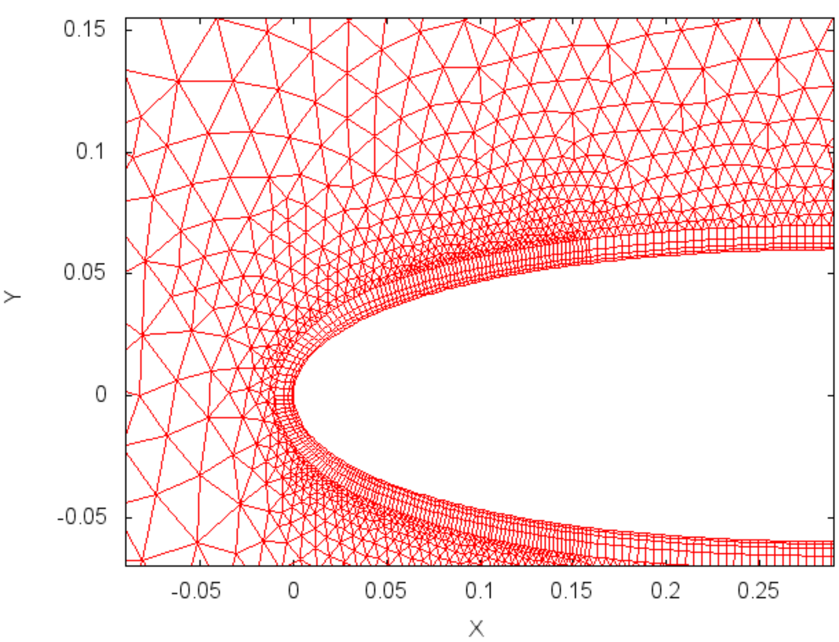


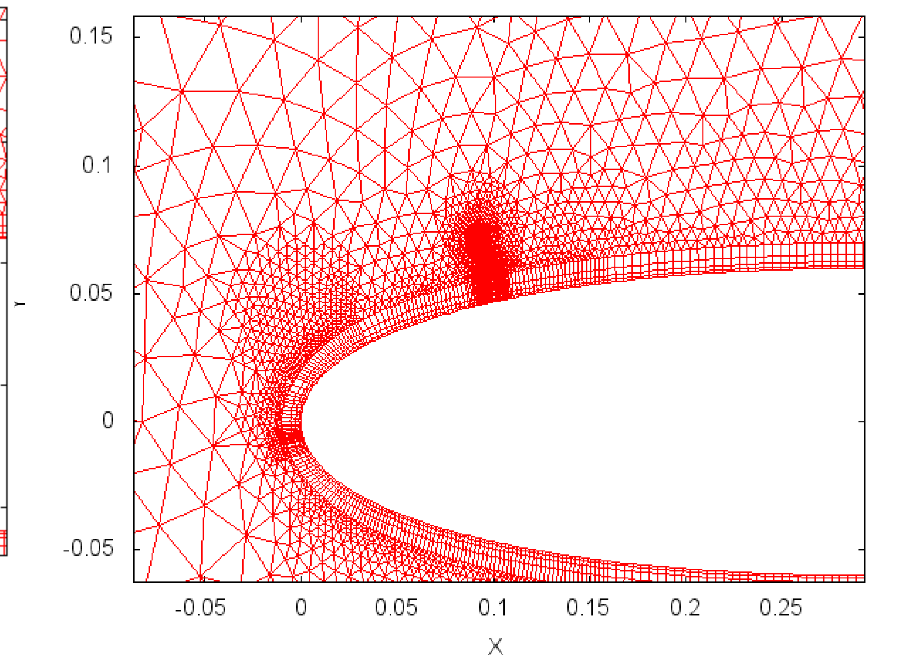




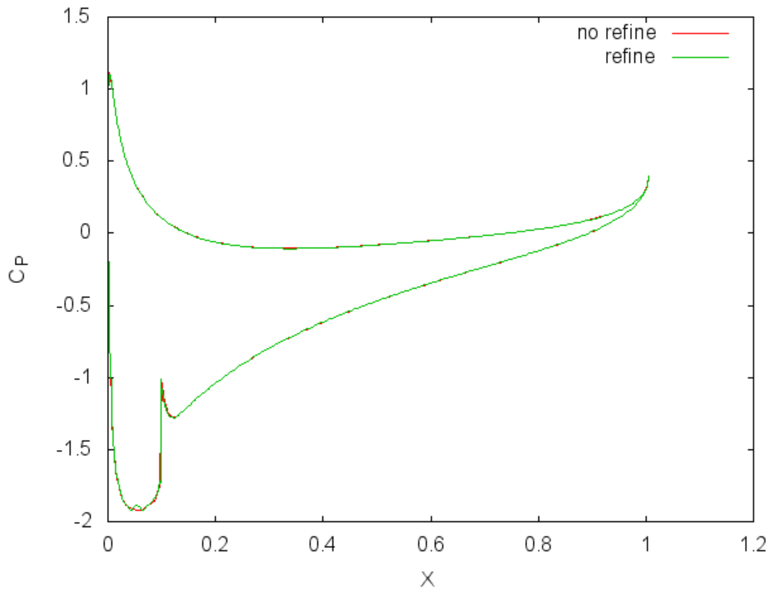
$Ma_{\infty} = 0.6, angle\ of\ inflow = 4.0°$

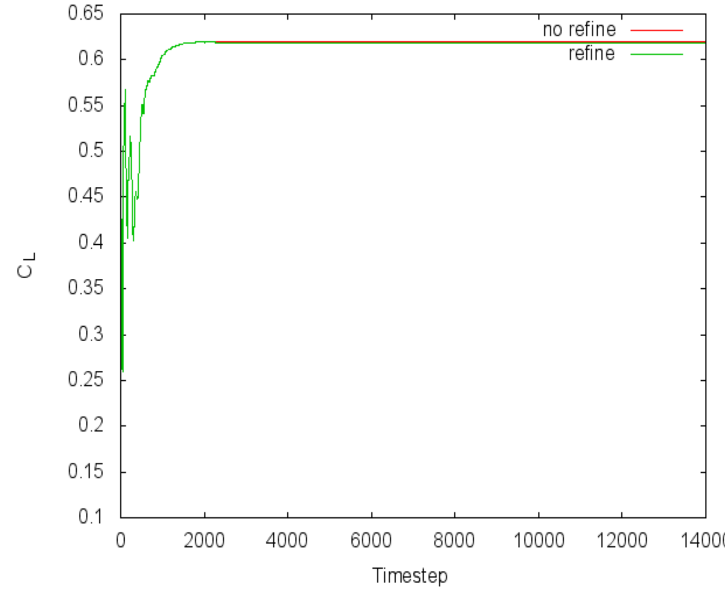
- Triangle and quadrilateral cells
- Initial grid: 10413 cells. Final grid = 36168 cells, CFL=10
- Criterion of Mach number and mean energy error per cell (5% of the initial grid selected)
- 6 levels of refinement

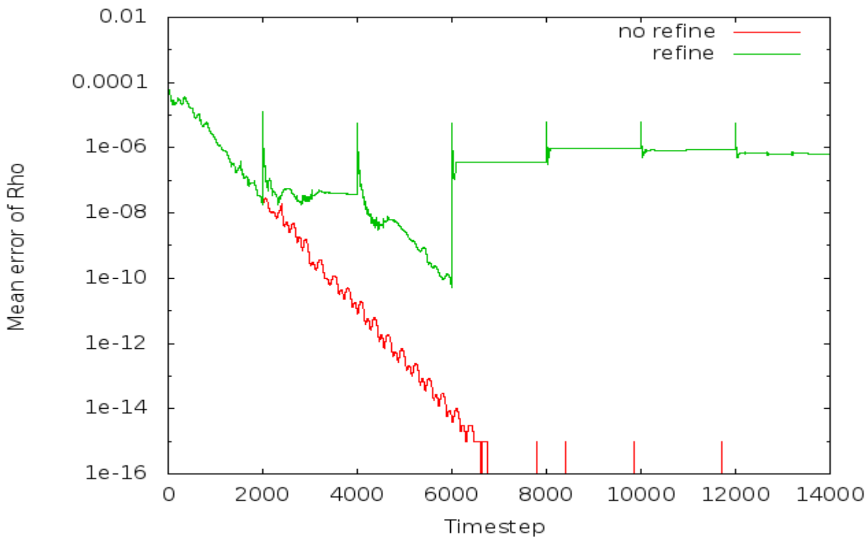


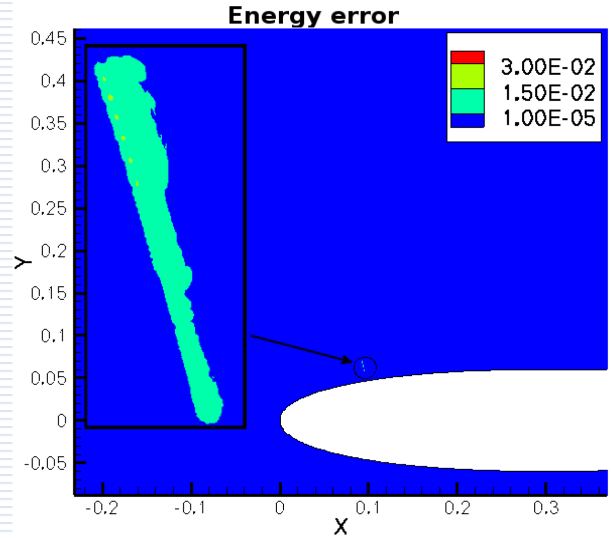


$Ma_\infty = 0.6, angle\ of\ inflow = 4.0°$



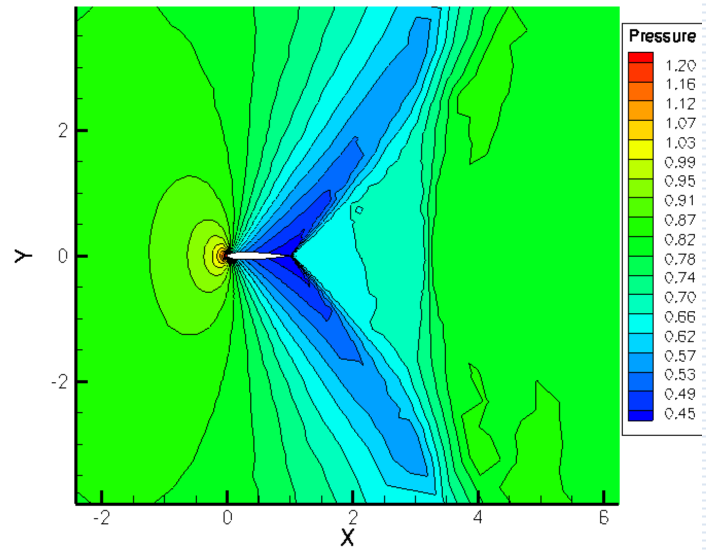






$Ma_\infty = 0.95, angle\ of\ inflow = 0.0°$

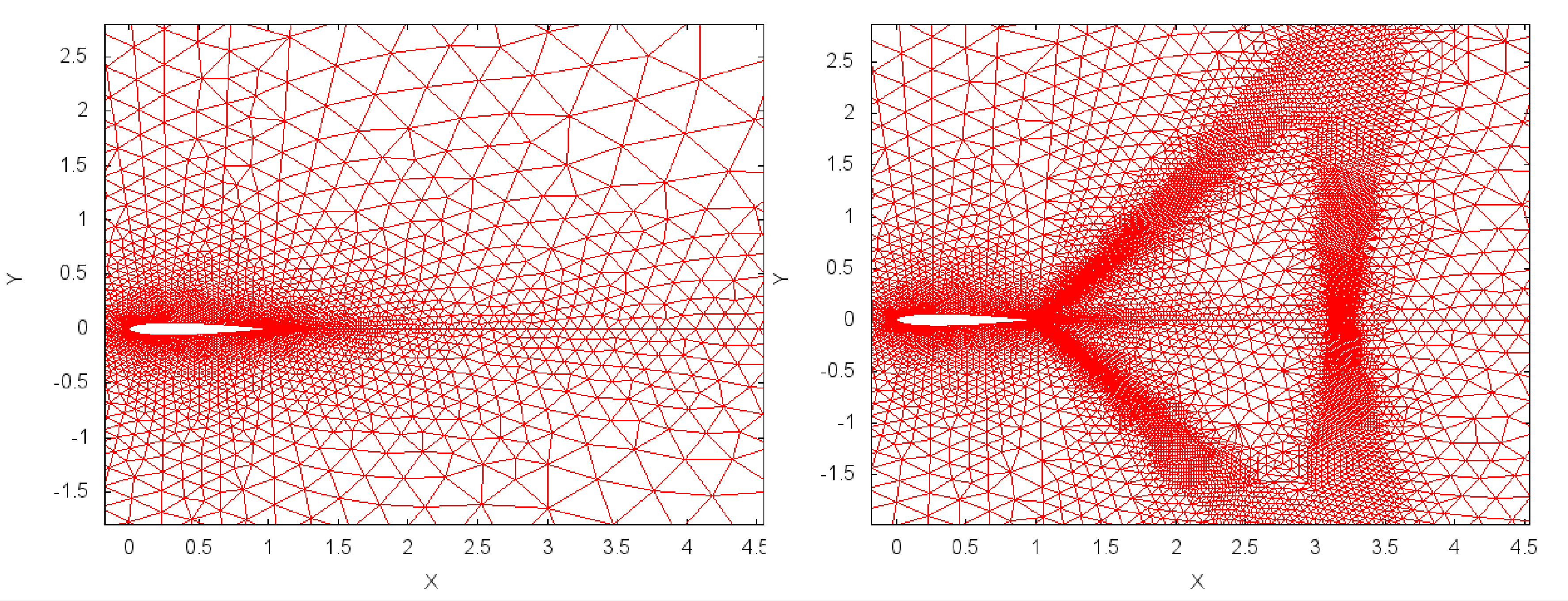
- Inviscid transonic flow around Naca0012 , $Ma_\infty = 0.95$, angle = 0.0°



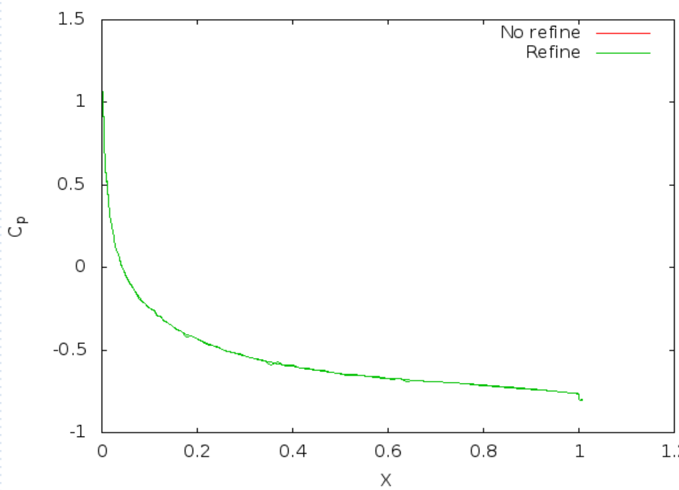
- Two shock waves
- Pressure variations. Criterion : Mach number, pressure, energy error per cells
- Selection from fluid criteria of the 8.2% of initial grid
- 6 levels of refinement

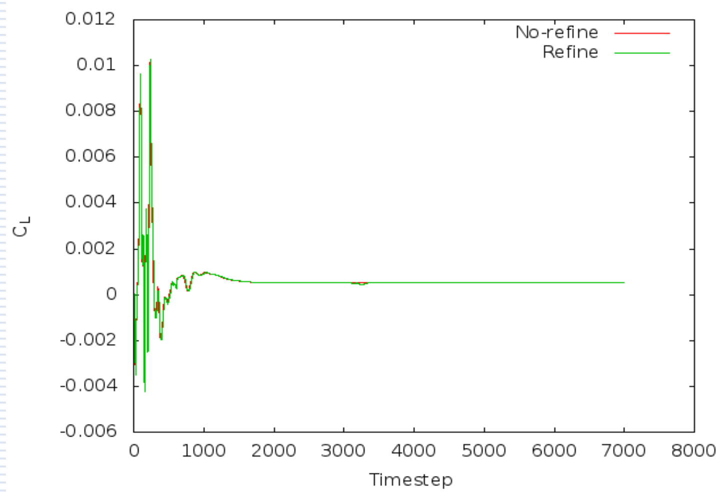
$Ma_\infty = 0.95, angle\ of\ inflow = 0.0°$

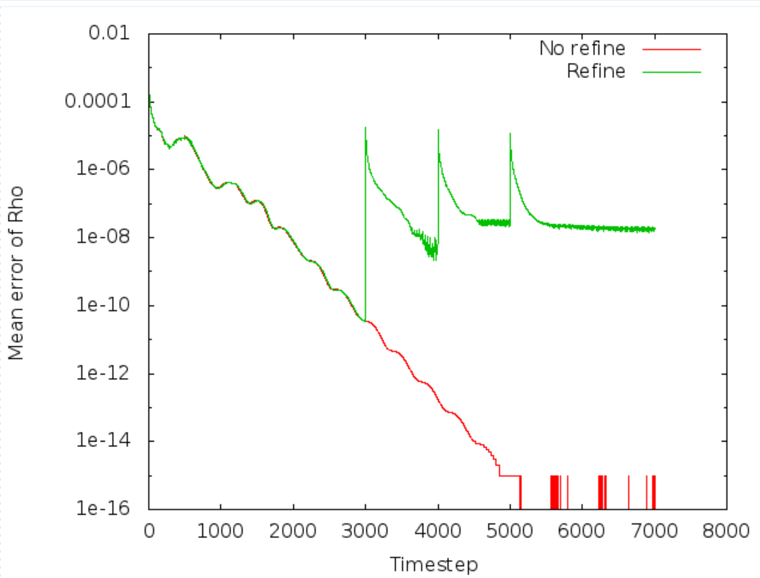
- Initial unstructured grid: 11629 triangles. Final grid: 44487 triangles (fig.) CFL=10

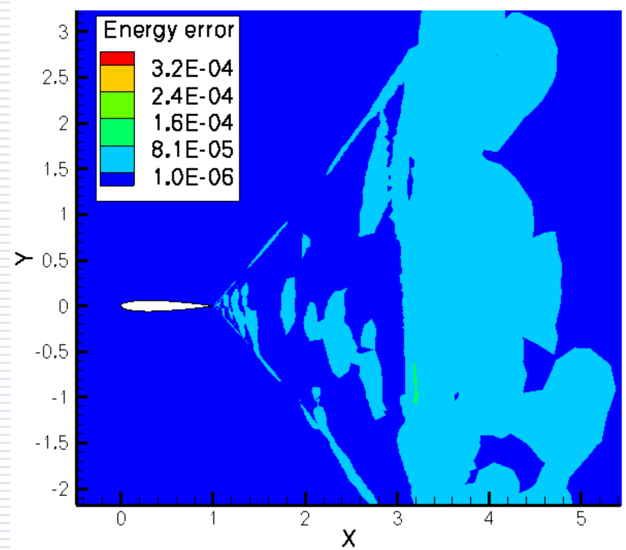


$Ma_\infty = 0.95, angle\ of\ inflow = 0.0°$



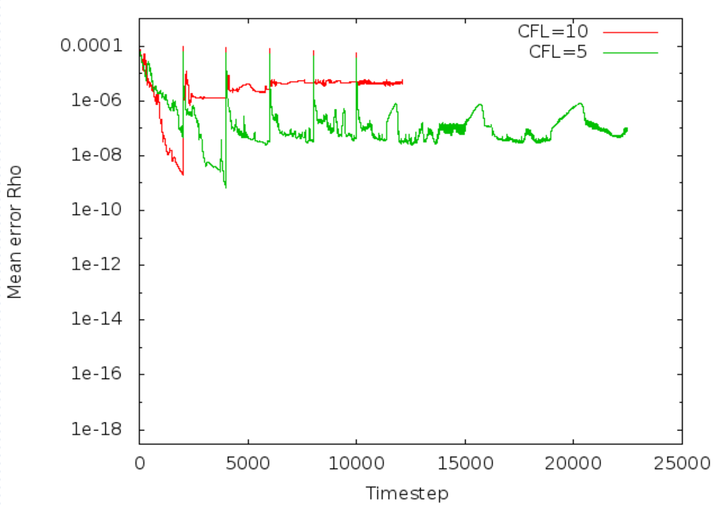


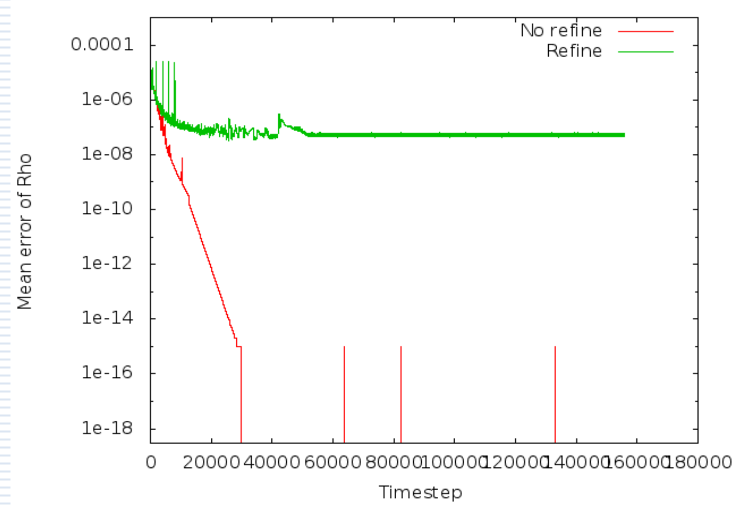


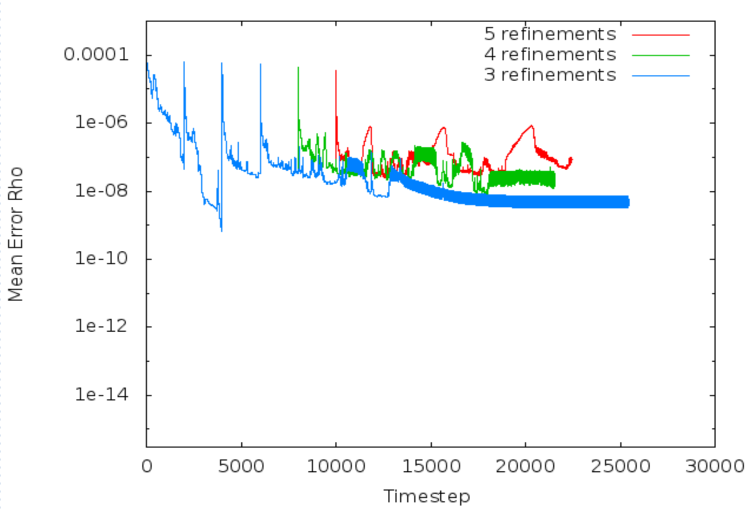


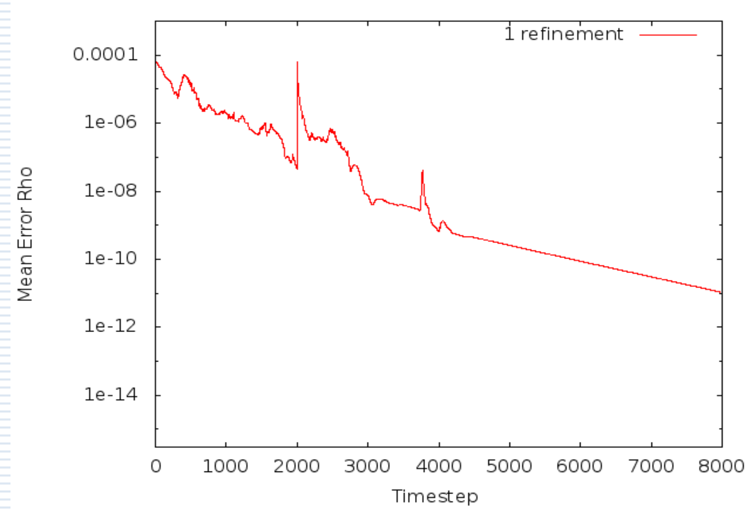
Some convergence parameters

- **Parameters: CFL number, coarse initial grid (13586 cells), levels of refinement**









Conclusions

- Development and validation of methods for mesh refinement in multiple steps with various criteria of the solver. Triangle and hybrid meshes
- Different test cases were tested for the flow domain where the method is applied. Local refinement
- Mesh changes defined at specific regions for each level from physical information. Mesh is modified at each of these regions and the rest grid remains invariant. Restriction of the convergence level after a limit. Grid smoothness is also affected at the transient regions. Geometry construction is taken care
- Pressure and lift coefficients improved
- Interpolation of variables with the simplest conservative scheme

Future implementations for improvement

- Apply the opposite procedure of coarsening
- Implementation of different interpolation schemes of higher order
- Re-meshing at carefully selected mesh regions of the grid if needed
- Implementation for parallel computing
- Refinement using hanging nodes

The end

Thank you for your time !