

CFD: Compressible Flows

High Performance Computing Seminar

Austen Duffy

April 8, 2008

CFD: Compressible Flows

◆ What is a Compressible Flow?

A compressible flow is one for which the Mach number exceeds 0.3. The Mach number is a dimensionless number that relates the speed of an object in a medium to that medium's speed of sound.

$$M = \frac{v_0}{v_s}$$

When the Mach number exceeds 0.3 there is a change in density of the medium (compression) with respect to the pressure.

CFD: Compressible Flows

- ◆ What are the differences between compressible and incompressible flows?

There are two main differences. First, in compressible flows a change in velocity leads to a change in temperature which is NOT negligible as it is in incompressible flows. Second, when the Mach number reaches or exceeds one, there is often the formation of a shock wave which the numerical method must be able to handle.

CFD: Compressible Flows



F/A-18F Super Hornet breaking the sound barrier

CFD: Compressible Flows

◆ The Importance of HPC for Compressible Flows

While high performance computing capabilities are immensely important to all CFD applications, they are indispensable for compressible flow simulations used in experimental design. Incompressible flow designs are capable of being physically tested (e.g. A tow tank for ships or wind tunnel for subsonic aircraft) at a reasonable cost, but for designs such as hypersonic and supersonic aircraft, CFD simulations may provide the best option due to the high cost and energy demands (and difficulty) of testing in supersonic and hypersonic wind tunnels.

The ultimate goal for these applications is the development of a true high resolution, highly accurate numerical wind tunnel.

CFD: Compressible Flows

- ◆ Types of interesting compressible flows:
 1. Transonic: Just below and above the speed of sound at Mach 0.8-1.2. In transonic flows Prandtl-Glauert singularities occur around aircraft, these are intense low pressure areas that become visible if enough water vapor is present.



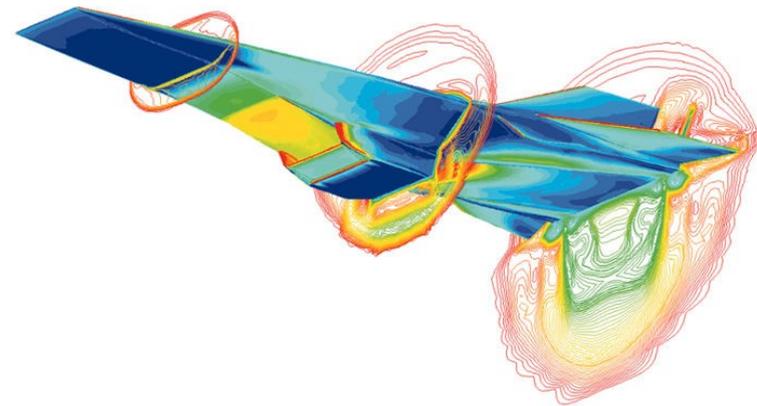
CFD: Compressible Flows

- ◆ Types of interesting compressible flows:
 2. Supersonic: Supersonic flows occur with Mach numbers that are roughly between 1 and 5. At supersonic speeds vapor cones form in front of aircraft and continue to travel at the speed of sound.



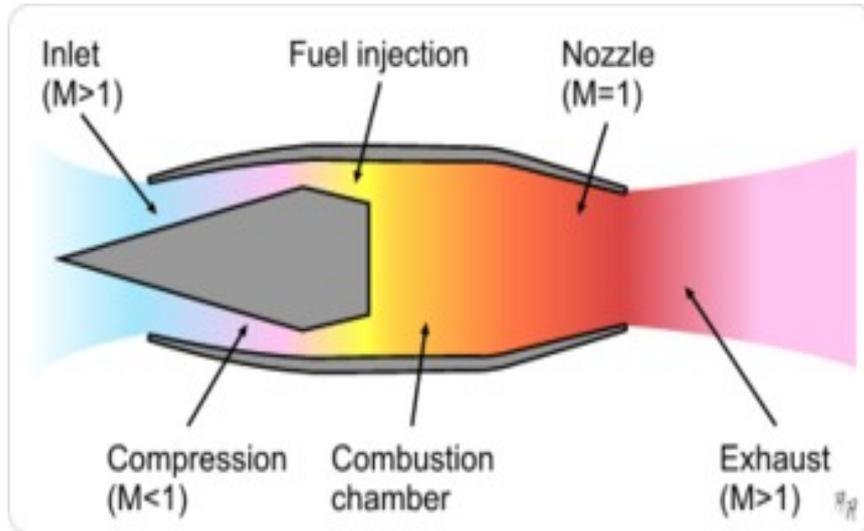
CFD: Compressible Flows

- ◆ Types of interesting compressible flows:
 3. Hypersonic: Flows much faster than the speed of sound with Mach numbers exceeding 5. These flows differ from supersonic in several areas such as thin shock layers and high temperature increases as a result of molecular dissociation and ionization.

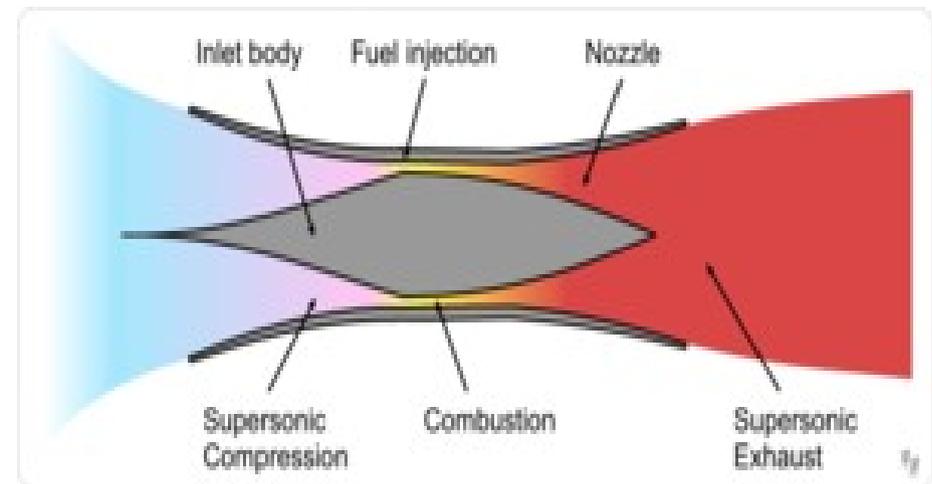


CFD: Compressible Flows

◆ Ramjet vs. Scramjet



Ramjet Engine: Only capable of speeds of up to Mach 5.



Scramjet Engine: Developed for hypersonic speeds above mach 5.

CFD: Compressible Flows

◆ Treatment of Shocks

Due to the presence of shocks in compressible flows, numerical methods are usually developed utilizing the Euler equations. These are derived from the Navier-Stokes equations by ignoring viscous terms, allowing shock waves to be treated as discontinuities.

The Euler equations solved in compressible flow problems are almost always written in a conservation form.

Note: There are two main ways to capture the flow features around a shock

1. Shock tracking (difficult)
2. Shock Capturing (easier to parallelize, better for AMR)

CFD: Compressible Flows

- ◆ Conservation form of the Euler equations

$$\frac{\partial \mathbf{m}}{\partial t} + \frac{\partial \mathbf{f}_x}{\partial x} + \frac{\partial \mathbf{f}_y}{\partial y} + \frac{\partial \mathbf{f}_z}{\partial z} = 0,$$

$$\mathbf{m} = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ E \end{pmatrix} \quad \mathbf{f}_x = \begin{pmatrix} \rho u \\ p + \rho u^2 \\ \rho uv \\ \rho uw \\ u(E + p) \end{pmatrix} \quad \mathbf{f}_y = \begin{pmatrix} \rho v \\ \rho uv \\ p + \rho v^2 \\ \rho vw \\ v(E + p) \end{pmatrix} \quad \mathbf{f}_z = \begin{pmatrix} \rho w \\ \rho uw \\ \rho vw \\ p + \rho w^2 \\ w(E + p) \end{pmatrix}.$$

CFD: Compressible Flows

◆ The Finite Volume Method

The finite volume method is used in many CFD codes. It is desirable as it can be used on both structured and unstructured meshes and most importantly here it is a conservative method, and is easily applied to PDE's written in integral conservation form.

This method works by converting volume integrals to surface integrals (divergence theorem) which can be evaluated as fluxes at the cell walls. The flux entering each cell must equal the flux leaving through the walls which makes the method conservative. On structured meshes it is easily parallelized, as each cell only needs information from neighboring cells and the order is predetermined. For unstructured meshes we still only need info from neighboring cells, but the setup is trickier to handle.

Reference: “Finite Volume Methods for Hyperbolic Systems” by Leveque

CFD: Compressible Flows

◆ The Problem of Two States: Riemann Solvers

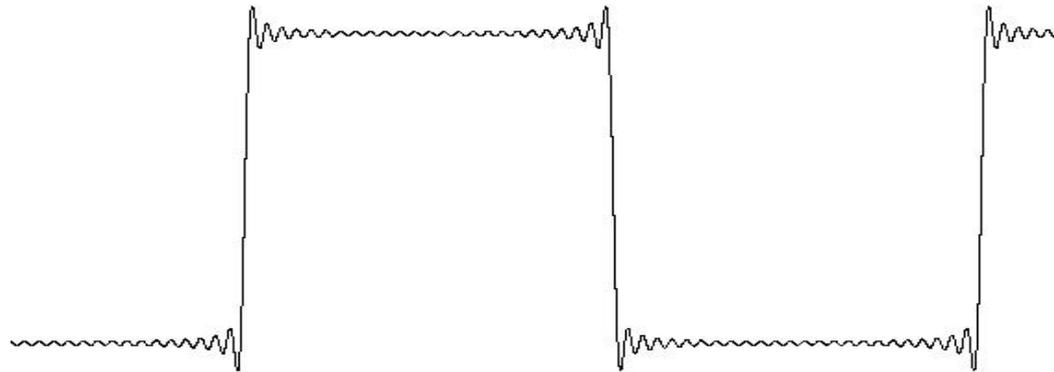
Riemann solvers are used extensively in CFD for solving Riemann problems which arise naturally in finite volume methods.

In finite volume methods, Riemann problems relate the states of neighboring cells using their characteristics.

CFD: Compressible Flows

◆ Gibbs Phenomena

Gibbs phenomena is apparent as spurious oscillations when attempting to use high order methods near a shock. One must use monotonic or methods which preserve monotonicity (i.e. TVD methods) to avoid these oscillations.



CFD: Compressible Flows

◆ Godunov's Theorem

Godunov proved that only first order linear schemes are monotonicity preserving and hence Total Variation Diminishing (TVD).

So we have a problem: How do we get high resolution when we can only avoid Gibbs phenomena by using a first order method?

CFD: Compressible Flows

◆ Slope and Flux Limiters

High resolution methods can be used which utilize flux limiters. These methods essentially switch between a high order method when the gradient is low and a low order method when the gradient is steep (e.g. when a shock is present). This allows the avoidance of Gibbs' phenomena in the solution while maintaining high resolution. There are numerous treatments of flux limiters used in accordance with finite volume (and finite difference) methods, but one well known and rather popular method is the MUSCL scheme (monotone upwind centered scheme for conservation laws).

CFD: Compressible Flows

◆ Reynolds Averaged Navier-Stokes

Another method used in compressible flow simulations, RANS are time averaged equations of motion for fluid flow that are derived from the Navier-Stokes equations using the Reynolds decomposition of the flow variables into a time averaged component and a fluctuating component i.e.

$$u(x, t) = \bar{u}(x) + u'(x, t)$$

And the resulting derivation yields

$$\rho \frac{\partial \bar{u}_i}{\partial t} + \rho \frac{\partial \bar{u}_j \bar{u}_i}{\partial x_j} = \rho \bar{f}_i + \frac{\partial}{\partial x_j} \left[-\bar{p} \delta_{ij} + \mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \overline{\rho u'_i u'_j} \right].$$

These equations are primarily used for modeling turbulent flow.

CFD: Compressible Flows

◆ Meshing: Structured Grid Methods

Chimera: A method used in many modern codes utilizing structured grids, Chimera uses a system of overlapping grid blocks of varying resolution, allowing complex geometries to be decomposed into a system of simple geometric grids. This is also known as an overset grid method, and can be used with various solvers.

CFD: Compressible Flows

◆ Dynamic Adaptive Mesh Refinement

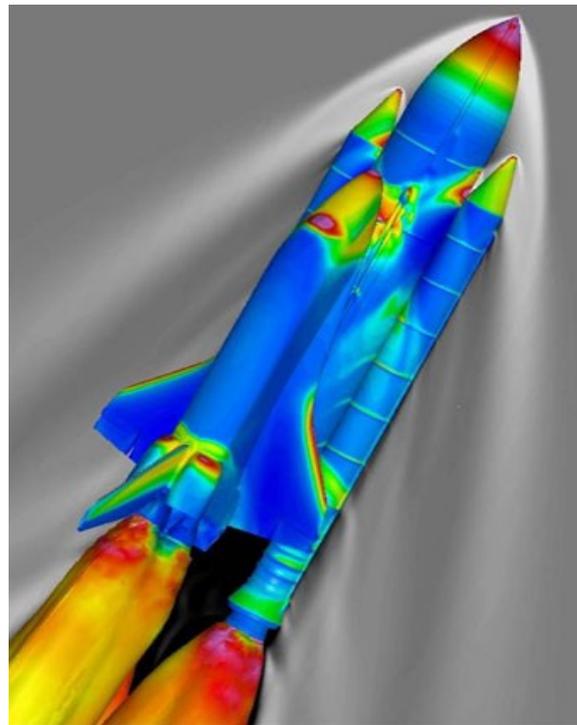
In Dynamic Adaptive Mesh Refinement, the mesh may be refined after a certain number of iterations. Various criteria may be used to determine where refinement is needed, for example one may want to refine an area where the local vorticity is high.

Dynamic Adaptive Refinement is also used to refer to refining areas by adding overset grids when deemed necessary

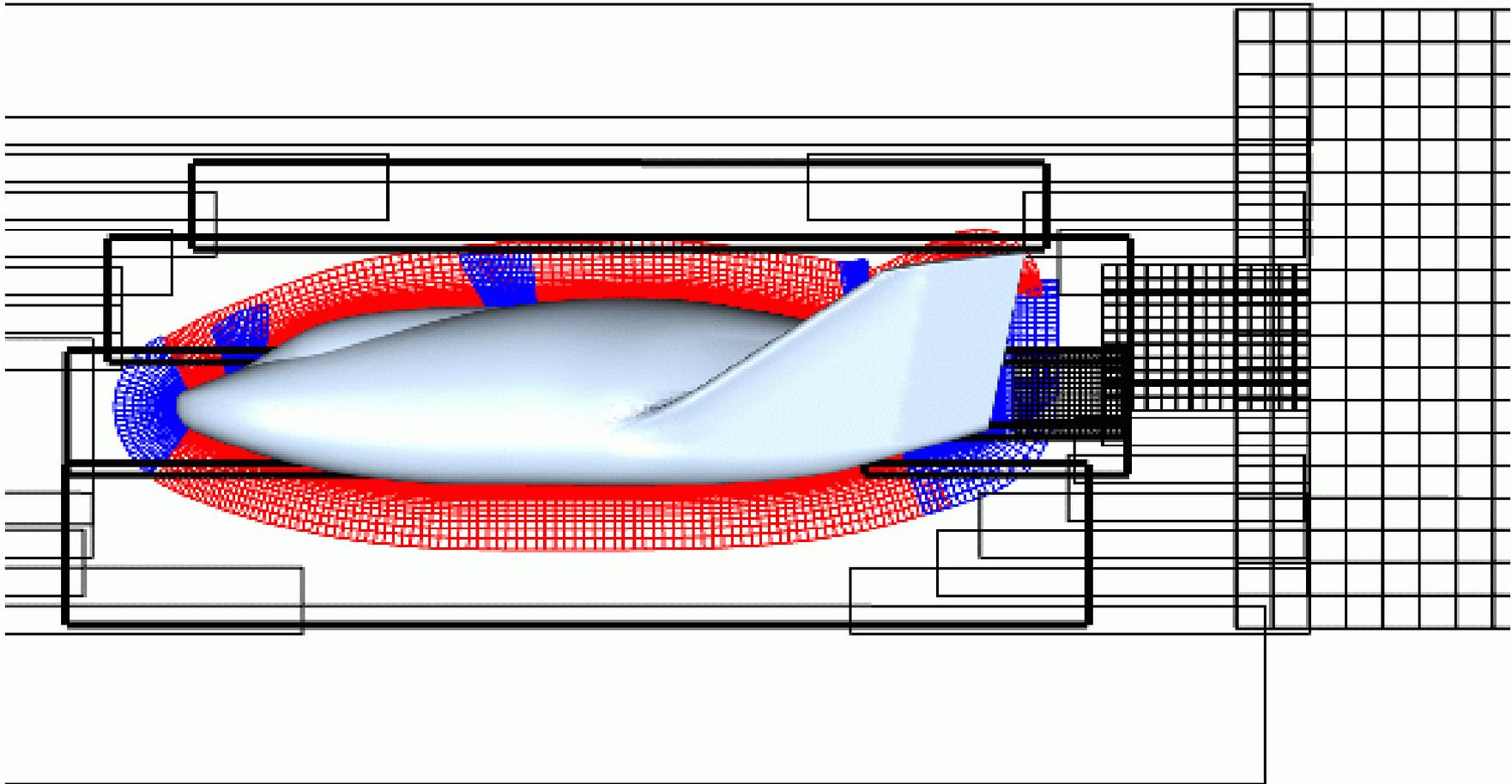
CFD: Compressible Flows

◆ OVERFLOW

NASA developed a code OVERFLOW (for OVERset grid FLOW solver) that is freely available to Department of Defense employees and U.S. Citizens who sign a non-disclosure agreement. It solves the Reynolds averaged Navier-Stokes (RANS) equations as opposed to the Euler equations.

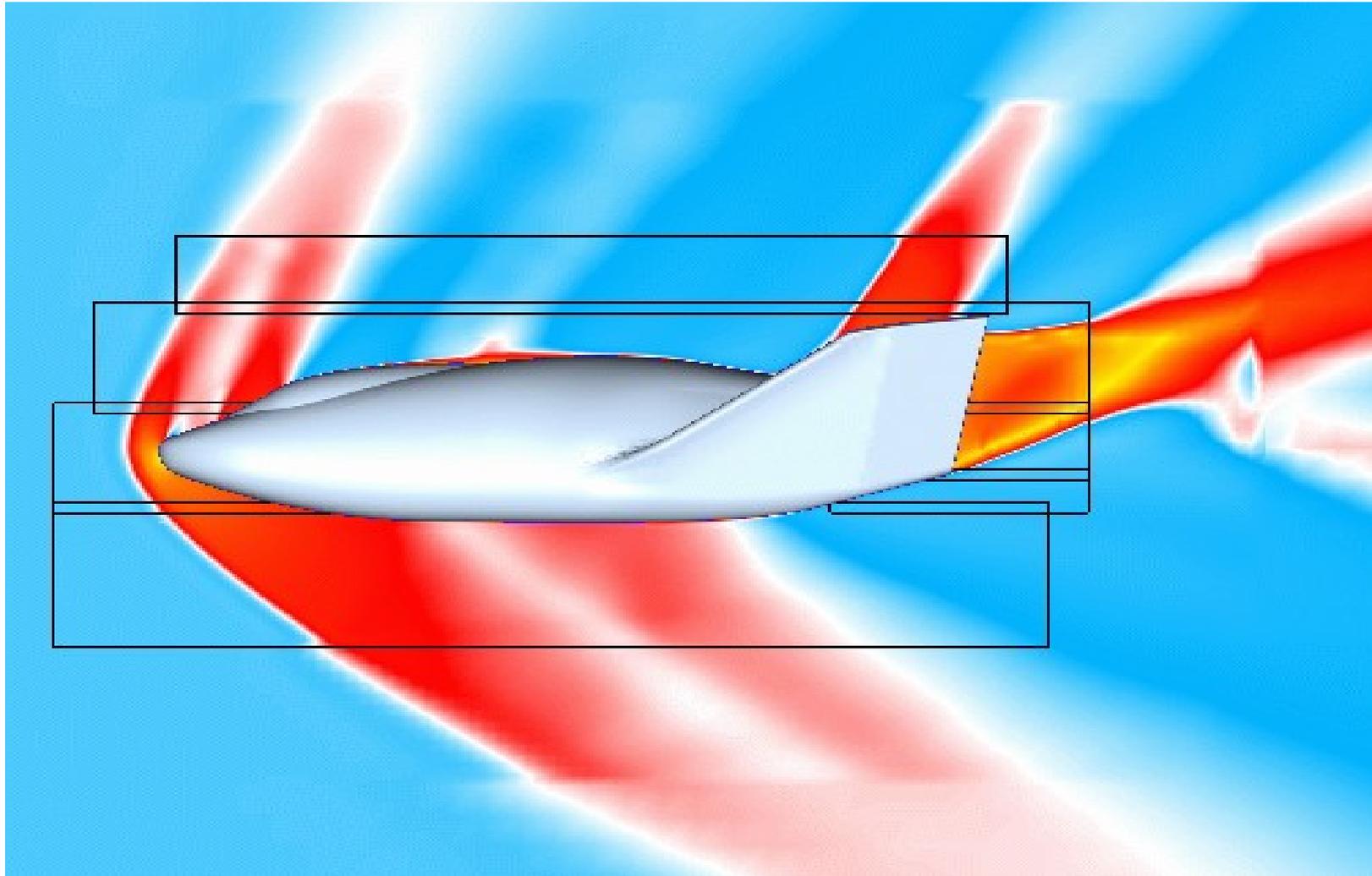


CFD: Compressible Flows



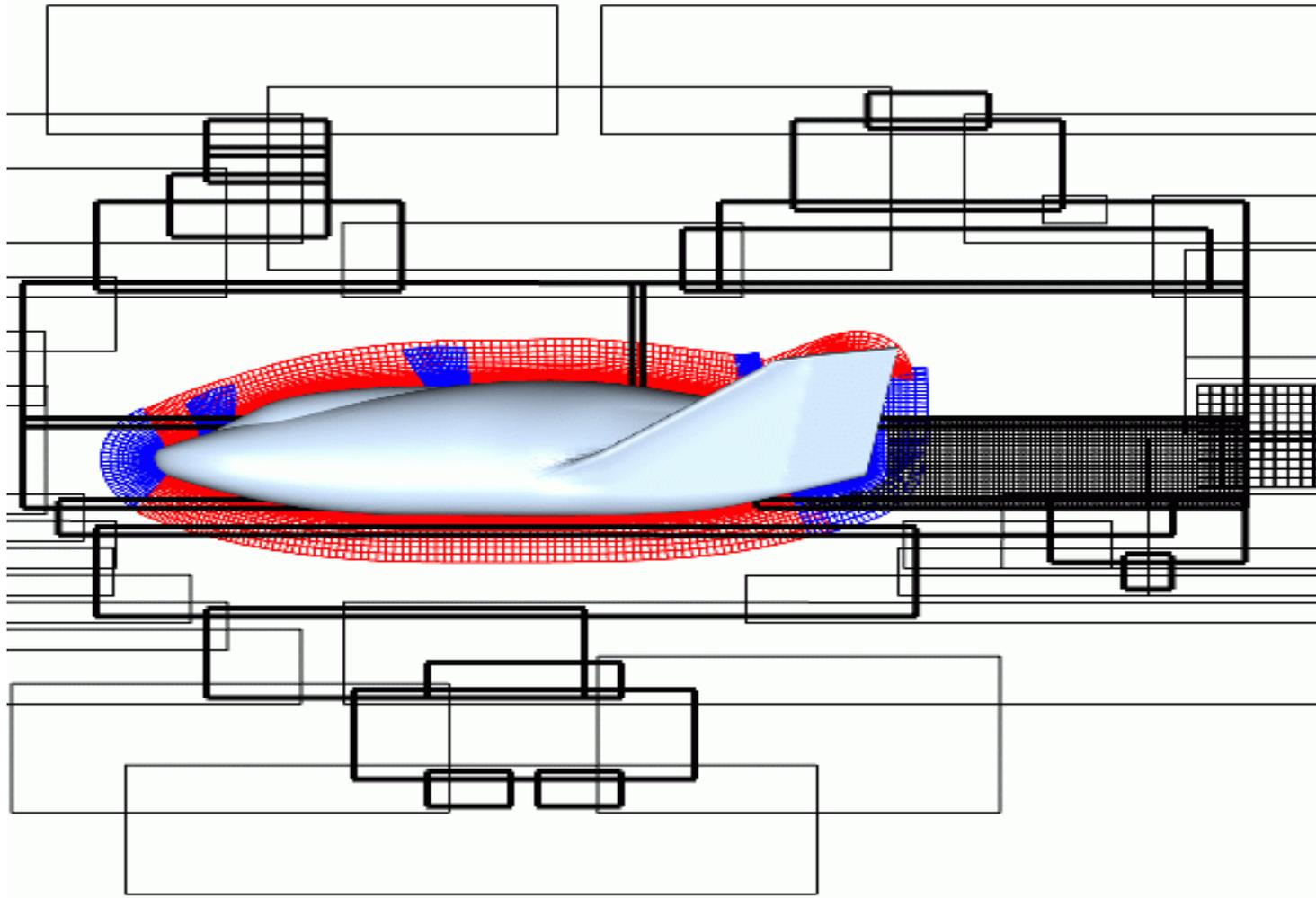
Chimera discretization of X-38 flight vehicle in NASA OVERFLOW-D code

CFD: Compressible Flows



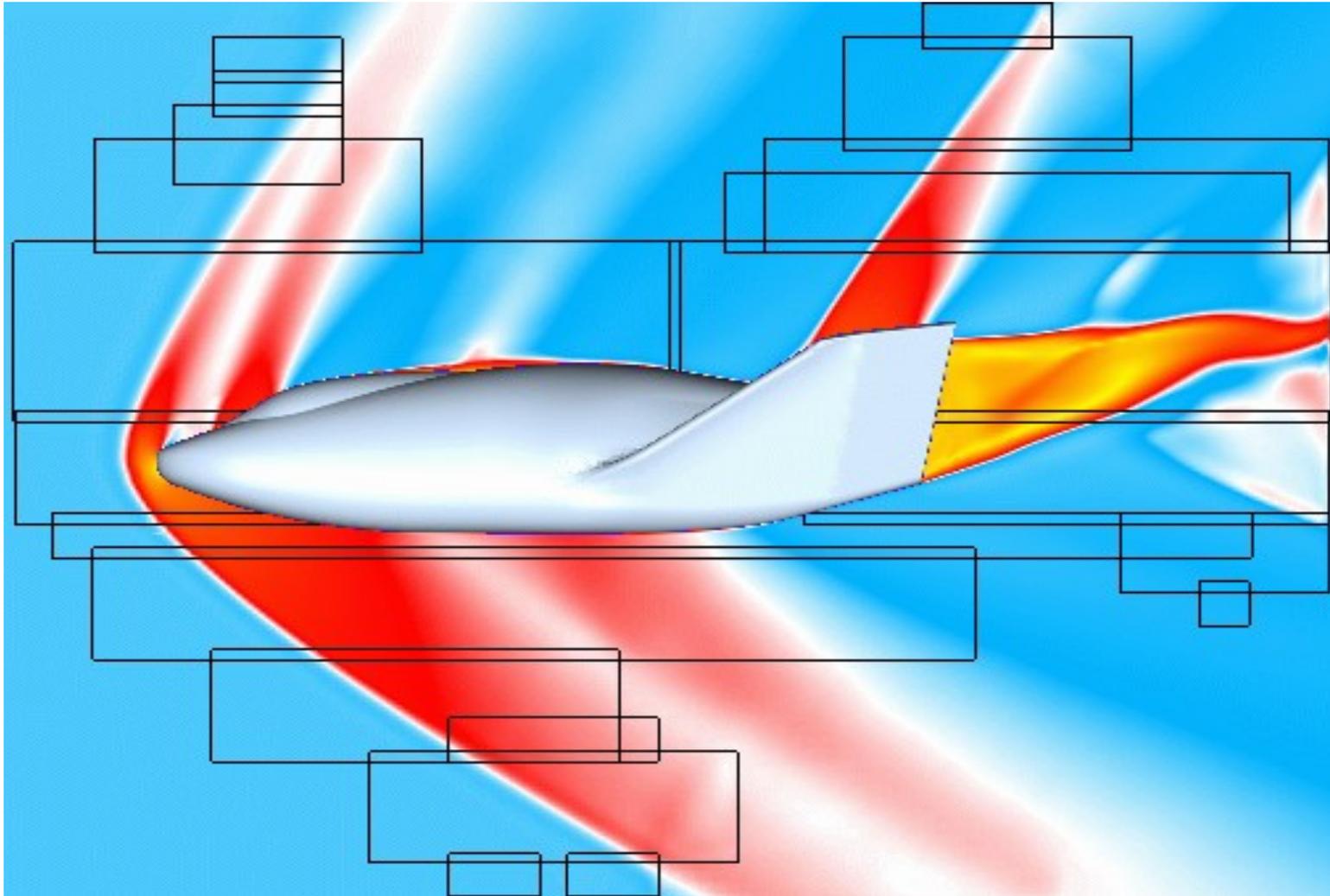
NASA OVERFLOW-D simulation of X-38 flight vehicle using Chimera

CFD: Compressible Flows



Grid after 1 adapt cycle (50 time steps between adapt cycles)

CFD: Compressible Flows



Simulation on adapted grid

CFD: Compressible Flows

◆ Conclusion

Compressible flow simulations are of vital importance to the aerospace engineering community which will always seek more accuracy and higher resolution creating a demand for faster codes and making the use of high performance computing strategies invaluable.

As these flows require immense numbers of grid points for high resolution, the tool in HPC of most importance is the parallelization of the grid structures utilizing AMR and overest grids.