OVERFLOW 2 Training Class

Pieter G. Buning

NASA Langley Research Center
pieter.g.buning@nasa.gov

12th Symposium on Overset Composite Grids
& Solution Technology
Georgia Institute of Technology
Atlanta, GA
October 6-9, 2014

Preliminaries

OVERFLOW website: http://overflow.larc.nasa.gov

- – Release notes

- – User's Manual

- – Procedure for requesting the code

- – Slides from 2010 training class (includes Bobby Nichols' morning session)

- – Reference papers

OVERFLOW email list:

https://lists.nasa.gov/mailman/listinfo/overflow

10/6/2014

Class Outline

- **Introduction**
- **OVERFLOW-D mode without grid motion**
- **OVERFLOW-D mode with grid motion**
- **Solution adaption**
- **Compiling and running OVERFLOW**
- **Utilities and test cases**

10/6/2014

Introduction

- Overset grid process
  - Compare OVERFLOW mode vs. OVERFLOW-D mode
  - Input files
  - Moving body simulation

Overset Grid Approaches

- OVERFLOW 2: two modes of operation
  - OVERFLOW mode with grid joining input from Pegasus 5 or SUGGAR
  - OVERFLOW-D mode using DCF
- OVERFLOW mode
  - All grids are created external to the flow solver (**grid.in**)
  - Pegasus 5 (or SUGGAR) used to cut holes and establish interpolation stencils (**XINTOUT**)
  - No moving body capability, no grid adaption
  - OVERFLOW namelist input
- OVERFLOW-D mode using DCF (Domain Connectivity Function)
  - Near-body grids are created external to the flow solver (**grid.in**)
  - X-rays of body surfaces used for cutting holes (**xrays.in**)
  - Additional namelist inputs
  - Geometry Manipulation Protocol (GMP) files to describe bodies and motion (**Config.xml, Scenario.xml**)
- OVERFLOW 2 can do either approach
  - Decision is based on whether additional namelists are present in input file

Example: Multi-Element Airfoil

- OVERFLOW mode
  - All grids supplied
  - Grid system from Pegasus 5
  - Holes are cut automatically, based on comparable cell sizes
  - Better quality grid system

- OVERFLOW-D mode
  - Only near-body grids supplied
  - Distance from surfaces specified for hole cutting
  - Holes cut by DCF inside OVERFLOW
  - Hole cutting is fast enough for moving body problems
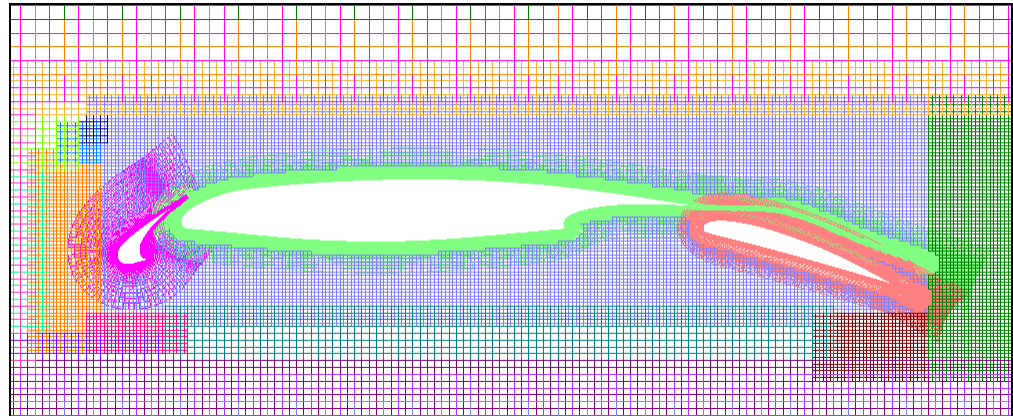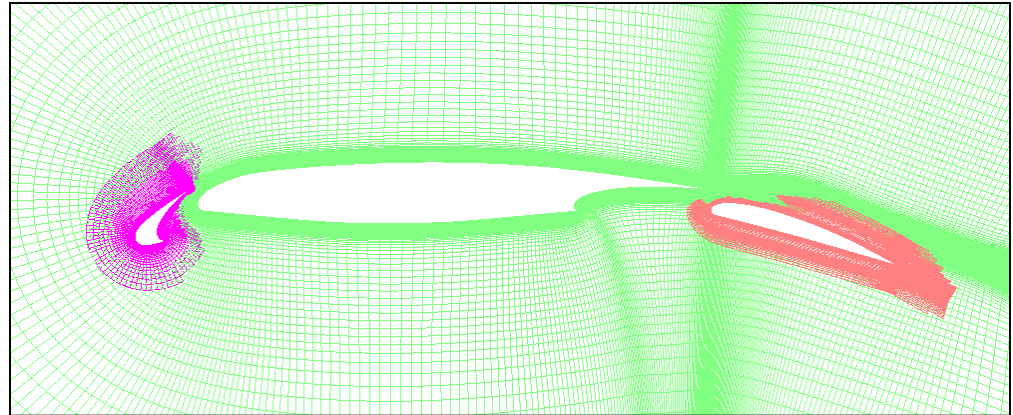
Input Files for OVERFLOW 2

What do you need to be able to run?

- OVERFLOW mode
  - **grid.in** (all grids)
  - **mixsur.inp** (input to force and moment preprocessor)
  - **XINTOUT** (Pegasus 5 hole cutting and interpolation stencils)
  - OVERFLOW namelist input
- OVERFLOW-D mode
  - **grid.in** (near-body grids)
  - **mixsur.inp** (input to force and moment preprocessor)
  - **xrays.in** (x-rays for hole cutting)
  - OVERFLOW namelist input
  - **Config.xml, Scenario.xml** (body properties and positioning)

Moving Body Simulation Process

- Pre-processing:
  - Near-body grid generation
  - Definition of force and moment integration surfaces
  - Creating X-rays for hole cutting
- OVERFLOW grid processing:
  - Off-body grid generation
  - Hole cutting and boundary interpolation stencils
- Moving body simulation
  - Body motion (GMP interface)
  - Time-advance scheme
  - Saving motion, forces, flow solution
- Post-processing
  - Non-trivial!

Flow Simulation Process

- Starting (and for grid adaption):
  - Read near-body grids, move to dynamic position(s)
  - Make off-body grids, and any near-body refinement regions
  - Interpolate flow solution onto new near- and off-body grids
  - Run DCF (cut holes, find interpolation stencils)
  - Advance flow solution one step
  - Compute forces and moments
- Every step:
  - Update near-body grid positions
  - Run DCF
  - Advance flow solution one step
  - Compute forces and moments

10/6/2014

OVERFLOW-D Mode Without Grid Motion

- **NAMELIST inputs**
- **Near-body grid generation**
- **Force and moment integration**
- **Generating X-rays**
- **Off-body grid generation**
- **Grid assembly with DCF**
- **Data surface grids**

OVERFLOW Namelist Input

- **$GLOBAL** (global inputs for OVERFLOW)
- **$OMIGLB** (global inputs for OVERFLOW-D)
- **$GBRICK** (off-body grid generation)
- **$BRKINP** (user-specified proximity regions)
- **$GROUPS** (grid splitting and load-balancing)
- **$XRINFO** (hole cutting)
- **$DCFGLB** (stencil quality and repair)
- **$FLOINP** (flow parameters)
- **$VARGAM** (variable gamma/multiple species)
- Other namelists per grid…

*Unique to OVERFLOW-D mode

OVERFLOW Namelists per Grid

- **$GRDNAM** (grid name)
- **$NITERS** (subiterations per grid)
- **$METPRM** (numerical method selection)
- **$TIMACU** (time accuracy)
- **$SMOACU** (smoothing parameters)
- **$VISINP** (viscous and turbulence modeling)
- **$BCINP** (boundary conditions)
- **$SCEINP** (species convection equations)

See **over2.2x/doc/namelist.pdf** for a detailed list of all input parameters and definitions

Near-Body Grid Generation

- Volume grids are generated from overset surface grids
- Use Chimera Grid Tools (CGT) or commercial package
- All near-body volume grids concatenated into **grid.in**
- Reference: *W.M. Chan, R.J. Gomez III, S.E. Rogers, and P.G. Buning, "Best Practices in Overset Grid Generation," AIAA 2002-3191, June 2002*
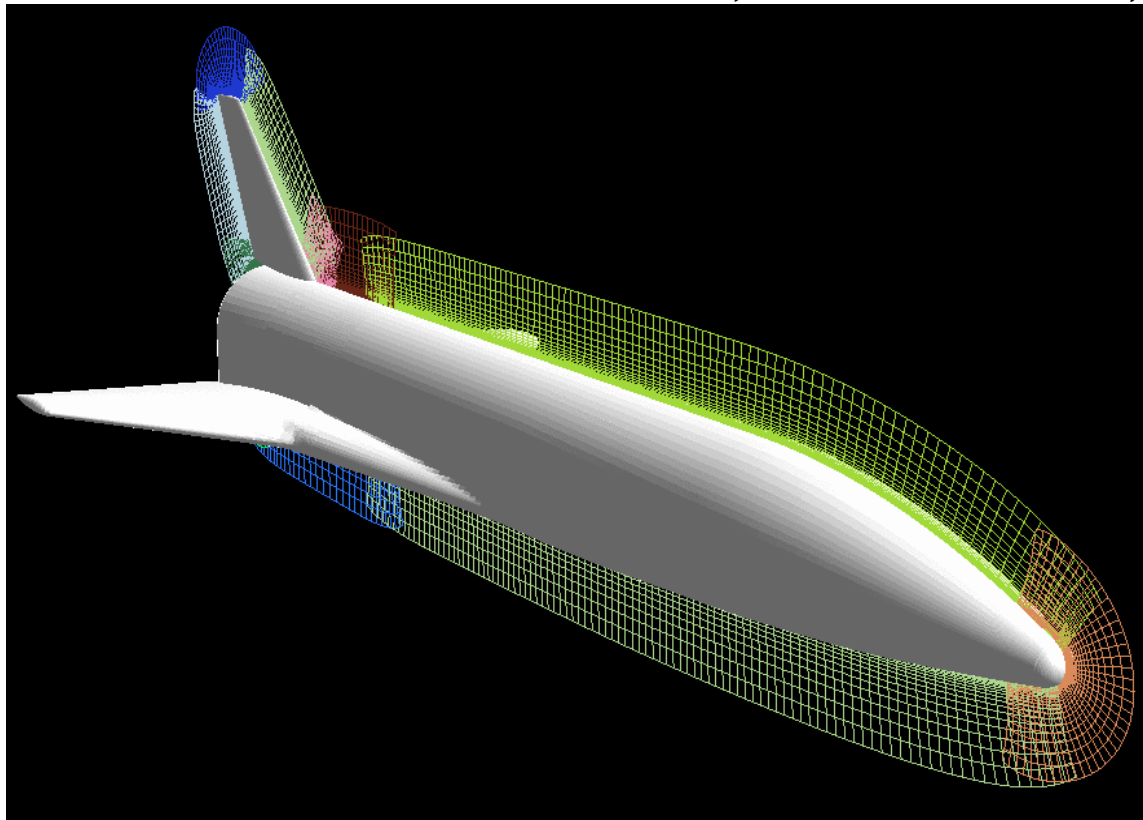
Near-Body Grid Generation

- Distance off wall (**S**) and outer grid spacing (Δ**S**) contribute to the size of the off-body grids
  - We will refer to **S** and Δ**S**, and how they affect the grid generation process, in following sections
- One philosophy:
  - Grow volume grids out until grid cells are roughly square
  - Grow out a total distance **S** which is about 10 times the outer cell size Δ**S**
  - This will determine the off-body grid spacing, and will contribute to the required X-ray spacing as well
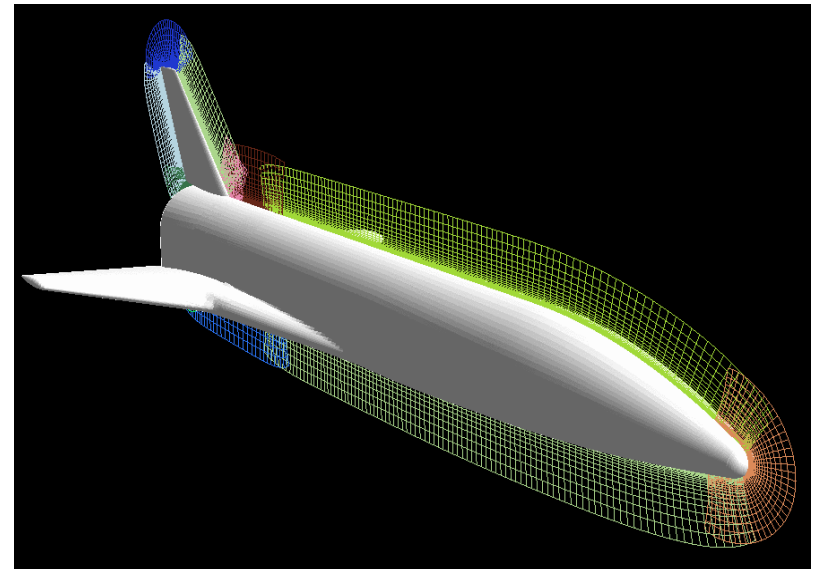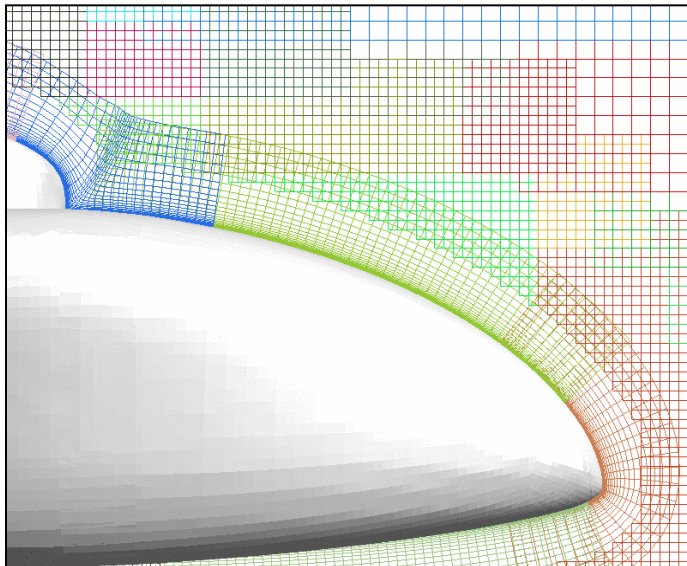
Force and Moment Integration

- Use the **mixsur** utility from CGT to generate combination of surface grids and triangular "zipper grids" for force and moment integration
  - Input file is commonly called **mixsur.inp**
  - Documentation and examples provided with CGT
- For 6-DOF simulations, aerodynamic forces will be used in OVERFLOW to update body position
  - Component (body) names in **mixsur.inp** must match GMP file component names
- Reference conditions (length, surface area, moment reference center)
  - Moment reference center will move with first body using this reference condition
  - Define separate reference conditions for each moving body
  - Example: 2 airfoils (one moves, the other is fixed)

```
0, 500, 500, -1, 0, 0 FSMACH,ALPHA,BETA,REY,GAMINF,TINF
2               NREF
1., 1., 0.25, 0., 0.    REFL,REFA,XMC,YMC,ZMC
1., 1., 0.25, 0., 0.    REFL,REFA,XMC,YMC,ZMC
```

Force and Moment Integration

- For **mixsur**, be sure to visually check resulting integration surfaces!
  - PLOT3D command files generated automatically
  - Look for missing triangles, tangled zipper grids
- **USURP** (Unique Surfaces Using Ranked Polygons) by David Boger (Penn State) is an open source alternative to **mixsur**
  - Same input file; output also recognized by OVERFLOW
  - Designed to overcome zipper grid problems
 - Distributed with Chimera Grid Tools
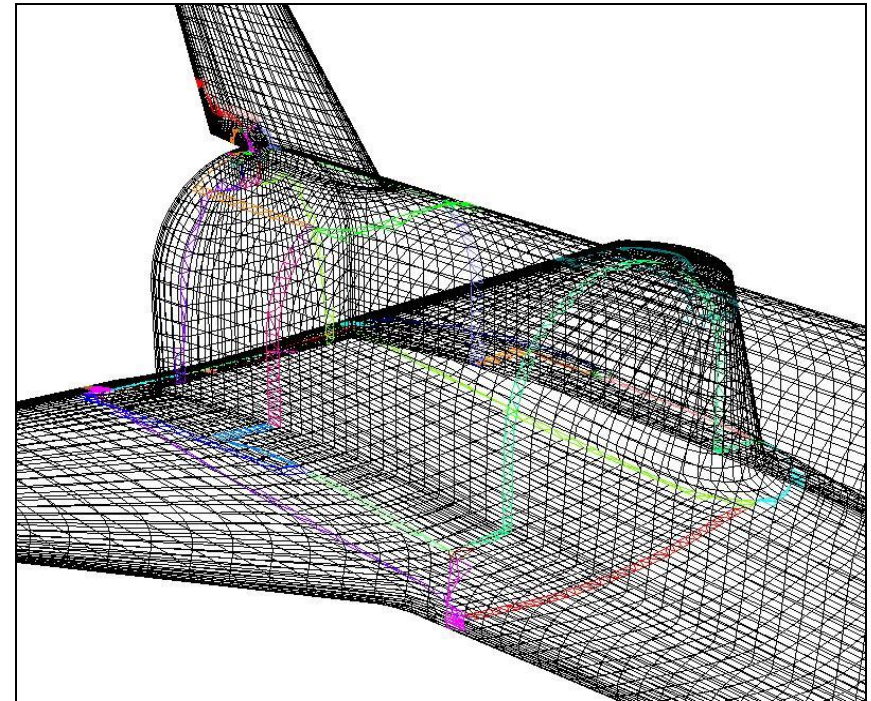
Creating X-Rays

- Creating X-rays
- Picking X-ray spacing
- Using OVERGRID to create X-rays
- X-ray number and Body ID
- Using **gen_x** to create X-rays
- Examples
- Notes and comments

Creating X-Rays

- An X-ray is an (x,y) array of z-value pierce-points of a body
  - These are used inside OVERFLOW for faster hole-cutting for grid connectivity
- Process relies (entirely) on Chimera Grid Tools (CGT)
- Create the **xrays.in** file before running OVERFLOW
- Use OVERGRID (interactive) or use **gen_x** (batch) in CGT

- Before you start, you will need to:
  - Generate a PLOT3D grid file of each body surface
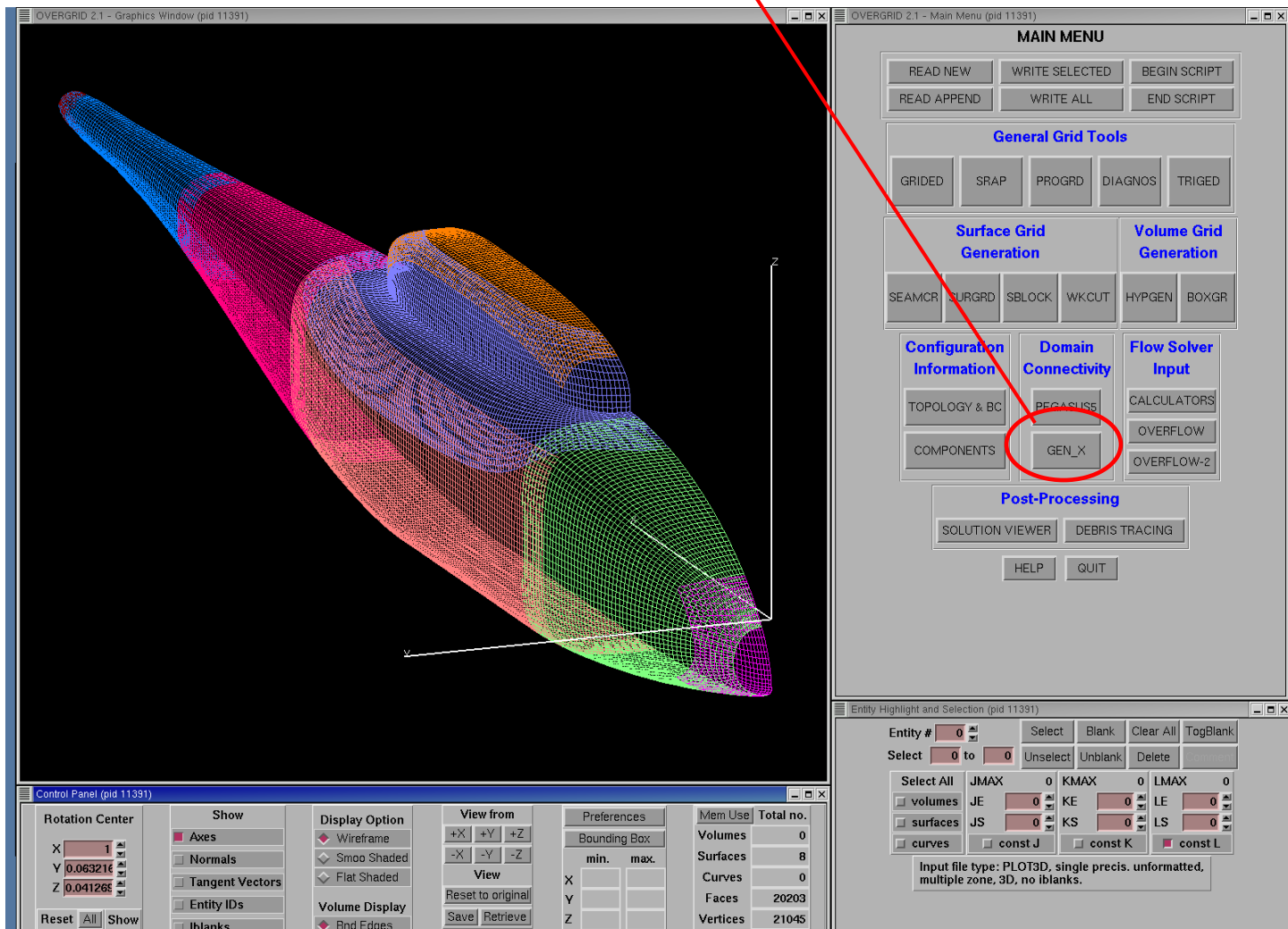 - Pick the X-ray spacing
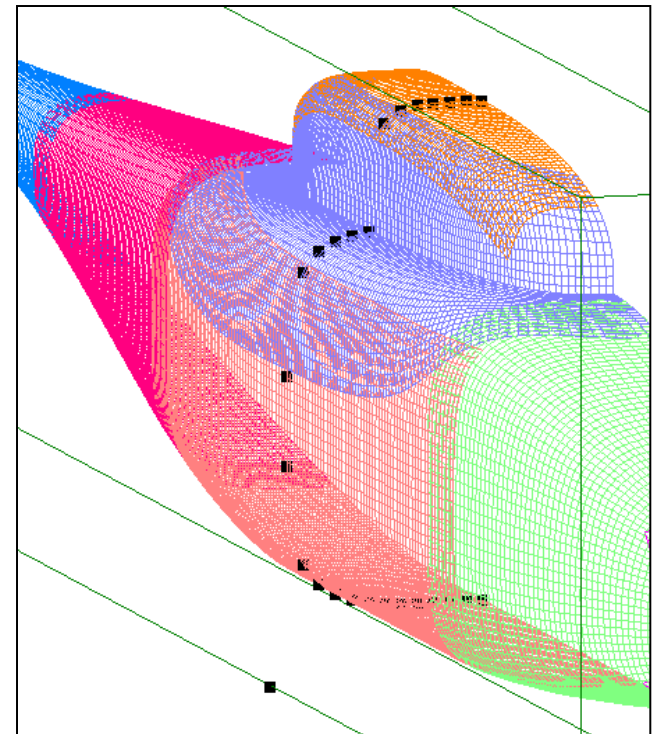
Picking X-Ray Spacing

- This is the "resolution" of the body surface for the hole-cutting operation
  - The X-rays need to represent the body geometry sufficiently well to cut holes in other grids
- For single-body applications, use ½ to 1 times the outer cell size of the near-body grids (**ΔS**)
  - Too-fine X-ray spacing slows down hole-cutting (very important for moving-body problems)
  - X-rays take memory in the flow solver (proportional to spacing squared)

- For bodies in close proximity, use 0.1 to 0.2 times the distance between bodies
  - Can use different x-rays (with different spacing) for different regions
- X-rays used for collision detection also need higher resolution
  - More on this later…

10/6/2014
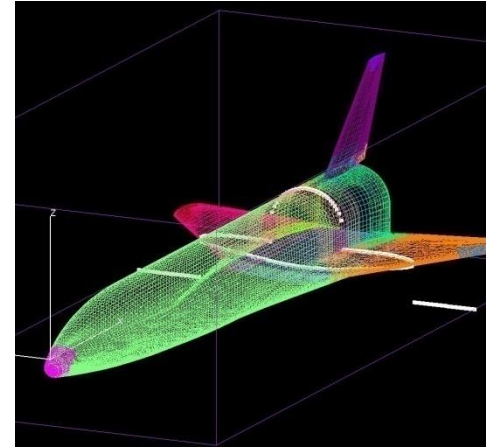
Using OVERGRID to Create X-Rays

- Start OVERGRID with the surface grid file
- Click "GEN_X" under "Domain Connectivity"

Using OVERGRID to Create X-Rays

- Enter X-ray spacing as "Image plane spacing"
  - Type <ENTER> to automatically adjust box boundaries

- Adjust box boundaries if needed
  - Ignore "add delta"
- Click "Make X-ray box"

- Click "GEN_X" to generate the X-rays

- Click "WRITE CURRENT" or "WRITE ALL" to save the X-rays to a file

CHIMERA GRID TOOLS GUI - OVERGRID

GEN_X - Domain Connectivity Pre-Processor

X-ray Management

| READ NEW | WRITE CURRENT | DELETE CURRENT |
| READ APPEND | WRITE ALL | DELETE ALL |

Total Number of X-rays   0

Current X-ray   0   Comp ID   1   Original

X-ray Box Boundaries

	min	max	add delta
X	-0.01015039	2.010158	0.0
Y	-0.010158	0.13659	0.0

Make X-ray box

Spacing/distance Options

◆ Absolute
◇ Fraction of global bounding box diagonal
◇ Fraction of X-ray box diagonal

Image plane spacing   0.010158

X-ray Viewer

Constant Plane   Max Index   Current Index   Coordinate
◆ X ◇ Y   1   1

| GEN_X | UNDO | HELP | QUIT |

X-Ray Number and Body ID

- X-rays are numbered sequentially and will be referred to by number in the OVERFLOW input

- Each X-ray is tied to a body, identified by "Comp(onent) ID" number (so when the body moves, the hole-cutting moves with it)

  – Body ID (Component ID) can be set here
  – Body ID=**n** refers to the **n**th component defined in the **Config.xml** file (discussed later)

- A text-input utility **xrayed** (part of CGT) allows manipulation of X-ray files

  – Combining X-ray files
  – Splitting files
  – Duplicating X-rays
  – Changing body IDs

10/6/2014
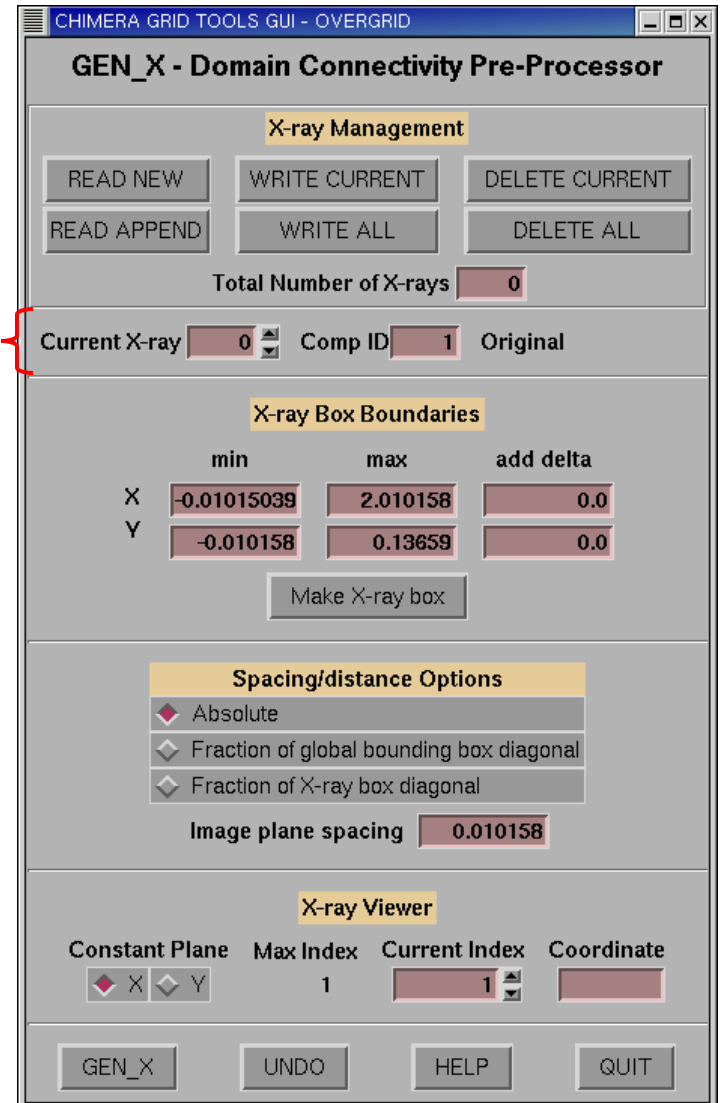
Using **gen_x** to Create X-rays

- **gen_x** is a text-input utility in Chimera Grid Tools (CGT)
- Documentation is included with CGT (excerpted here)
  - Input files:
    - PLOT3D surface grid file
    - Input parameter file:

```
surface_grid_filename
1                  ISOPT(1/2/3)
0.01               DS
0                  DELTA
1                  NCROPS
1                  IDBODY
-0.1, 1.1, 0, 1.5, -1, 1          XMIN,XMAX,YMIN,YMAX,ZMIN,ZMAX
```

(Ignore DELTA parameter)

  - Output files:
    - X-ray file **gen_x.xry**
    - Output messages
  - Execution:
    - gen_x < input_parameter_file > output_messages_file
- X-rays can be read into OVERGRID for viewing

Example: Axisymmetric External Tank

- For 2D or axisymmetric geometries, X-rays only need to bound the center (y=0) grid plane
  - Create the surface grid to represent the geometry within ± the X-ray spacing of y=0
  - Set the X-ray bounding box y limits to ± the X-ray spacing
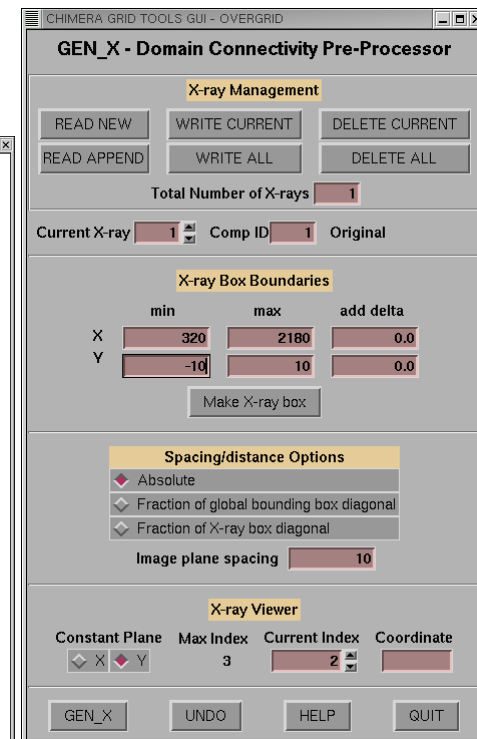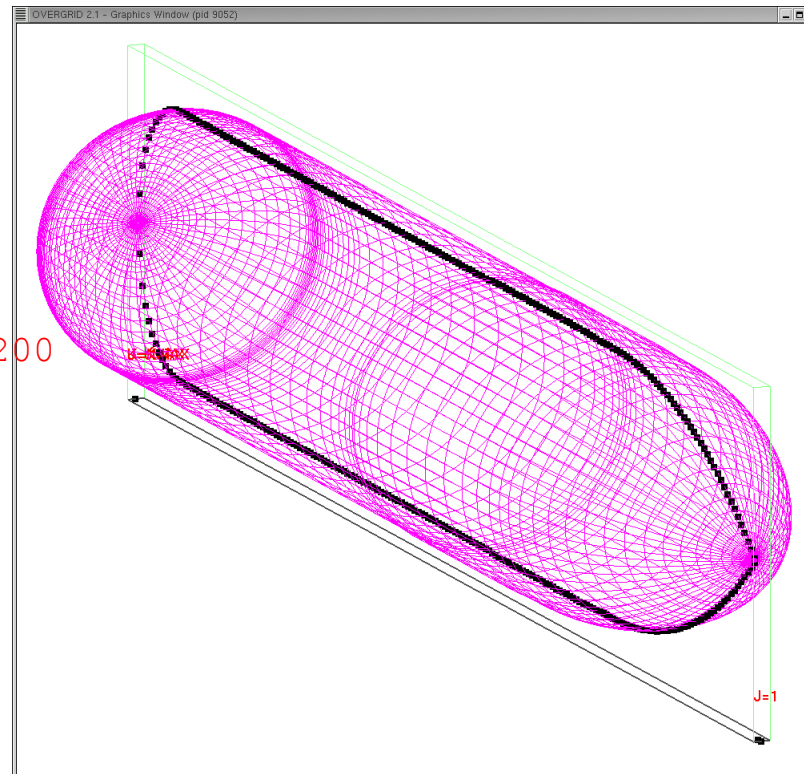 - Comparable **gen_x** input:

```
et.srf
1   ISOPT(1/2/3)
10  DS
0   DELTA
1   NCROPS
1   IDBODY
320,2180,-10,10,-200,200
```
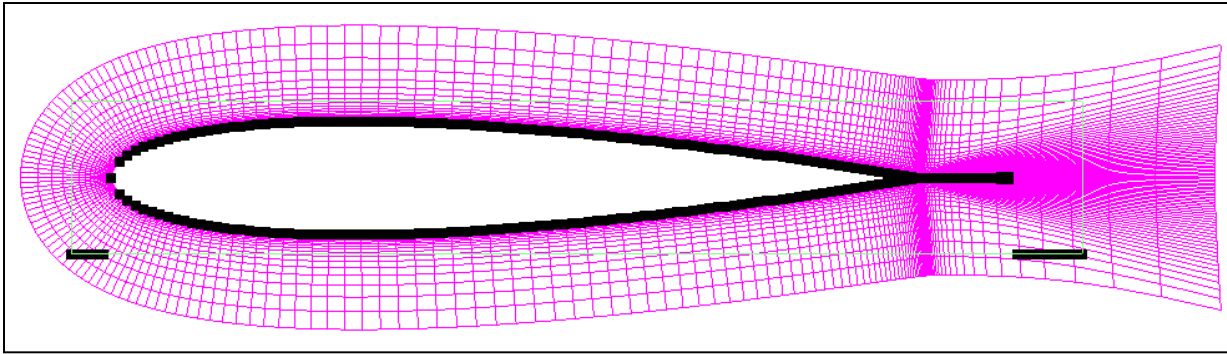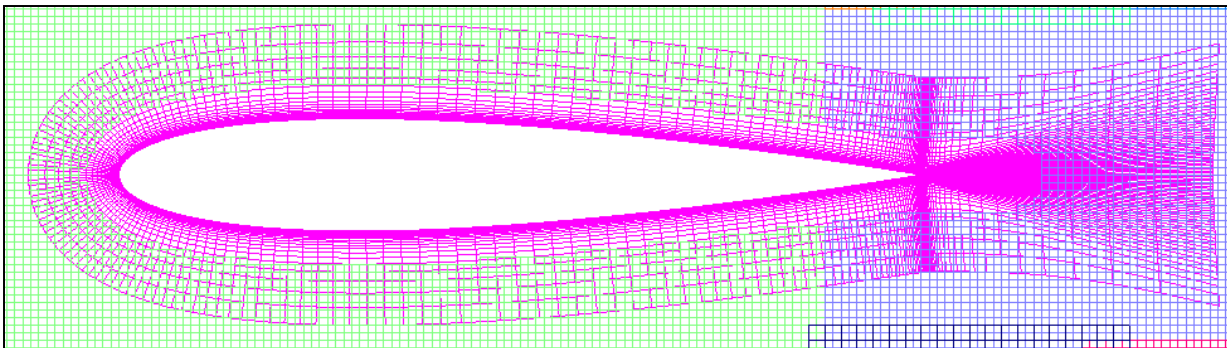
Example: 2D Airfoil

- For airfoils and wings, include a thin section of the C-grid wake with the surface grid
  - Use L=2 (or K=2) surface for finite thickness wake
 - Allows X-ray to cut other grids out of refined wake region

Airfoil grid and X-ray, showing extension into wake

Resulting hole in off-body grids

Notes and Comments

- "Duplicated" X-rays are useful in some cases
  - For example when multiple bodies are different only in position
  - Special format in X-ray file does not take additional space
 - X-rays can be duplicated using **xrayed** utility
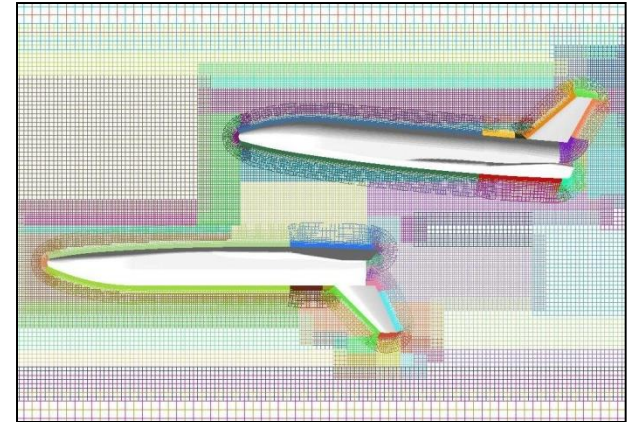
- Remember that when creating X-rays, surface grids for different bodies have to be in different files. Resulting X-ray files then have to be merged (again, using **xrayed**).

10/6/2014

Notes and Comments

- If a user-generated box grid is added, an X-ray must be generated to cut off-body grids from the inside of the box
  - A surface grid file must be created using interior surfaces of the box grid, for example constant planes of J,K,L=8 and -8

Sample capsule plus wake box, with X-rays for the wake box. Capsule will cut a hole in the wake box; wake box will cut a hole in off-body grids.
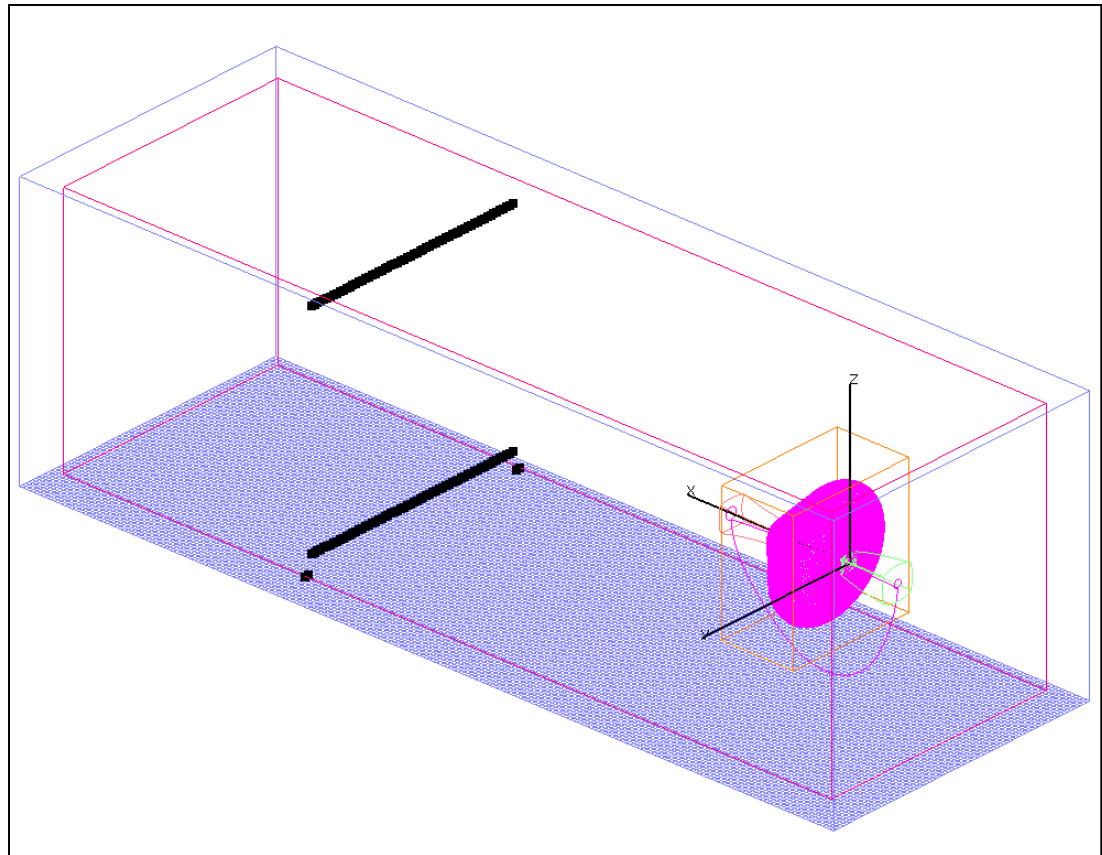
Automatic Off-Body Grid Generation

- Function of off-body grids

- Basic controls

- Matching near-body and off-body grid spacing

- Specifying additional refined regions

- Controlling the rate of grid coarsening

- Specifying symmetry planes, ground planes, etc.

- Far-field boundary conditions

- Examples

- Notes and comments

10/6/2014

Function of Off-Body Grids

- Level-1 (finest) off-body grids:
  - Surround (all) near-body grids
  - Fill user-specified regions
  - Solution adaption (if used)
- Level-2 and coarser grids fill in to the far-field boundary

10/6/2014
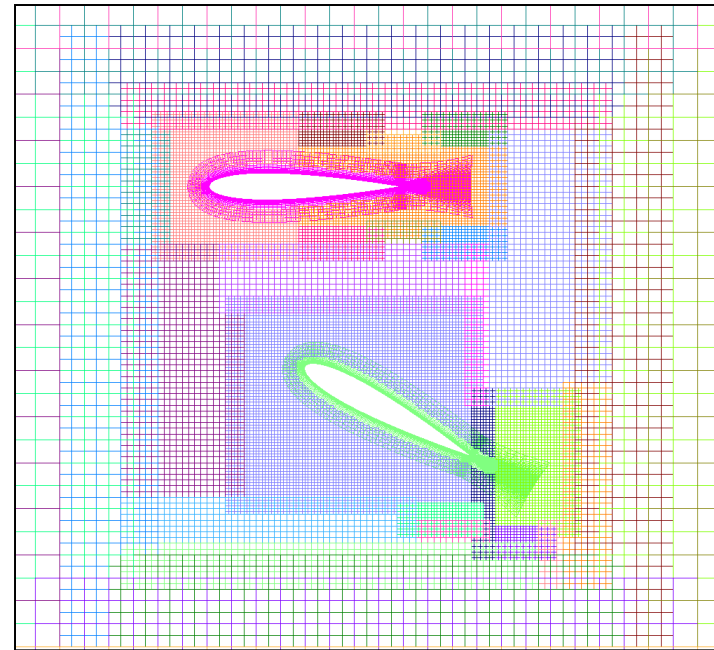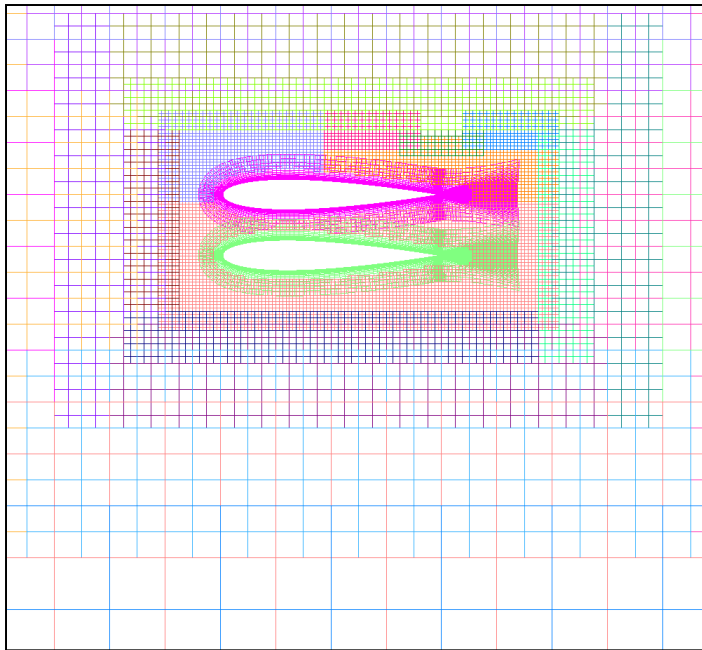
Basic Controls

- Basic controls (input parameters in **$GBRICK**):
  - **DS** – spacing for level-1 (finest) off-body grids
    - This parameter is critical for (a) proper communication with near-body grids, (b) resolving off-body flow gradients, and (c) controlling overall number of grid points.
  - **DFAR** – distance to (all) outer boundaries
  - **CHRLEN** – characteristic body length (*no longer used*)
    - Default is 1, use (major) dimension of body
  - **XNCEN,YNCEN,ZNCEN** – center of off-body grid system
    - Default is center of near-body grids
    - Must be specified for moving body problems

10
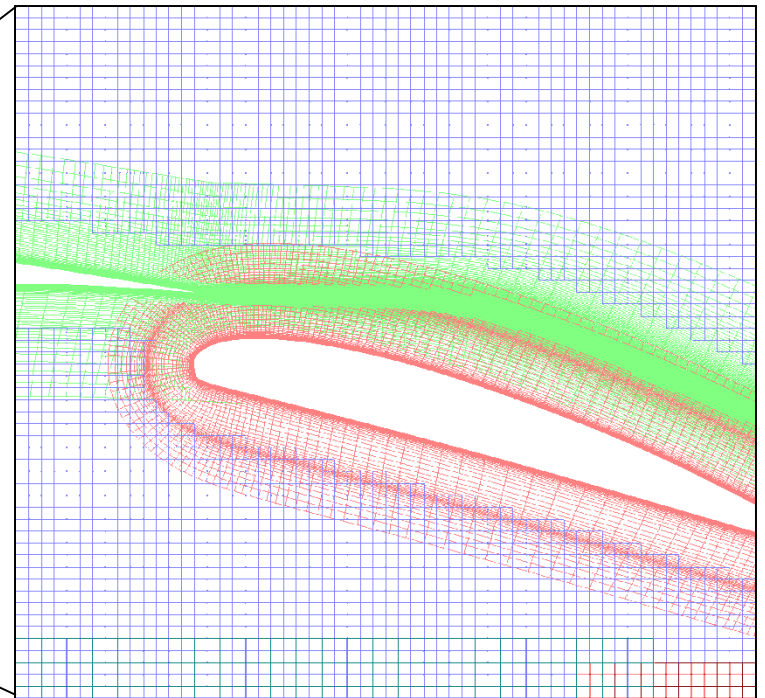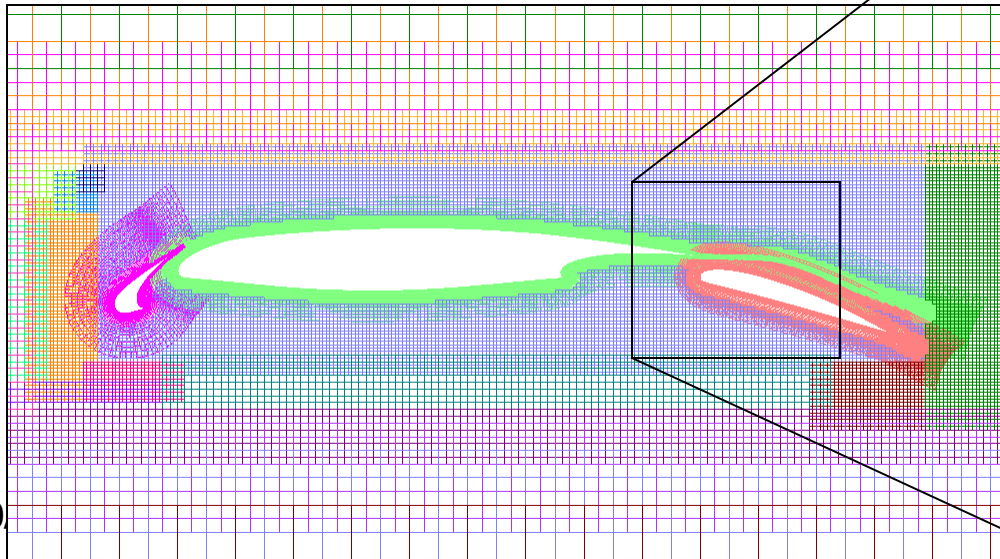
Matching Near-Body and Off-Body Grid Spacing

- How to pick **DS** (or, how far to grow near-body grids)?
  - **DS** should match **ΔS** (outer boundary spacing of near-body grids)
  - **DS** (and **ΔS**) should be sized to resolve off-body flow gradients
  - Near-body grids should extend out about **10xDS** from the body surface
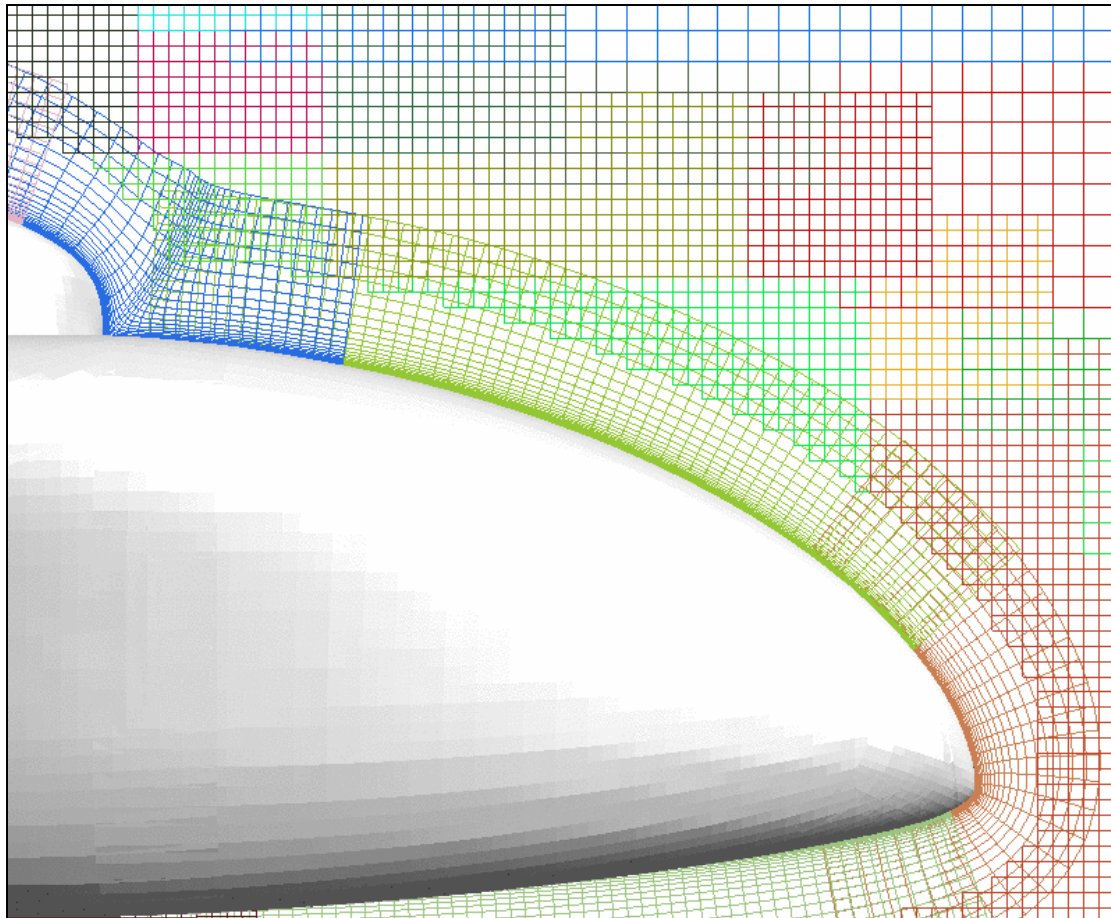
Off-Body Process Output

- Output indicates number of off-body grids generated for each level of coarseness (level-1 is finest)

```
................. START BRICK ....................

   Off-body grids generated with    2 fringe points.


   LEVEL  1:   GENERATED     35 GRIDS
   LEVEL  2:   GENERATED     44 GRIDS
   LEVEL  3:   GENERATED      5 GRIDS
   LEVEL  4:   GENERATED      5 GRIDS
   LEVEL  5:   GENERATED      5 GRIDS
   LEVEL  6:   GENERATED      5 GRIDS
   LEVEL  7:   GENERATED      5 GRIDS


.................. END BRICK ....................
```

10/6/2014

Specifying Additional Refined Regions

- Additional regions of level-1 grids (input parameters in **$BRKINP**):
  - **NBRICK** – number of user-specified proximity regions
    - If **NBRICK** is positive, these regions add to geometry regions
    - If **NBRICK** is negative, these regions *replace* geometry regions
  - **(XBRKMIN,XBRKMAX, YBRKMIN,YBRKMAX, ZBRKMIN,ZBRKMAX)** – min/max of proximity region
  - **IBDYTAG** – proximity region will move with motion of the indicated body
    - Example where you don't want this, e.g., wakes
    - Use **IBDYTAG=0** for no body movement
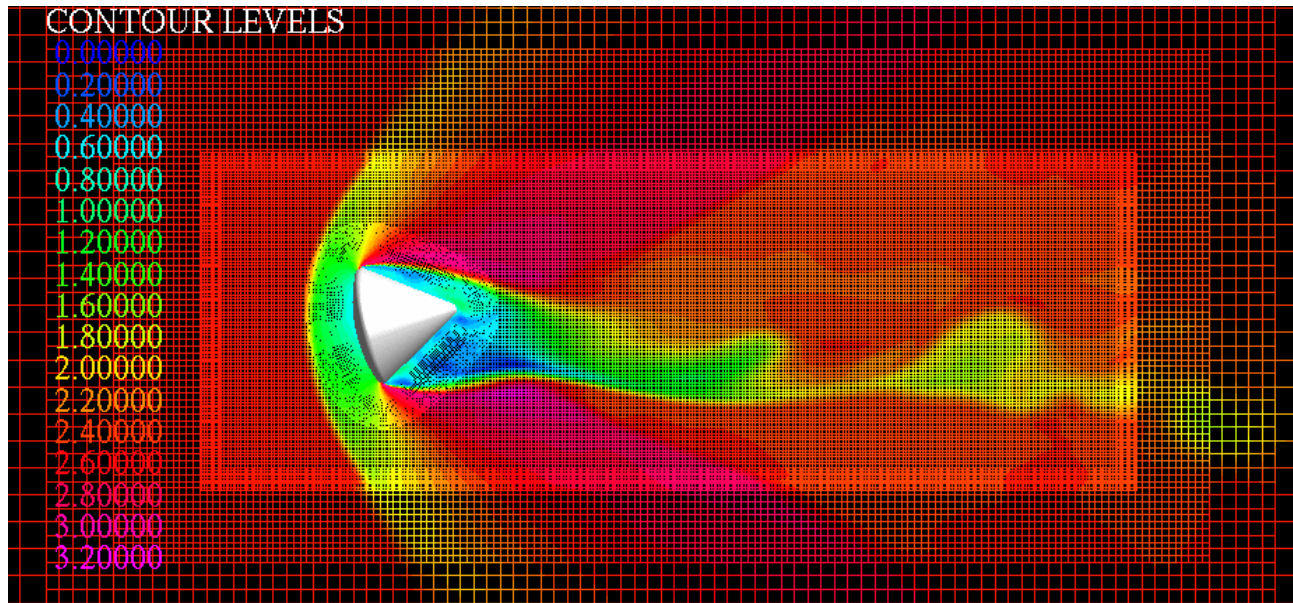
Controlling the Rate of Grid Coarsening

- Effect of **MINBUF** (in **$GBRICK**):
    - Default **MINBUF=4** gives minimum overlap between successively coarser off-body grids
    - Larger values give more gradual coarsening, but use more grid points

    - 2-airfoil example:
        - **MINBUF=4** (if geometry were 3D, off-body grids would have 2 million points)
        - **MINBUF=8** (3D off-body grids would have 3 million points)

MINBUF=4

MINBUF=8

10/6/2014
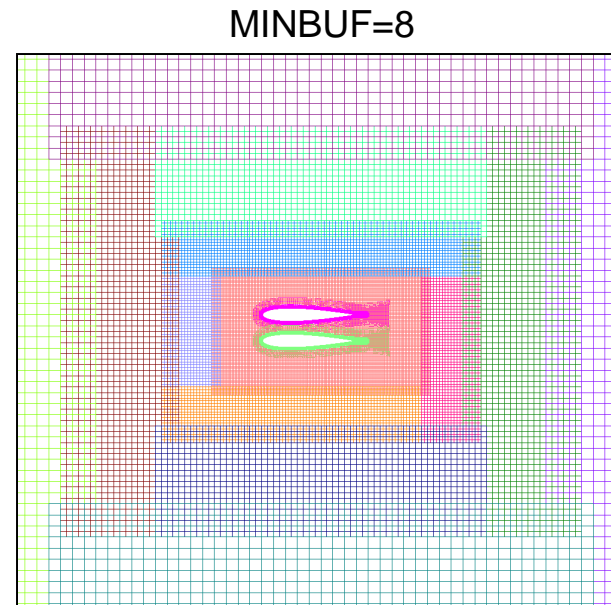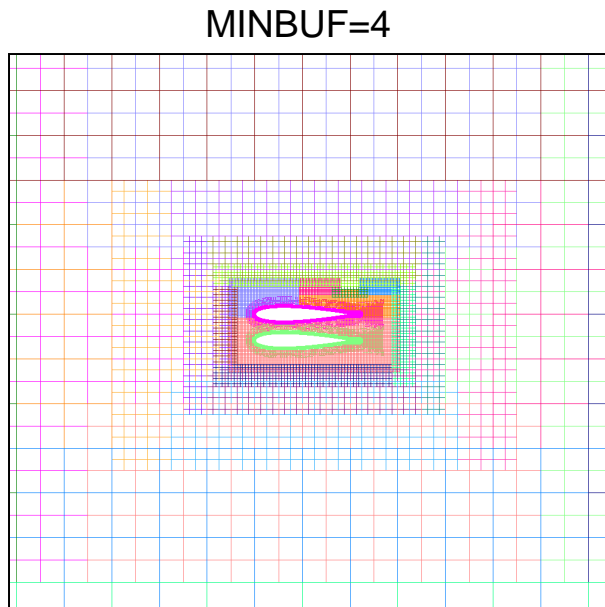
Specifying Symmetry Planes, Ground Planes, etc.

- Special planes (input parameters in **$GBRICK**):
  - Used to set a ground plane, symmetry plane, inflow plane, etc.
  - **I_XMIN=1** – use value of **P_XMIN** as off-body grid X(minimum)
  - **I_XMIN=0** – default is to use **DFAR** to set X (minimum)
  - Same for **I_XMAX, I_YMIN, I_YMAX, I_ZMIN, I_ZMAX,** and **P_XMAX, P_YMIN, P_YMAX, P_ZMIN, P_ZMAX**
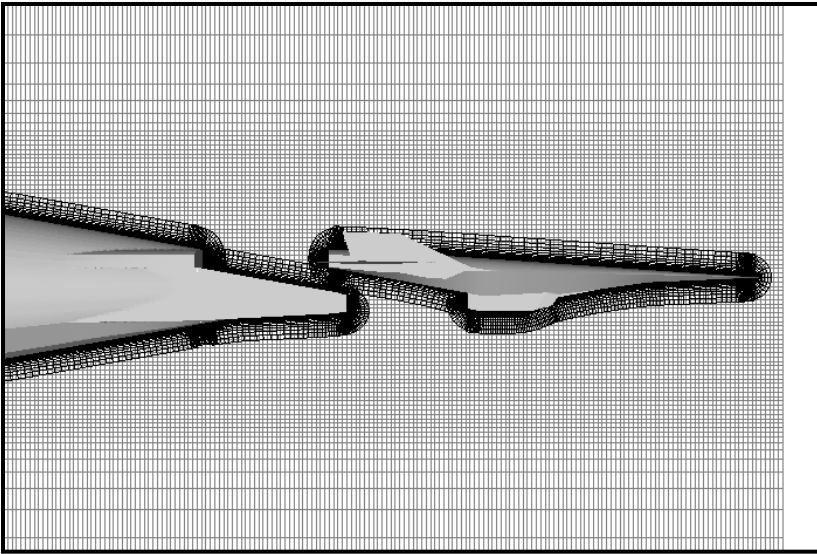
Hyper-X supersonic inflow plane:
I_XMIN=1, P_XMIN=-20
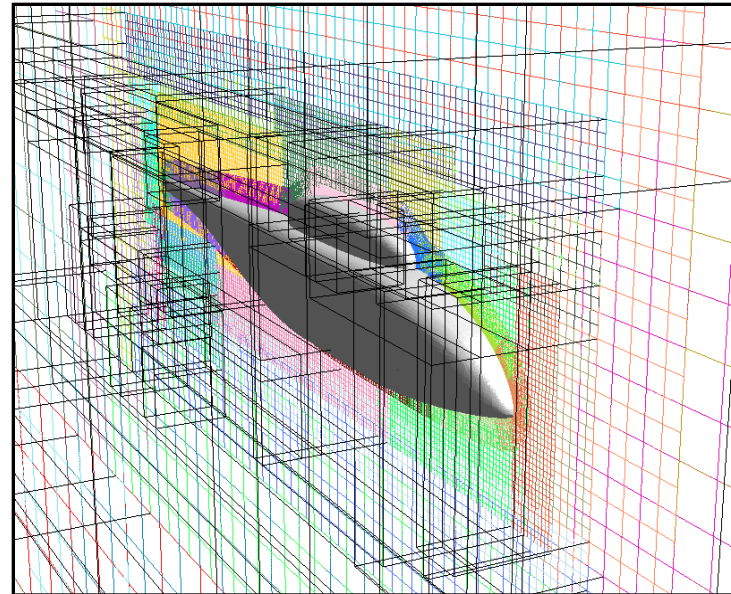
Symmetry plane for helicopter fuselage:
I_YMIN=1, P_YMIN=0

Far-Field Boundary Conditions

- Far-field boundary conditions (input parameters in **$OMIGLB**):
  - **IBXMIN** – boundary condition type for off-body grid system X (minimum) boundary
  - Same for **IBXMAX, IBYMIN,IBYMAX, IBZMIN,IBZMAX**
  - A limited number of boundary conditions are implemented:
    - Inflow/outflow conditions: BC types 30,35,37,40,41,47,49
    - 2D or axisymmetric condition (y only): BC types 21,22
    - Axis condition (z only, and combined with axisymmetric in y): BC type 16
    - Symmetry conditions: BC types 11,12,13,17
    - Inviscid wall: BC type 1
  - Default is free-stream/characteristic condition (BC type 47)

10/6/2014

Example 1: Helicopter Fuselage

- 100-inch long body, symmetry at y=0, far-field at 1000 inches
  - Set off-body symmetry boundary condition with **IBYMIN=17**
  - Make off-body grids start at y=0 by using **I_YMIN, P_YMIN**

  $OMIGLB IBYMIN=17, … $END

  $GBRICK

      DS=1, DFAR=1000,

      XNCEN=50, YNCEN=0, ZNCEN=0,

      I_YMIN=1, P_YMIN=0,

      $END

  $BRKINP $END
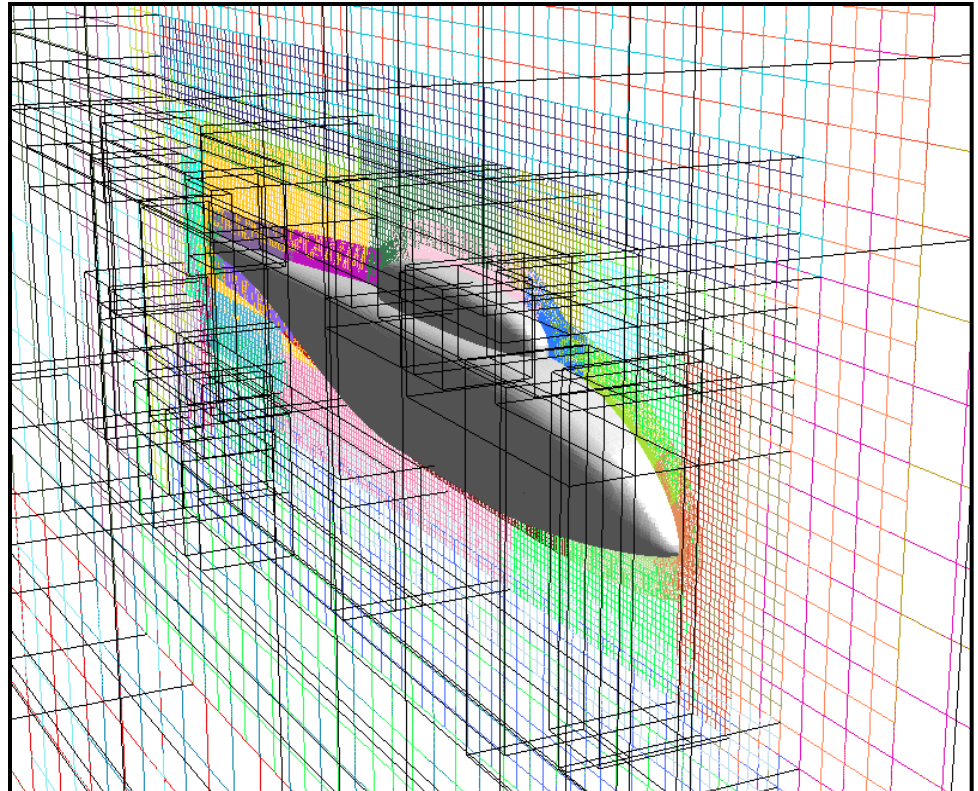
Example 2: Airfoil With Refined Shock Grid

- 2D airfoil, chord=1, far-field at 100 chords
  - Use **$BRKINP** to add a refined level-1 region for shock
  - Since **IBDYTAG=1**, this region is tied to motion of the airfoil

$OMIGLB IBYMIN=21, … $END

$GBRICK
    DS=0.01, DFAR=100,
    XNCEN=0.5, YNCEN=0, ZNCEN=0,
    $END

$BRKINP
    NBRICK=1,
    XBRKMIN=0.5, XBRKMAX=0.9,
    YBRKMIN=0, YBRKMAX=0,
    ZBRKMIN=0, ZBRKMAX=1,
    IBDYTAG=1,
    $END
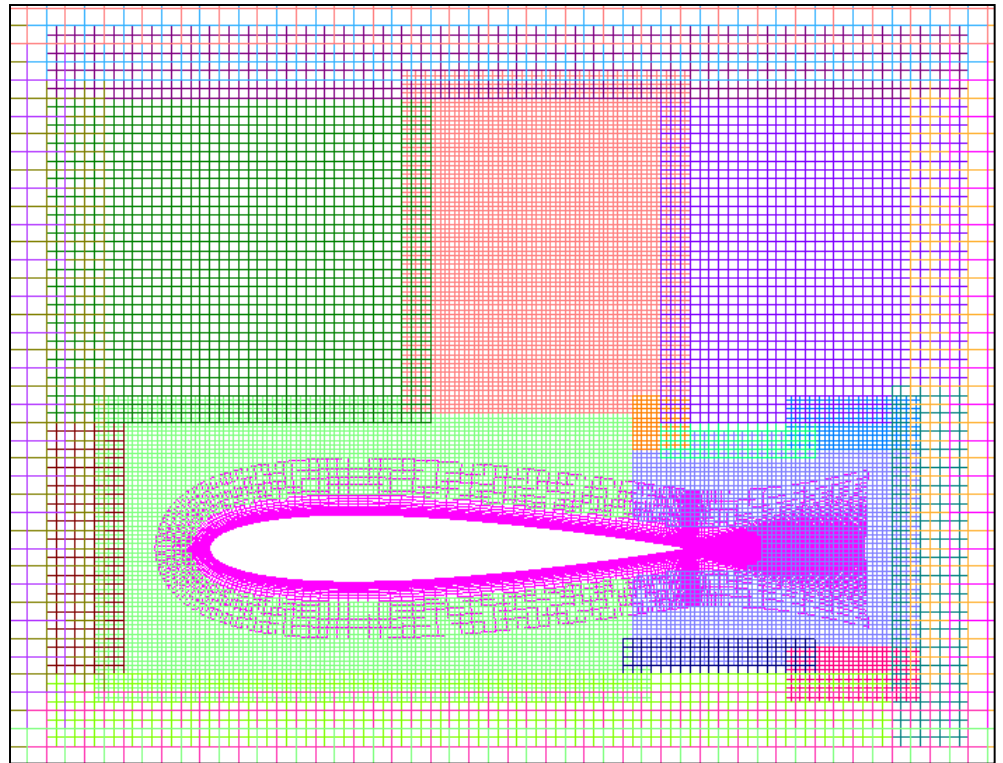
Example 3: Oscillating Airfoil

- Airfoil forced oscillation problem
  - Use **$BRKINP** to make level-1 region big enough to capture expected body motion, so that off-body grids will not need to be regenerated during moving-body run

```
$OMIGLB  IBYMIN=21, …  $END
$GBRICK
    DS=0.01, DFAR=100,
    XNCEN=0.5, YNCEN=0, ZNCEN=0,
    $END
$BRKINP
    NBRICK= -1,
    XBRKMIN= -0.3, XBRKMAX= 1.5,
    YBRKMIN= 0, YBRKMAX= 0,
    ZBRKMIN= -0.8, ZBRKMAX= 0.8,
    IBDYTAG= 0,
    $END
```


Without **$BRKINP**

With **$BRKINP**

Example 3: Oscillating Airfoil

- Airfoil forced oscillation problem
  - Use **$BRKINP** to make level-1 region big enough to capture expected body motion, so that off-body grids will not need to be regenerated during moving-body run
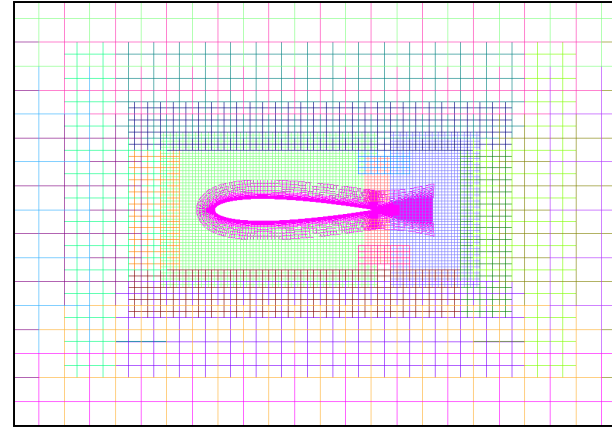
```
$OMIGLB  IBYMIN=21, …  $END
$GBRICK
    DS=0.01, DFAR=100,
    XNCEN=0.5, YNCEN=0, ZNCEN=0,
    $END
$BRKINP
    NBRICK= -1,
    XBRKMIN= -0.3, XBRKMAX= 1.5,
    YBRKMIN= 0, YBRKMAX= 0,
    ZBRKMIN= -0.8, ZBRKMAX= 0.8,
    IBDYTAG= 0,
    $END
```
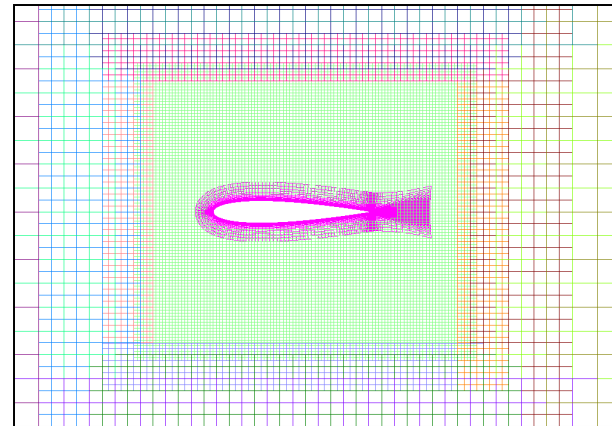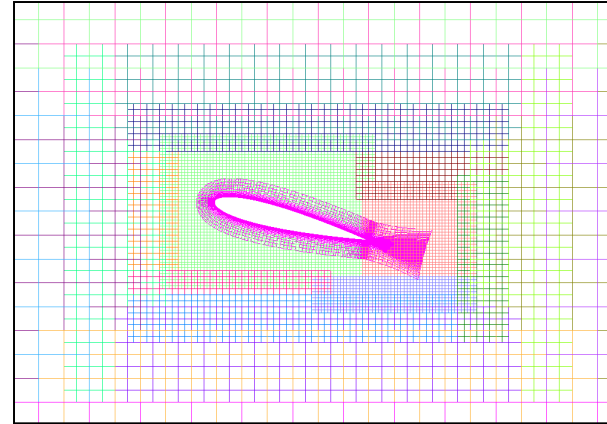

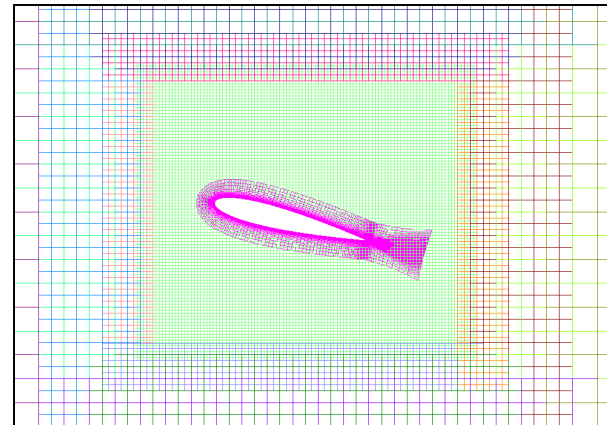Without **$BRKINP**

With **$BRKINP**

Notes and Comments

- **OBGRIDS=FALSE** – no off-body grids created
  - For single-grid problems, or where background grids are already supplied
- Files created: **brkset.save, brkset.restart**
  - Needed for moving body or solution adaption restarts to generate consistent off-body grids and near-body refinement regions
- Residual history for off-body grids is combined into a single entry in **resid.out**, **turb.out**, **species.out**
  - Instead of one entry per grid level, there is one entry representing *all* levels of off-body grids
  - Entry contains L_2 and L_∞-norms of RHS and ΔQ
  - Entry lists (x,y,z) instead of (j,k,l) location of L_∞-norm (off-body grids only)
  - Especially appropriate for moving body or solution adaption problems, where the number of off-body grids changes every adapt cycle

DCF: Hole Cutting and Grid Assembly

- Using X-rays to cut holes
- Choosing **XDELTA**
- Orphan points and donor quality
- Multiple fringe interpolation
- Viscous stencil repair
- Examples
- Notes and comments

10/6/2014

Using X-Rays to Cut Holes

- Specifying X-ray cutters (input parameters in multiple **$XRINFO**):
  - **IDXRAY** – X-ray number
  - **IGXLIST** – list of grids to be cut
    - Special grid number "-1" refers to (all) off-body grids
  - Or use **IGXBEG,IGXEND** – starting/ending grids to be cut
  - **XDELTA** – offset of hole from body surface
- Example: $XRINFO IDXRAY=1, IGXLIST=-1, XDELTA=0.05 $END
  - Use the first X-ray in **xrays.in** file, cut a hole in the off-body grids, 0.05 grid units off the X-ray surface:
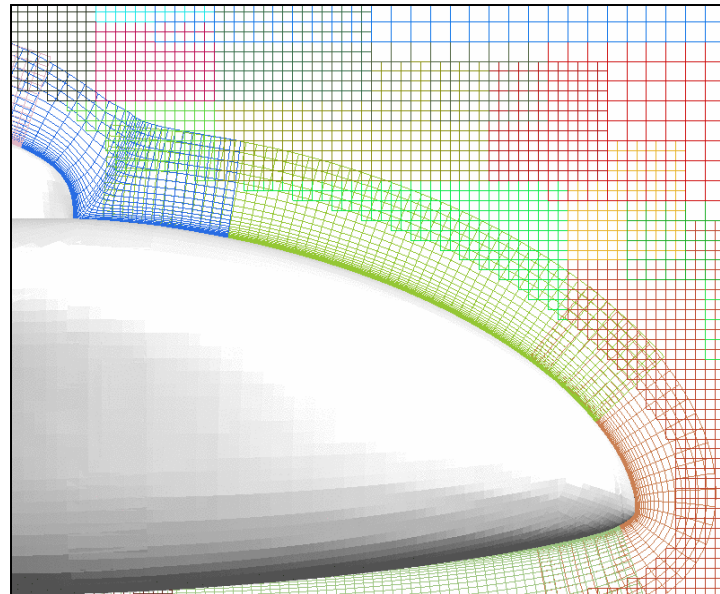
Using X-Rays to Cut Holes

- Example: multi-element airfoil

  $XRINFO IDXRAY=1, IGXLIST=-1, XDELTA=0.02 $END

  $XRINFO IDXRAY=2, IGXLIST=-1, XDELTA=0.02 $END

  $XRINFO IDXRAY=3, IGXLIST=-1, XDELTA=0.02 $END

  – Slat, main, and flap X-rays (X-rays 1,2,3) cut holes in off-body grids

  $XRINFO IDXRAY=1, IGXLIST=2, XDELTA=0.005 $END

  – Slat X-ray cuts hole in main grid (grid 2)

  $XRINFO IDXRAY=2, IGXLIST=1,3, XDELTA=0.005 $END

  – Main X-ray cuts hole in slat and flap grids (grids 1,3)

  $XRINFO IDXRAY=3, IGXLIST=2, XDELTA=0.005 $END

  – Flap X-ray cuts hole in main grid (grid 2)
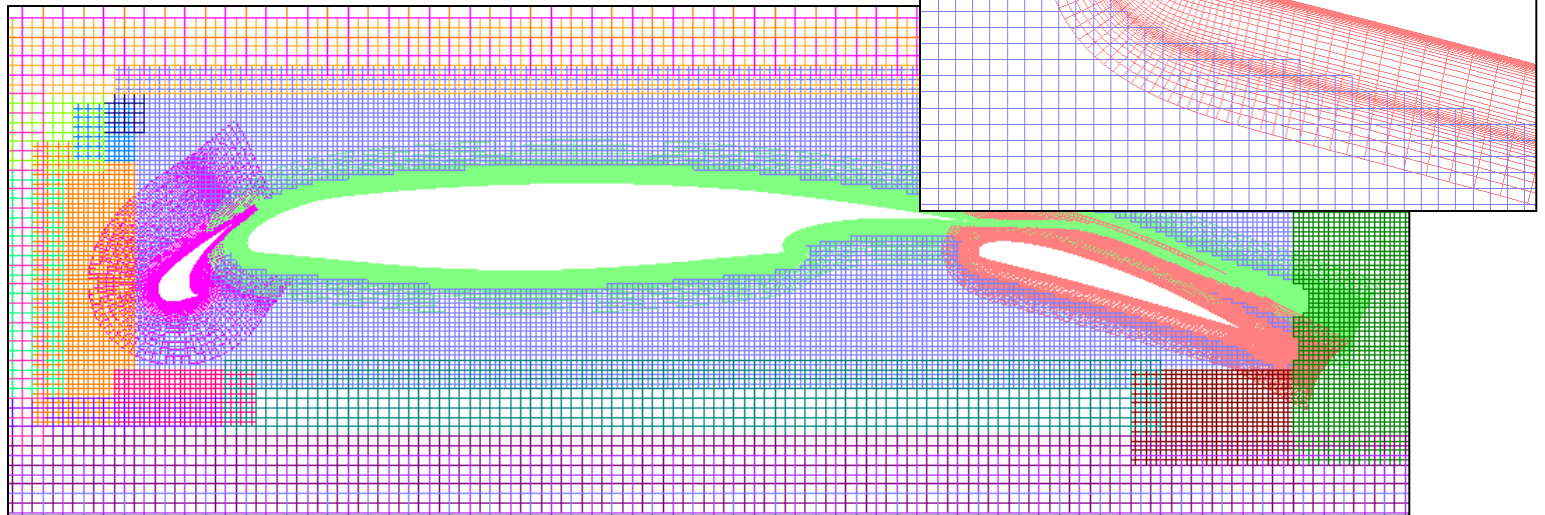
Choosing **XDELTA**

- Holes should be cut to keep coarser grids out of high-gradient regions (such as boundary layers)

- Holes should be cut so that grids have similar resolution in overlap regions, and have sufficient overlap for interpolation of boundary data

- When cutting holes in off-body grids, choose **XDELTA** to be 5 times **DS**, *in from the outer boundary* of the near-body grids, or **XDELTA = S – 5xDS**
  - This is often about half the distance to the surface
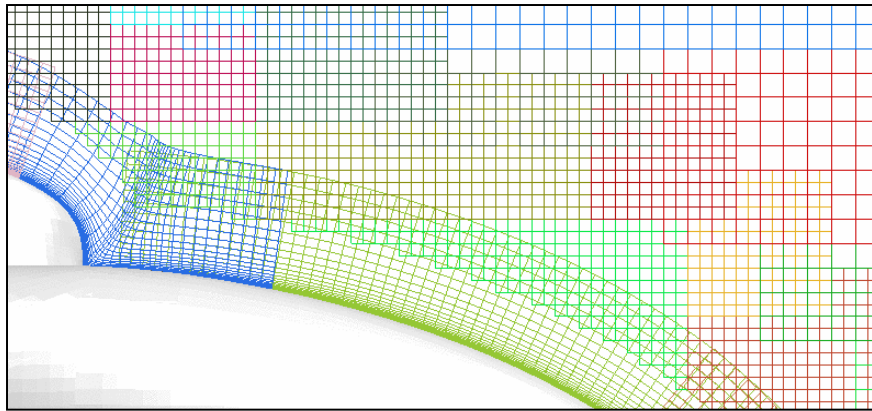
- When cutting holes in nearby bodies, **XDELTA** must be less than half the expected minimum distance between the bodies to avoid orphan points
  - Can use different values for different cutters

Orphan Points and Donor Quality

- Some overset grid definitions (thanks to Ralph Noack):
  - **Blanked-out points** – points inside bodies or holes, where the solution is not computed or is ignored
  - **Fringe points** – inter-grid boundary points where solution values are obtained via interpolation from another grid
  - **Donor points** – points contributing to interpolation stencils
  - **Orphan points** – fringe points without valid donors; resulting from hole cutting failure (no possible donor) or only poor quality donors are available (insufficient overlap)
- Donor stencil quality (input parameter in **$DCFGLB**):
  - "Quality" of the donor stencil refers to how much of the interpolated information has to come from donor points that are interior to the flow solution, i.e., not fringe points themselves
  - **DQUAL=1** – donor stencils must consist of only field points (default)
  - **DQUAL=0** – stencils which include all fringe points may be accepted
    - This is *not* a good idea—the simulation may simply pass boundary data back and forth between grids
  - **DQUAL=0.1** is generally acceptable

Multiple Fringe Interpolation

- Single/double/triple fringe refers to the number of layers of interpolated points at overlapped grid boundaries. These are analogous to ghost points.
    - For single fringe interpolation, the flow solver must revert to lower-order differencing at the boundaries
    - Double fringe interpolation supports 5-point stencil differencing (up to **FSO=3**)
    - Triple fringe interpolation supports 7-point stencil differencing (up to **FSO=5**)
- **Appropriate number of fringe points is *always* highly recommended,** even though the code will run fewer fringe points
- Input parameter in **$OMIGLB**:
    - **LFRINGE=2** – double fringe; any double fringe points that are orphan points are "repaired," i.e., changed back to field points (default)
    - **LFRINGE=-2** – double fringe; do not revert orphan points to field points
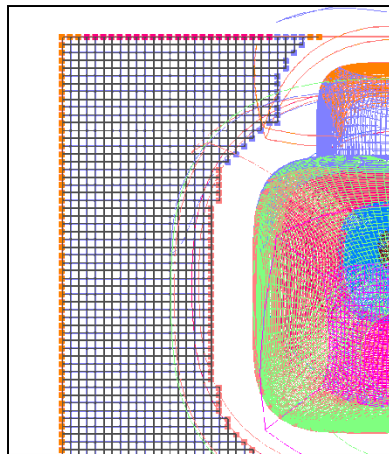
Single fringe   Double fringe
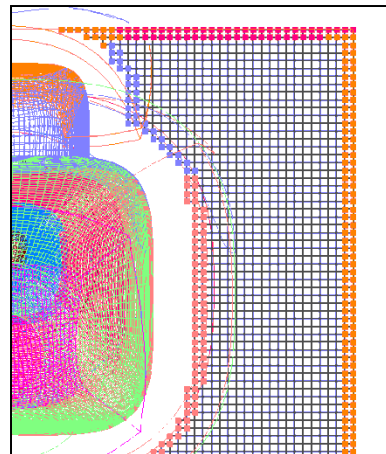
Triple (and Higher) Fringe Interpolation

- Higher-order schemes in OVERFLOW need more than double fringe
  - 4th-order central, 5th-order WENO schemes need **LFRINGE=3**
  - 6th-order central needs **LFRINGE=4**
  - Default **LFRINGE** is based on numerical scheme
  - **LFRINGE** can be changed whenever grid connectivity is recomputed (DCF process)
- Off-body grids need more overlap as well
  - Use **OFRINGE** in **$GBRICK** to specify number of fringe points for off-body grids
  - Default **OFRINGE** is based on numerical scheme
  - But, **OFRINGE** cannot be changed without regenerating off-body grids
  - If you plan to use a higher-order scheme later, set **OFRINGE** now
- Orphan points cause fringe layers to degrade gradually
  - Triple fringe will locally change to double fringe, then to single fringe until orphans are resolved or converted to field points

Viscous Stencil Repair

- Viscous stencil repair (input parameters in **$DCFGLB**):
  - **MORFAN** – enable/disable viscous stencil repair (1/0)
  - **NORFAN** – number of points above a viscous wall subject to viscous stencil repair
  - Viscous stencil repair is needed to handle bad interpolations when overlapping surface grids lie on the same curved surface. If not corrected, this can result in orphan points (convex surfaces) or interpolations too high in the boundary layer (concave surfaces).
  - WARNING: Interpolation stencils for boundary points within NORFAN points of a viscous surface will be modified, using the assumption that all viscous walls have the same grid distribution in the normal direction. QUALITY OF REPAIRED STENCILS IS NOT CHECKED.
  - A better scheme is needed!

DCF Output

- Output from DCF process indicates the number of multiple fringe and viscous stencils repaired, and the final number of orphans

```
...............      START DCFCRT      .................


  WARNING:  USING VISCOUS STENCIL REPAIR WITHIN     6 POINTS OF A WALL.
   Interpolation stencils for boundary points within NORFAN points of a
   viscous surface will modified, using the assumption that all viscous
   walls have the same grid distribution in the normal direction.
  WARNING:  QUALITY OF REPAIRED STENCILS IS NOT CHECKED.

  WARNING:       278 viscous stencils/orphans repaired in DCFCRT
  NO orphan points found in DCFCRT

  ORPHAN POINT SUMMARY:
    *Numbers are approximate due to grid splitting.
     Points in overlap region may be counted twice.

           Initial  Visc Stencils   Visc Orphans  Multi-Fringe      Final
     Grid  Orphans    Repaired         Repaired   Orphs Repaired   Orphans
     --------------------------------------------------------------------
       1*       0       214              0             0               0
       2*      36        28             36             0               0

...............      END DCFCRT      .................
```
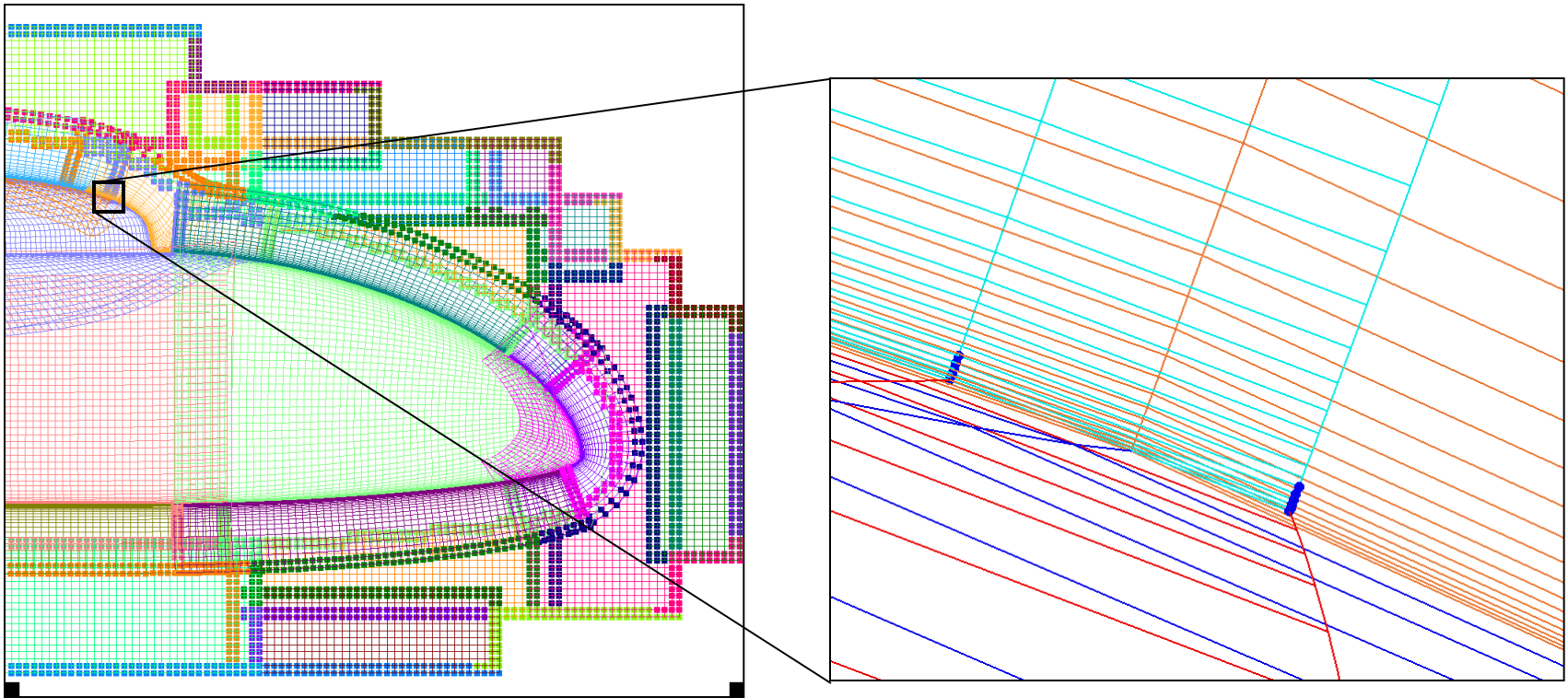
Example 1

- Helicopter fuselage

  $OMIGLB LFRINGE=2, … $END
  $DCFGLB DQUAL=0.3, MORFAN=1, NORFAN=6, $END
  $XRINFO IDXRAY=1, IGXLIST=-1, XDELTA=0.035, $END

10/6/2014

Example 2

- Airfoil drop
  - For bodies that are very close to each other, very small values of **XDELTA** may be needed

$OMIGLB LFRINGE=2, … $END **$DCFGLB DQUAL=0.3, $END** **$XRINFO IDXRAY=1, IGXLIST=2,-1, XDELTA=0.04, $END** **$XRINFO IDXRAY=2, IGXLIST=1,-1, XDELTA=0.04, $END**

$OMIGLB LFRINGE=2, … $END **$DCFGLB DQUAL=0.3, $END** **$XRINFO IDXRAY=1, IGXLIST=-1, XDELTA=0.04, $END** **$XRINFO IDXRAY=2, IGXLIST=-1, XDELTA=0.04, $END** **$XRINFO IDXRAY=1, IGXLIST=2, XDELTA=0.0, $END** **$XRINFO IDXRAY=2, IGXLIST=1, XDELTA=0.0, $END**
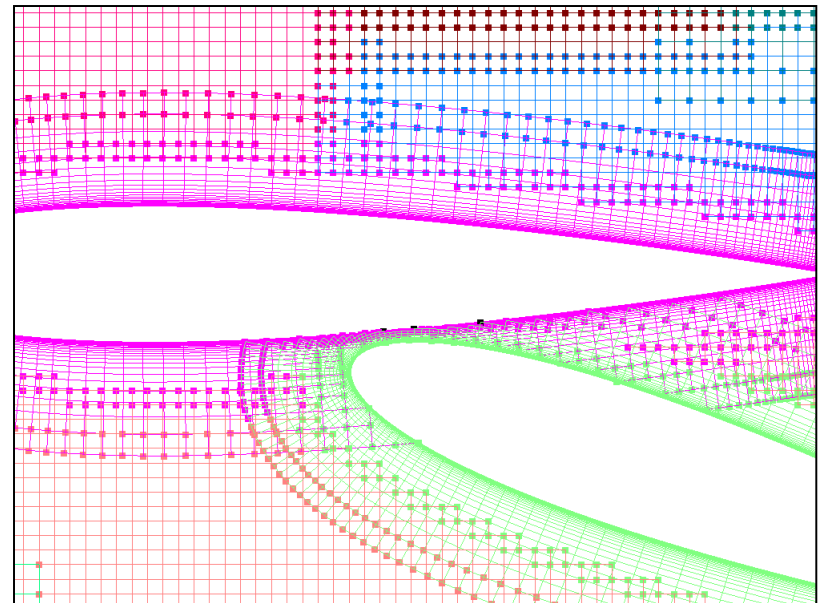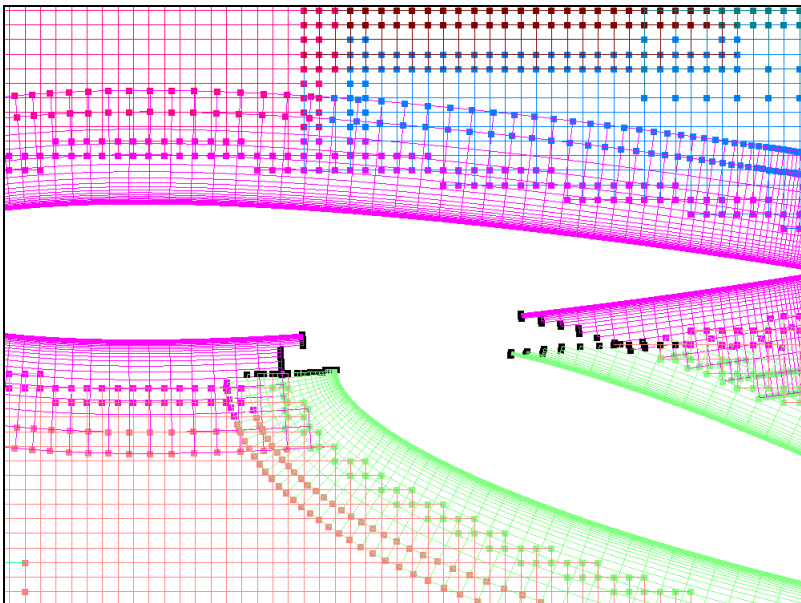
Notes and Comments

- It is OK to have "some" orphan points
  - But you should understand why, and where they are in the grid system
  - Be careful of compromising grid quality because you don't want to refine the off-body grids, or don't want to fix the near-body grids
- Orphan points become much harder to control in moving body problems
  - Have to anticipate grid movement
- OVERFLOW "fills" orphan points (and *all* hole points) with average of neighboring point values
- Input parameter **IRUN** in **$OMIGLB** allows test run of DCF:
  - **IRUN=1** – just do off-body grid generation (write **x.save** file)
  - **IRUN=2** – do off-body grid generation *and* DCF (write **x.save**)
  - **IRUN=0** – do a complete run, including flow solver

Data Surface Grids

- Can be used to extract acoustic data surfaces, velocity profiles, pressure tap locations, 2D slices, etc.

- Any "1D" or "2D" (mx1x1 or mxnx1) grid in the **grid.in** file will be treated by DCF as a "data surface grid"
  - Flow solution at all points will be interpolated from other grids
  - Grid and solution will be saved in usual files (**x.save**, **q.save**)
  - Can also write these out explicitly using **$SPLITM**

10/6/2014

OVERFLOW-D Mode With Grid Motion

- General moving body process
- Off-body grid adaption to geometry
- GMP files **Config.xml** and **Scenario.xml**
- Non-dimensionalization of dynamics quantities
- Time step specification
- Simulating collisions
- Output information for moving body problems
- Visualizing body motion in OVERGRID
- Some references

General Moving Body Process

- General process (input parameters in **$OMIGLB**):
  - **DYNMCS=.TRUE.** – enable body dynamics (default is FALSE)
  - **I6DOF=2** – Prescribed and/or 6-DOF motion for different components. Specified via the GMP interface (**Config.xml** and **Scenario.xml** files) (**$SIXINP** is ignored). **This is the recommended (and supported) option for moving body problems.**
    - **I6DOF=1** – 6-DOF body motion, specified via **$SIXINP** namelist input.
    - **I6DOF=0** – User-specified motion, controlled by user-supplied USER6 subroutine.
  - **NADAPT** – number of steps between adaption (regeneration) of the grid system
    - **NADAPT=-n** – (off-body) grids adapt to geometry only
    - **NADAPT=0** – grids will not be regenerated during solution process
    - **NADAPT=n** – grids adapt to geometry and flow solution (see next section)

General Moving Body Process

- DCF (hole-cutting and interpolation stencil-finding) is done every step
  - Want interpolation stencils to change less than one cell per step for time accuracy
  - Estimate maximum velocity of fringe points and compare to donor grid cell size
  - This sets maximum desired physical time step
- Look at a simple example:
  - We have level-1 boxes
  - We have a near-body grid inside the boxes
  - The body is moving, the boxes are not
- What happens in OVERFLOW?
  - Body motion is computed
  - Body is moved
  - DCF is performed
  - Flow solution is advanced
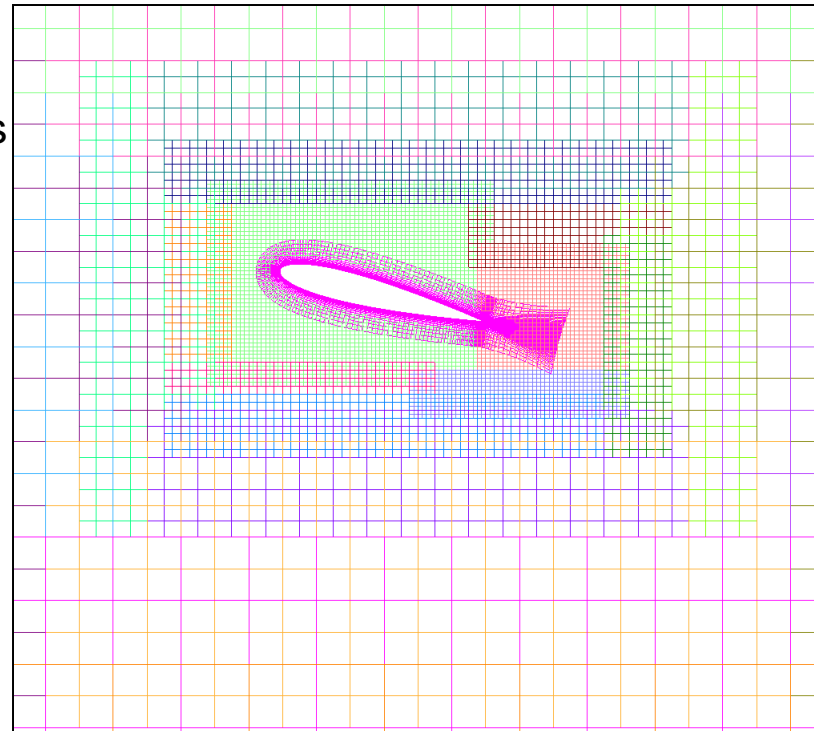
Off-Body Grid Adaption to Geometry

- As body moves, near-body grid gets close to the edge of the level-1 boxes
- Off-body grids must be regenerated, and the flow solution transferred (interpolated) to the new off-body grids
- **NADAPT=-n** gives the number of time steps between off-body grid adaption
  - Usually every 20-50 steps (based on time step **DTPHYS** and **MINBUF**)
    - Check this by running sample cases
  - In some cases we can avoid this by creating a larger level-1 grid (e.g., for pitching airfoil problem)

10/6/2014
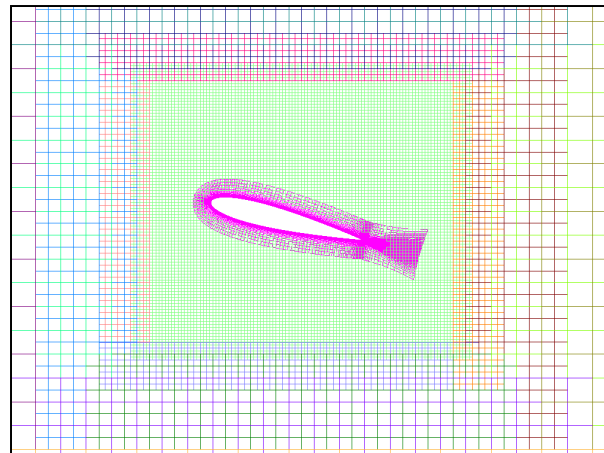
Geometry Manipulation Protocol Files: Config.xml

- Defines body (component) names and the associated grids
- Specifies any initial body transforms to assemble components into their starting positions
- Components and their transforms can be defined hierarchically
- Example:

```
<?xml version='1.0' encoding='utf-8'?>
<Configuration AngleUnit="degree">
    <Component Name="wing" Type="struc">
        <Data> Grid List=1-5 </Data>
    </Component>
    <Component Name="aileron" Parent="wing" Type="struc">
        <Data> Grid List=6,7 </Data>
        <Transform>
            <Rotate Center="0.7,0,0" Axis="0,-1,0" Angle="10"/>
        </Transform>
    </Component>
</Configuration>
```

GMP Files: **Scenario.xml**

- Prescribed motion: specify
  - Start time and duration
  - Translation and rotation rates
- Example (prescribed motion):

```
<?xml version='1.0' encoding='utf-8'?>
<Scenario Name="Forced Oscillation" AngleUnit="degree">
   <Prescribed Component="aileron" Start="0", Duration="0">
      <Rotate Center="0.7,0,0" Axis="0,-1,0"
       Speed="20.*2.*pi/100.*cos(2.*pi/100.*t+pi/2.)"/>
   </Prescribed>
</Scenario>
```

GMP Files: **Scenario.xml**

- 6-DOF motion: specify
  - Start time and duration
  - Component inertial properties
  - Applied forces
  - Motion constraints
- Example (constrained 6-DOF motion):

```
<?xml version='1.0' encoding='utf-8'?>
<Scenario Name="Constrained Motion" AngleUnit="degree">
    <Aero6dof Component="aileron" Start="0", Duration="0">
        <InertialProperties Mass="1.0" CenterOfMass="0.7,0,0"
         PrincipalMomentsOfInertia="0,2,0"/>
        <Constraint Rotate="1,0,1" Frame="body" Start="0"/>
        <Constraint Translate="1,1,1" Frame="body" Start="0"/>
    </Aero6dof>
</Scenario>
```

GMP Files: **Config.xml** and **Scenario.xml**

- For 6-DOF problems, GMP component names *must* match component names in **mixsur.inp** (force and moment calculation)
- GMP files can be created in a text editor or using OVERGRID
  - Be careful that the motion illustrated in OVERGRID is the same as that in OVERFLOW (should be OK with CGT 2.1)
- Some GMP capabilities do not work in OVERFLOW:
  - Moments of inertia do not change with moving parts

Non-Dimensionalization of Dynamics Quantities

- This is critical!
- Non-dimensionalizations in the flow solver are easy (free-stream density $\rho^*_\infty=1$, free-stream speed-of-sound $c^*_\infty=1$)
- Non-dimensionalizations for all dynamics and time-accurate information are based on V_{ref} rather than c_∞
  - V_{ref} is defined as $V_{ref}=$**REFMACH**$^*c_\infty$
  - V_{ref} is the same as V_∞ if **REFMACH** is not explicitly specified in **$FLOINP**
    - **REFMACH** defaults to **FSMACH**
    - **REFMACH** may be different from **FSMACH**, for example for hover problems (**FSMACH=0**)
  - This includes **DTPHYS**; all quantities in GMP files; and output forces and moments, velocities and angular rates
  - Some quantities can be very large (or small)

Non-Dimensionalization of Dynamics Quantities

- Non-dimensionalizations of dynamic quanities are thus based on
  - Length: L=1 grid unit
  - Time: L/V_{ref}
  - Mass: $\rho_\infty L^3$
- Indicating non-dimensional quantities with a *:
  - Length: len*   = len / L
  - Mass: m*   = m / $(\rho_\infty L^3)$
  - Velocity: V*   = V / V_{ref}
  - Time: t*   = t (V_{ref}/L)
  - Acceleration: a*   = a (L/V_{ref}^2)
  - Force: F*   = F / $(\rho_\infty V_{ref}^2 L^2)$
  - Moment of inertia: I*   = I / $(\rho_\infty L^5)$
  - Angular velocity: ω*   = ω (L/V_{ref})
  - Moment: M*   = M / $(\rho_\infty V_{ref}^2 L^3)$

Non-Dimensionalization Example: Airfoil Drop

- Assume standard sea-level conditions:
  - ρ_∞ = 0.002378 slug/ft^3
  - c_∞ = 1117 ft/sec
  - Gravity = 32.2 ft/sec^2
- Pick airfoil properties:
  - chord = 1 ft
  - weight = 30 lb (heavy!)
- Flow conditions:
  - Mach = 0.2
  - Re/chord = 1 million
- From these we have:

dimensional		non-dimensional		
L	= 1 ft	L*	= 1	(grid is in chords)
Vref	= 223.4 ft/sec	Vref*	= 1	
g	= 32.2 ft/sec^2	g*	= 645x10^{-6}	
Wt	= 30 lb	Wt*	= 0.2528	
mass	= 0.9317 slug	mass*	= 392	
Iyy	= 0.05054 slug-ft^2	Iyy*	= 21.25	

- And pick (so that 400 steps is 0.1 sec):
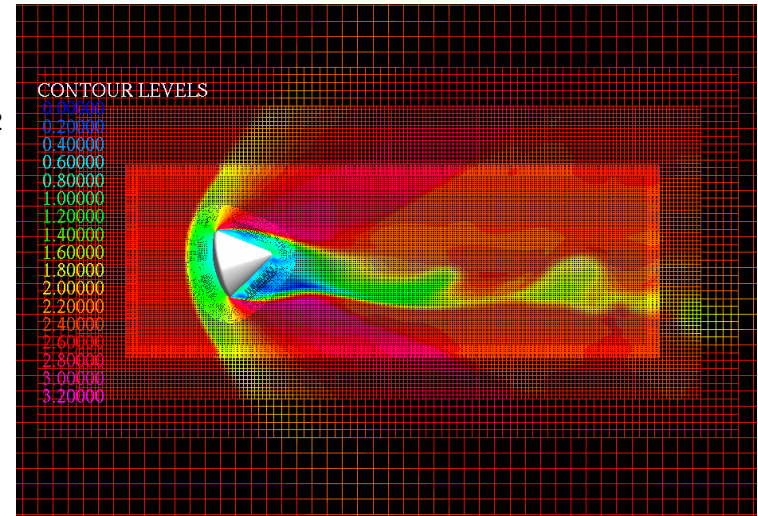
Δt	= 0.00025 sec	Δt*	= 0.05585	
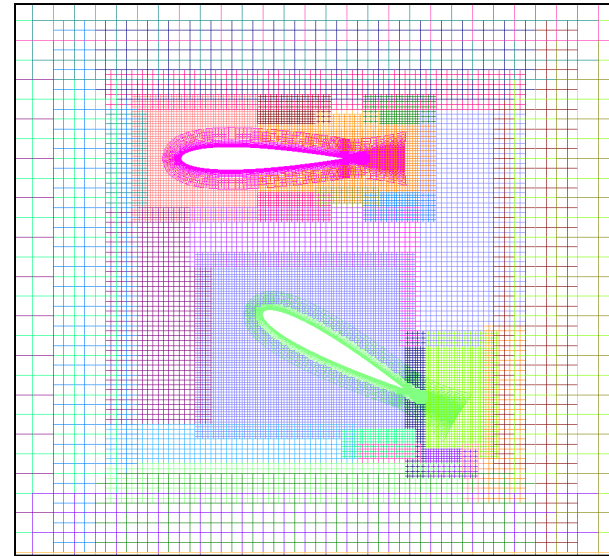
Non-Dimensionalization Example:
Apollo Ballistic Range Model

- Ballistic range model properties:
  - diameter = 63 mm
  - mass = 575.9 g
  - $(I_{xx}, I_{yy}, I_{zz}) = (0.1833, 0.1761, 0.1761) \times 10^6$ g-mm^2
- Assume standard sea-level conditions:
  - $\rho_\infty = 1.226$ g/mm^3
  - $c_\infty = 0.3405 \times 10^6$ mm/sec
  - Gravity = 9807 mm/sec^2
  - $\mu_\infty = 1.781$ g/mm-sec
- Flow conditions:
  - Mach = 2.5
  - Re/mm = 58,610/mm
- From these we have:

CONTOUR LEVELS
0.10000
0.20000
0.40000
0.60000
0.80000
1.00000
1.20000
1.40000
1.60000
1.80000
2.00000
2.20000
2.40000
2.60000
2.80000
3.00000
3.20000

dimensional		non-dimensional		
L	= 1 mm	L*	= 1	(grid is in mm)
Vref	= 0.8512×10^6 mm/sec	Vref*	= 1	
g	= 9807 mm/sec^2	g*	= 13.53×10^{-9}	
mass	= 575.9 g	mass*	= 469.7×10^6	
Ixx	= 0.1833×10^6 g-mm^2	Ixx*	= 149.5×10^9	
Iyy = Izz = 0. 1761×10^6 g-mm^2		Iyy* = Izz*	= 143.6×10^9	

GMP Example: Capsule Forced Oscillation

- **Config.xml** file is common to forced oscillation, free oscillation, and ballistic range (6-DOF) simulation
  - Grids 1-3 are the capsule, grid 4 is the wake box
  - 4 deg initial angle
  - Capsule CG is at (21,0,0)

```
<?xml version='1.0' encoding='utf-8'?>
<Configuration AngleUnit="degree">
   <Component Name="Capsule" Type="struc">
      <Data> Grid List=1-3 </Data>
      <Transform>
         <Rotate Center="21.0,0,0" Axis="0,1,0" Angle="4.0"/>
      </Transform>
   </Component>
   <Component Name="Box" Type="struc">
      <Data> Grid List=4 </Data>
      <Transform>
         <Rotate Center="21.0,0,0" Axis="0,1,0" Angle="0.0"/>
      </Transform>
   </Component>
</Configuration>
```

GMP Example: Capsule Forced Oscillation

- **Scenario.xml** file for forced oscillation
  - Time period for 1 oscillation is 20100 (non-dimensionalized)

```xml
<?xml version='1.0' encoding='utf-8'?>
<Scenario Name="Forced Oscillation" AngleUnit="degree">
    <Prescribed Component="Capsule" Start="0" >
        <Rotate Center="21.0,0,0" Axis="0,1,0"
         Speed="4.*2.*pi/20100.*cos(2.*pi/20100.*t+pi/2.)"
         Frame="parent" />
    </Prescribed>
</Scenario>
```

GMP Example: Capsule Free Oscillation

- **Scenario.xml** file for free oscillation
  - Capsule inertial properties
  - Constraints: only allow rotation about y

```
<?xml version='1.0' encoding='utf-8'?>
<Scenario Name="Free Oscillation" Gravity="0,0,-13.53e-9"
 AngleUnit="degree">
    <Aero6dof Component="Capsule" Start="0">
        <InertialProperties Mass="469.7e6" CenterOfMass="21.0,0,0"
         PrincipalMomentsOfInertia="149.5e9, 143.6e9, 143.6e9"
          <PrincipalAxesOrientation Axis="1,0,0" Angle="0"/>
        </InertialProperties>
        <Constraint Start="0" Translate="1,1,1" Rotate="1,0,1"/>
    </Aero6dof>
</Scenario>
```

GMP Example: Capsule Ballistic Range Shot

- **Scenario.xml** file for ballistic range simulation
    - Capsule inertial properties
    - Box "flies" with capsule, but no rotations allowed
    - In **mixsur.inp**, "Box" component is defined to be the same as "Capsule"

```xml
<?xml version='1.0' encoding='utf-8'?>
<Scenario Name="Ballistic Range" Gravity="0,0,-13.53e-9"
 AngleUnit="degree">
   <Aero6dof Component="Capsule" Start="0">
      <InertialProperties Mass="469.7e6" CenterOfMass="21.0,0,0"
       PrincipalMomentsOfInertia="149.5e9, 143.6e9, 143.6e9">
       <PrincipalAxesOrientation Axis="1,0,0" Angle="0"/>
      </InertialProperties>
</Aero6dof>
<Aero6dof Component="Box" Start="0">
      <InertialProperties Mass="469.7e6" CenterOfMass="21.0,0,0"
       PrincipalMomentsOfInertia="149.5e9, 143.6e9, 143.6e9">
       <PrincipalAxesOrientation Axis="1,0,0" Angle="0"/>
      </InertialProperties>
      <Constraint Rotate="1,1,1" Start="0" Frame="body" />
   </Aero6dof>
</Scenario>
```

Time-Step Specification for Time-Accurate Simulations

- Dual time-stepping (input parameters in **$GLOBAL**):
    - **DTPHYS** – physical time-step (non-dimensionalized by V_{ref})
    - **TPHYS** – to reset simulation time, e.g., **TPHYS=0** when starting dynamics
    - **FSONWT** – selection of multi-step time-advance scheme
        - 1—BDF1 (1st-order)
        - 2—BDF2 (2nd-order) (default)
        - 2.5—BDF2OPT (2nd-order)
    - **NDIRK** – selection of multi-stage time-advance scheme
        - 0—Default (Euler implicit BDF1, BDF2, BDF2OPT based on **FSONWT**)
        - 1—Euler explicit (1-stage, 1st-order)
        - 2—Euler implicit BDF1 or BDF2 (1-stage, 1st- or 2nd-order)
        - 3—RK3 explicit (3-stage, 3rd-order)
        - 4—RK4 explicit (4-stage, 4th-order)
        - 5—ESDIRK3 implicit (3-stage, 3rd-order)
        - 6—ESDIRK4 implicit (5-stage, 4th-order)

Time Step Specification for Time-Accurate Simulations

- Dual time-stepping (input parameters in **$GLOBAL**):
  - **NITNWT** – (maximum) number of Newton/dual subiterations
    - 3 for simplest problems ("rough" time-accuracy)
    - 10-20 for general problems
    - 40+ for difficult problems
    - Affected by overset grid boundaries (explicit boundaries slow information transfer)
    - Affected by boundary conditions (like C-grid wakes)
  - **ORDNWT** – order of convergence for early cutoff of subiterations
    - >0 – L_∞-norm(RHS) convergence over first subiteration
    - <0 – L_∞-norm(RHS) convergence over temporal error estimate
    - $=0$ – do not limit subiterations (default)

Newton/Dual Subiteration

- Subiteration is used for several functions:
  - Drives out left-hand side factorization error
  - Converges explicit boundary conditions
  - Converges unsteady simulation to the next physical time step
- Choose physical time step **DTPHYS**
  - For dual time-stepping, also choose local time-stepping (**ITIME, DT, CFLMIN, CFLMAX**) and multigrid
  - For Newton subiteration, local time-step is the same as **DTPHYS** (set **ITIME=0**)
  - Goal is to converge subiterations at least 2 orders of magnitude (ad hoc rule)
- Subiteration convergence can be improved by increasing **NITNWT** and/or decreasing **DTPHYS**
  - Reducing **DTPHYS** also improves physical time-accuracy
- Generally, choose **DTPHYS**:
  - To resolve physical scales (<u>at least</u> 100-200 steps per cycle)
  - To have sufficiently resolved body motion for accurate DCF
  - To have adequate drop in subiteration residual

Converging Newton/Dual Subiterations

- In residual history files (**resid.out**, etc.) there is one entry (for each grid, or off-body grids) per subiteration
- First subiteration right-hand side (RHS) residual represents the unsteady forcing function
  - If this is decreasing (converging), the flow is becoming more steady
- The drop in RHS residual from first to last subiteration represents the numerical accuracy of computing the unsteady flow
  - This should be at least 2 orders-of-magnitude (unless the flow is steady)
  - Try using **ORDNWT=2** to do this
- If selected grids are not converging as well as others, try setting **ITER=2** for those grids
- A 2-order-of-magnitude drop indicates that the time advance is *numerically* converged; it does not guarantee that the physical time-step is small enough to resolve physical processes
- Use OVERPLOT from CGT to plot **resid.out** –type files, as well as subiteration convergence

Converging Newton/Dual Subiterations
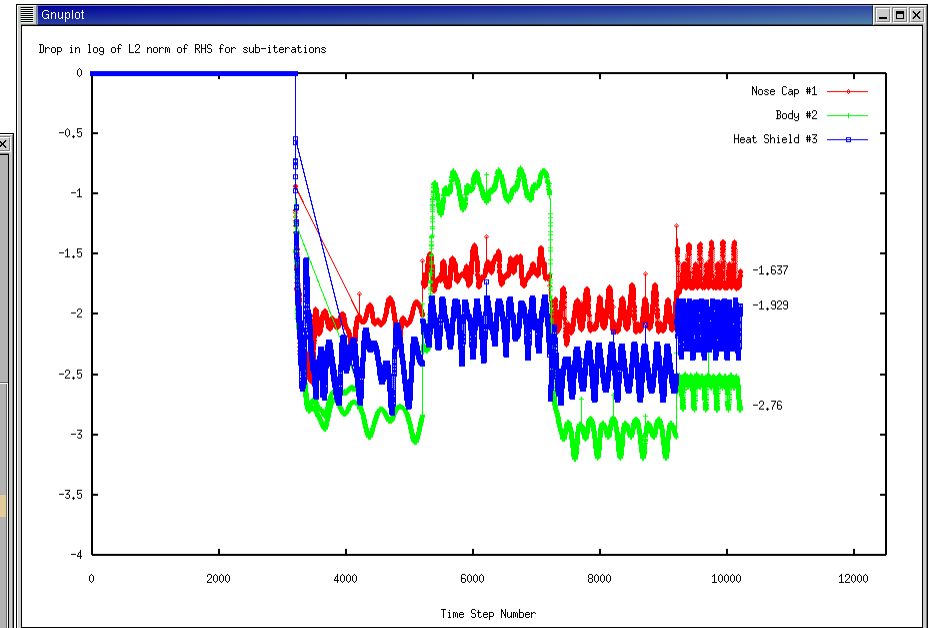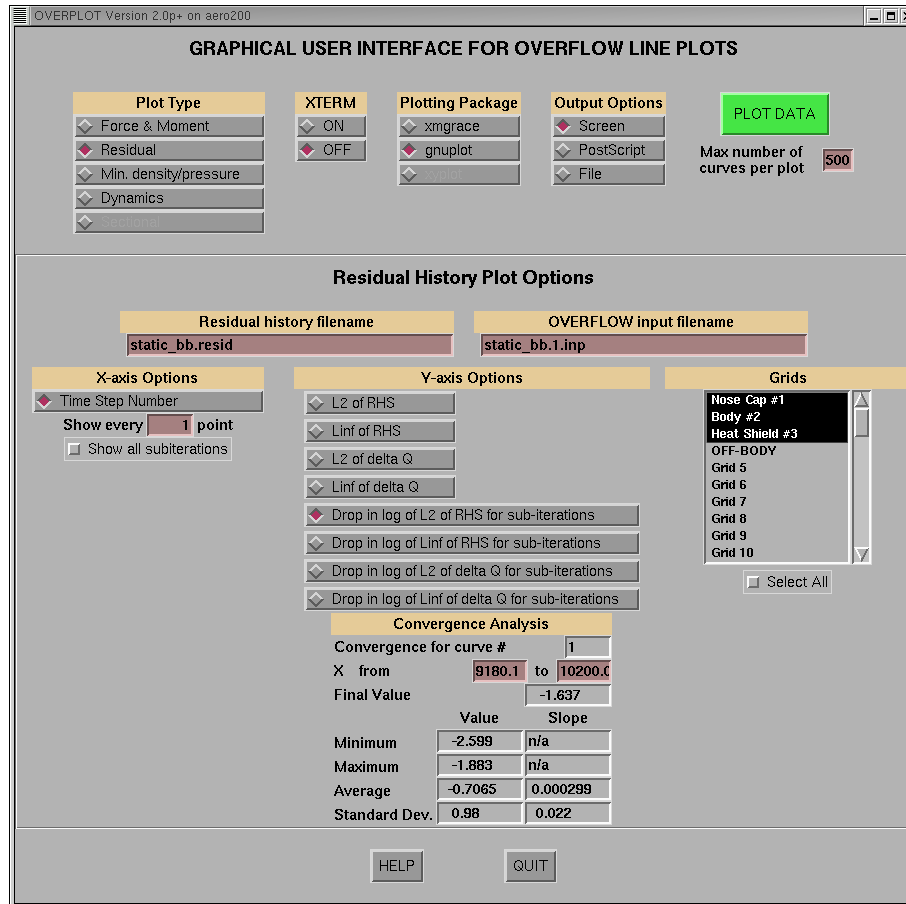
Simulating Collisions

- Contact between bodies is detected by using X-ray hole-cutting applied to surface grids of other bodies
- Contact detection is enabled (per body) by adding grid "0" to **IGXLIST** in X-ray cutter(s)
- Example: airfoil drop

  $XRINFO IDXRAY=1, IGXLIST=-1, XDELTA=0.04, $END
  $XRINFO IDXRAY=2, IGXLIST=-1, XDELTA=0.04, $END
  $XRINFO IDXRAY=1, IGXLIST=2,0, XDELTA=0.0, $END
  $XRINFO IDXRAY=2, IGXLIST=1,0, XDELTA=0.0, $END

- Accurate geometric representation of collisions may require much finer X-rays than hole-cutting
  - To keep DCF process from becoming very slow, can make collision X-rays separate from DCF X-rays
- **R_COEF** in **$OMIGLB** sets (global) coefficient of restitution
- Time of contact is accurate only to within **DTPHYS**

Collision Example: Airfoil Drop

- Two collisions
- Off-body grids follow airfoil
- Airfoil remains inside level-1 grid (**NADAPT** is OK)
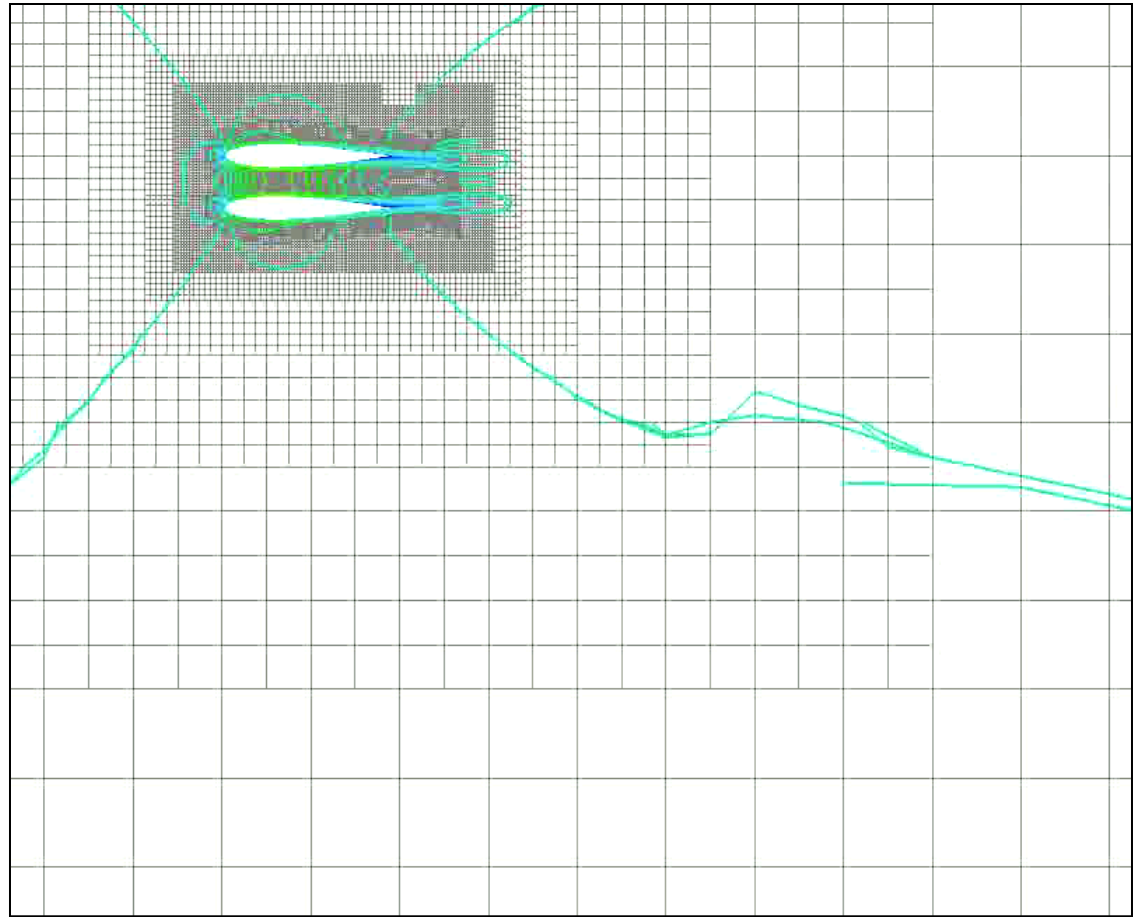
Output Information for Moving Body Simulations

- Input parameters in **$GLOBAL**:
  - **NSAVE** – grid system, flow solution, and 6-DOF restart information is saved every **NSAVE** steps, as **x.step#, q.step#, sixdof.step#**
  - **NFOMO** – force and moment coefficients are written to fomoco.out every **NFOMO** steps (automatically set to 1 for 6-DOF simulations)
- Namelist **$SPLITM**: write subsets of grid and solution every n steps (similar to CGT utilities SPLITMX, SPLITMQ)
  - **XFILE,QFILE,QAVGFILE** – specify base names for grid, solution, and/or Q-average data (if blank, don't write); step# will be appended to base name
  - **NSTART,NSTOP** – start/stop step numbers for writing output files (use -1 for "last")
  - **IPRECIS** – output file precision (0—default,1—single, 2—double)
  - **IG(subset#)** – subset grid number; use **IG()=-1** for cut of all off-body grids
  - **JS,JE,JI,KS,KE,KI,LS,LE,LI(subset#)** – subset ranges and increments
  - **CUT(subset#),VALUE(subset#)** – off-body grid cut type ("x", "y", or "z") and corresponding x, y, or z value
  - Can have multiple **$SPLITM** namelists for multiple files

Output Information for Moving Body Simulations

- History files:
  - **fomoco.out** – force and moment coefficients per component, per step (same as for static problems, except moment reference center moves with body)
  - **animate.out** – body ID, physical time, body position and orientation (quaternion notation), velocity and rotation rates, aero forces and moments (not coefficients)
  - **contact.out** – lists step #, body IDs, contact point and normal vector, reaction impulse, and linear and angular velocity changes (this is more for debugging collisions)
  - Note that OVERRUN script concatenates these files into **basename.{fomoco,animate,contact}**
- Use OVERPLOT from CGT to plot **fomoco.out, animate.out** –type files
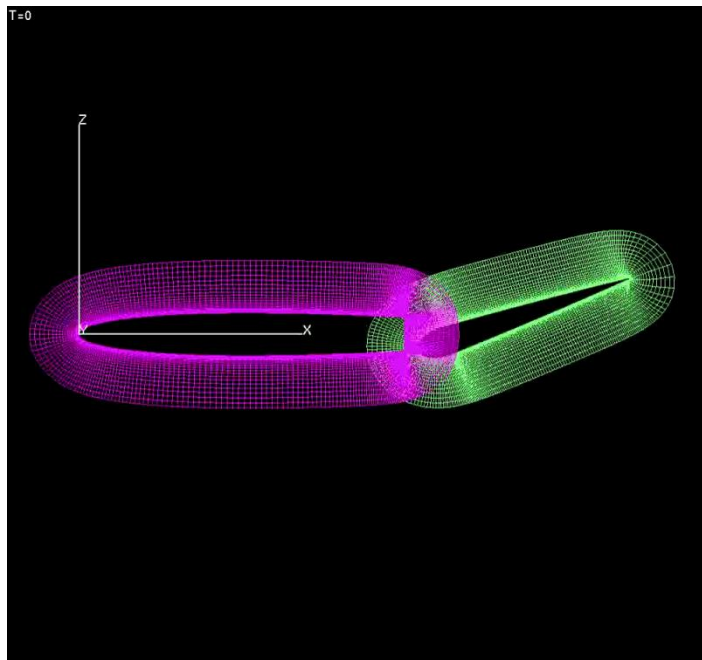
Sample Trajectory Plot in OVERPLOT

Visualizing Body Motion in OVERGRID

- Prescribed motion can be visualized in OVERGRID by reading in (surface grids or) **grid.in, Config.xml** and **Scenario.xml**
  - Start OVERGRID with surface grids or grid.in
  - Click "COMPONENTS"
  - On COMPONENTS menu,
    - Click Read "Config" ("OK")
    - Click Read "Scenario" ("OK")
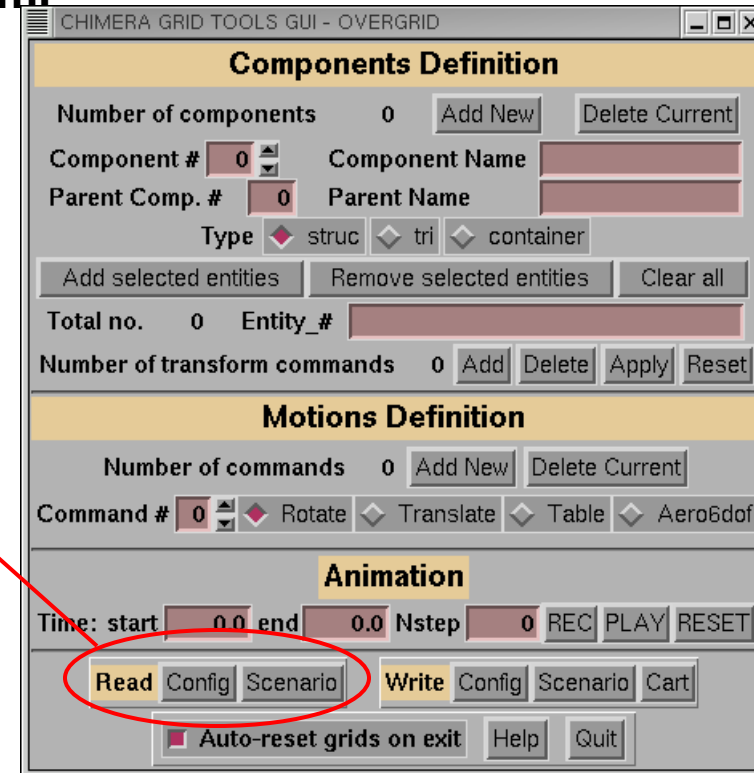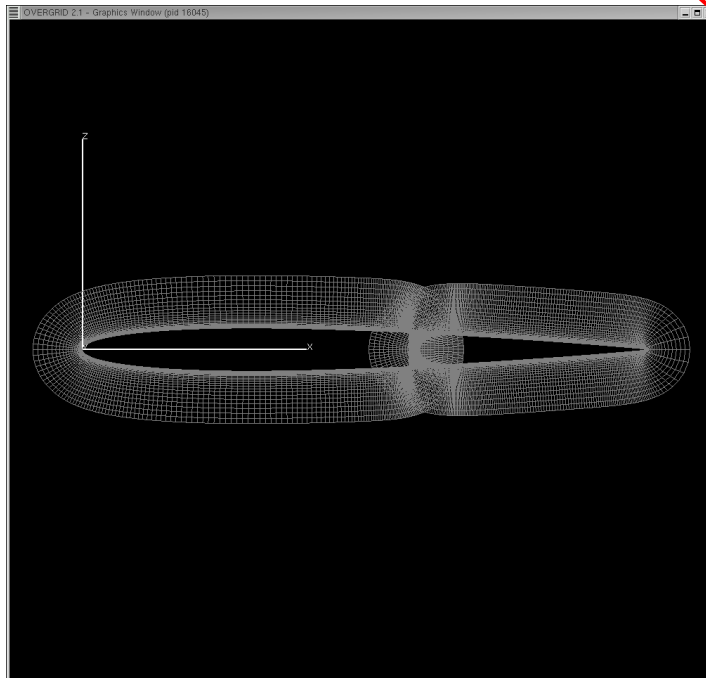
Visualizing Body Motion in OVERGRID

- Menu shows information on each component
  - Component names and hierarchy
  - Initial transforms from **Config.xml**
  - Prescribed motions from **Scenario.xml**
- Enter animation information
  - Start/end time and number of steps
  - Click "PLAY"

Visualizing Body Motion in OVERGRID

- For visualizing 6-DOF motion (after the OVERFLOW simulation is complete) read in **basename.animate**:
  - Click "Add New" motion command
  - Click "Table"
  - Type in animate filename and click "Read"
 - Click "PLAY"
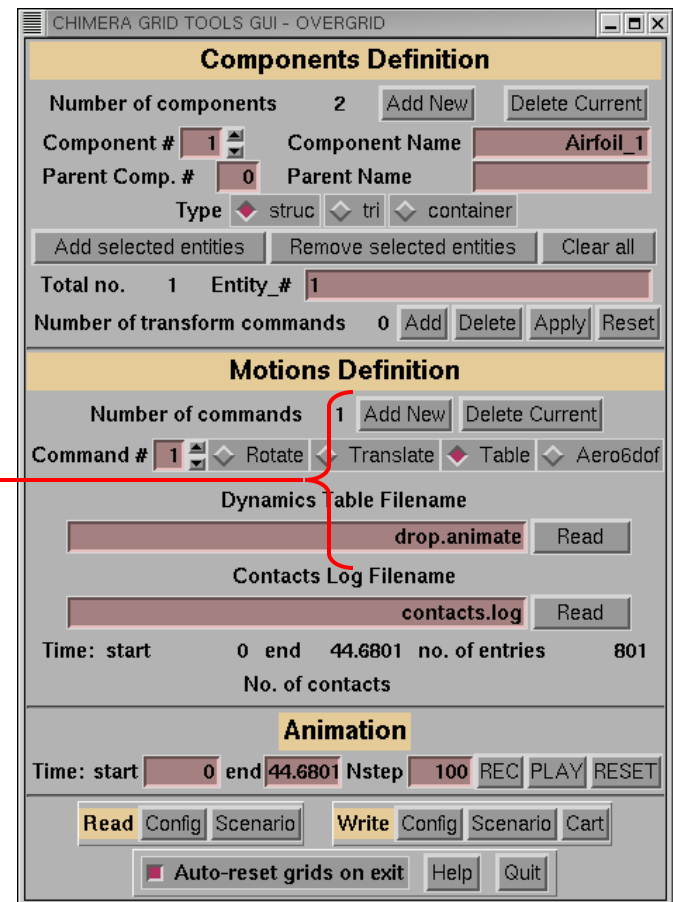
Some References

- GMP interface
  - S.M. Murman, W.M. Chan, M.J. Aftosmis, and R.L. Meakin, "An Interface for Specifying Rigid-Body Motions for CFD Applications," AIAA 2003-1237, Jan. 2003

- Solution adaption
  - P.G. Buning and T.H. Pulliam, "Cartesian Off-body Grid Adaption for Viscous Time-Accurate Flow Simulations," AIAA 2011-3693, June 2011
  - R.L. Meakin, "An Efficient Means of Adaptive Refinement Within Systems of Overset Grids," AIAA 95-1722, June 1995
  - R.L. Meakin, "On Adaptive Refinement and Overset Structured Grids," AIAA 97-1858, June 1997

- Hole cutting using X-rays
  - R.L. Meakin, "Object X-Rays for Cutting Holes in Composite Overset Structured Meshes," AIAA 2001-2537, June 2001

- Off-body grid generation
  - R.L. Meakin, "Automatic Off-Body Grid Generation for Domains of Arbitrary Size," AIAA 2001-2536, June 2001

- Collision dynamics
  - R.L. Meakin, "Multiple-Body Proximate-Flight Simulation Methods," AIAA 2005-4621, June 2005

Solution Adaption for
Near-Body and Off-Body Grids

- Allows off-body refinement grids that are finer than level-1, near-body refinement grids that are finer than original grid

- Refinement levels are labelled -1, -2, etc., and have grid spacing of **DS**/2, **DS**/4, etc. (off-body), and ½, ¼, etc. of original spacing (near-body)

- Can be used with or without grid motion, for steady-state or time-accurate simulations
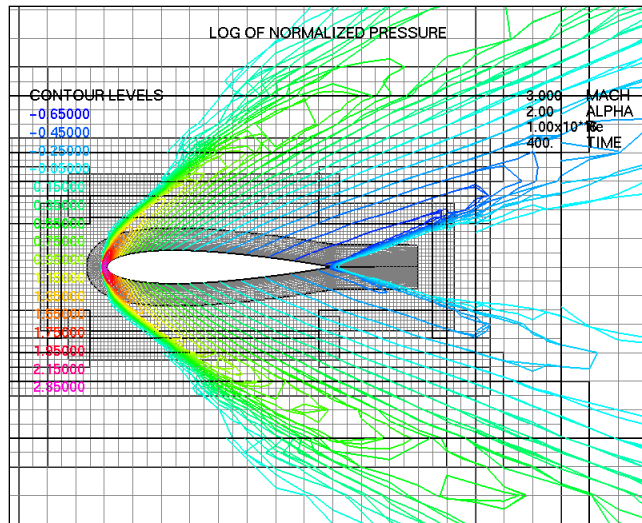
400 steps, no adaption
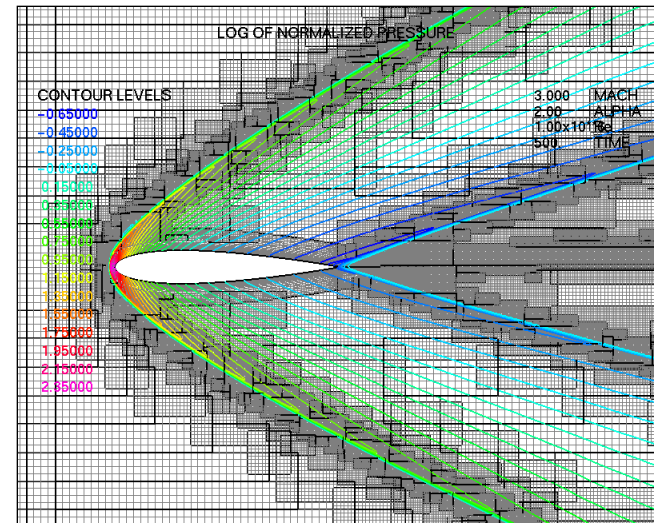
Additional 100 steps,
adapting every 10 steps

10/6/2014

Solution Adaption for Near- and Off-Body Grids

- Original and finer grids are cut by X-rays

- Refinement grids blank out regions in coarser level grids
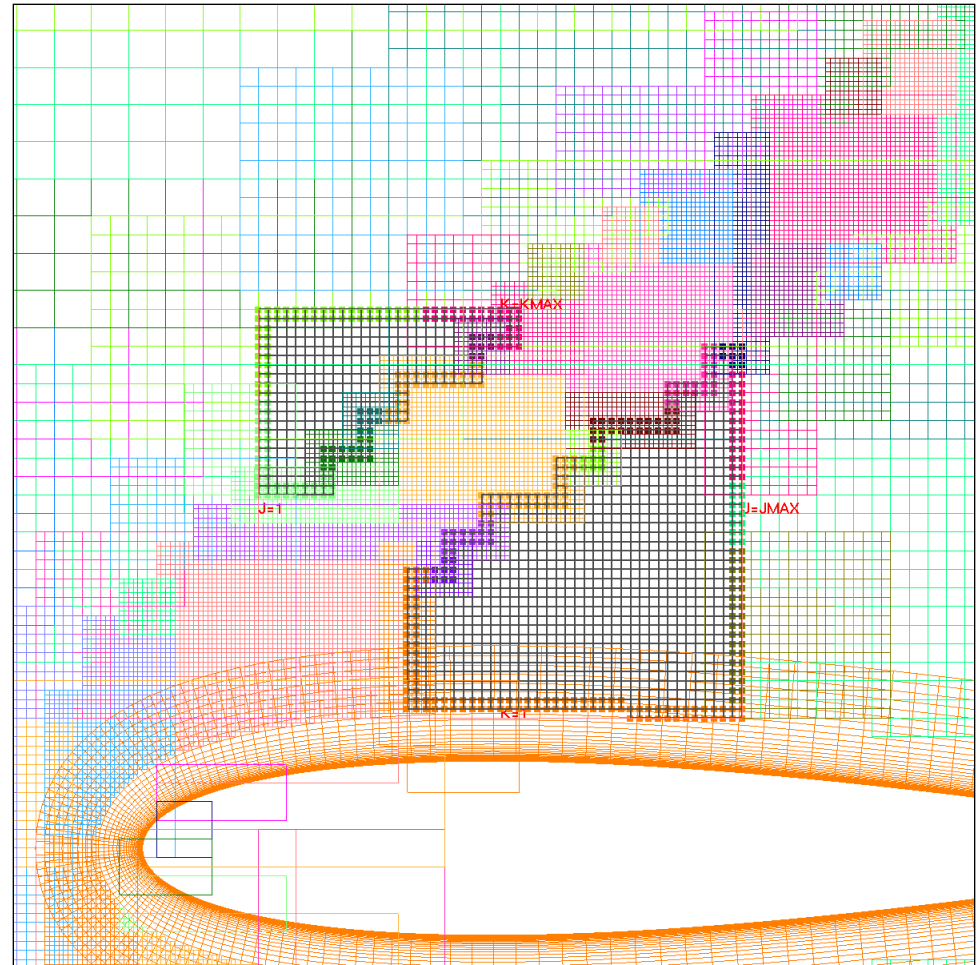
Input Parameters

- Basic control parameters (in **$OMIGLB**)
  - **NADAPT=n** – adapt solution every n steps
    - 0—do not adapt
    - >0—adapt to geometry and sensor function
    - <0—adapt to geometry only
  - **NREFINE=m** – allow up to m levels of refinement
  - **NBREFINE=k** – allow up to k levels of refinement for near-body grids
  - **ETYPE** – sensor function for adaption
    - 0—undivided 2nd-difference of Q variables (squared)
    - 1—vorticity magnitude
    - 2—undivided vorticity magnitude
  - **EREFINE/ECOARSEN** – refine above/coarsen below these function values
  - **SIGERR** – shortcut method to set **EREFINE** and **ECOARSEN**
    - **EREFINE**$=(1/8)^{\text{SIGERR}}$, **ECOARSEN**$=(1/8)^{\text{SIGERR}+2}$

Airfoil Example: Effect of **NREFINE**

$OMIGLB NADAPT=10, NREFINE=0,
    ETYPE=0, SIGERR= 5, $END

- No grids finer than level-1
- Results in 84 grids and 185K points
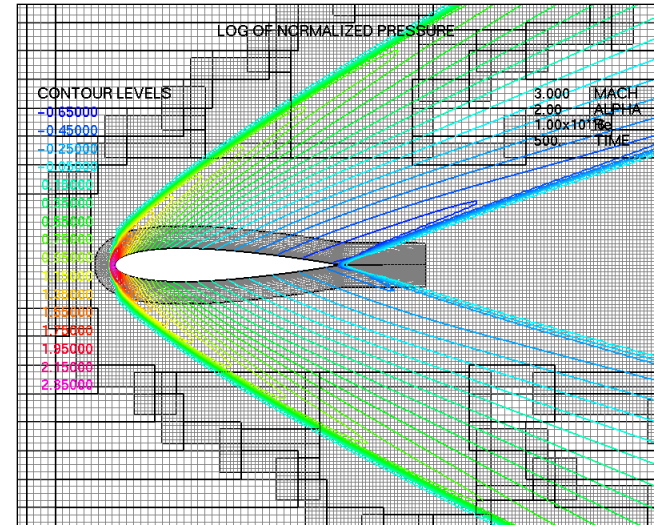
$OMIGLB NADAPT=10, NREFINE=2,
    ETYPE=0, SIGERR= 5, $END

- Two levels of refinement
- Results in 342 grids and 754K points
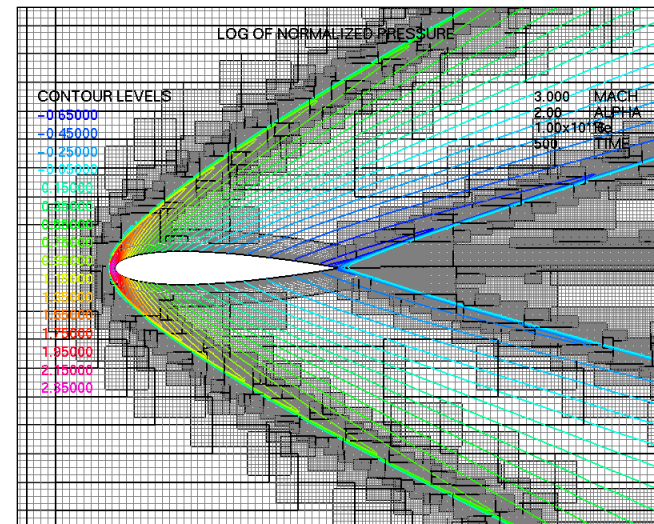
Input Parameters

- Parameters to restrict refinement regions (in **$BRKINP**)

  - **(XREFMIN,XREFMAX, YREFMIN,YREFMAX, ZREFMIN,ZREFMAX)** – min/max of region to limit grid refinement

  - **REFLVL=n** – limit grid refinement level to ≥n for this region

  - **REFINOUT** – grid refinement level is limited "INSIDE" or "OUTSIDE" the specified region

  - **IBDYREF** – limit region min/max box is tied to this Body ID for motion transformations (0 for no motion)

- Example: no adaption outside specified region

  $OMIGLB NREFINE=2, ..., $END

  $BRKINP

  XREFMIN=-0.5, XREFMAX=1.5,

  ZREFMIN=-0.5, ZREFMAX=0.5,

  REFLVL=2, REFINOUT='OUTSIDE',

  $END
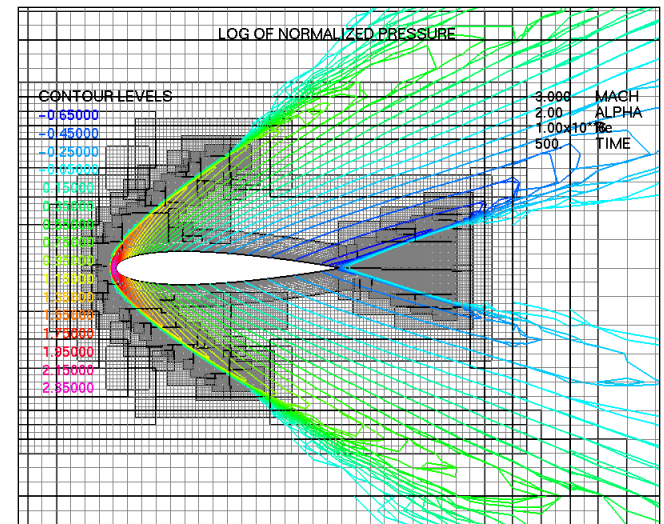
Notes and Comments

- For time-accurate simulations, adjust NADAPT to make sure that adapted regions keep up with flow features and geometry

- Because the adaption can generate a large number of small grids, time-accurate simulations may need (more) subiterations to ensure good communication across grid boundaries

10/6/2014

Visualizing Moving and/or Adapting Grids and Solutions using OVERGRID

- Moving or adapting (surface or 2D) grids can be visualized in OVERGRID by using the SOLUTION button (under "Viewers and Special Modules")
  - Start OVERGRID with x.save (or some grid)
  - Click "SOLUTION"
  - On SOLUTION menu,
    - Click "Adaptive grids"
    - Click "multiple"
    - Adjust Root name, start, end
    - Click "Read"
    - Same for Q

    - Select Function
    - Click "PLAY" or step through frames
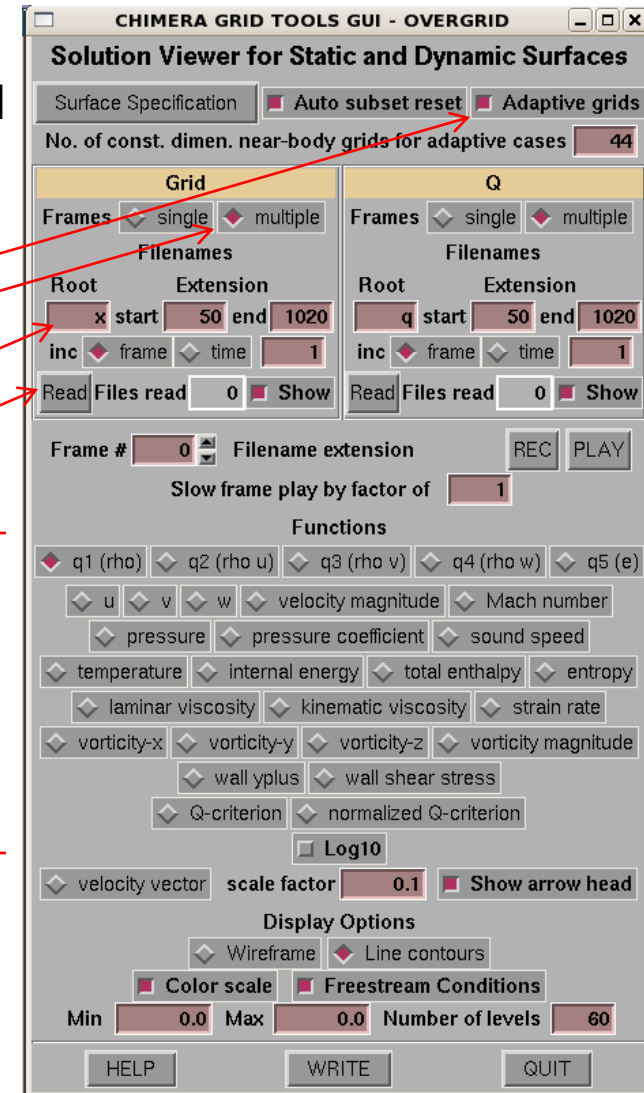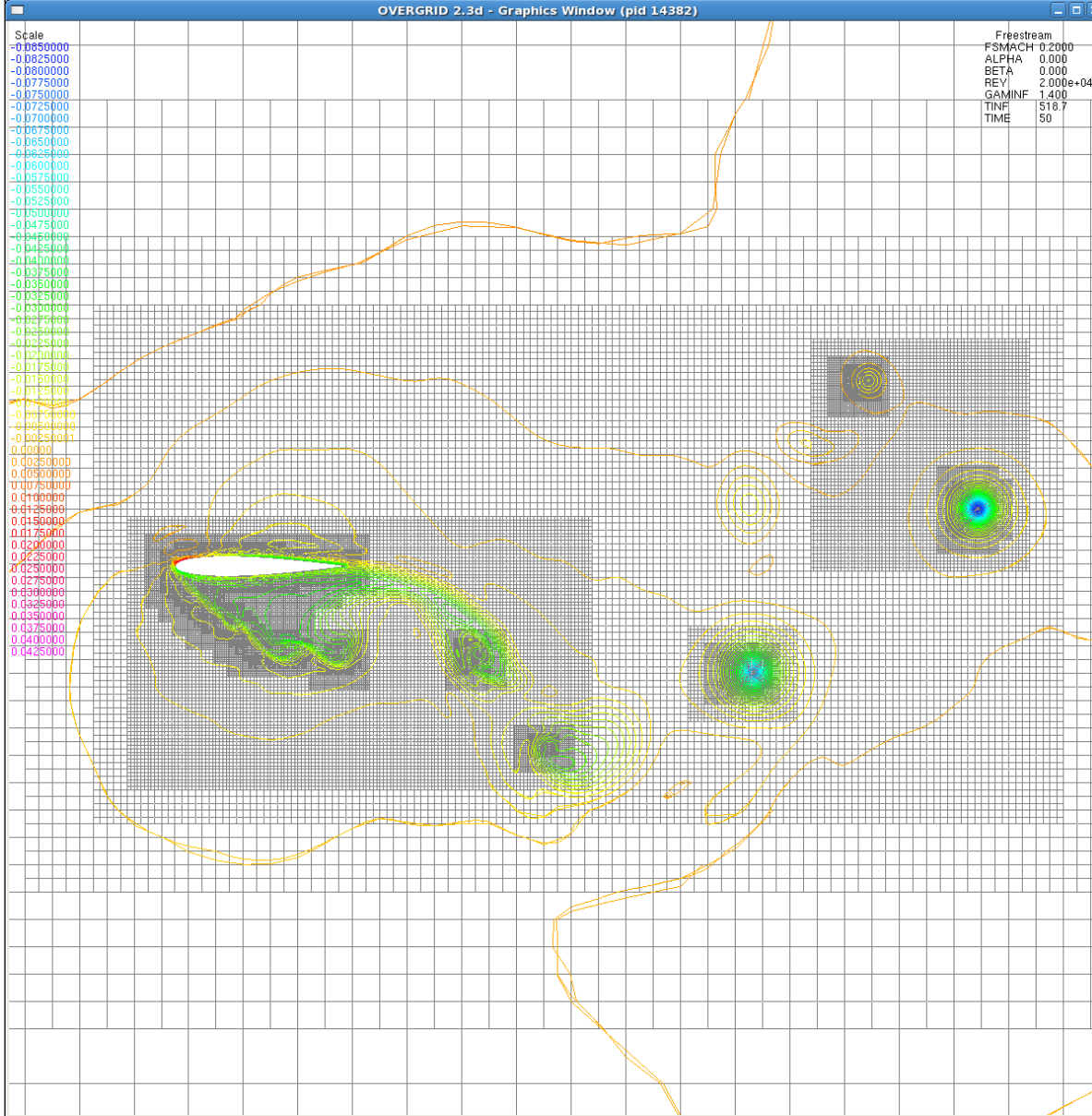
CHIMERA GRID TOOLS GUI - OVERGRID

Solution Viewer for Static and Dynamic Surfaces

Compiling and Running OVERFLOW

- Unpacking and compiling
- Execution scripts
- Parallel processing and MPI load-balancing
- Hints and warnings
- Utility codes
- Test cases included with OVERFLOW

Unpacking and Compiling

- Code comes as a gzipped tar file:

  `tar zxvf over2.2x.tar.gz`

  - Documentation is in **over2.2x/doc**
  - Test cases are in **over2.2x/test**
  - Tools are in **over2.2x/tools/***

- Flow solver makefiles **Makefile, Makefilempi**
  - Makefiles for tools, other utilities too
  - **Make.sys** contains compiler options for a large variety of machines
    - These are current suggested options only
    - User's responsibility to check and update
    - Set for "big-endian" file I/O
  - Sample **makeall** script to compile and install flow solver and tools: single and double precision, MPI and non-MPI (e.g., `makeall intel`)
  - For MPI compilation and running, **.cshrc** (or equivalent) must define environment variable MPI_ROOT, for example

    `setenv MPI_ROOT /usr/local/mpich`

Execution Scripts

- Execution scripts **overrun** and **overrunmpi**
  - Moves *.save** files to *.restart** before starting
  - Highlights warnings and errors
  - Creates a log file with the time/date, machine name, executable name, and namelist input file name
  - Concatenates output history files upon completion
  - Expected namelist input file of the form **basename.n.inp**

    ```
    overrun basename n

    overrunmpi –n <ncpus> -t <threads> \
       –machinefile <hostfile> basename n
    ```

  - I strongly recommend using these scripts, unless OVERFLOW is built into another process

Parallel Processing Options

- MPI (Message Passing Interface)
  - Useful for PC clusters (how many depends on network)
  - Good efficiency for shared memory machines with special MPI library (SGI Altix)
  - Parallelized on groups of grids (more on load-balancing later)
  - 3 general flavors, OpenMPI, MPICH, LAM, available on the web
- OpenMP
  - Useful for shared memory machines (8 CPUs or less)
  - Useful for multi-core machines (but memory bandwidth may limit performance)
  - Parallelized on grid planes (doesn't work for 2D or axisymmetric problems)
- Hybrid parallelization (mix of MPI and OpenMP)
  - Depends on system; MPI library must allow multiple threads

10/6/2014

MPI Load-Balancing

- Number of groups == number of processes in the MPI run
- Default load-balancing scheme:
  - Based on equal distribution of grid points between processes (target group size)
  - Grids are split in half (with overlap added) until each grid is less than half the target group size
  - Grids are distributed, from largest to smallest, to current smallest group
  - This scheme works quite well for grid systems with large numbers of grids, and reasonably well for smaller systems
  - Some pathological cases:
    - 1 grid, 2 processes (grid is split into 4 instead of 2)
    - 1 grid, 3 processes (grid is split into 8, load-balance is 3/8,3/8,2/8)
  - Note that grid splitting introduces additional explicit boundaries, which affects convergence behavior

10/6/2014

MPI Load-Balancing

- Controlling load-balancing (input parameters in **$GROUPS** and **$GLOBAL**):
  - **Use of the following inputs is rarely needed**
  - **USEFLE=.TRUE.** – use previous timing information in **grdwghts.restart** for distributing grids to groups (FALSE – use default load-balancing scheme)
    - Same as **GRDWTS** in **$GLOBAL**
  - **WGHTNB** – weighting factor for near-body grids vs. off-body grids in default load-balancing scheme (for example if viscous terms are turned off in off-body grids)
  - **MAXNB** – control splitting of near-body grids
    - **MAXNB=0** – use automatic splitting algorithm
    - **MAXNB>0** – specified (weighted) size limit
    - **MAXNB<0** – do not split grids
    - Same as **MAX_GRID_SIZE** in **$GLOBAL**
  - **MAXGRD** – control splitting of off-body grids (same options as **MAXNB**)
  - **IGSIZE** – maximum group size during grid adaption (default is 10Mpts)
  - Example: pathological case 1 (single grid (1 million points), 2 processes)
    - $GROUPS MAXNB=600000, $END
    - Grid will be split once, with both halves smaller than 600,000 pts
    - Each process will get one piece

Load-Balancing Diagnostics

- What grid splitting was done for load-balancing?

```
Target (weighted) near-body grid size from grouping:          12862
Checking near-body grids...
Original number of near-body grids:      2
Final    number of near-body grids:      8


Target (weighted)  off-body grid size from grouping:          14752
Checking  off-body grids...
Original number of  off-body grids:    30
Final    number of  off-body grids:    31
```

- Detailed list of split grids is also given in the output

Load-Balancing Diagnostics

- What is the resulting grouping of grids?

```
Load balance will be based on grid size.

    Summary of work distribution for     4 groups:

Group      Kpts        %load    Grid list
   1         30         100      4    8   11   17   14   22   21   33   31   34
                                39
   2         29          99      6    7   12   19   13   18   20   32   26   37
   3         30         100      1    9    3   23   24   28   30   35   38
   4         30         100      2   10    5   15   25   29   27   36   16


Predicted parallel efficiency is    100%,

   based on a maximum of     30K grid points per group
    compared to an average of     30K points (weighted)


Estimated parallel speedup is      4.0
```

Load-Balancing Diagnostics

- What is the actual load-balance?

```
GROUP TIMING SUMMARY    (Time each group spent in OVERFL)
 (*) STEP loop, (/) Chimera BC, (a) Adapt, (D) DCF(non-idle), (s) Grid,Q save


             0            25            50            75           100
             |-----------|------------|------------|-----------|
Group:   1   |************************************///aaaaDDs    97%
Group:   2   |*************************************///aaaaDD    98%
Group:   3   |************************************///aaaaDD    97%
Group:   4   |************************************///aaaaDDs   99%


  Overall Measured Parallel Efficiency:  97.9%
```

Load-Balancing – What to Look For

- Predicted parallel efficiency is low

```
Predicted parallel efficiency is     75%
```

- Not able to split or group grids effectively
- Some grids may not be split because of boundary conditions (axis, C-grid wake)
- Change the number of CPUs or manually split problem grid

- Histogram shows groups are not well balanced

```
Group:    1  |*******************************************///s    97%
Group:    2  |*********************///                           47%
Group:    3  |*********************///                           47%
Group:    4  |*********************///s                          49%
```

- Group 1 is sharing the CPU with another process
- Eliminate other process or use a different CPU

- Large amount of time (~50%) spent exchanging Chimera BCs

```
Group:    1  |***********************///////////////////////s    98%
Group:    2  |***********************////////////////////////    96%
Group:    3  |***********************////////////////////////    97%
Group:    4  |***********************////////////////////////s   99%
```

- Network is too slow to permit efficient use of this many CPUs
- Use fewer CPUs

Load-Balancing – What to Look For

- Measured parallel efficiency is less than predicted efficiency (5-10%)

```
Predicted parallel efficiency is     96%

    . . .
              0             25            50            75            100
              |-----------|------------|------------|-----------|
Group:    1   |*********************************//DDD           87%
Group:    2   |********************************//DDD            85%
Group:    3   |*******************************//DDD             82%
Group:    4   |********************************///DDD           84%
Group:    5   |********************************//DDD            84%

    . . .
Group:   26   |*******************************************//DDD     99%
Group:   27   |******************************************//DDD      97%
Group:   28   |*******************************************//DDD     98%
Group:   29   |******************************************//DDD      96%
Group:   30   |******************************************//DDD      96%
Group:   31   |******************************************//DDD      97%
Group:   32   |******************************************//DDD      97%


 Overall Measured Parallel Efficiency:   92.4%
```
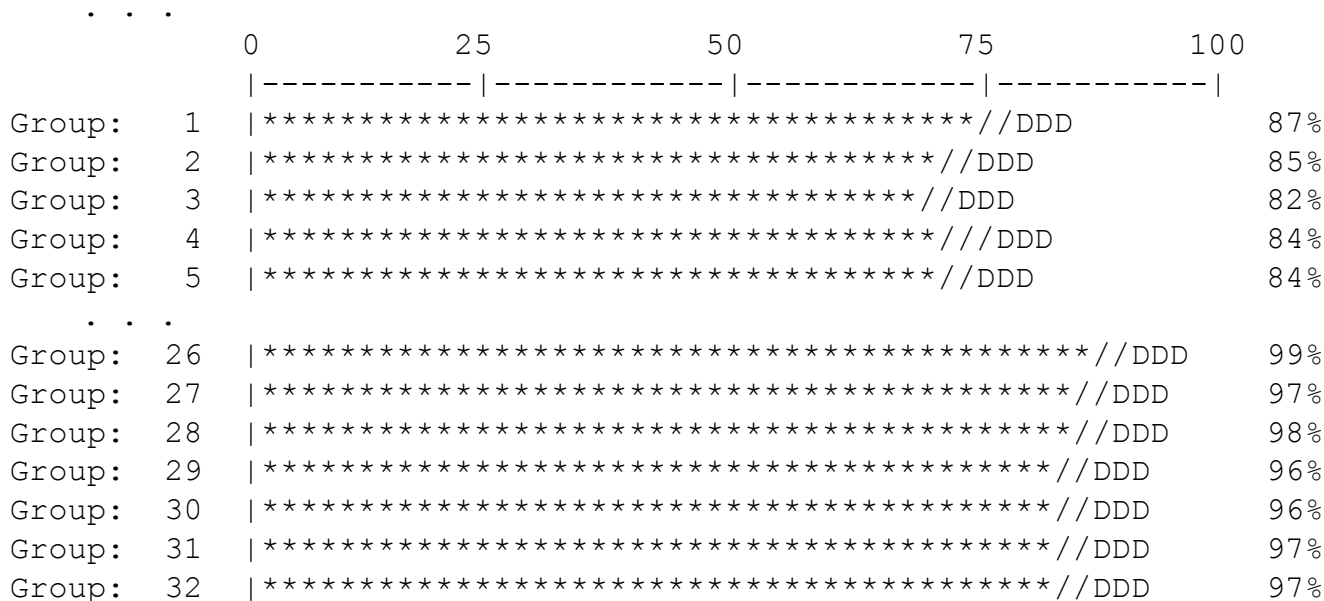
  – Set **USEFLE=.TRUE.** to use timing from previous run for load-balancing
  – This may improve the performance SOME

Hints and Warnings

- Unexplained errors while reading **grid.in** or **q.restart** file: check that all input files are the correct precision, correct "endian", and match the executable being run

- Unexplained segmentation violation while running (Intel Linux machines?): available stack memory has been exceeded, add "limit stacksize unlimited" in **.cshrc** file

- "overflow killed" message on console: process ran out of memory, check problem size

Utility Codes

- **Chimera Grid Tools** (CGT version 2.1)
  - Grid generation and manipulation utilities
  - Scripting process for grid generation and assembly
  - Force & moment integration: **mixsur, overint, USURP**
  - Post-processing utilities, OVERPLOT
  - OVERGRID user interface
  - Primary author is William Chan (NASA Ames)

- **Pegasus 5** (version 5.2)
  - Overset grid joining code and utilities
  - Primary author is Stuart Rogers (NASA Ames)

Both are available from NASA Ames Research Center
(*https://www.nas.nasa.gov/cgi-bin/software/start.html*)

Debug Options

- **DEBUG=1** – turbulence information quantities
  - Surface quantities: wall spacing, y^+, turbulence index
  - Field quantities: μ_t, vorticity, damping functions, k, ω, etc.
  - Different quantities per model—see OVERFLOW 2.2 manual, Section 6.1
  - Data output in "fake" q file **q.turb**
- **DEBUG=2** – time step information
  - Field quantities: Δt, J,K,L, and overall CFL#
  - Data output in "fake" q file **q.time**
- **DEBUG=3** – flow solver residual information
  - Field quantities: flow solver residuals (right-hand side before time-step scaling)
  - Data output in "fake" q file **q.resid**
- **DEBUG=4** – solution adaption information
  - Field quantities: sensor function, coarsen/refine marker array
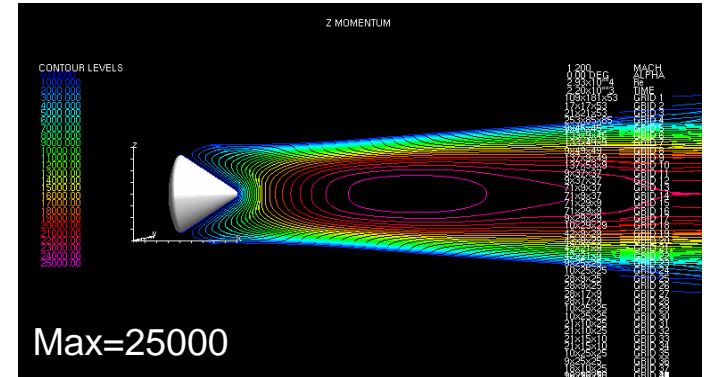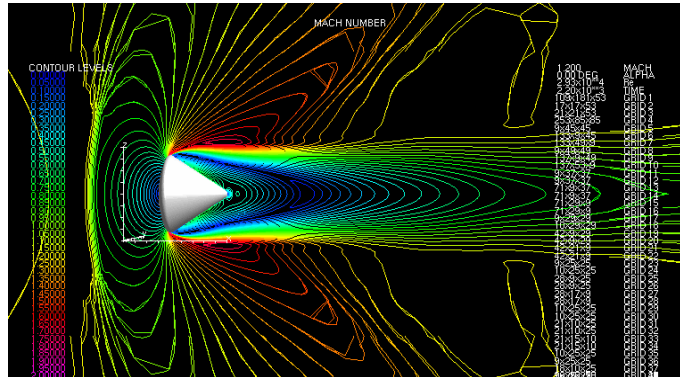 - Data output in "fake" q file **q.errest**
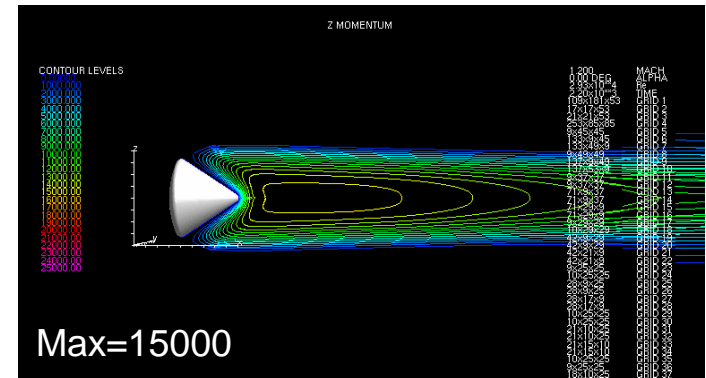
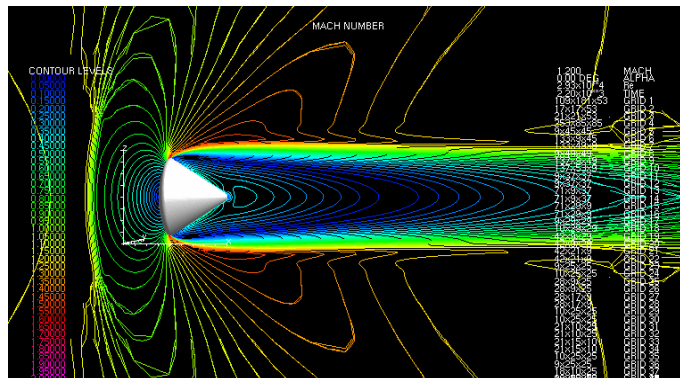# Apollo Static Aero, Mach 1.2
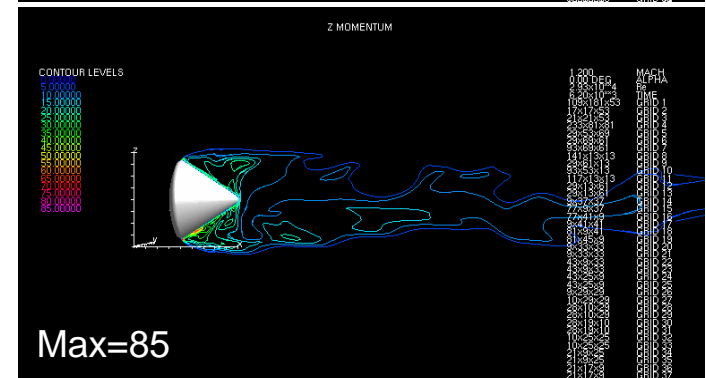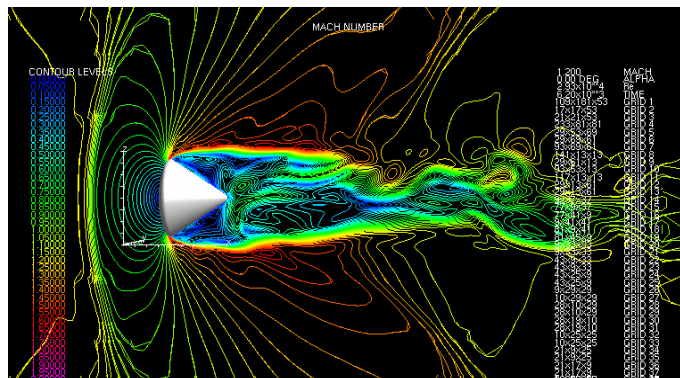
Turbulence model:

Mach number

Turbulent eddy viscosity

Spalart-Allmaras

Baldwin-Barth

Laminar outside boundary layer

Max=25000

Max=15000

Max=85

Test Cases Included With OVERFLOW

- Simple 2D cases (steady flow, single grid):
    - flat_plate, flat_plate_high_re
    - flat_plate_wf (tests wall function skin friction)
    - shear_layer
    - driven_cavity_2d (low-Mach preconditioning test case)
    - curved_wall_2d (tests turbulence model curvature corrections)
    - 3gas (simple multiple species convection case)
- Transonic 2D or axisymmetric cases (steady flow, single grid):
    - bump (axisymmetric bump, shock-induced separation)
    - naca, naca4412, naca_ogrid
    - et_axi, srb_axi
- Hypersonic 2D cases (steady flow, single grid):
    - cylinder, cyl_holden (2D Mach 8,16 flow)

10/6/2014

Test Cases, Continued

- 2D multiple grids:
    - af3_96 (multi-element airfoil)
    - cascade
- 2D moving body cases:
    - airfoil_drop_2d
    - rotating_paddle_2d
    - pitching_airfoil_2d
- Propulsion cases:
    - nozzle (rocket nozzle inflow/outflow boundary conditions)
    - eggers, seiner (axisymmetric plume flows)
    - powered_nacelle (jet engine inflow/outflow boundary conditions)
    - normal_jet_2d (simple jet-in-crossflow)
- Classical time-accurate cases:
    - shock_tube
    - vortex_convection, vortex_convection_HiO, lambVortex_convection
    - stokes_1st_problem (impulsively started plate)
    - oscillating_sphere (acoustic test case)

Test Cases, Continued

- Subsonic/transonic 3D (steady, single grid):
  - m2129_s_duct (S-duct inlet)
  - rotating_disk (infinite rotating plate)
  - onera_m6 (classic transonic wing test cases)
  - inf_swept (infinite swept wing)
  - ogive_cylinder
- Subsonic/transonic 3D (steady, multiple grid):
  - wingbody (AGARD test case)
  - bizjet (assembling and running a wing/body/pylon/nacelle)
  - robin_sym (helicopter fuselage, illustrates some numerical problems)
- Off-body grid adaption cases:
  - airfoil_adapt (new)
  - normal_jet_adapt (simple jet-in-crossflow) (new)

10/6/2014

Questions?