OVERFLOW 2 Training Class

Morning Session

Robert H. Nichols
University of Alabama at Birmingham
CREATE-AV API Principal Developer
robert.nichols@arnold.af.mil

Pieter G. Buning
NASA Langley Research Center
pieter.g.buning@nasa.gov

10th Symposium on Overset Composite Grids
& Solution Technology
NASA Ames Research Center
September 20-23, 2010

Class Outline - Morning

- OVERFLOW 2.2 capabilities
- CFD nomenclature overview
- Running in OVERFLOW mode
- NAMELIST Input
  - Inviscid fluxes
  - Implicit solvers
  - Boundary conditions
  - Species equations
  - Turbulence models
  - Unsteady flow outputs

Capabilities

Capabilities

- Implicit time marching solution algorithms
- Node-centered structured mesh
- OVERSET
- Gas models
  - Perfect gas
  - Variable gamma (currently not correct)
  - Low Mach number preconditioning (not all flux algorithms)
  - Multi-species (non-reacting)
- Moving body
  - Grid assembly (DCF)
  - Prescribed motion (GMP)
  - 6dof (GMP or internal or user specified)
  - Force integration (FOMOCO or USURP)
  - Collision modeling
  - Automatic off-body Cartesian grid generation (DCF)
  - Off-body grid refinement based on body motion or flow field
- Parallel performance enhancement
  - Parallel with MPI and/or OPENMP
  - Auto grid decomposition for load balance

Capabilities (cont.)

- Turbulence models
  - Baldwin-Lomax
  - Baldwin-Barth
  - Spalart-Allmaras (SA-DES, SA-DDES, SARC,ASARC)
  - k-ω
  - SST (SST-DES, SST-DDES, SST-MS, SSTRC,ASSTRC)
- Boundary conditions
  - Slip and no-slip wall
  - Constant temperature wall
  - Topology bc's (overlap, slit, polar axis)
  - Characteristic inflow/outflow
  - Nozzle inflow
  - Actuator Disk
  - Mass flow
  - Wall functions
  - And much, much more

Convergence Acceleration Methods

- Newton subiteration
- Dual time-stepping subiteration
- Multigrid
- Grid sequencing
- Local time step
- dq Limiter

CFD Nomenclature Overview

Navier-Stokes Equations

Differential form:

$$\frac{\partial \vec{q}}{\partial t} + \frac{\partial \vec{E}}{\partial \xi} + \frac{\partial \vec{F}}{\partial \eta} + \frac{\partial \vec{G}}{\partial \zeta} = 0$$

Conserved variables:

$$\vec{q} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho\left(e + 1/2(u^2 + v^2 + w^2)\right) \end{bmatrix}$$

Implicit Discrete Unfactored Form (1st or 2nd Order Time):

LHS

$$\boxed{\left[I+\frac{\Delta t}{1+\theta}\left(\delta_\xi A+\delta_\eta B+\delta_\zeta C\right)\right]}\Delta q^{n+1}=$$

$$\boxed{\left[\frac{\theta}{1+\theta}\Delta q^n-\frac{\Delta t}{1+\theta}RHS^n\right]}$$

RHS

Inviscid and viscous flux terms

$$A=\frac{\partial\vec{E}}{\partial\vec{q}},B=\frac{\partial\vec{F}}{\partial\vec{q}},C=\frac{\partial\vec{G}}{\partial\vec{q}}\qquad RHS^n=\frac{\partial\vec{E}^n}{\partial\xi}+\frac{\partial\vec{F}^n}{\partial\eta}+\frac{\partial\vec{G}^n}{\partial\zeta}$$

LHS Approximations

ADI Factorization (block tridiagonal matrix system):

$$\boxed{\left[I+\frac{\Delta t}{1+\theta}\partial_\xi A\right]}\boxed{\left[I+\frac{\Delta t}{1+\theta}\partial_\eta B\right]}\boxed{\left[I+\frac{\Delta t}{1+\theta}\delta_\zeta C\right]}\Delta q^{n+1}=$$
$$\boxed{\left[\frac{\theta}{1+\theta}\Delta q^n-\frac{\Delta t}{1+\theta}RHS^n\right]+Error}$$

Diagonalized Scheme (scalar pentadiagonal matrix scheme):

$$\boxed{X_\xi\left[I+\frac{\Delta t}{1+\theta}\partial_\xi \Lambda_\xi\right]X_\xi^{-1}}\boxed{X_\eta\left[I+\frac{\Delta t}{1+\theta}\partial_\eta \Lambda_\eta\right]X_\eta^{-1}}\boxed{X_\zeta\left[I+\frac{\Delta t}{1+\theta}\delta_\zeta \Lambda_\zeta\right]X_\zeta^{-1}}\Delta q^{n+1}=$$
$$\boxed{\left[\frac{\theta}{1+\theta}\Delta q^n-\frac{\Delta t}{1+\theta}RHS^n\right]+Error}$$

X_ξ = Eigenvector of A $\qquad$ Λ_ξ = Eigenvalues of A

Factorization Error:

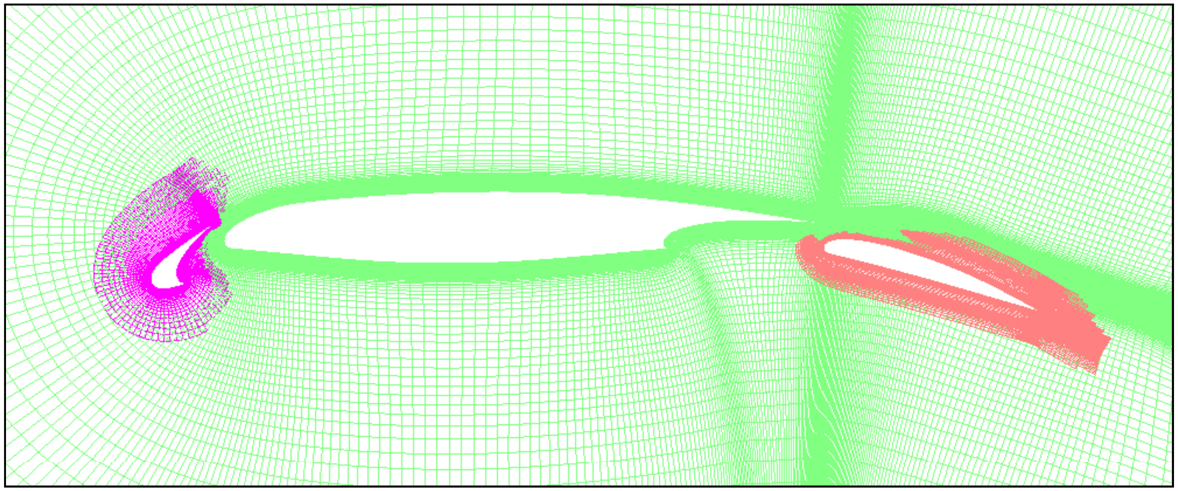
$$Error=\left[\boxed{\left(\frac{\Delta t}{1+\theta}\right)^2}\left(\delta_\xi A\delta_\eta B+\partial_\xi A\delta_\zeta C+\delta_\eta B\delta_\zeta C\right)+\boxed{\left(\frac{\Delta t}{1+\theta}\right)^3}\left(\partial_\xi A\delta_\eta B\delta_\zeta C\right)\right]\Delta q^{n+1}$$

LHS Approximation Summary

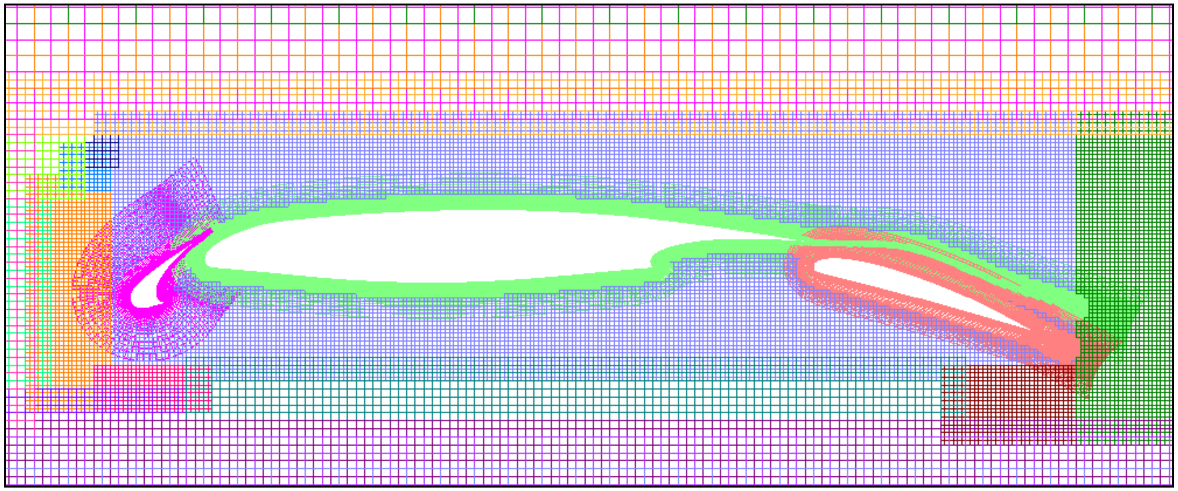
- Unfactored SSOR
  - More memory (store entire Jacobian matrix)
  - Relaxation method to invert matrix
    - Slower
    - Most stable
  - No factorization error
- ADI block tridiagonal
  - Fast (fits well in cache)
  - Small memory (solve 1 direction at a time)
  - Factorization error can cause instability for large time steps
- ADI diagonalized
  - Extremely fast
  - Smallest memory (solve 1 direction at a time)
  - Least stable
  - Factorization error can cause instability for large time steps

Running in OVERFLOW Mode

Overset Methodology



External grid assembly using PEGASUS 5



Internal grid assembly and Cartesian blocks generated using DCF

Required Input Files

- Single grid solution
  - **grid.in** plot3d grid file (single or multi grid format)
  - **over.namelist** namelist input file
  - Optional initial solution file **q.restart**
  - Optional force and moment files **mixsur.fmp**, **grid.ibi**, and **grid.ptv** (fomoco) or **panels_weight.dat** (usurp)

- Multiple grid solution
  - **grid.in** plot3d grid file (multi grid format)
  - **over.namelist** namelist input file
  - **INTOUT** or **XINTOUT** overset communication file
  - Optional initial solution file **q.restart**
  - Optional force and moment files **mixsur.fmp**, **grid.ibi**, and **grid.ptv** (fomoco) or **panels_weight.dat** (usurp)

Restart and Solution Files

- Normally solution written to **q.save** plot3d format file
  - **q.save** file **is overwritten** with latest output
- Option to write solution to **q.<istep>** plot3d format files
- Solution file for restart is named **q.restart**
  - Restart file now contains information needed for true 2^{nd} order time restart if running 2^{nd} order time
  - Restart file **q.restart** is not automatically generated on restart

Initializing Solution Files

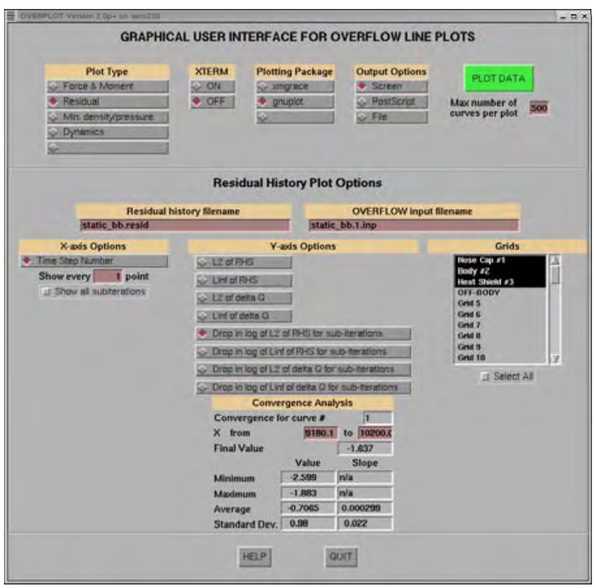
- Code will initialize to free stream input conditions in NAMELIST (initial **q.restart** not required)
- Users may write their on **q.restart** file
- Code will scale restart file when NAMELIST input values are different from those in the **q.restart** file (M_∞, α, β)
  – Allows start from existing solution file
  – Allows high speed solutions to be started from lower speed initial file

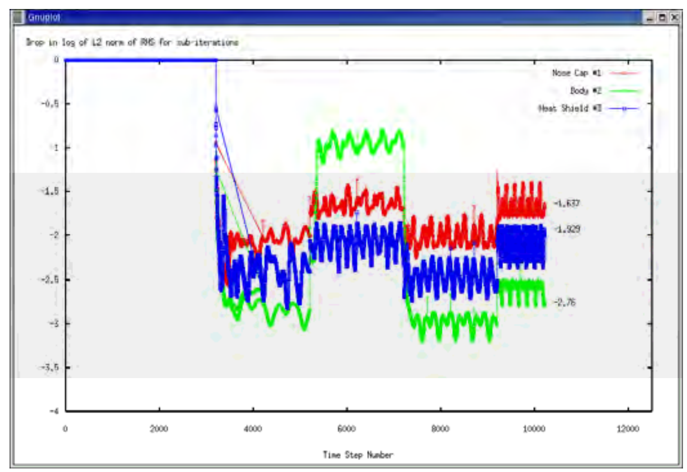
Running the Code

- Serial code
  - **./overflow**
- MPI code
  - **mpirun –np <ncpus> ./overflowmpi**
- Serial run script (Input file **basename.inp** or **basename.n.inp**)
  - **overrun basename n**
- MPI run script
  - **overrunmpi –np <ncpus> -machinefile <hostfile> basename n**
- Run scripts perform the following tasks
  - Move ***.save** output files to ***.restart** input files
  - Highlight warnings and errors
  - Creates a log file with time/date, machine name, executable name, and NAMELIST input file name
  - Concatenates output history files upon completion

Output Files

- Solution files
  - **q.save** or **q.<istep>**
  - **q.avg** time averaged solution file
- History files (view with **overplot**)
  - **resid.out** - residual history file
  - **rpmin.out** – min ρ, min p, γ, number of reverse flow points, number of supersonic points, and max μ_t
  - **turb.out** - residual history file for turbulence equations
  - **species.out** - residual history file for species equations
  - **fomoco.out** – force and moment history file
  - **timers.out** – timing information for run





Standard Out File

Compile information:

```
OVERFLOW/OVERFLOW-D  --  OVERLAPPED GRID FLOW SOLVER
   ( MPI VERSION )
         VERSION 2.1s   8 July 2008


   Compiled for SINGLE PRECISION


Compile time: Tue Sep 16 14:07:40 CDT 2008

Code was compiled with the following:
F90     = mpif77
F90FLAGS= -Mnoopenmp -fastsse -Ktrap=fp
CC      = mpicc
CFLAGS  = -fastsse -Ktrap=fp
CPP     = /lib/cpp -traditional
CPPFLAGS= -DUSE_MPI -DNOCPU_TIME

Current time: Sep 19 01:33:54 2008
```

Standard Out File (cont.)

NAMELIST Input Summary with checks:

```
** ASSUMING THIS IS AN OVERFLOW-D RUN.

THIS IS A PARALLEL RUN WITH    16 GROUPS.

GLOBAL PARAMETERS                         ($GLOBAL)
  USE grdwghts.dat FOR LOAD BALANCING?   (GRDWTS) = F
  MAXIMUM GRID SIZE              (MAX_GRID_SIZE) =           0
  RUNNING CDISC INVERSE DESIGN?         (CDISC ) = F
  SUPPRESS WRITING q.bomb FILE?         (NOBOMB) = F
  CONSERVE MEMORY?                 (CONSERVE_MEM) = F
  DEBUG OPTION (0-3)                    (DEBUG ) =     0
  NUMBER OF STEPS                       (NSTEPS) =       12000
  READ RESTART FILE?                    (RESTRT) = T
  SAVE RESTART FILE EVERY               (NSAVE )           50 STEPS
  2ND-ORDER Q RESTART OPTION       (SAVE_HIORDER) =           2
  START Q AVERAGING AT STEP        (ISTART_QAVG) =        2300
  COMPUTE FORCE/MOMENT COEFS EVERY      (NFOMO )            10 STEPS
  TURBULENCE MODEL TYPE                 (NQT)    =   205
  NUMBER OF SPECIES                     (NQC)    =     0
  USE MULTIGRID?                        (MULTIG) = F
  USE FULL MULTIGRID?                   (FMG)    = F
  NO. OF GRID LEVELS (IF MULTIG=.T.)    (NGLVL)  =     3
  NO. OF FMG CYCLES (IF FMG=.T.)        (FMGCYC) =     0    0
```

Standard Out File (cont.)

Grid Summary with checks:

```
GRID SIZE FOR GRID     1:

  NUMBER OF POINTS IN J  (JD    ) =   201
                      K  (KD    ) =   111
                      L  (LD    ) =    51

CHECKING TIME STEP SPECIFICATION FOR GRID     1:

  RUNNING TIME-ACCURATE WITH NEWTON SUBITERATIONS
    WITH DTPHYS (BASED ON V_REF) =  0.20000
                (BASED ON C_INF) =  0.21053

CHECKING BOUNDARY CONDITIONS FOR GRID     1:

    1)  BOUNDARY CONDITION TYPE#     1     DIRECTION     3
     INVISCID ADIABATIC SOLID WALL (PRESSURE EXTRAPOLATION)
     DIR= 3    J-RANGE=    1   10    K-RANGE=    1  111    L-RANGE=    1    1
    2)  BOUNDARY CONDITION TYPE#     5     DIRECTION     3
     VISCOUS ADIABATIC SOLID WALL  (PRESSURE EXTRAPOLATION)
     DIR= 3    J-RANGE=   11   41    K-RANGE=    1  111    L-RANGE=    1    1
```

Standard Out File (cont.)

Grid Splitting Summary:

```
NEAR-BODY/OFF-BODY GRID LEVEL SUMMARY:

  Level     #Grids    First     Last
 ------------------------------------
 near-body       2        1        2

................ START GROUPR ....................


 Target (weighted) near-body grid size from grouping:          51179
 Checking near-body grids...
 Original number of near-body grids:     2
   Splitting grid     1 at J =   101
   Splitting grid     1 at K =    56
   Splitting grid     1 at J =    52
   Splitting grid     1 at K =    30
   Splitting grid     1 at J =    28
   Splitting grid     2 at J =    61
   Splitting grid     2 at L =    41
   Splitting grid     2 at J =    32
   Splitting grid     2 at K =    26
   Splitting grid     3 at K =    56
```

Standard Out File (cont.)

Load Balance Summary by Groups:

Load balance will be based on grid size.

Summary of work distribution for 16 groups:

Group	Kpts	%load	Grid list		
1	151	100	20	13	11
2	151	100	35	12	2
3	151	100	32	3	8
4	151	100	37	16	22
5	151	100	43	17	26
6	151	100	46	19	39
7	151	100	45	36	24
8	151	100	48	18	9
9	150	100	1	34	47
10	150	100	29	33	40
11	150	100	28	7	41
12	150	100	5	4	38
13	150	100	42	21	27
14	150	100	30	44	23
15	150	100	31	15	10
16	150	100	6	14	25

Predicted parallel efficiency is 100%,

based on a maximum of 151K grid points per group
compared to an average of 150K points (weighted)

Estimated parallel speedup is 16.0

Standard Out File (cont.)

Residual Summary:

```
     FOR GRID     1  AT STEP      12030   L2NORM =  0.80239815E-05
 FLOW SOLVER/TURB MODEL/SPECIES CONVERGENCE RATES:
0.9995    1.0214    0.0000
     FOR GRID     2  AT STEP      12030   L2NORM =  0.92307528E-05
 FLOW SOLVER/TURB MODEL/SPECIES CONVERGENCE RATES:
1.0041    0.9737    0.0000
     FOR GRID     1  AT STEP      12040   L2NORM =  0.76590641E-05
 FLOW SOLVER/TURB MODEL/SPECIES CONVERGENCE RATES:
0.9954    1.0133    0.0000
     FOR GRID     2  AT STEP      12040   L2NORM =  0.91043430E-05
 FLOW SOLVER/TURB MODEL/SPECIES CONVERGENCE RATES:
0.9986    1.0114    0.0000
     FOR GRID     1  AT STEP      12050   L2NORM =  0.78810453E-05
 FLOW SOLVER/TURB MODEL/SPECIES CONVERGENCE RATES:
1.0029    0.9666    0.0000
     FOR GRID     2  AT STEP      12050   L2NORM =  0.85000529E-05
 FLOW SOLVER/TURB MODEL/SPECIES CONVERGENCE RATES:
0.9932    1.0180    0.0000
   Wrote 2nd-order restart file -- q.save
   Elapsed simulation time (based on V_ref):   0.2410016E+04
   Wrote restart file -- q.avg
   Elapsed simulation time (based on V_ref):   0.2410016E+04
   q.avg data collected over       9751 steps
```

Standard Out File (cont.)

Timer Summary on Completion of Run:

```
----------------------------------------------
TIMED FOR    12000 LEVEL-1 STEPS
----------------------------------------------


----------------------------------------------
BREAKDOWN FOR TOTAL      (TIMER NUMBER   1)
----------------------------------------------
  N    TIMER          %          TIME
 --- ----------    ---------    ------------
  1  TOTAL         100.00      1.8575E+05
  2  OVERGL          0.00      2.4827E-03
  3  OVERSZ          0.01      1.0730E+01
  4  OVERST          0.00      3.2468E+00
  5  OVERFL         99.99      1.8573E+05
  6  OVERDO          0.00      1.4612E+00
  7  OTHER           0.00      7.1377E-06
  8  test            0.00      0.0000E+00
  9  test            0.00      0.0000E+00

----------------------------------------------
BREAKDOWN FOR OVERFL     (TIMER NUMBER   5)
----------------------------------------------
  N    TIMER          %       TIME/STEP        MAX/STEP
 --- ----------    ---------    -----------       -----------
  5  OVERFL        100.00      1.5478E+01       1.5478E+01
 11  CBCXCH          8.45      1.3080E+00       1.4018E+00
 12  FLOW_SOLVE     87.85      1.3597E+01       1.3706E+01
 13  FOMOCO          0.00      0.0000E+00       0.0000E+00
 14  FLOW_Idle       2.08      3.2176E-01       4.7814E-01
 15  SAVE            1.42      2.1947E-01       2.5384E-01
 16  SIXDOF          0.00      0.0000E+00       0.0000E+00
```

Standard Out File (cont.)

Group Timing Summary on Completion of Job:

```
GROUP TIMING SUMMARY   (Time each group spent in OVERFL)
 (*) Flow Solver, (/) Chimera BC, (a) Adapt, (D) DCFCRT, (s) Grid & Q save


            0          25           50           75          100
            |-----------|------------|------------|-----------|
Group:   1  |*********************************************////s   98%
Group:   2  |*********************************************////    98%
Group:   3  |********************************************////     97%
Group:   4  |********************************************////     97%
Group:   5  |********************************************////s    97%
Group:   6  |********************************************////     97%
Group:   7  |*********************************************////    97%
Group:   8  |********************************************////     98%
Group:   9  |*********************************************////    98%
Group:  10  |*********************************************////    98%
Group:  11  |*********************************************////    98%
Group:  12  |*********************************************////    99%
Group:  13  |*********************************************////    98%
Group:  14  |*********************************************////    99%
Group:  15  |*********************************************////    99%
Group:  16  |*********************************************////    99%

 Overall Measured Parallel Efficiency:  97.9%
------------------------------------------------------------------------


 Current time: Sep 21 05:09:40 2008
```

Debug Options

- Turbulence model diagnostics (**DEBUG**=1)
  - Baldwin Lomax **q.turb** file

Q Value	*Q1*	*Q2*	*Q3*	*Q4*	*Q5*
(J,K,LS,_)	F_{max}	y^+	$\lvert\Omega\rvert$	μ_w	-
(J,K,L,_)	$F(y)$	y	$\lvert\Omega\rvert$	μ_t	F_{max} *location*

  - Spalart Allmaras **q.turb** file

Q Value	*Q1*	*Q2*	*Q3*	*Q4*	*Q5*
(J,K,LS,_)	f_{v1}	y^+	$\lvert\Omega\rvert$	μ_t	*Turbulence index*
(J,K,L,_)	f_{v1}	y	$\lvert\Omega\rvert$	μ_t	*Transition factor*

  - SST **q.turb** file

Q Value	*Q1*	*Q2*	*Q3*	*Q4*	*Q5*
(J,K,LS,_)	ω	y^+	F_1	μ_t	k
(J,K,L,_)	ω	y	F_1	μ_t	k

- Time step **q.time** diagnostics file (**DEBUG**=2)

Q Value	*Q1*	*Q2*	*Q3*	*Q4*	*Q5*
(J,K,L,_)	Δt	CFL_j	CFL_k	CFL_l	CFL_{max}

- Residual **q.resid** diagnostics file (**DEBUG**=3)

Q Value	*Q1*	*Q2*	*Q3*	*Q4*	*Q5*
(J,K,L,_)	r_{Q1}	r_{Q2}	r_{Q3}	r_{Q4}	r_{Q5}

Namelist Inputs

NAMELIST Input

&GLOBAL /	&METPRM
&FLOINP /	&TIMACU /
&VARGAM /	&SMOACU /
&OMIGLIB /	&VISINP /
&DCFGLIB /	&BCINP /
&GBRICK /	&SCEINP /
&BRKINP /	&ADSNML /
&GROUPS /	&SPLITM /
&XRINFO /	&CDNTNS /
&GRDNAM /	&EROTOR /
&NITERS /	&SIXINP /

Color Codes:

Required once per run.

Required once per run for OVERFLOW-D mode. May be omitted if not using bricks and DCF.

Required for every grid.

Optional output control. Once per run.

Optional rotorcraft coupling input.

Required for every grid for moving body runs using internal 6DOF. May be omitted for static grid cases, GMP, or prescribed motion problems.

NAMELIST Inputs

- NAMELIST must follow standard format specifications
  - Old format $NAME $END
  - New format &NAME /
- All NAMELIST values have defaults
- Input values can be inherited from the previous grid
- Order of grids in NAMELIST must correspond to order of grids in **grid.in!!!**

Namelist Input for Euler 2D Wing

```
&GLOBAL  NSTEPS = 500,
/
&FLOINP  ALPHA  = 2.0, FSMACH = 0.8,
/

&GRDNAM  NAME   = ‘WING’,  /
&NITERS /
&METPRM /
&TIMACU /
&SMOACU /
&VISINP /
&BCINP
       IBTYP  =  1, 47, 10, 21,
       IBDIR  =  2, -2,  1,  3,
       JBCS   =  1,  1,  1,  1,
       JBCE   = -1, -1,  1, -1,
       KBCS   =  1, -1,  1,  1,
       KBCE   =  1, -1, -1, -1,
       LBCS   =  1,  1,  1,  1,
       LBCE   = -1, -1, -1,  1,
/
&SCEINP /
```

Inheriting Defaults in Namelist

```
&GRDNAM  NAME   = ‘Grid1’,  /
&NITERS /
&METPRM IRHS = 5, /
&TIMACU /
&SMOACU /
&VISINP /
&BCINP
…
/
&SCEINP /

&GRDNAM  NAME   = ‘Grid2’,  /
&NITERS /
&METPRM /
&TIMACU /
&SMOACU /
&VISINP /
&BCINP
…
/
&SCEINP /
```

Grid1 sets IRHS=5

Grid2 uses IRHS=5

```
&GRDNAM  NAME   = ‘Grid3’,  /
&NITERS /
&METPRM IRHS = 4, /
&TIMACU /
&SMOACU /
&VISINP /
&BCINP
…
/
&SCEINP /

&GRDNAM  NAME   = ‘Grid4’,  /
&NITERS /
&METPRM /
&TIMACU /
&SMOACU /
&VISINP /
&BCINP
…
/
&SCEINP /
```

Grid3 sets IRHS=4

Grid4 uses IRHS=4

Inviscid Flux Algorithms

RHS Options

- Central Difference (2^{nd} – 6^{th})(IRHS=0)
- Symmetric Yee (2^{nd})(IRHS=2)
- Upwind AUSM^{+} (3^{rd} – 5^{th}) (IRHS=3)
- Upwind Roe (3^{rd} – 5^{th}) (IRHS=4)
- Upwind HLLC (3^{rd} – 5^{th}) (IRHS=5)
- Upwind HLLE++ (3^{rd} – 5^{th}) (IRHS=6)
- Low Mach preconditioning for 2^{nd} order central, HLLC, and Roe

Central Difference Scheme (2nd)

- Smoothing required because of odd-even decoupling
  - 4th order smoothing away from shocks (DIS4)
  - 2nd order smoothing near shocks (DIS2)
- Smoothing options:
  - F3D dissipation scheme (IDISS=1)
  - ARC3D dissipation scheme (IDISS=2)
  - TLNS3D dissipation scheme (IDISS=3)
  - Matrix dissipation scheme (IDISS=4)

```
&METPRM
        IRHS = 0, IDISS=3,
/
&SMOACU
        DIS2 = 2.0, DIS4 = 0.04, FSO = 2.0,
/
```

Roe Scheme

- Up to 5th order upwind in space (FSO)
- Flux limiter options:
  - Koren (ILIMIT=1)
  - Minmod (ILIMIT=2)
  - Van Albada (ILIMIT=3)
  - WENO
- Limiter fix for carbuncles and strong shocks (DELTA)
- Preconditioned option

```
&METPRM
        IRHS = 4,
/
&SMOACU
        DELTA = 1.0, FSO = 3.0,
/
```

HLLC Scheme

- Up to 5th order upwind in space (FSO)
- Flux limiter options:
  - Koren (ILIMIT=1)
  - Minmod (ILIMIT=2)
  - Van Albada (ILIMIT=3)
  - WENO
- Limiter fix for strong shocks (DELTA)
- Preconditioned option

```
&METPRM
        IRHS = 5, ILIMIT = 1,
/
&SMOACU
        DELTA = 1.0, FSO = 3.0,
/
```

HLLE++ Scheme

- Up to 5th order upwind in space (FSO)
- Flux limiter options:
  - Koren (ILIMIT=1)
  - Minmod (ILIMIT=2)
  - Van Albada (ILIMIT=3)
  - WENO
- Limiter fix for strong shocks (DELTA)

```
&METPRM
        IRHS = 6, ILIMIT = 1,
/
&SMOACU
        DELTA = 1.0, FSO = 3.0,
/
```

High Order Upwind Schemes

- 5th order upwind in space (FSO=5.0)
- Based on WENO
- Use with AUSM, Roe, HLLC, or HLLE++
- Flux limiter options:
  - WENO (ILIMIT = anything but 4)
  - Mapped WENO (ILIMIT = 4)
- Limiter fix for strong shocks (DELTA)
- Requires triple fringe interpolation boundaries

```
&METPRM
        IRHS = 5, ILIMIT = 4,
/
&SMOACU
        DELTA = 1.0, FSO = 5.0,
/
```

High Order Central Schemes

- 4^{th} or 6^{th} order in space
- Use smoothing or filtering
- Smoothing controlled using DIS2 or DIS4
- Requires triple fringe interpolation boundaries

FSO = 2*	**2^{nd} order central with 4/2 dissipation**
FSO = 3*	**4^{th} order central with 4/2 dissipation**
FSO = 4	**4^{th} order central with 6/2 dissipation**
FSO = 5*	**6^{th} order central with 6/2 dissipation**
FSO = 6	**6^{th} order central with 8/2 dissipation**

Smoothing:

```
&METPRM
        IRHS = 0, /
&SMOACU
        DIS2 = 0.0, DIS4 = 0.005,
        FSO = 5.0, SMOO = 0, /
```

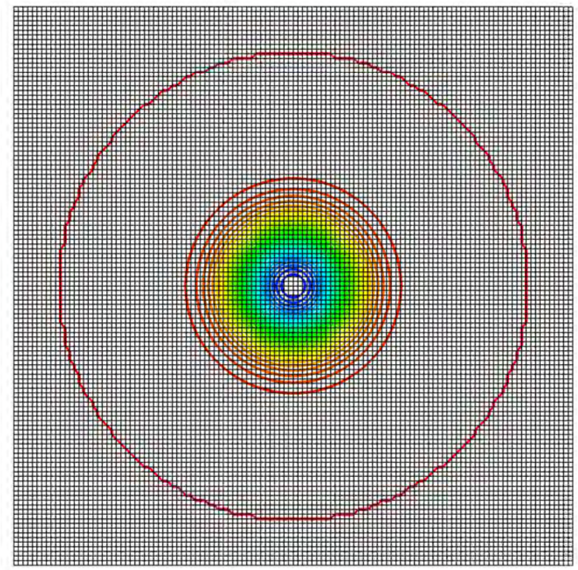
Filtering:

```
&METPRM
        IRHS = 0, /
&SMOACU
        DIS2 = 0.0, DIS4 = 0.005,
        FSO = 6.0, SMOO = 0,
        FILTER = 5,/
```

*Recommended

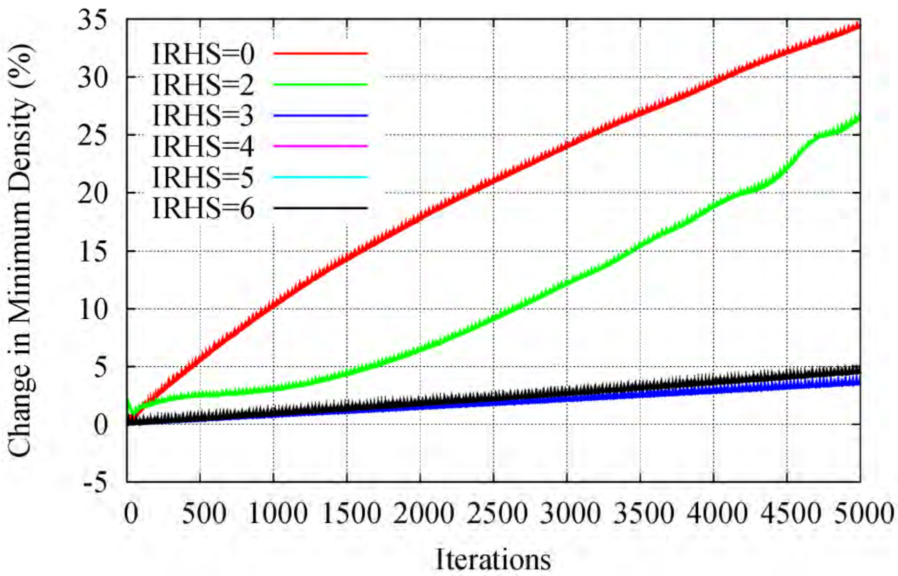
Isentropic Vortex Convection

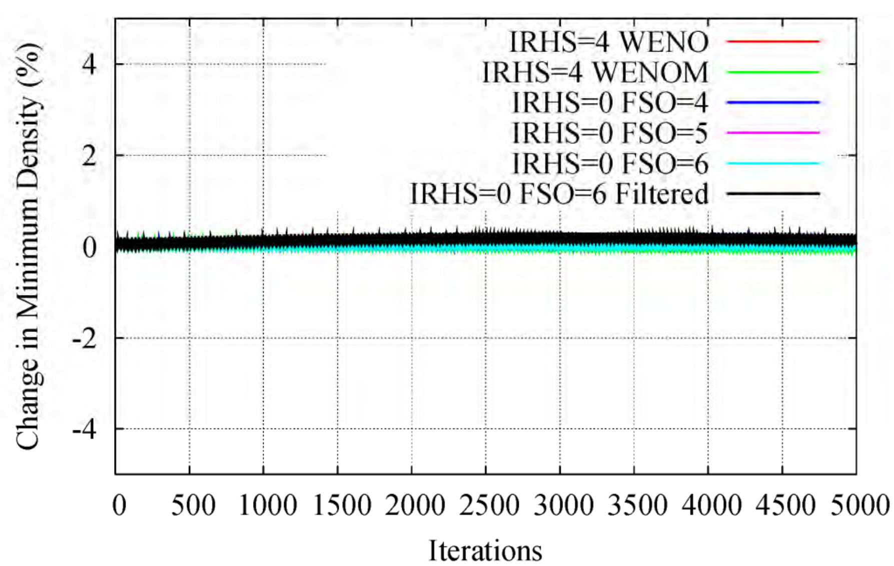
- Prescribed starting vortex
- Free stream Mach = 0.5
- Periodic BC's in flow direction
- DTPHYS=0.01
- 1000 time steps per grid cycle
- 2nd order time with 10 Newton subiterations



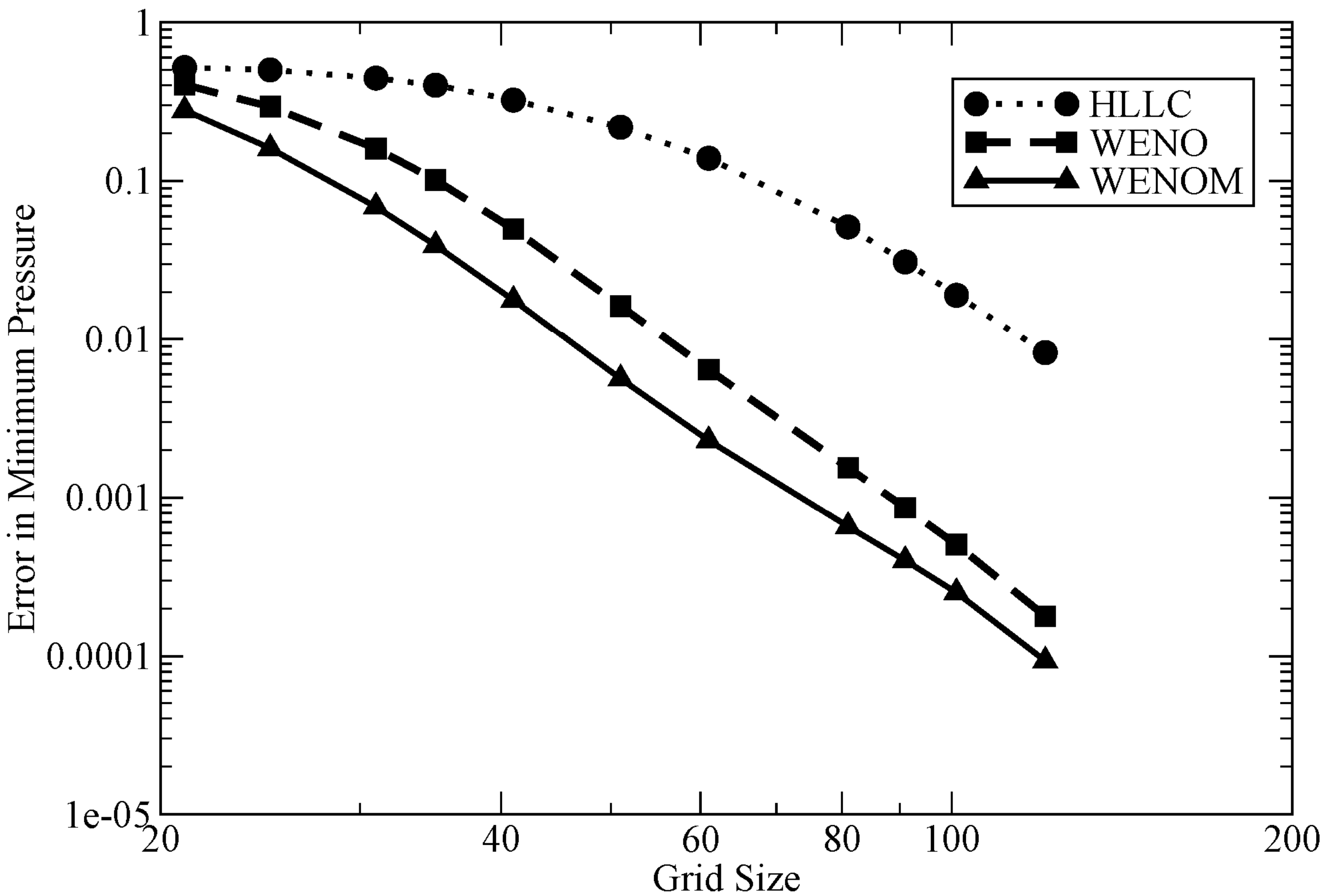
121x121 Grid

Flux Scheme Dissipation

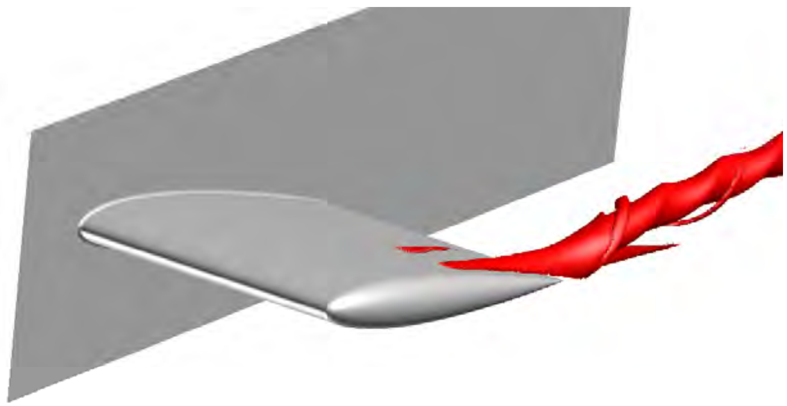


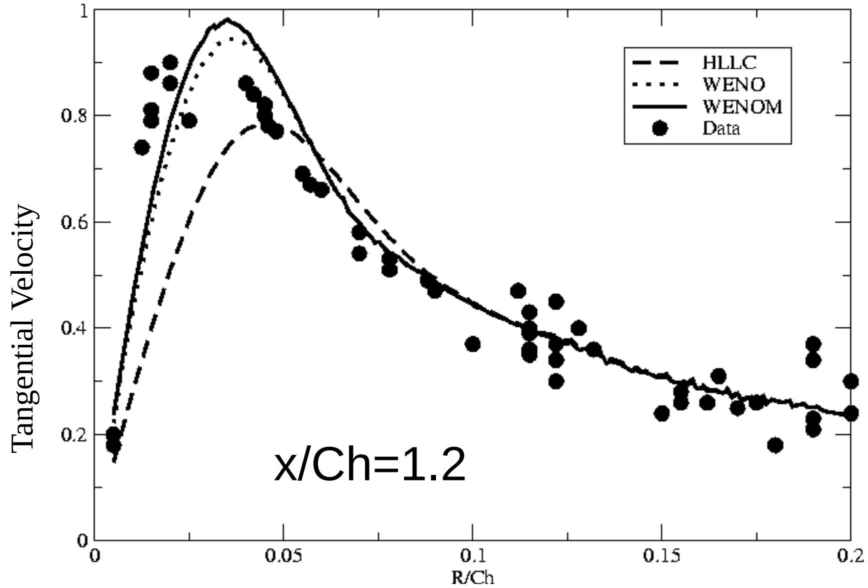


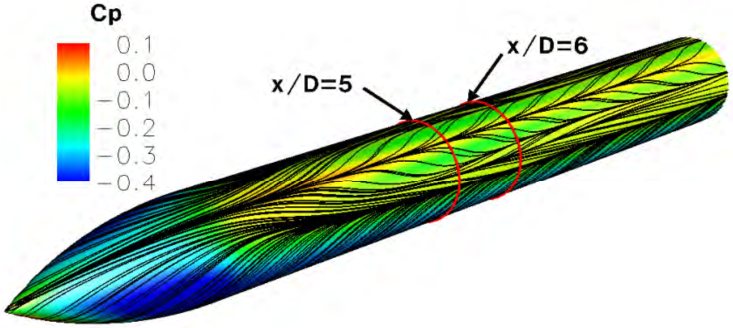
Grid Convergence

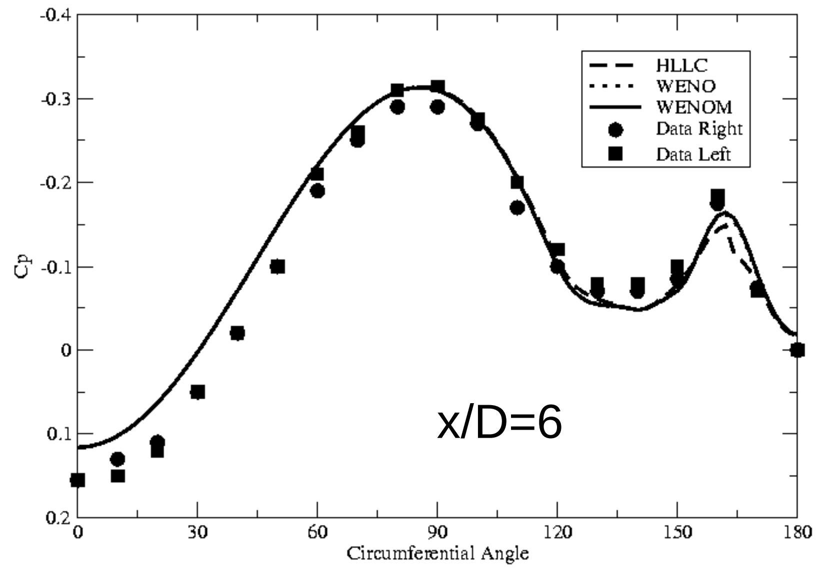


3D Vortex Preservation



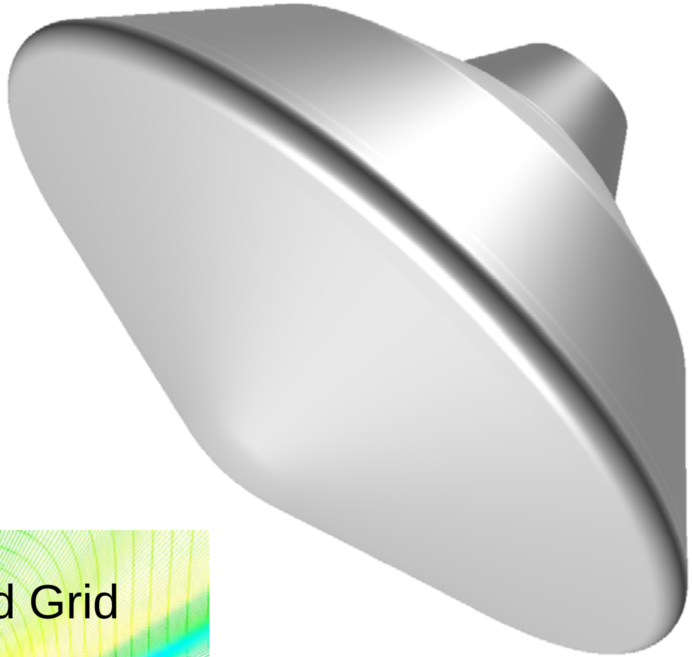


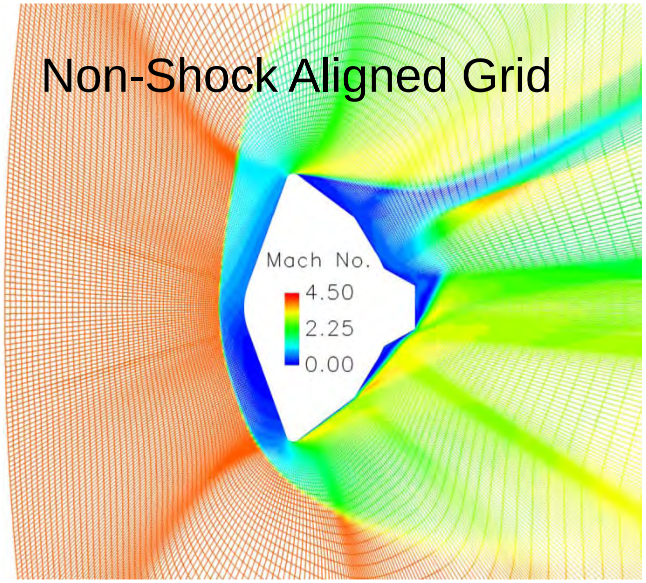




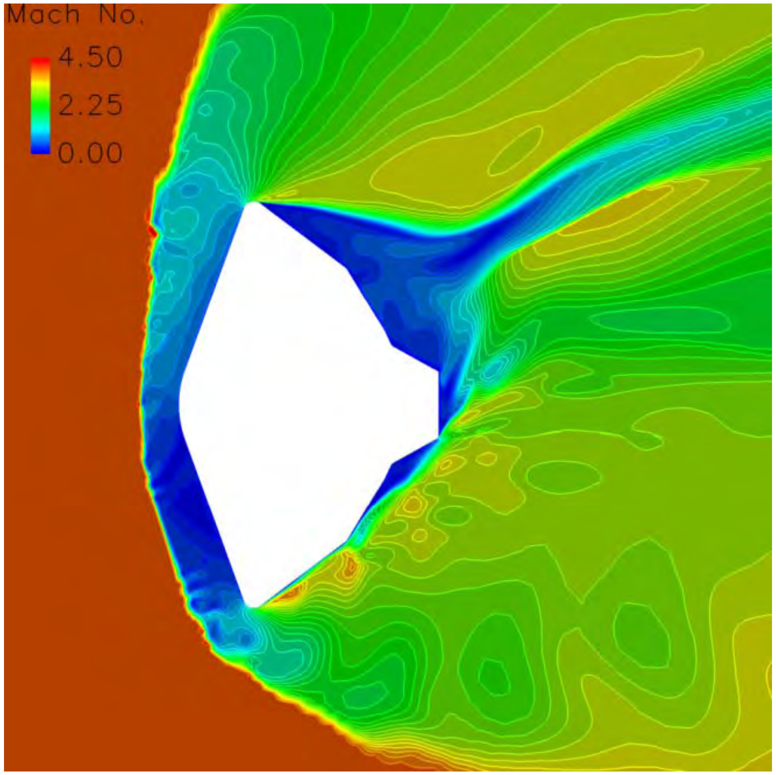
3D Capsule Test Case

- M_∞=4.02
- γ=1.246
- Re=2.49x10^5
- α=16^o
- SST Turbulence Model

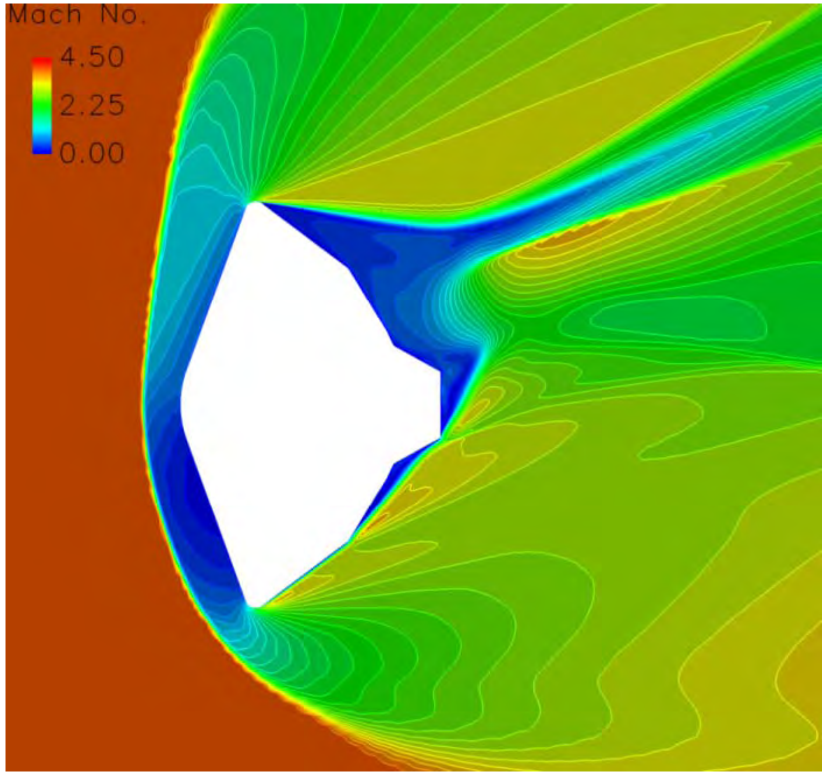




3D Capsule Symmetry Plane

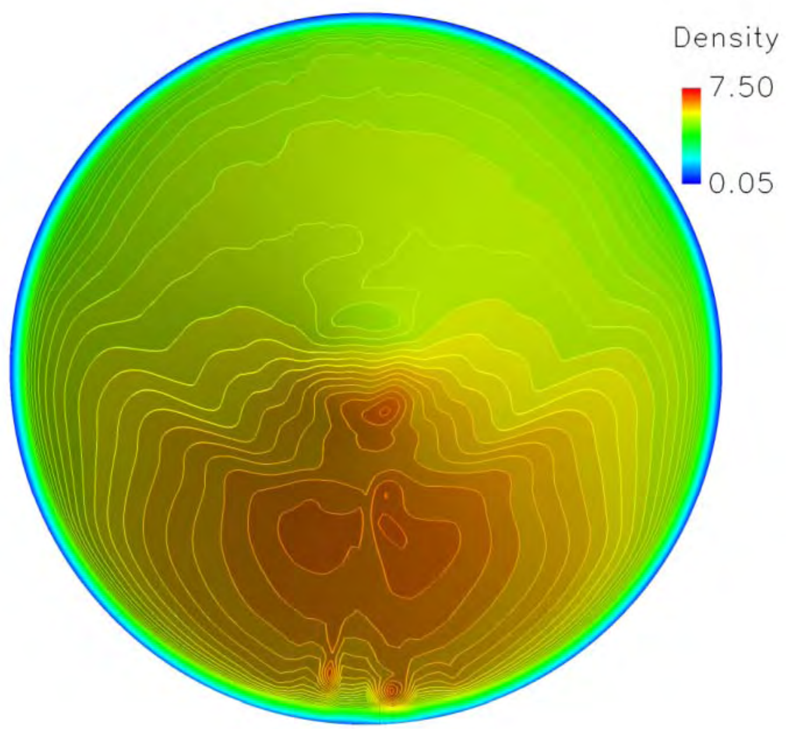


HLLC Delta=1

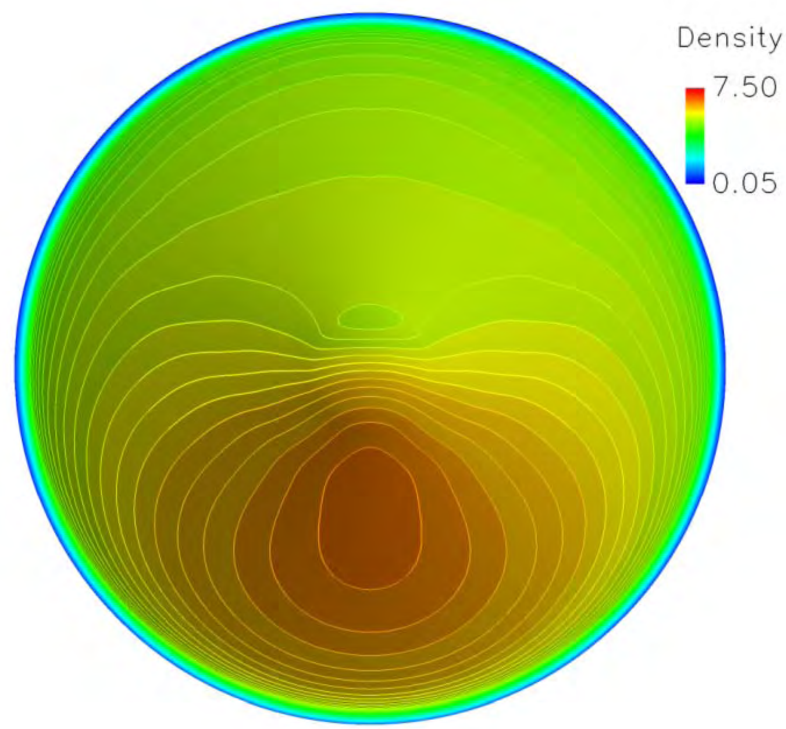


HLLE++ Delta=5

3D Capsule Windward Surface

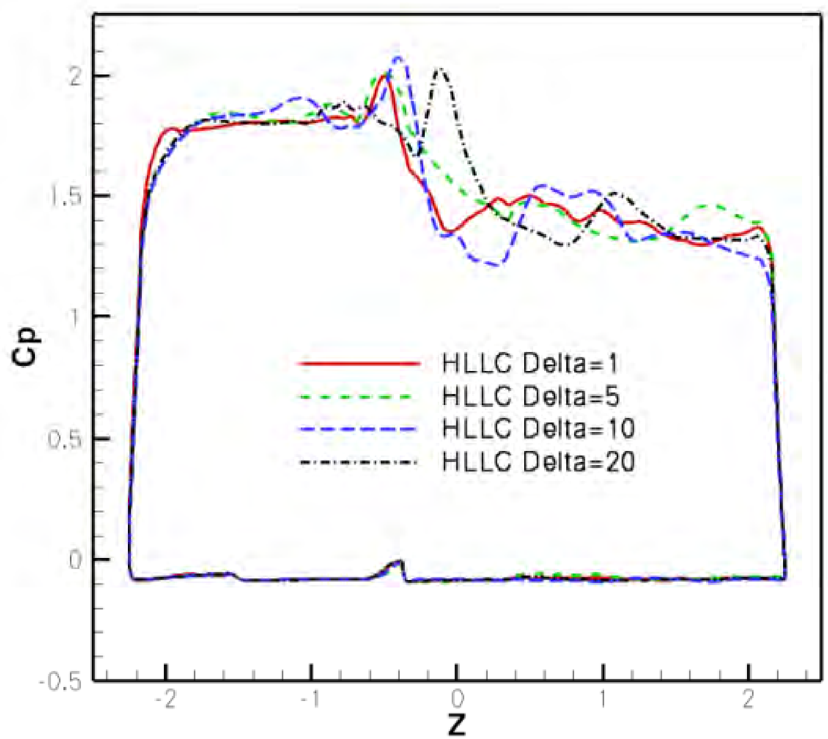


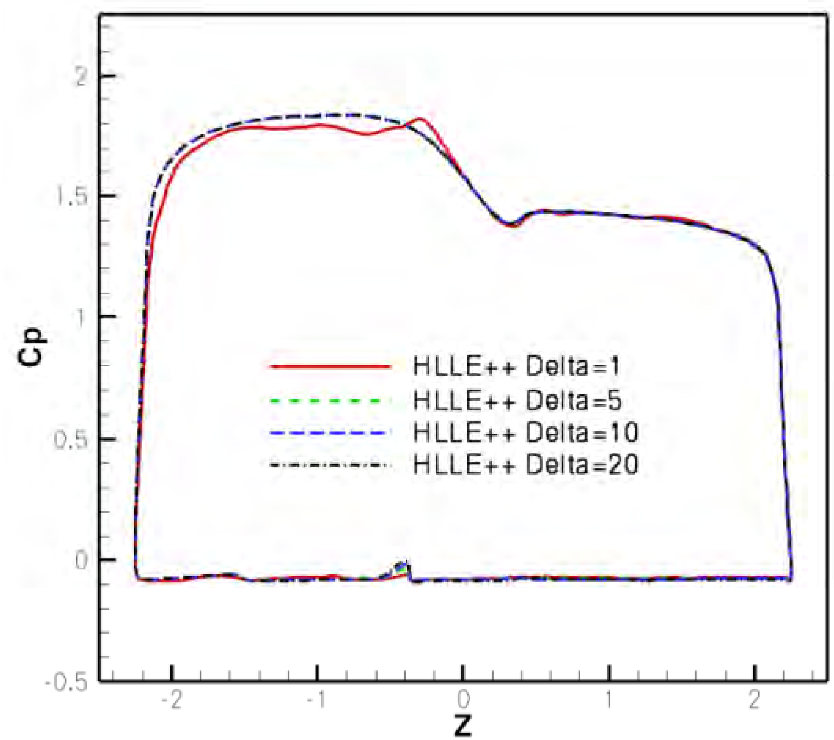
HLLC Delta=1



HLLE++ Delta=5

3D Capsule Symmetry Plane





Low Mach Number Preconditioning

Low Mach Number Preconditioning

- Eigenvalues of the inviscid fluxes (u,u+a,u-a) become stiff as u$\rightarrow$ 0
- Modify equation set with preconditioning matrix Γ_p to rescale eigenvalues
- Must also modify RHS fluxes to use scaled eigenvalues
- Must use dual time step for time accurate simulations

$$\boxed{\Gamma_p \frac{\partial \vec{q}_p}{\partial \tau}} + \frac{\partial \vec{q}}{\partial t} + \frac{\partial \vec{E}}{\partial x} + \frac{\partial \vec{F}}{\partial y} + \frac{\partial \vec{G}}{\partial z} = 0$$

Eigenvalues

Navier-Stokes:

$$\lambda = \begin{bmatrix} U \\ U \\ U \\ U+C \\ U-C \end{bmatrix}$$

Smith-Weiss Preconditioned:

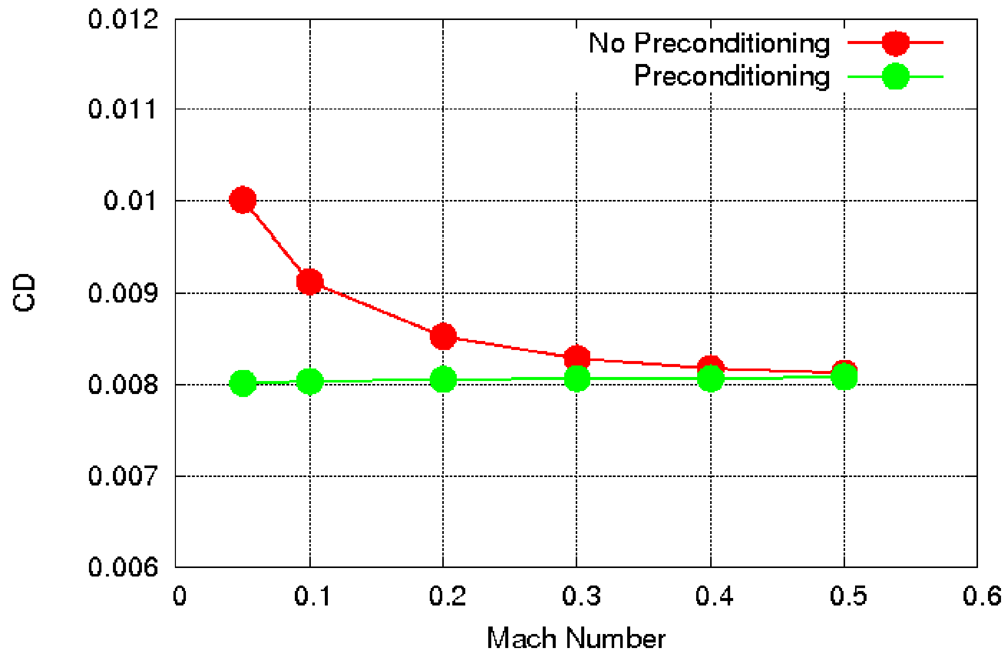
$$\beta_{\min} = 3M_{ref}^2$$

$$\beta = MAX\left[MIN\left(M^2,1\right),\beta_{\min}\right]$$

$$\lambda = \begin{bmatrix} U \\ U \\ U \\ 0.5U\left(\beta+1\right)-\sqrt{\left(0.5U\left(\beta-1\right)\right)^2+\beta C^2} \\ 0.5U\left(\beta+1\right)+\sqrt{\left(0.5U\left(\beta-1\right)\right)^2+\beta C^2} \end{bmatrix}$$

NACA 0012

- $Re_c=1.0x10^7$, $\alpha=0^o$
- IRHS = 5
- FSO = 3.0
- ILHS = 4
- Default BIMIN for preconditioned results



Flux Schemes Relative Timings*

2nd and 3rd Order Methods

Central IRHS=0	Yee IRHS=2	AUSM IRHS=3	Roe IRHS=4	HLLC IRHS=5	HLLE++ IRHS=6
1.00	1.68	1.91	1.38	1.58	1.71

High Order Methods

Central FSO=3	Central FSO=4	Central FSO=5	Central FSO=6	WENO	WENOM
1.04	1.06	1.15	1.19	2.66	3.63

*Sensitive to processor choice

RHS Hints

1. Default explicit smoothing (DIS4=0.04) is high for central algorithm for stability – accuracy can often be improved by reducing DIS4. Recommended range is $0.005 \leq DIS4 \leq 0.04$.
2. HLLE++ is the best choice for high speed flows with non-grid aligned grids.
3. DELTA can normally be set to 1.0 (default = no fix) for upwind algorithms. If help is needed, try setting DELTA ~ 2-10 for supersonic and hypersonic flows.
4. Preconditioning can improve accuracy of the code for low speed applications ($M<0.25$). Preconditioning destroys time accuracy – must use dual time stepping for time accurate applications. Sensitive to choice of BIMIN.
5. Riemann solvers are happy to have shock in the first cell off the wall. Two approaches to push off shock:
   a.) Run central difference with grid sequencing for a few steps and then switch to upwind.
   b.) Start solution with transonic Mach and increase free stream Mach to desired value.

Viscous Fluxes

Viscous Terms

- Option to include selected thin layer terms
- Option to include only the cross terms for selected thin layer terms
- Option to include all viscous terms (VISC=.TRUE.)
- Option to select wall functions (automatic)

```
&VISINP
        VISC = .TRUE., WALLFUN = .TRUE.,
/
```

Implicit Solvers

Implicit Solvers

- ADI block tridiagonal solver (Beam & Warming)
  - Central difference inviscid flux jacobians + 2nd order smoothing (ILHS=0)
  - Steger-Warming inviscid upwind flux jacobians (ILHS=5)
- F3D solver (ILHS=1)
- ADI Pulliam-Chaussee pentadiagonal solver (ILHS=2)
  - Central difference + 2nd/4th order smoothing
  - Preconditioned
- LU-SGS solver (ILHS=3)
- D3ADI diagonalized solver (ILHS=4)
  - Preconditioned
- NXAIR unfactored SSOR solver (ILHS=6,7)
  - Preconditioned

Implicit Solvers

Tridiagonal ADI:

```
&METPRM
    ILHS = 0,
/
```

Diagonal ADI - Central:

```
&METPRM
    ILHS = 2,
/
```

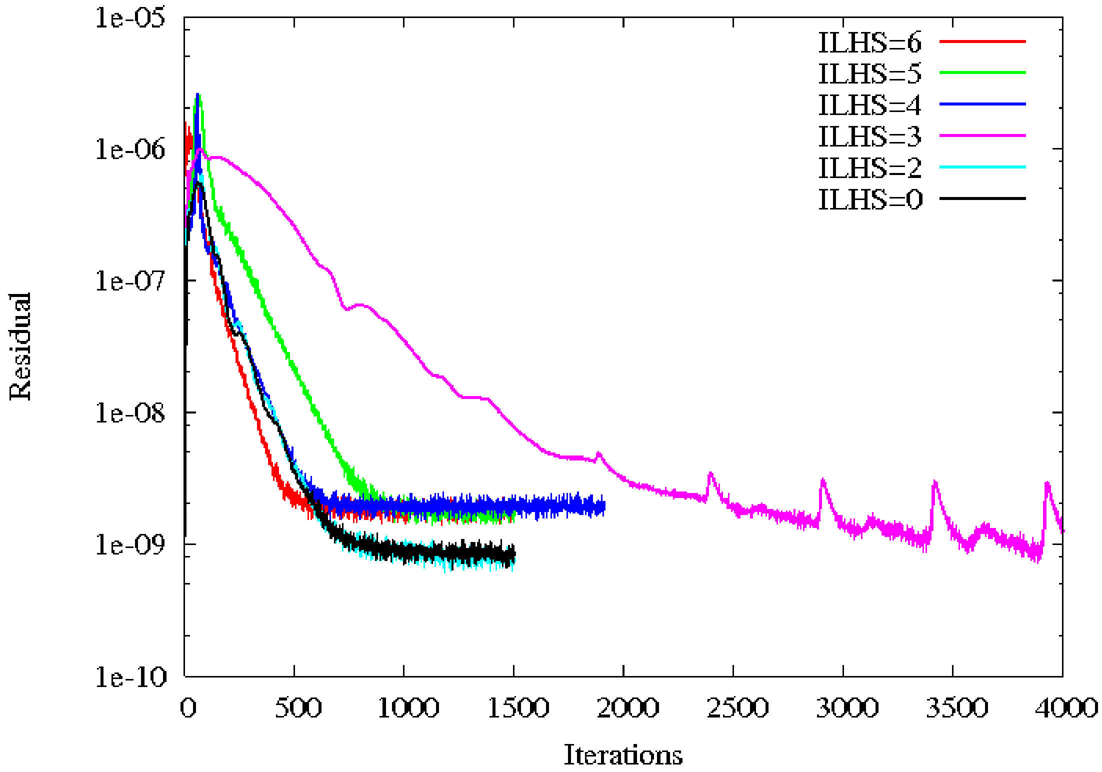
SSOR:

```
&METPRM
    ILHS = 6, ILHSIT = 10,
/
```

Diagonal ADI - Upwind:

```
&METPRM
    ILHS = 2, IDISS =2,
/
&SMOACU
  DIS2 = 10.0, DIS4 = 0.1,
  SMOO = 0.0
/
```

Axisymmetric Bump Convergence



ILHS	IRHS	Time to Converge (sec.)
0	0	30.5
2	0	21.0
3	0	83.3
4	5	10.0
5	5	32.5
6	5	33.4

Solver Relative Timings*

B&W C ILHS=0	F3D ILHS=1	Diag. ILHS=2	LU-SGS ILHS=3	D3ADI ILHS=4	B&W U ILHS=5	SSOR ILHS=6	SSOR ILHS=7
3.94	5.31	1.27	1.00	3.41	4.08	9.38	9.85

*Sensitive to processor and grid size

*51x51x51 Grid

*Full Viscous Terms

Subiterations

Subiteration Strategies

$$\left[I + \frac{\Delta t}{1+\theta}\left(\delta_\xi A + \delta_\eta B + \delta_\zeta C\right)\right]\Delta q^{n+1,m+1} =$$

$$-\left[\left(q^{n+1,m} - q^n\right) - \frac{\theta}{1+\theta}\Delta q^n + \frac{\Delta t}{1+\theta} RHS^{n+1,m}\right]$$

Subiteration update of q

n = iteration counter

m = subiteration counter

Recalculate RHS each subiteration with latest q

Newton Method:

Δt = Constant

Dual Time Stepping:

- Use local Δt
- Locally converge inner iteration (m)

Subiteration NAMELIST Input

Dual Time Stepping:

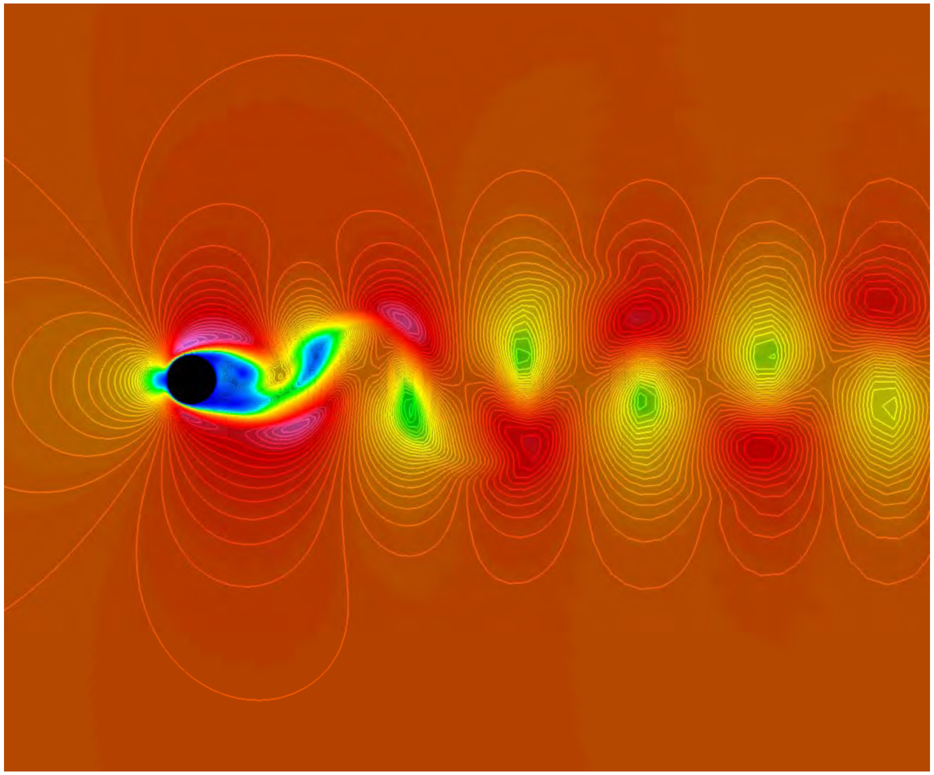
```
&GLOBAL
    NITNWT=5, FSONWT=2.0, DTPHYS=1.0,
    ORDNWT = 0,
/
&TIMACU
      ITIME=1, DT=0.1, CFLMIN=10,
/
```

Newton Time Stepping:

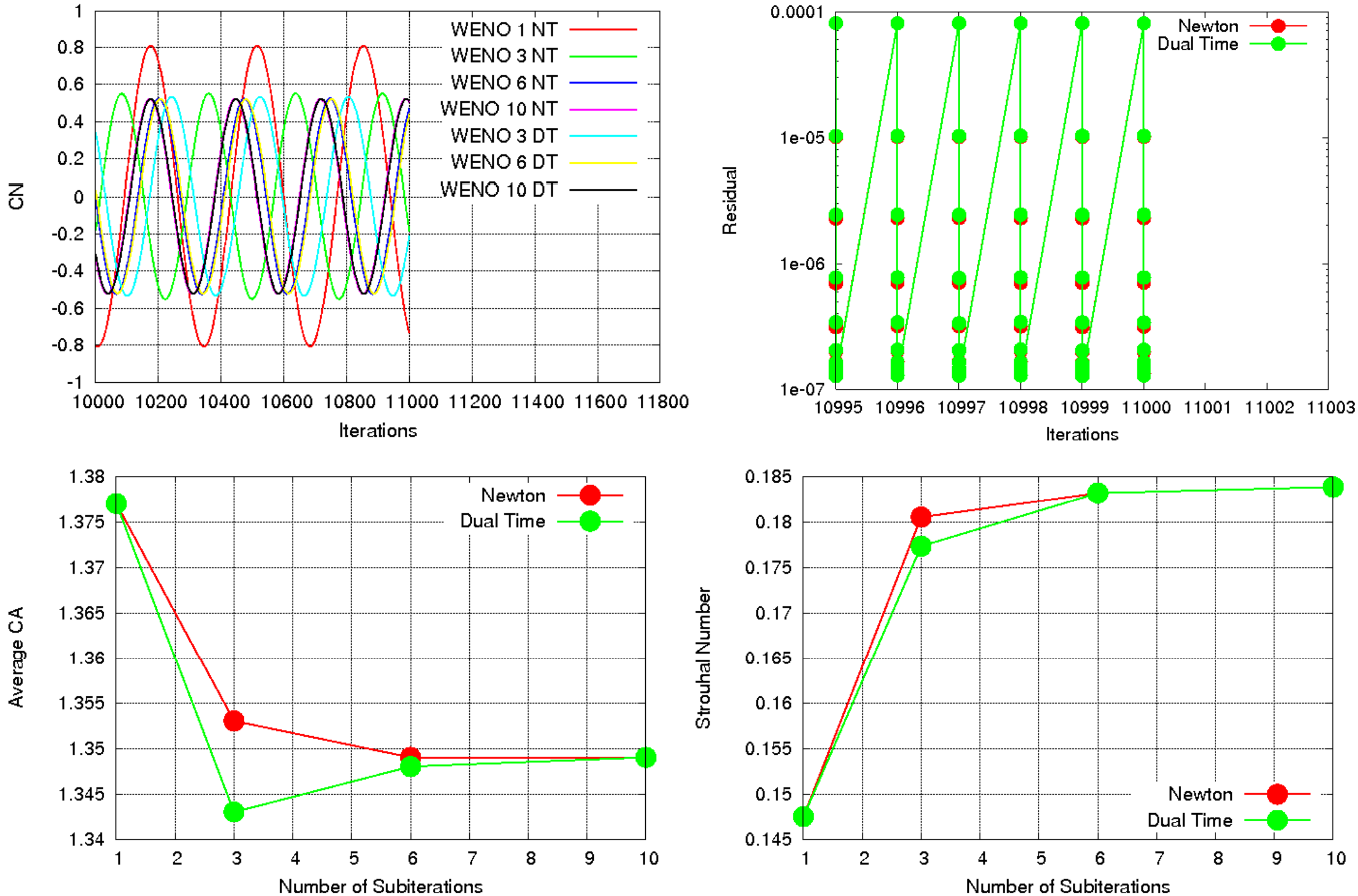
```
&GLOBAL
    NITNWT=5, FSONWT=2.0, DTPHYS=1.0,
    ORDNWT=3,
/
&TIMACU
      ITIME=0,DT=0.0,
/
```

Laminar Cylinder

- $M_\infty = 0.2$, $Re_D = 150$
- DTPHYS = 0.02 (9.123×10^{-5} sec.)
  - 272 time steps per lift cycle
  - 136 time steps per drag cycle
- 5^{th} order WENO inviscid fluxes
- Upwind Tridiagonal Solver
- 2^{nd} order time
- 401x201 grid



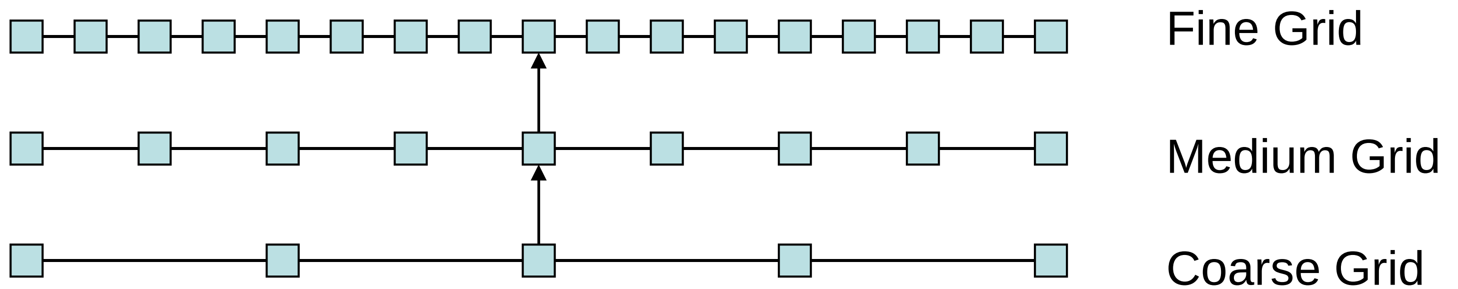
Laminar Cylinder



Convergence Acceleration

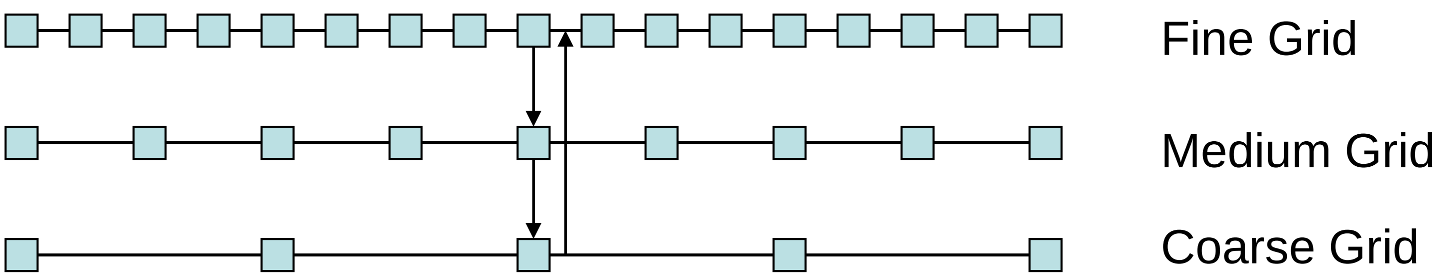
Grid Sequencing

- Get an improved initial solution quickly on coarser grids
- Works best for grids with $2^m n_{coarse}+1$ points
  - m = number of levels
  - n_{coarse} = number of points in coarse grid
- Not all grids will sequence
  - Too few points in fine grid
  - Highly skewed fine grid
- Overset updates on fine grid only
- Can jump over holes on coarser grids



Multigrid

- Construct solution at each time step from solutions on coarse and fine grid levels
- Currently use a "V" multigrid cycle
- Quickly dissipate low frequency error
- Transport turbulence models and species equations are not solved with multigrid
- Works best for grids with $2^m n_{coarse}+1$ points
  - m = number of levels
  - n_{coarse} = number of points in coarse grid
- Not all grids will sequence
  - Too few points in fine grid
  - Highly skewed fine grid
- Can jump over holes on coarser grids
- Overset updates on fine grid only



Convergence Acceleration NAMELIST Input

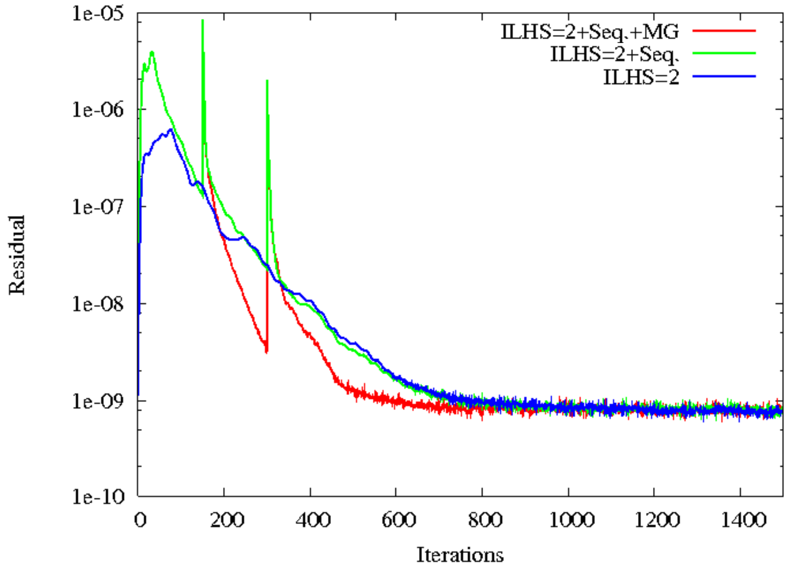
Grid Sequencing:

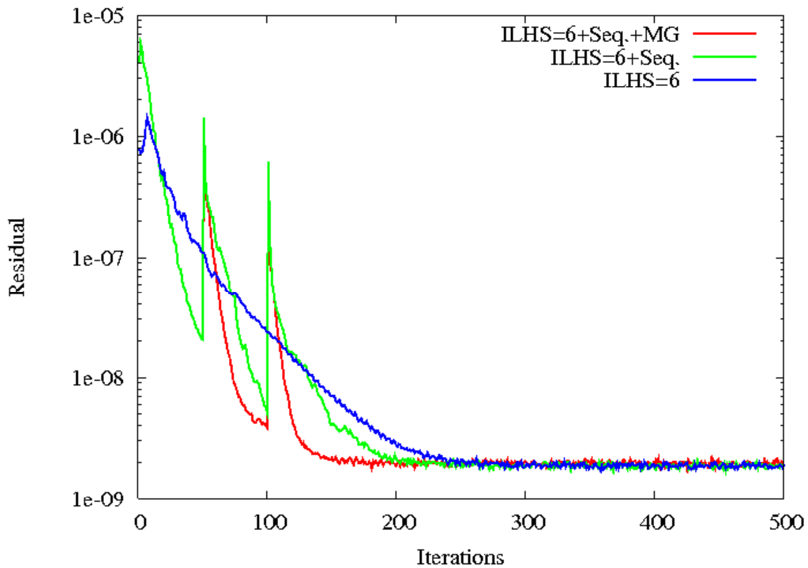
```
&GLOBAL
    FMG = .TRUE., NGLVL=3,
    FMGCYC = 150,150,
/
```

Multigrid:

```
&GLOBAL
    MULTIG = .TRUE., NGLVL=3,
/
```

Axisymmetric Bump Convergence





ILHS	Time to Converge (sec.)
2*	26.3
2 +Seq.	20.5
2+Seq.+MG	22.8
6**	51.3
6+Seq.	37.4
6+Seq.+MG	52.8

* Local time step, IRHS=0

** 3 Newtons, global time step, IRHS=5

LHS Hints

1. Recommended values for default diagonalized solver (ILHS=2) when used with upwind algorithms (IRHS=3,4,5): IDISS=2, SMOO=0.0, DIS2=10.0, DIS4=0.1
2. Recommend use of ILHS=2 or 4 (fastest) or ILHS=6 (most stable)
3. Use local time stepping and diagonalized solvers (ILHS=2,4) for low speed preconditioning. Use dual time stepping for time accurate solutions.
4. Use local time stepping and dual time stepping with diagonalized solvers. Recommended time step ITIME=1, DT=0.1, CFLMIN=10.
5. SSOR solver (ILHS=6) often does not need local time stepping. Best when used with Newton subiteration and second order time (FSONWT=2.0, NITNWT=3, DTPHYS=1.0)
6. Use grid sequencing (FMG=.TRUE., NGLVL=3, FMGCYC=150,150) when possible to accelerate solution convergence.
7. DT nondimensionalized by a_∞. DTPHYS nondimensionalized by V_∞. (DTPHYS = M_∞*DT)

Boundary Conditions

Boundary Condition Inputs

- Select BC type with IBTYP
- Specify direction with IBDIR
  - 1 for +J, 2 for +K, 3 for +L
  - -1 for –J, -2 for –K, -3 for -L
- Specify region with JBCS, JBCE, KBCS, KBCE, LBCS, LBCE
  - -1 for last point
  - -n for nend-n+1
- BCPAR1, BCPAR2 for BC specific values
- All boundaries must have a BC or be interpolated – if not the BC checker will stop the code

Boundary Condition Hints

- Walls (IBTYP=1-8)
  - Pressure extrapolation more stable and less sensitive to non-normal grid lines
  - Momentum equation more accurate (requires grid lines to be near normal to surface)
- Symmetry planes
  - IBTYP=11-13 requires reflection plane (implicit)
  - IBTYP=17 does not require reflection plane (explicit slip wall)
- Polar axis (IBTYP=14-16) $f_0=f_1+\frac{1}{2}\alpha(f_1-f_2)$
  - α=1 (default) is 1st order extrapolation (more accurate, less stable)
  - α=0 is 0th order extrapolation (less accurate, more stable)
  - α set by BCPAR1

Boundary Condition Hints (Cont.)

- IBTYP=21 - 2D – 3 planes ±1 in Y
- IBTYP=22 – Axi in Y, rotate about X ±1^o
- IBTYP=41 – Nozzle inflow
  - BCPAR1=p_o/p_{oinf}
  - BCPAR2=T_o/T_{oinf}
- IBTYP=141 – Uniform nozzle inflow
  - BCPAR1=p_o/p_{oinf}
  - BCPAR2=T_o/T_{oinf}
- IBTYP=47 – Riemann outflow with free stream for incoming information
- IBTYP=51-59 C-grid and fold-over bcs
  - Be careful with topology

Boundary Condition Input

```
&BCINP
    IBTYP =  5,   17,   30,   141,   21,
    IBDIR  = -2,    2,    -1,       1,     3,
    JBCS  =   1,     1,    -1,       1,     1,
    JBCE  =  -1,   -1,    -1,       1,    -1,
    KBCS =  -1,    1,       1,       1,     1,
    KBCE =  -1,    1,    -1,      -1,    -1,
    LBCS  =   1,    1,     1,       1,      1,
    LBCE  =  -1,  -1,    -1,      -1,     1,
    BCPAR1(4) = 1.0,
    BCPAR2(4) = 1.0,
 /
```

Species Equations

Species Transport Equations

$$
\begin{aligned}
&\frac{\partial \rho c_i}{\partial t}+\frac{\partial \rho U c_i}{\partial \xi}+\frac{\partial \rho V c_i}{\partial \eta}+\frac{\partial \rho W c_i}{\partial \zeta}=\frac{\partial}{\partial \xi}\left[\left(\frac{\mu}{\sigma_L}+\frac{\mu_t}{\sigma_T}\right)\left(\xi_x^2+\xi_y^2+\xi_z^2\right)\frac{\partial c_i}{\partial \xi}\right]+\\
&\frac{\partial}{\partial \eta}\left[\left(\frac{\mu}{\sigma_L}+\frac{\mu_t}{\sigma_T}\right)\left(\eta_x^2+\eta_y^2+\eta_z^2\right)\frac{\partial c_i}{\partial \eta}\right]+\frac{\partial}{\partial \zeta}\left[\left(\frac{\mu}{\sigma_L}+\frac{\mu_t}{\sigma_T}\right)\left(\zeta_x^2+\zeta_y^2+\zeta_z^2\right)\frac{\partial c_i}{\partial \zeta}\right]+\\
&\frac{\partial}{\partial \xi}\left[\left(\frac{\mu}{\sigma_L}+\frac{\mu_t}{\sigma_T}\right)\left(\xi_x\eta_x+\xi_y\eta_y+\xi_z\eta_z\right)\frac{\partial c_i}{\partial \eta}\right]+\frac{\partial}{\partial \xi}\left[\left(\frac{\mu}{\sigma_L}+\frac{\mu_t}{\sigma_T}\right)\left(\xi_x\zeta_x+\xi_y\zeta_y+\xi_z\zeta_z\right)\frac{\partial c_i}{\partial \zeta}\right]+\\
&\frac{\partial}{\partial \eta}\left[\left(\frac{\mu}{\sigma_L}+\frac{\mu_t}{\sigma_T}\right)\left(\xi_x\eta_x+\xi_y\eta_y+\xi_z\eta_z\right)\frac{\partial c_i}{\partial \xi}\right]+\frac{\partial}{\partial \eta}\left[\left(\frac{\mu}{\sigma_L}+\frac{\mu_t}{\sigma_T}\right)\left(\eta_x\zeta_x+\eta_y\zeta_y+\eta_z\zeta_z\right)\frac{\partial c_i}{\partial \zeta}\right]+\\
&\frac{\partial}{\partial \zeta}\left[\left(\frac{\mu}{\sigma_L}+\frac{\mu_t}{\sigma_T}\right)\left(\xi_x\zeta_x+\xi_y\zeta_y+\xi_z\zeta_z\right)\frac{\partial c_i}{\partial \xi}\right]+\frac{\partial}{\partial \zeta}\left[\left(\frac{\mu}{\sigma_L}+\frac{\mu_t}{\sigma_T}\right)\left(\eta_x\zeta_x+\eta_y\zeta_y+\eta_z\zeta_z\right)\frac{\partial c_i}{\partial \eta}\right]
\end{aligned}
$$

c_i = Species Mass Fraction

σ_L = Laminar Schmidt Number

σ_T = Turbulent Schmidt Number

Species Transport Equations

For each species c_i:

$$c_p - c_v = R$$

$$\frac{c_p}{R} = a_0 + a_1 T + a_2 T^2 + a_3 T^3 + a_4 T^4$$

Mass-Averaged Properties:

$$1 = \sum_{i=1}^{ngas} c_i \qquad R_{mix} = \sum_{i=1}^{ngas} c_i R_i \qquad c_{pmix} = \sum_{i=1}^{ngas} c_i c_{pi}$$

$$c_{vmix} = \sum_{i=1}^{ngas} c_i c_{vi} \qquad \gamma_{mix} = \frac{c_{pmix}}{c_{vmix}}$$

$$\overline{T} = \frac{T}{\gamma_\infty T_\infty} = \frac{\gamma_{mix} - 1}{R_{mix}} \overline{e}_i = \frac{\overline{e}_i}{c_{vmix}}$$

Species Solution Options

- Differencing of species convection terms:
  – Central difference (IUPC=0 – Requires DIS2C and DIS4C)
  – Upwind difference (IUPC=1)
  – HLLC upwind difference (IUPC=2)
- Spatial order for convection terms:
  – 2 for central
  – 1-3 for upwind
  – 1-5 for HLLC
- LHS options:
  – ADI (ITLHIC = 1)
  – SSOR (ITLHIC > 1)

Constant γ, 1 Species

```
&GLOBAL
        NQC = 0,
/
&VARGAM
        IGAM=0,
/
&SCEINP
/
```

No species equation solved, so don't need &SCEINP

Variable γ, 1 Species

```
&GLOBAL
        NQC = 1,
/
&VARGAM
        IGAM=1,
        ALT0(1)=3.5,ALT1(1)=0,ALT2(1)=0,ALT3(1)=0,ALT4(1)=0,
        AUT0(1)=3.5,AUT1(1)=0,AUT2(1)=0,AUT3(1)=0,AUT4(1)=0,
/
&SCEINP
/
```

No species equation solved, so don't need &SCEINP

Variable γ, 2 Species

```
&GLOBAL
        NQC = 0,
/
&VARGAM
        IGAM=2,
        HT1=10.0,HT2=20.0,
        SCINF=1.0,0.0,
        SMW=1.0,1.2,
        ALT0(1)=3.5,ALT1(1)=0,ALT2(1)=0,ALT3(1)=0,ALT4(1)=0,
        AUT0(1)=3.5,AUT1(1)=0,AUT2(1)=0,AUT3(1)=0,AUT4(1)=0,
        ALT0(2)=3.5,ALT1(2)=0,ALT2(2)=0,ALT3(2)=0,ALT4(2)=0,
        AUT0(2)=3.5,AUT1(2)=0,AUT2(2)=0,AUT3(2)=0,AUT4(2)=0,
/
&SCEINP
/
```

No species equation solved, so don't need &SCEINP

Variable γ, ≥2 Species

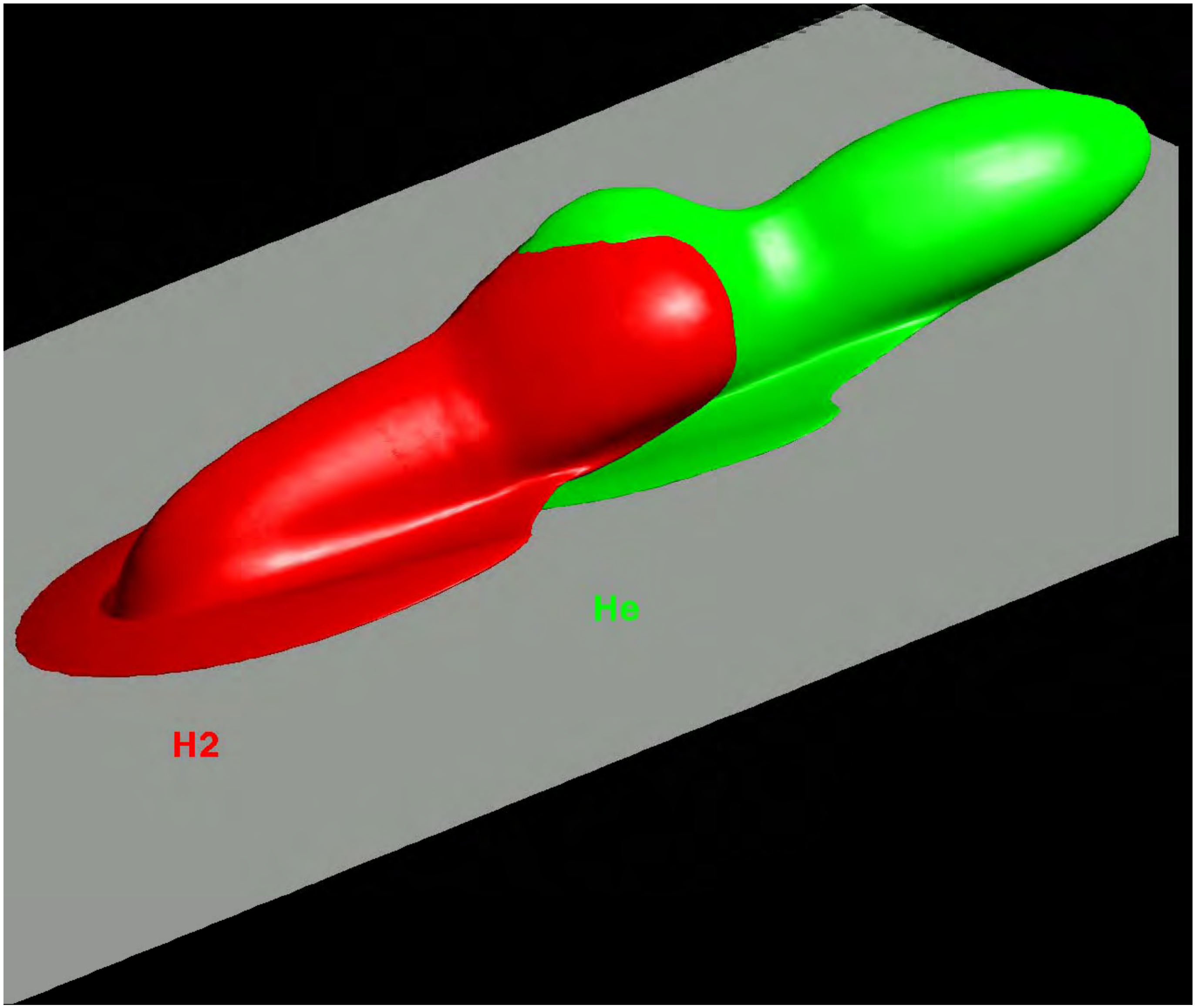
```
&GLOBAL
        NQC = 3, /
&VARGAM
        SCINF = 1.0,0.0,0.0,

        SMW(1) = 28.0, SIGL(1) = 1.0, SIGT(1) = 1.0,
        ALT0(1)=3.5,ALT1(1)=0,ALT2(1)=0,ALT3(1)=0,ALT4(1)=0,
        AUT0(1)=3.5,AUT1(1)=0,AUT2(1)=0,AUT3(1)=0,AUT4(1)=0,

        SMW(2) = 2.0, SIGL(2) = 2.0, SIGT(2) = 1.0,
        ALT0(2)=3.5,ALT1(2)=0,ALT2(2)=0,ALT3(2)=0,ALT4(2)=0,
        AUT0(2)=3.5,AUT1(2)=1,AUT2(2)=1,AUT3(2)=1,AUT4(2)=1,

        SMW(3) = 4.0, SIGL(3) = 0.9, SIGT(3) = 1.0,
        ALT0(3)=2.52,ALT1(3)=0,ALT2(3)=0,ALT3(3)=0,ALT4(3)=0,
        AUT0(3)=2.52,AUT1(3)=0,AUT2(3)=0,AUT3(3)=0,AUT4(3)=0,
/
&SCEINP
        ITLHIC = 10, IUPC=2,FSOC=3, /
```

N_2

H_2

He

H2 and HE Sonic Jets into N2



Species Hints

1. Recommend the HLLC flux with the SSOR solver (ITLHIC = 10, IUPC=2,FSOC=3)
2. Variable γ currently broken – pressure calculations inside code are not correct. Constant γ OK.
3. Free stream molecular weights and γ calculated based on NAMELIST inputs SCINF and SMW. SMW may be ratio or actual molecular weight. GAMINF is ignored.
4. Code initializes species to SCINF. Must modify initial q file or use specified input BC (BC 45) to get species into desired locations in grid.
5. Species concentration calculated based on local total enthalpy when using IGAM=2.
6. Most post-processors are constant γ. Use post-processing tool **vgplot** to get pressure, temperature, Mach number, enthalpy, γ, and species mass fraction (c_i) for plotting.

Turbulence Models

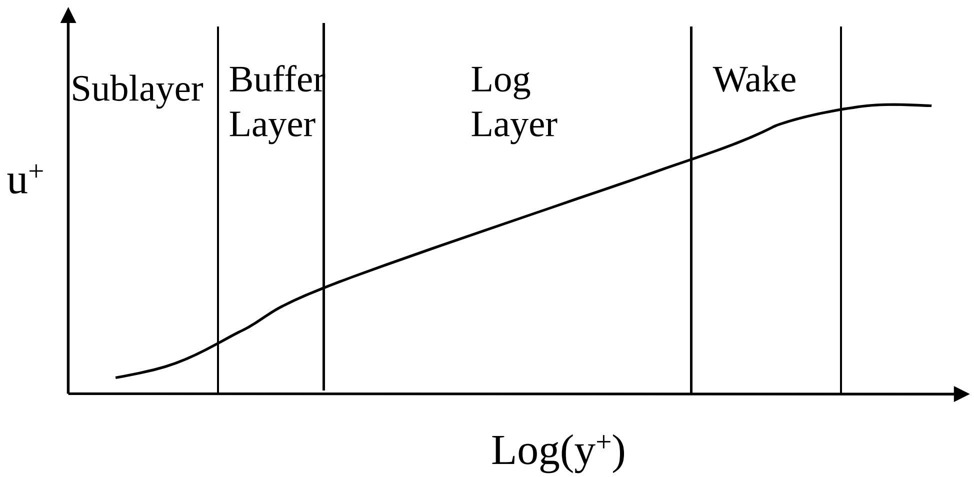
Turbulence Models

- Baldwin-Lomax (with wake model) (NQT=0)
- Baldwin-Barth 1 equation transport model (NQT=100)
- Spalart-Allmaras 1 equation transport model with trip line specification (NQT=101)
- Spalart-Allmaras 1 equation transport model (NQT=102)
- SA-DES hybrid RANS/LES (NQT=103)
- k-ω (1988) 2 equation transport model
  - DDADI LHS(NQT=202)
  - SSOR LHS (NQT=203)
- SST with compressibility correction 2 equation transport model
  - DDADI LHS (NQT=204)
  - SSOR LHS (NQT=205)
- Hybrid RANS/LES (SA and SST)
  - DES (IDES=1)
  - DDES (IDES=2)
  - MS (IDES=3)
- Rotational and curvature corrections (SA and SST)
  - SARC and SSTRC (IRC=1)
  - ASARC and ASSTRC (IRC=2)
- SST-MS hybrid RANS/LES (NQT=207)
- Wall functions for BB, SA, k-ω, and SST transport models

Wall Distance Calculations

- NAMELIST **$GLOBAL** parameters **WALLDIST** and **NWALL** used to control how and when wall distance is calculated
- **WALLDIST** options:
  - **WALLDIST = 0** – Read precomputed wall distance from file *walldist.da*t (PLOT3D function file format)
  - **WALLDIST = +/-1** – Simple computation of wall distance (only uses walls contained within the grid, ignores iblank)
  - **WALLDIST = +/-2** – Global wall distance computation
  - If **WALLDIST** is negative, write wall distance file *walldist.dat*
- **NWALL** – Recompute wall distance every **NWALL** steps
  - Currently ignored
  - Global wall distance currently computed on startup and after grid adaptation

Baldwin Lomax - Walls



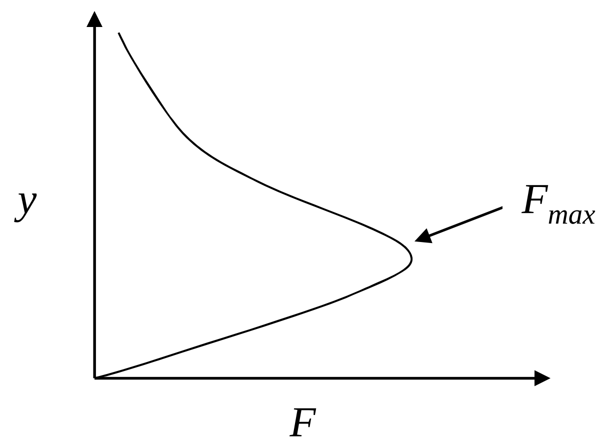
$$(\mu_t)_{inner} = \rho L_m^2 |\Omega|$$

$$(\mu_t)_{outer} = \rho K C_{cp} F_{wake} F_{kleb}$$

$$F_{kleb}(y) = \left[1 + 5.5\left(\frac{C_{kleb}\, y}{y_{\max}}\right)^6\right]^{-1}$$

$$F_{wake} = \min\left\{ y_{\max} F_{\max}, C_{wk}\, y_{\max} U_{diff}^2 / F_{\max} \right\}$$

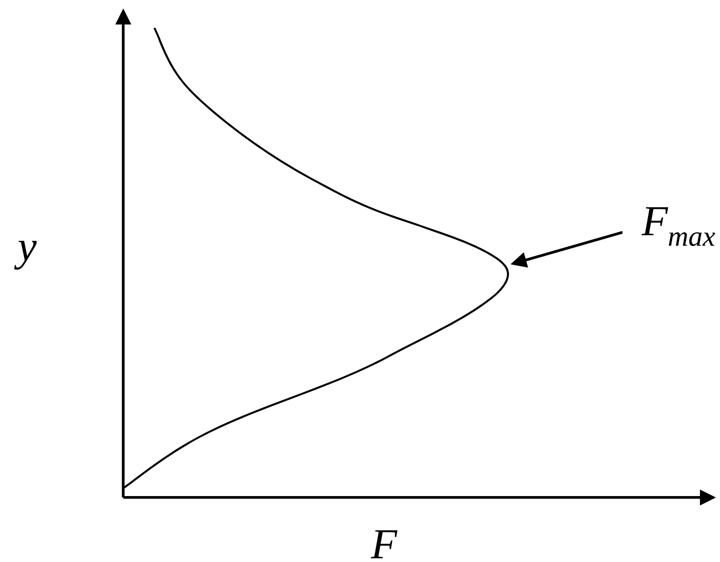
$$F(y) = y|\Omega|\left[1 - \exp\left(\frac{-y^+}{A^+}\right)\right]$$



Baldwin Lomax – Shear Layers

$$\mu_t = \rho K C_{cp} F_{wake} F_{kleb}$$

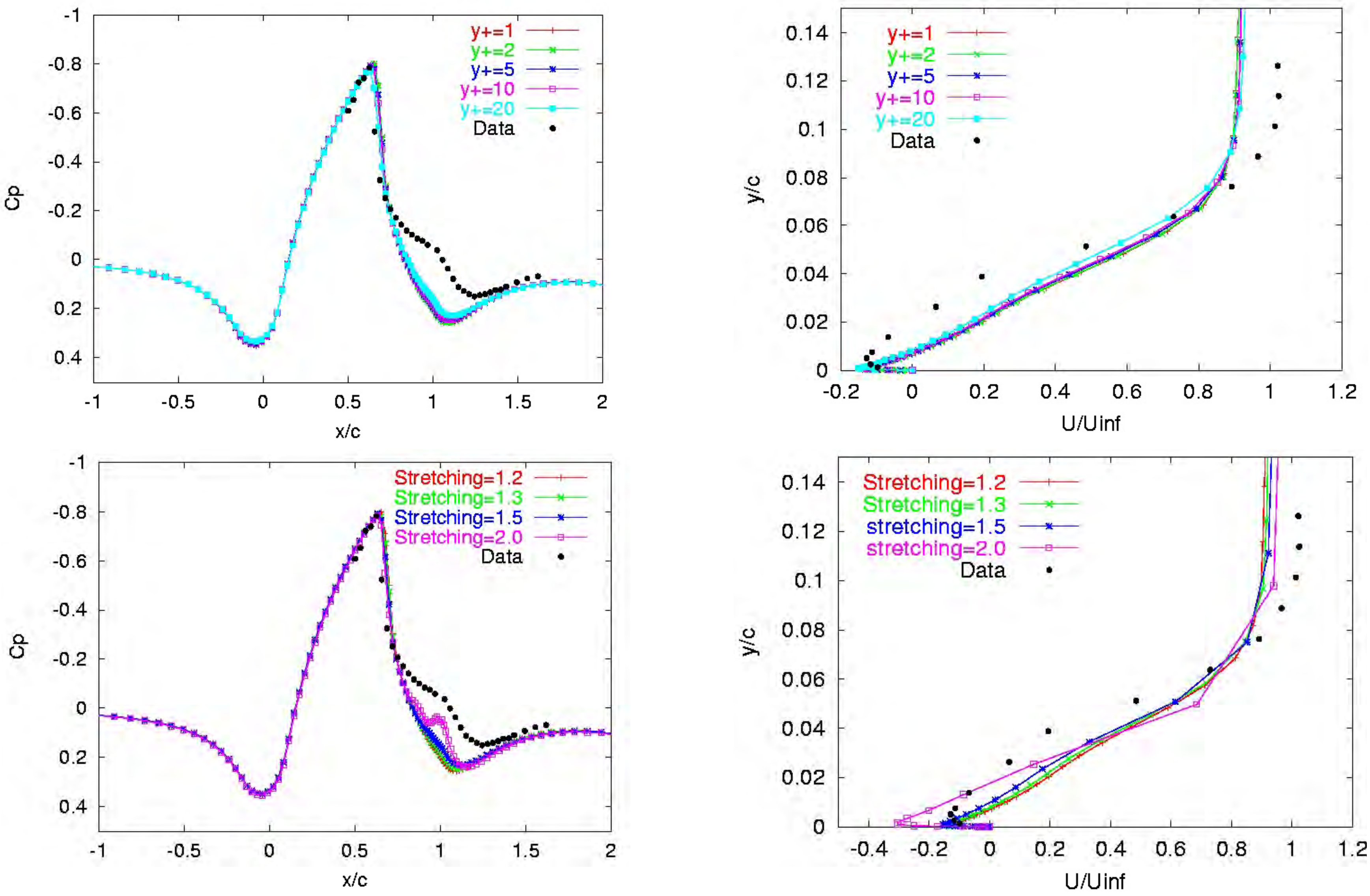
$$F(y) = y|\Omega|$$



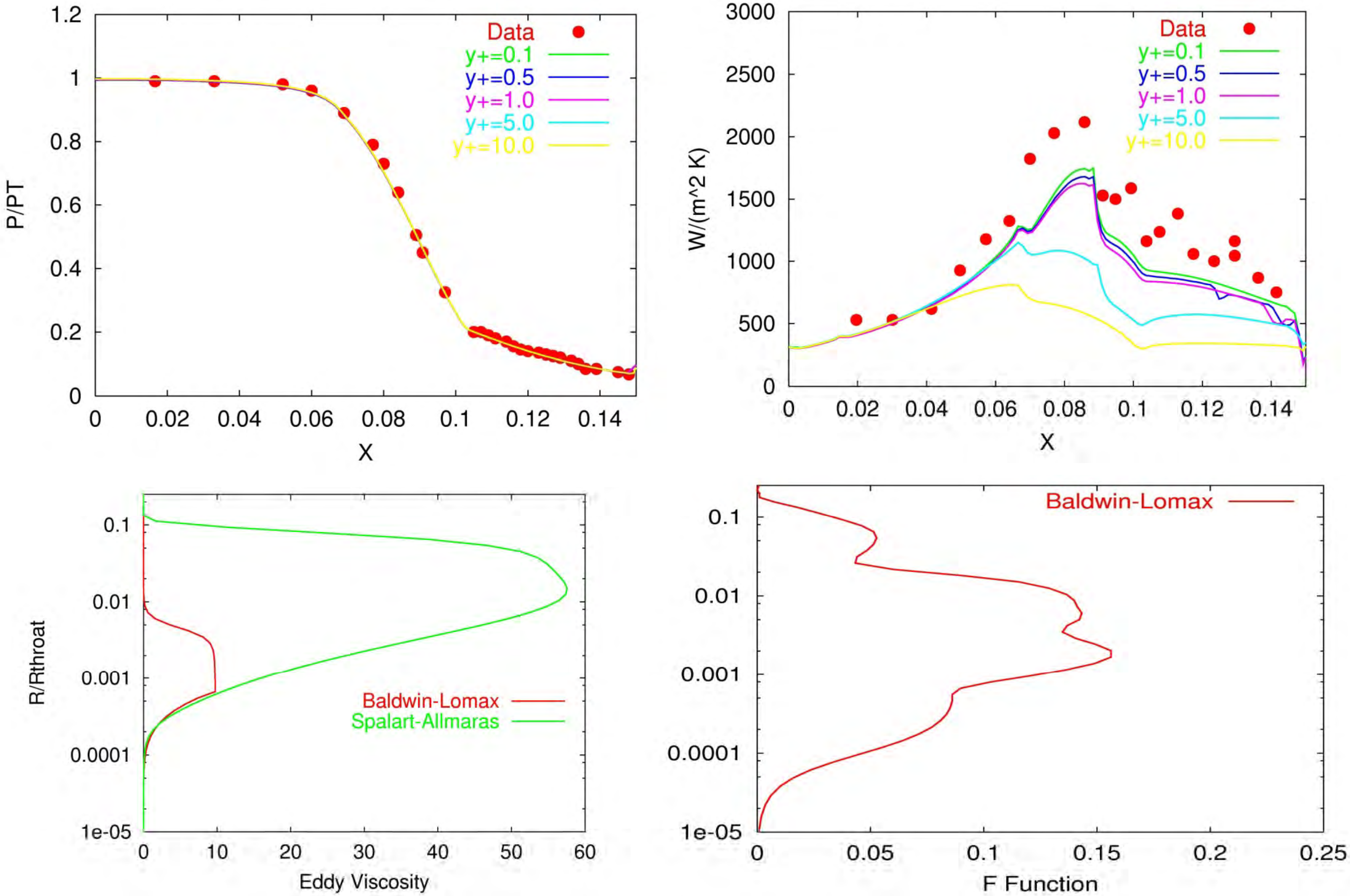
$$F_{Kleb} = \left(1 + 5.5 \left(C_{Kleb} \frac{\left| y - y_{|\Omega|_{\max}} \right|}{\left(\frac{u_{dif}}{|\Omega|_{\max}} \right)} \right)^6 \right)^{-1}$$

$$F_{wake} = \frac{u_{dif}^2}{|\Omega|_{\max}}$$

Baldwin Lomax - Axi Bump



Baldwin Lomax - Nozzle



Viscous Regions

- Used with Baldwin-Lomax to define where to apply the model
- Used with transport modes to specify boundary layer transition location
- Usage similar to boundary condition specification

Turbulent Region Types (ITTYP)

Type	*Description*
1	Baldwin-Lomax boundary layer model
11	Baldwin-Lomax shear layer model
102	**1- or 2-equation laminar region (zero production)**
103	Spalart-Allmaras boundary layer trip line

Baldwin Lomax Viscous Region

```
&GLOBAL
          NQT = 0,
/
&VISINP
          VISC = .TRUE.,
          ITTYP   =   1,   11,   11,
          ITDIR    =   2,    2,    2,
          JTLS     =  20,    1,  -19,
          JTLE     = -20,   19,    1-,
          KTLS     =   1,    1,    1,
         KTLE      =  -1,   -1,   -1,
          LTLS     =   1,    1,    1,
          LTLE     =  -1,   -1,   -1,
          TLPAR1 =  -1,    1,    1,
/
```

Baldwin-Lomax Application Hints

1. **The Baldwin-Lomax model requires that the *F* function be well defined. This normally requires that at least three points be located within the sublayer (y^+<10).**
2. **The *F* function should be determined on lines normal to the flow direction.**
3. **The first point off the wall should be located about y^+<*5* for pressure distributions, y^+<2 to obtain reasonable skin friction values, and y^+<0.5 for heat transfer.**
4. **The grid stretching normal to the wall should not exceed 1.3.**
5. **Improved heat transfer results can be obtained by using a constant spacing for the first three cells off the wall.**
6. **In order to reduce the probability of finding a second peak well off the wall, it is usually good to limit the number of points over which the *F* function is calculated.**
7. **Care should be taken not to divide viscous regions such as boundary layers when dividing the computational domain for blocked or chimera applications since the entire velocity profile is required to properly define the F_{max} and U_{diff} quantities.**

Spalart-Allmaras One-Equation Model

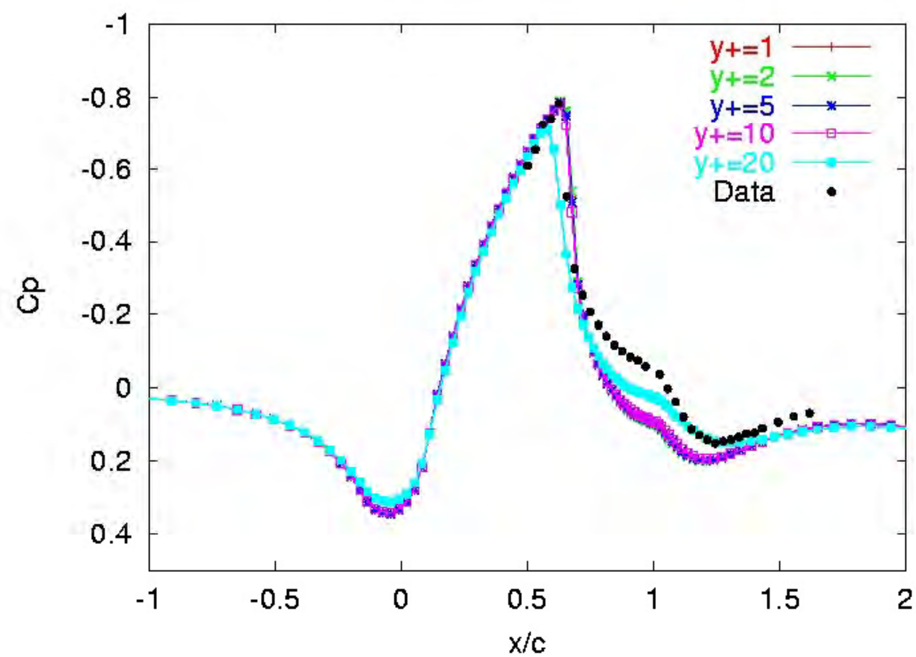
$$\underbrace{\frac{\partial \tilde{\nu}}{\partial t} + U_i \frac{\partial \tilde{\nu}}{\partial x_i}}_{\text{Advection}} = \underbrace{\frac{1}{\sigma}\left[\nabla \cdot \left(\left(\nu + \tilde{\nu}\right)\nabla \tilde{\nu}\right) + C_{b2}\left(\nabla \tilde{\nu}\right)^2\right]}_{\text{Diffusion}} + \underbrace{P(\tilde{\nu})}_{\text{Production}} - \underbrace{D(\tilde{\nu})}_{\text{Dissipation}}$$

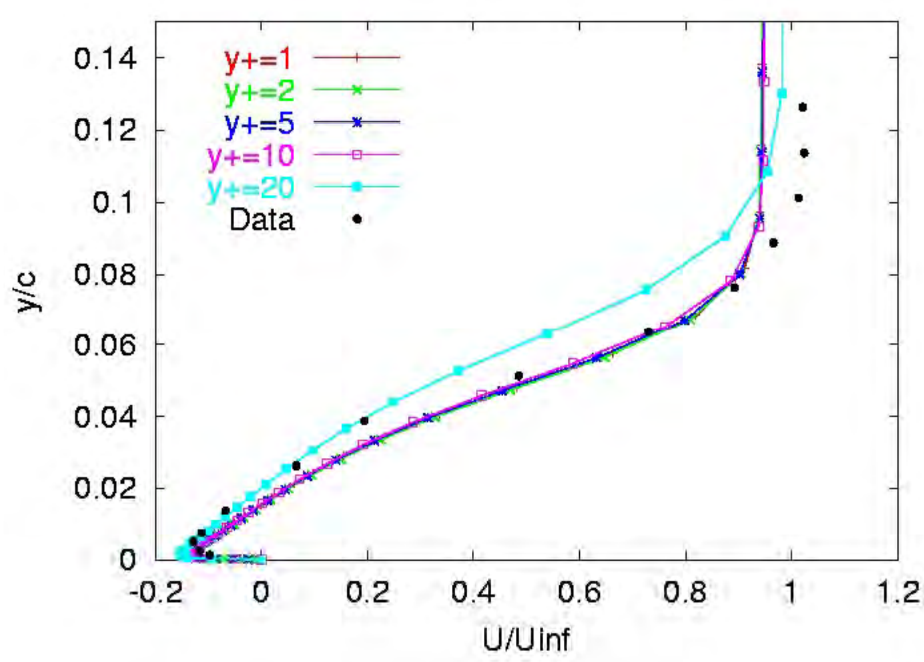
$$P(\tilde{\nu}) = C_{b1}\left(S + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2}\right)$$

$$D(\tilde{\nu}) = C_{w1} f_w \left(\frac{\tilde{\nu}}{d}\right)^2$$

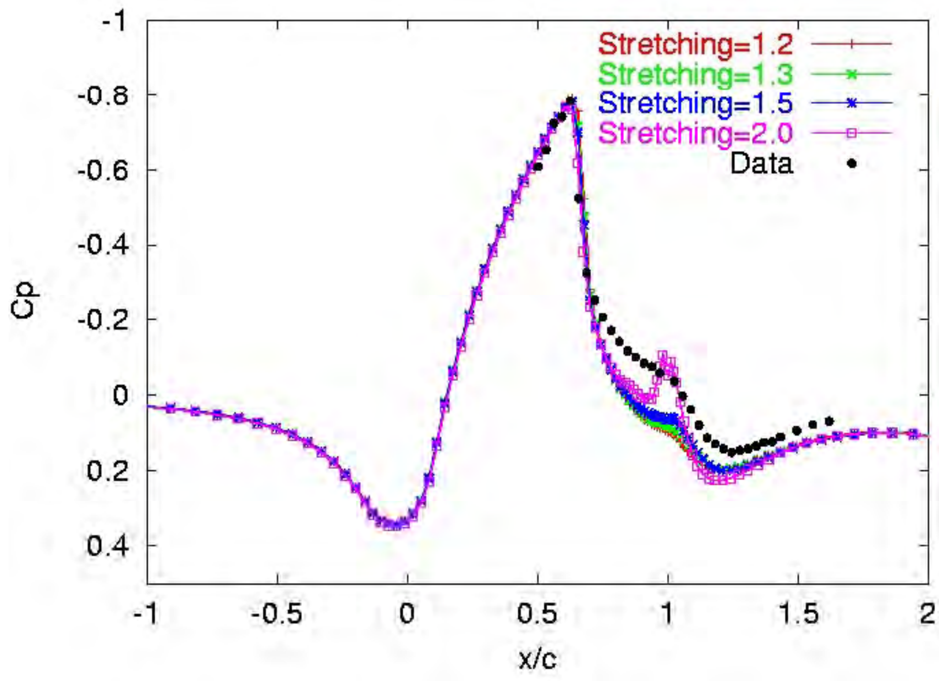
$$\nu_t = \tilde{\nu} f_{v1}$$

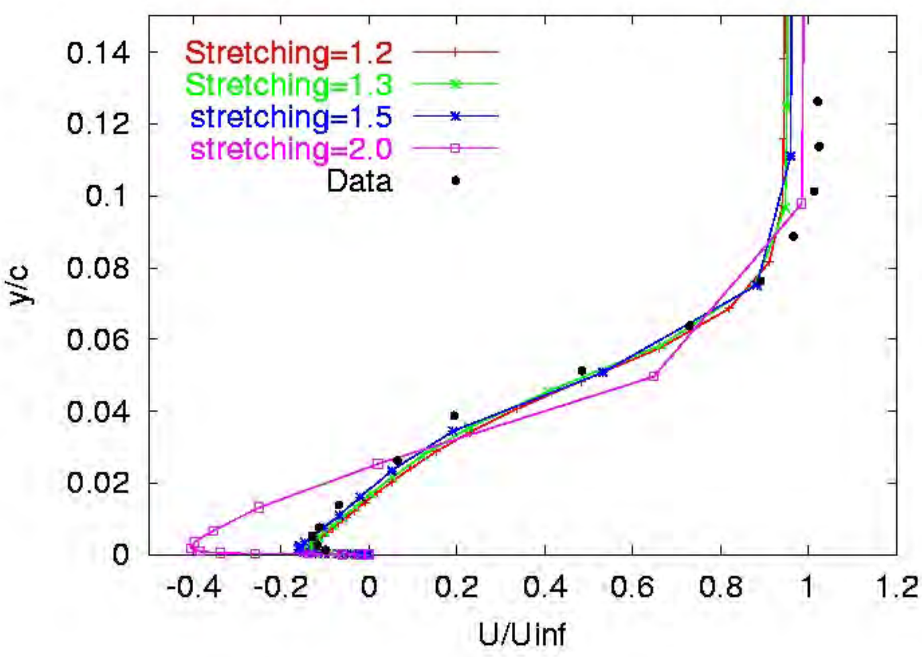
Effect of Initial Wall Spacing
Axi Bump - SA Model



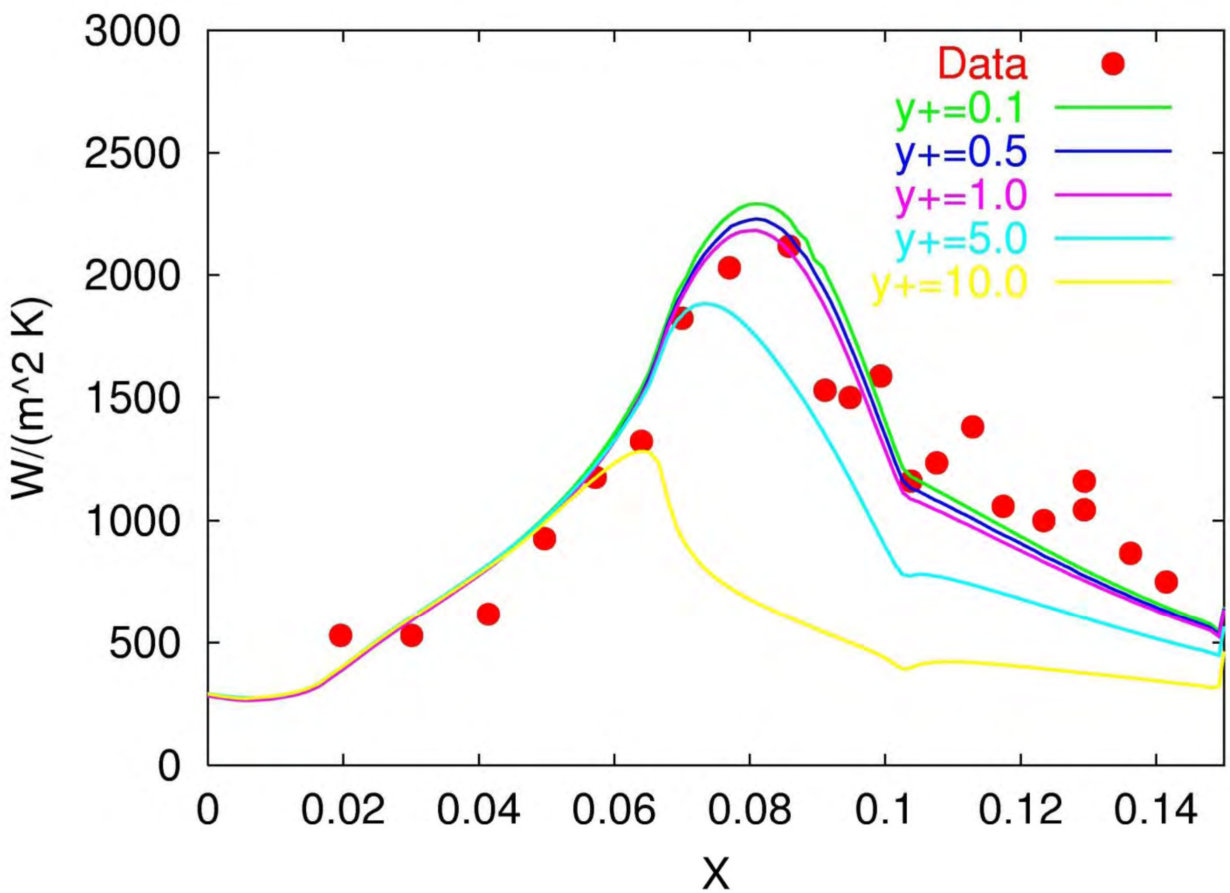


Effect of Grid Stretching Ratio
Axi Bump - SA Model

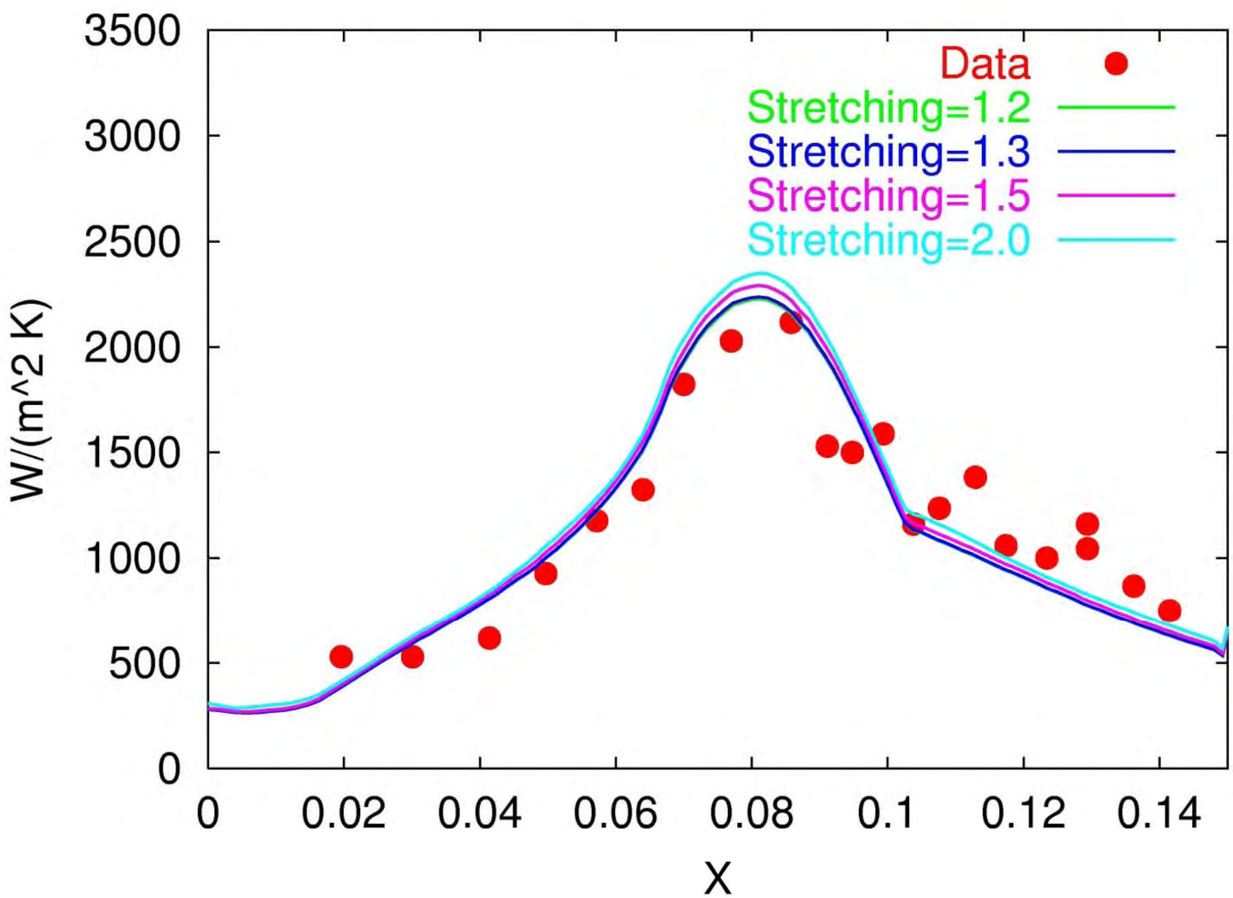




Effect of Initial Wall Spacing on Nozzle Heat Transfer - SA



Effect of Grid Stretching on Nozzle Heat Transfer - SA



Spalart-Allmaras Application Hints

1. The first point off the wall should be located about y^+=1 to obtain reasonable skin friction values and about y^+=0.5 for heat transfer.
2. The grid stretching normal to the wall should not exceed 1.3.
3. The eddy viscosity should be limited so that it will not run away in some complex applications. Generally a limit of ν_t/ν=200,000 is acceptable.
4. Care should be taken not to divide viscous regions such as boundary layers when dividing the computational domain for blocked or overset applications since the model requires the distance from the nearest wall.
5. This model tends to smear out three-dimensional vortical flows (rotation and curvature corrections can help).
6. The model can overdamp some unsteady flows.
7. The model contains no corrections for compressibility and will overpredict the growth rate of high speed shear layers.

k-ε Transport Equations

$$\boxed{\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i}} = \boxed{\frac{\partial}{\partial x_i}\left[\left(\nu + \frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]} + \boxed{P_k} - \boxed{\varepsilon}$$

Advection Diffusion Production Dissipation

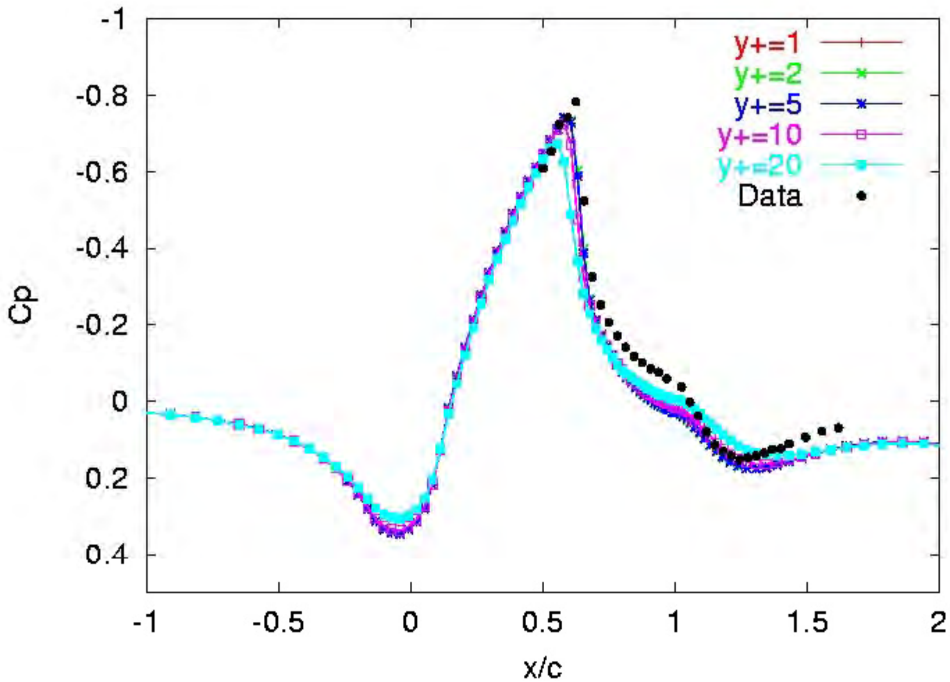
$$\boxed{\frac{\partial \varepsilon}{\partial t} + U_i \frac{\partial \varepsilon}{\partial x_i}} = \boxed{\frac{\partial}{\partial x_i}\left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right]} + \boxed{C_{\varepsilon 1}\frac{\varepsilon}{k}P_k} - \boxed{C_{\varepsilon 2}\frac{\varepsilon^2}{k}}$$

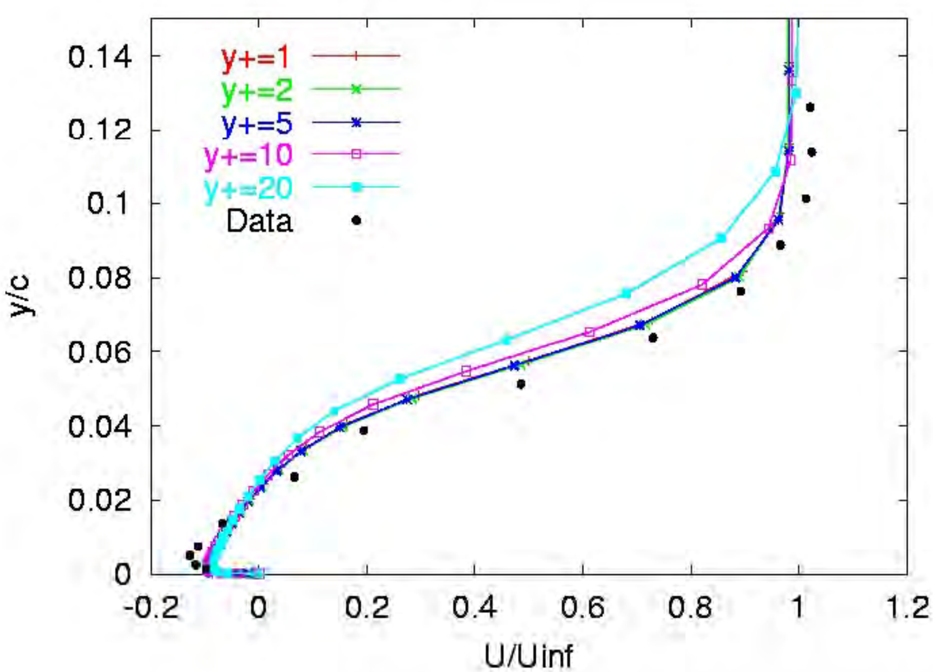
$$P_k = \nu_t\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j} - \frac{2}{3}k\frac{\partial U_i}{\partial x_i} \qquad \nu_t = C_\mu \frac{k^2}{\varepsilon}$$

Two-Equation Model Variants

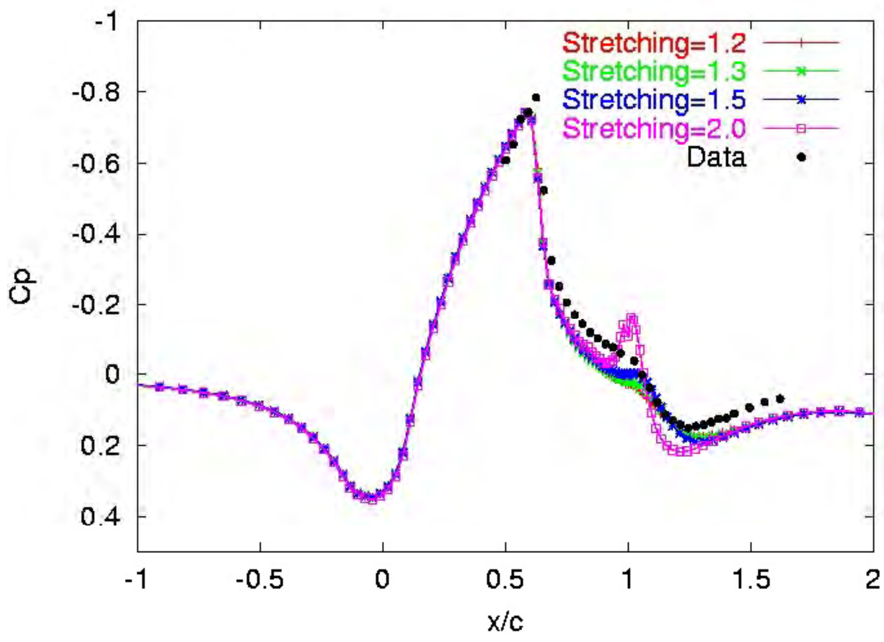
- *k*-ε
  - Require wall-damping terms
  - Good in shear layers
  - Lots of corrections available (compressibility, roughness, etc.)
  - Not as good near walls
- Wilcox *k*-ω $\left(\omega = \frac{\varepsilon}{C_\mu k}\right)$
  - Additional cross-diffusion term $\left(\nu + \frac{\nu_t}{\sigma_\varepsilon}\right) \frac{\partial \varepsilon}{\partial x_i} \frac{\partial k}{\partial x_i}$
  - Good near walls
  - Pretty good in adverse pressure gradients
  - Sensitive to far field value of ω
- Mentor's Shear Stress Transport (SST)
  - Blended model with Bradshaw's shear stress relationship in boundary layer (improves performance in adverse pressure gradients)
  - *k*-ω near wall
  - *k*-ε away from wall

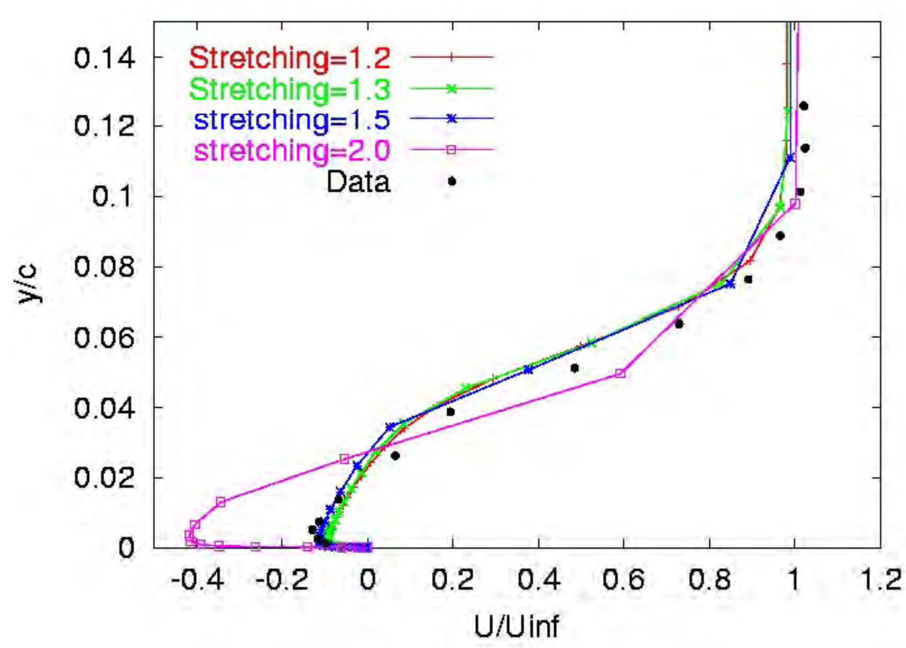
Effect of Initial Wall Spacing
Axi Bump - SST Model



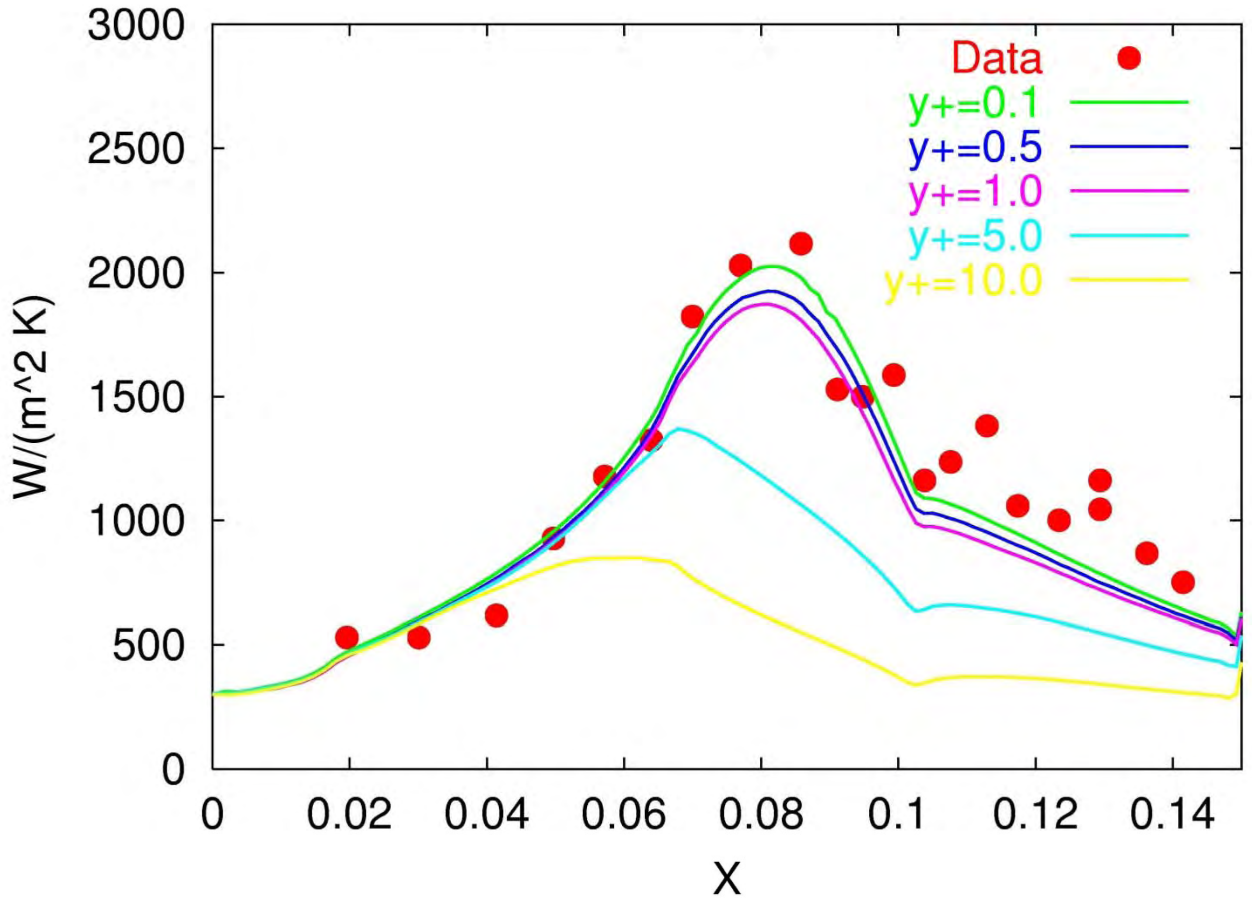


Effect of Grid Stretching Ratio
Axi Bump - SST Model

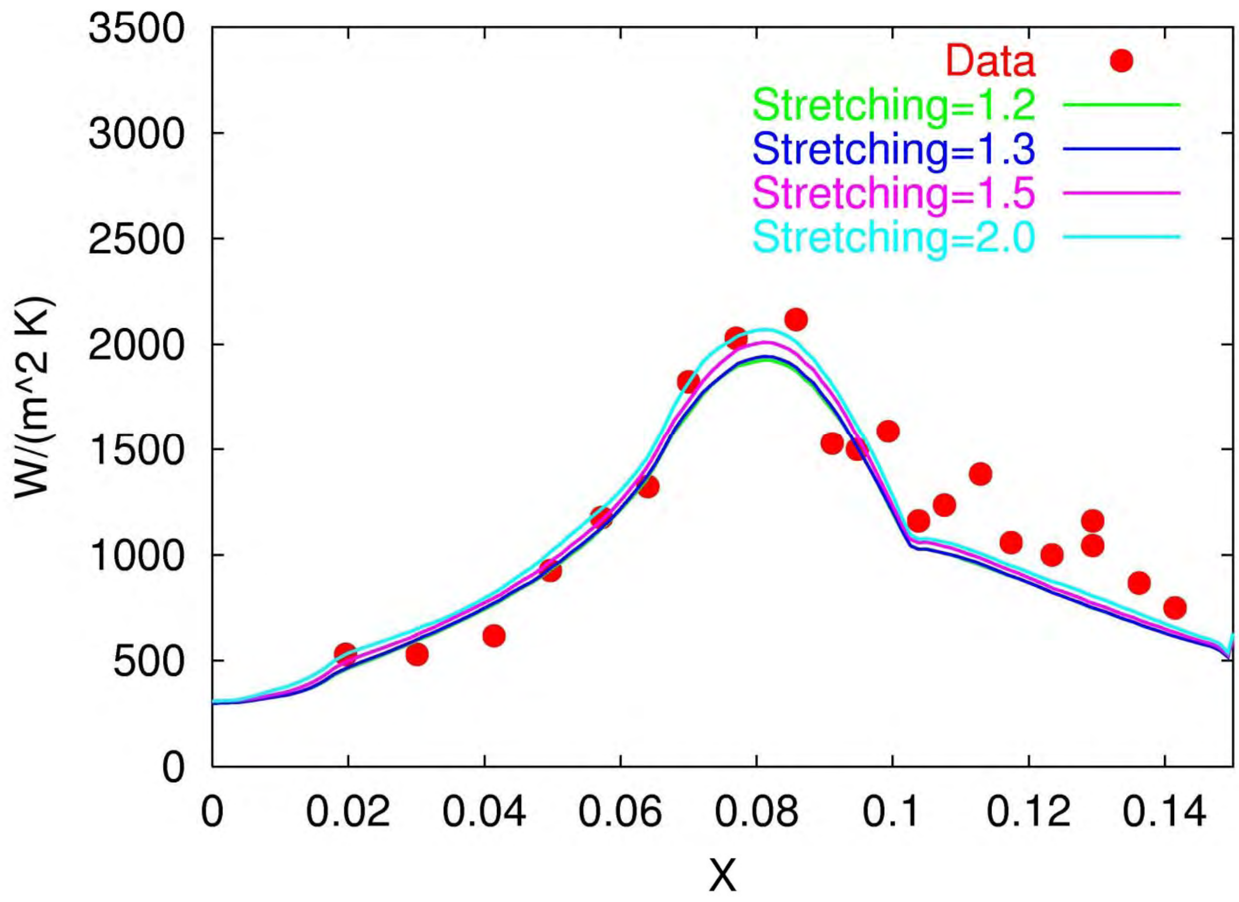




Effect of Initial Wall Spacing on Nozzle Heat Transfer - SST



Effect of Grid Stretching on Nozzle Heat Transfer - SST



Sarkar Compressibility Correction
SST Model

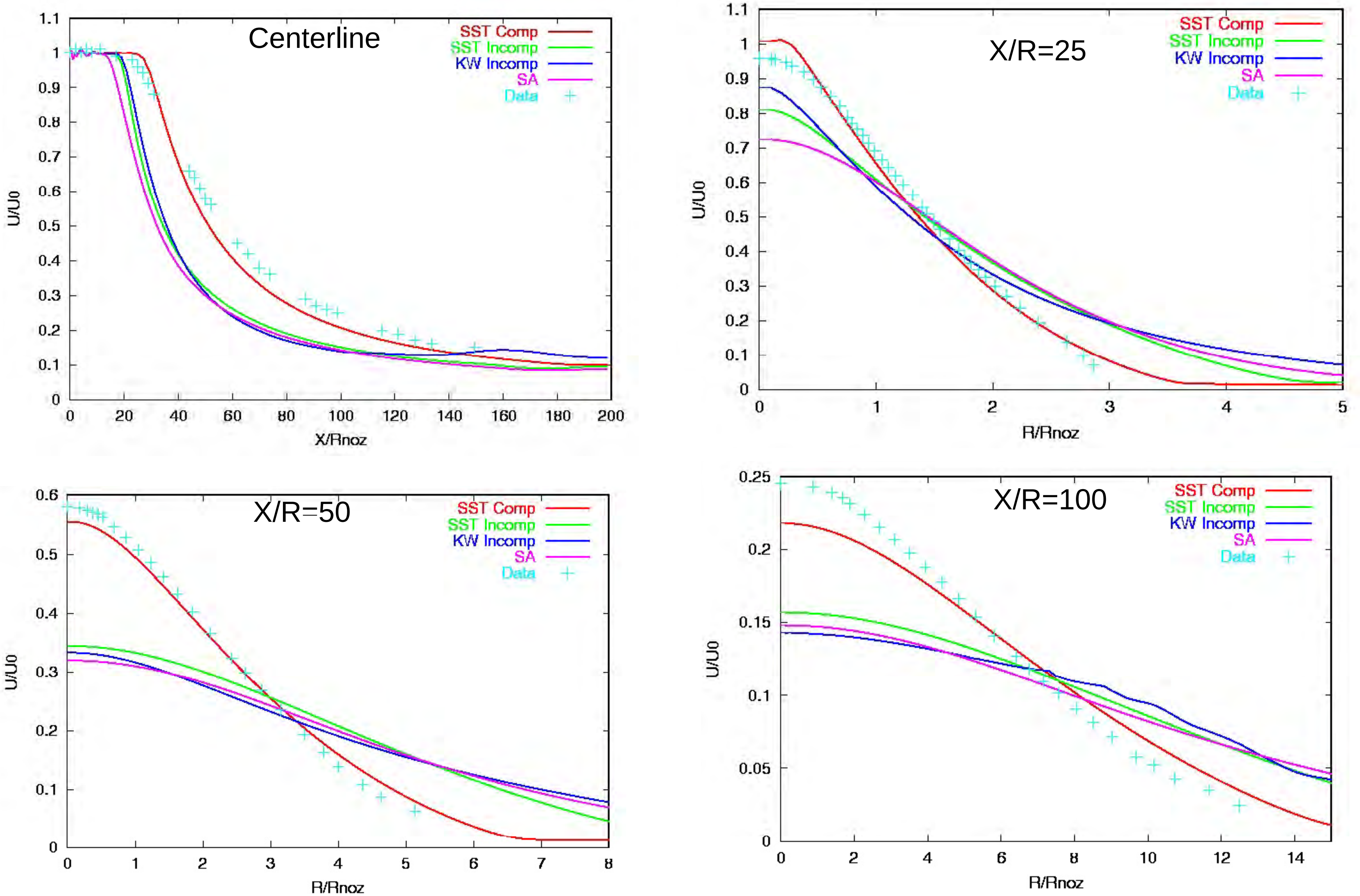
$$\frac{\partial \rho k}{\partial t}+\frac{\partial \rho U_i k}{\partial x_i}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+\rho P_k-\rho\left(\varepsilon+\boxed{\varepsilon_c}\right)+\boxed{\overline{p''d''}}$$

$$\varepsilon_c=\alpha_1 M_t^2 \varepsilon$$

$$\overline{p''d''}=-\alpha_2 \rho P_k M_t^2+\alpha_3 \rho \varepsilon M_t^2$$

$$\alpha_1=1.0, \alpha_2=0.4, \alpha_3=0.2 \qquad M_t=\sqrt{\frac{2k}{\gamma RT}}$$

Compressibility Corrections
Supersonic Jet (M=2.22)



2-Equation Transport Model Application Hints

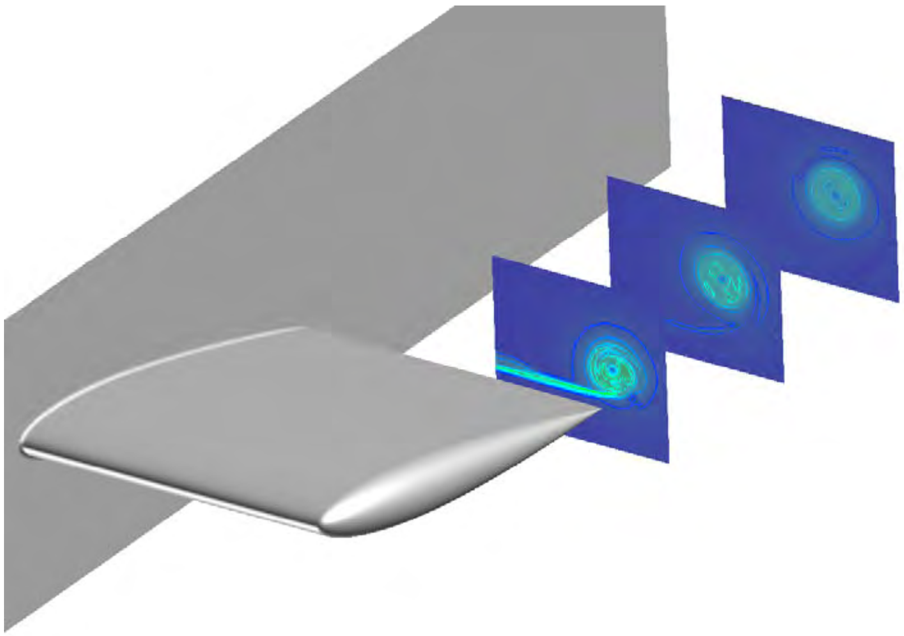
1. **The first point off the wall should be located about y+=1 to obtain reasonable skin friction values and about y+=0.5 for heat transfer.**
2. **The grid stretching normal to the wall should not exceed 1.3. A constant spacing should be used for the first three cells off the wall for heat transfer calculations.**
3. **The eddy viscosity should be limited so that it will not run away in some complex applications. Generally a limit of ν_t/ν=200,000 is acceptable.**
4. **Care should be taken not to divide viscous regions such as boundary layers when dividing the computational domain for blocked or overset applications since the model requires the distance from the nearest wall.**
5. **This model can overdamp some unsteady flows.**
6. **Compressibility corrections should be included for high-speed shear layer flows.**

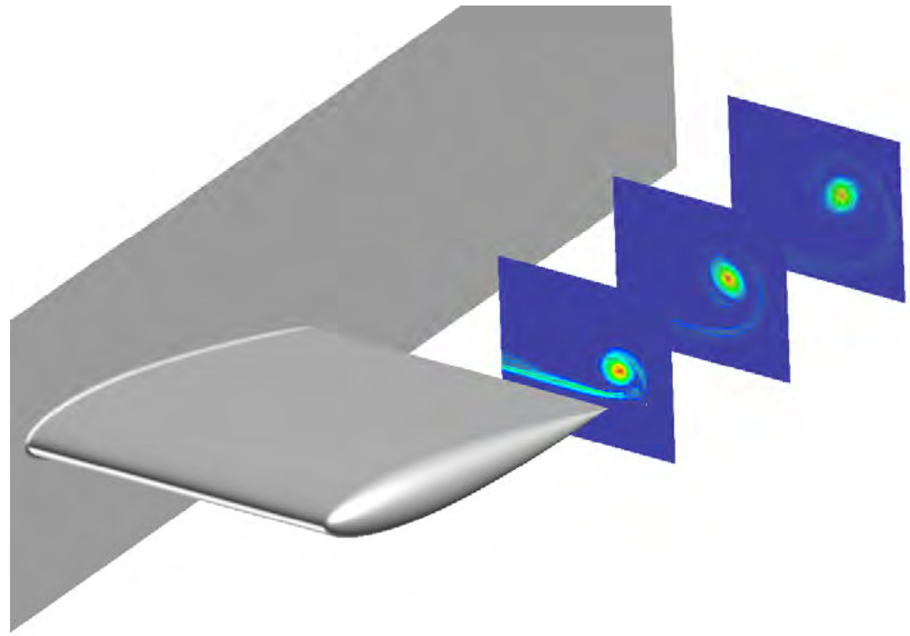
Rotation and Curvature Corrections

Why Do We Need R&C Corrections?

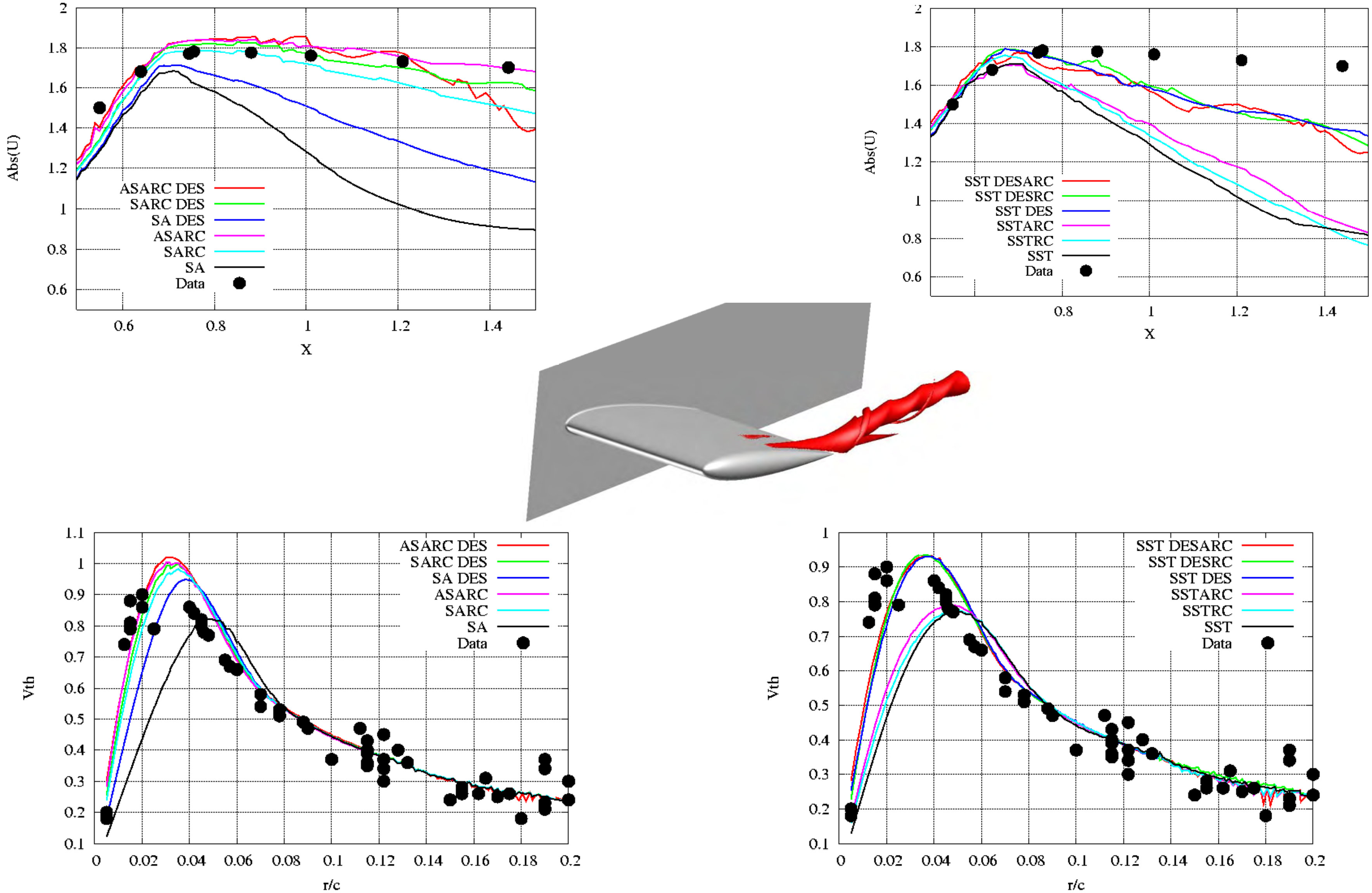
- SA and SST turbulence models are based on isotropic turbulence assumption
  - Curvature is a non-isotropic effect
- SA model uses vorticity in the production term
  - Vorticity reaches a local maximum in vortex core
  - Eddy viscosity also reaches a local maximum in the vortex core and overdamps the vortex core
- SST uses strain in the production term
  - Strain reaches a local minimum in a vortex core

Vorticity and Strain Magnitude





Wing Tip Vortex



Wall Functions

What are wall functions?

Solid wall boundary conditions based on curve fits from some point inside the boundary layer to the wall requiring functional expressions for:

- Velocity, pressure, and temperature
- Turbulence transport variables
- Wall shear stress and heat transfer
- Species

Why use wall functions?

- Improve solution turnaround by reducing the number of points in a solution
- Simplify grid generation
- Improve numerical stability
- Improve wall approximation with multigrid or grid sequencing methods

Profiles for determining τ_w and q_w

Spalding's equation with the outer velocity profile of White and Christoph

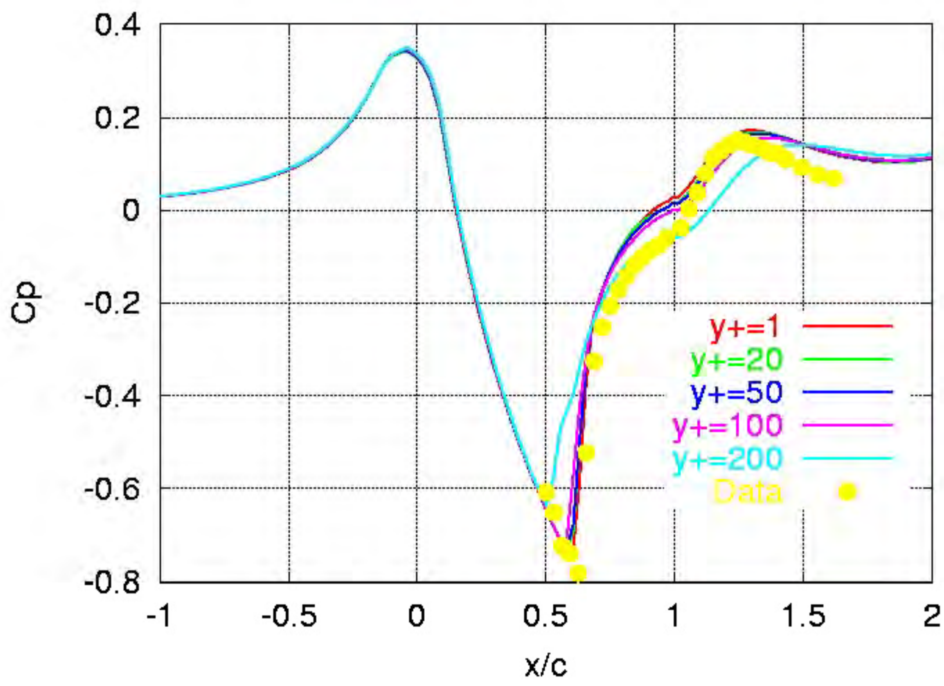
$$y^+ = u^+ + y^+_{White} - e^{-\kappa B}\left[1 + \kappa u^+ + \frac{(\kappa u^+)^2}{2} + \frac{(\kappa u^+)^3}{6}\right]$$

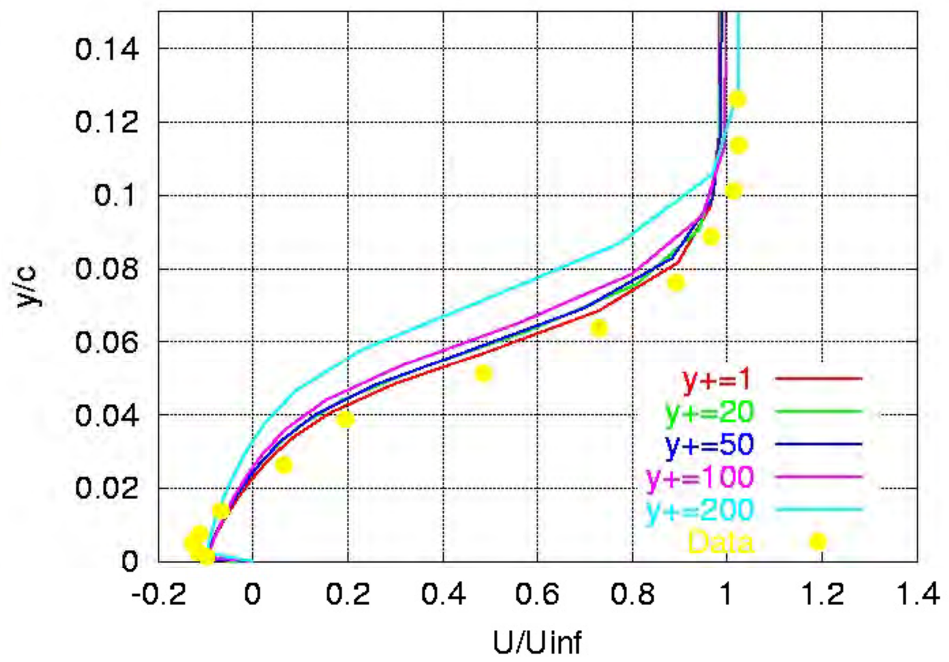
$$y^+_{White} = exp\left\{\frac{\kappa}{\sqrt{\Gamma}}\left[sin^{-1}\left(\frac{2\Gamma u^+ - \beta}{Q}\right) - \varphi\right]\right\} exp(-\kappa B)$$

Crocco-Busemann temperature profile

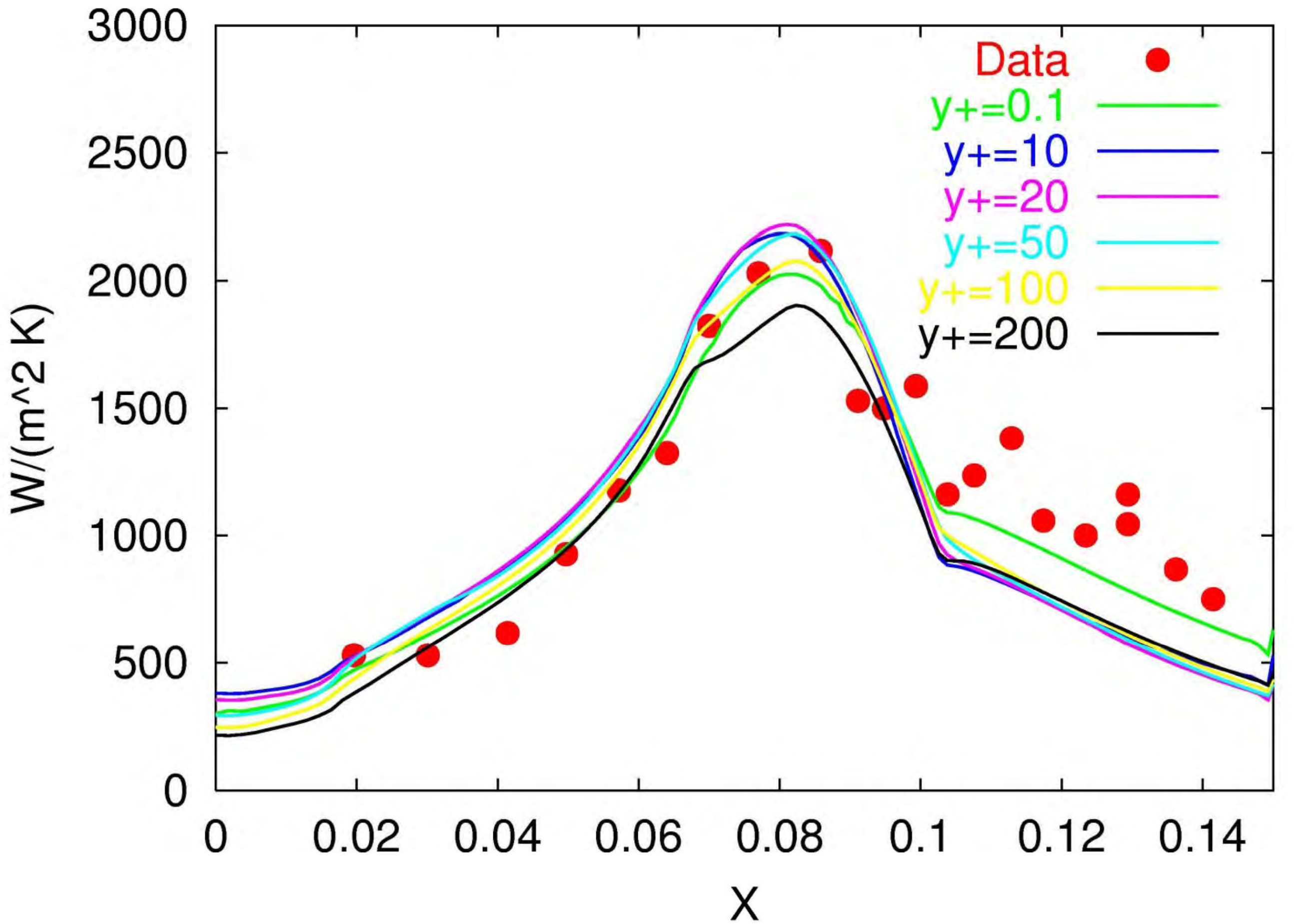
$$T = T_w\left[1 + \beta u^+ - \Gamma(u^+)^2\right]$$

Effect of Initial Spacing on Ames Axi Bump – SST Model with Wall Functions





Effect of Initial Grid Spacing on Nozzle Heat Transfer – SST with Wall Functions



Wall Function Application Hints

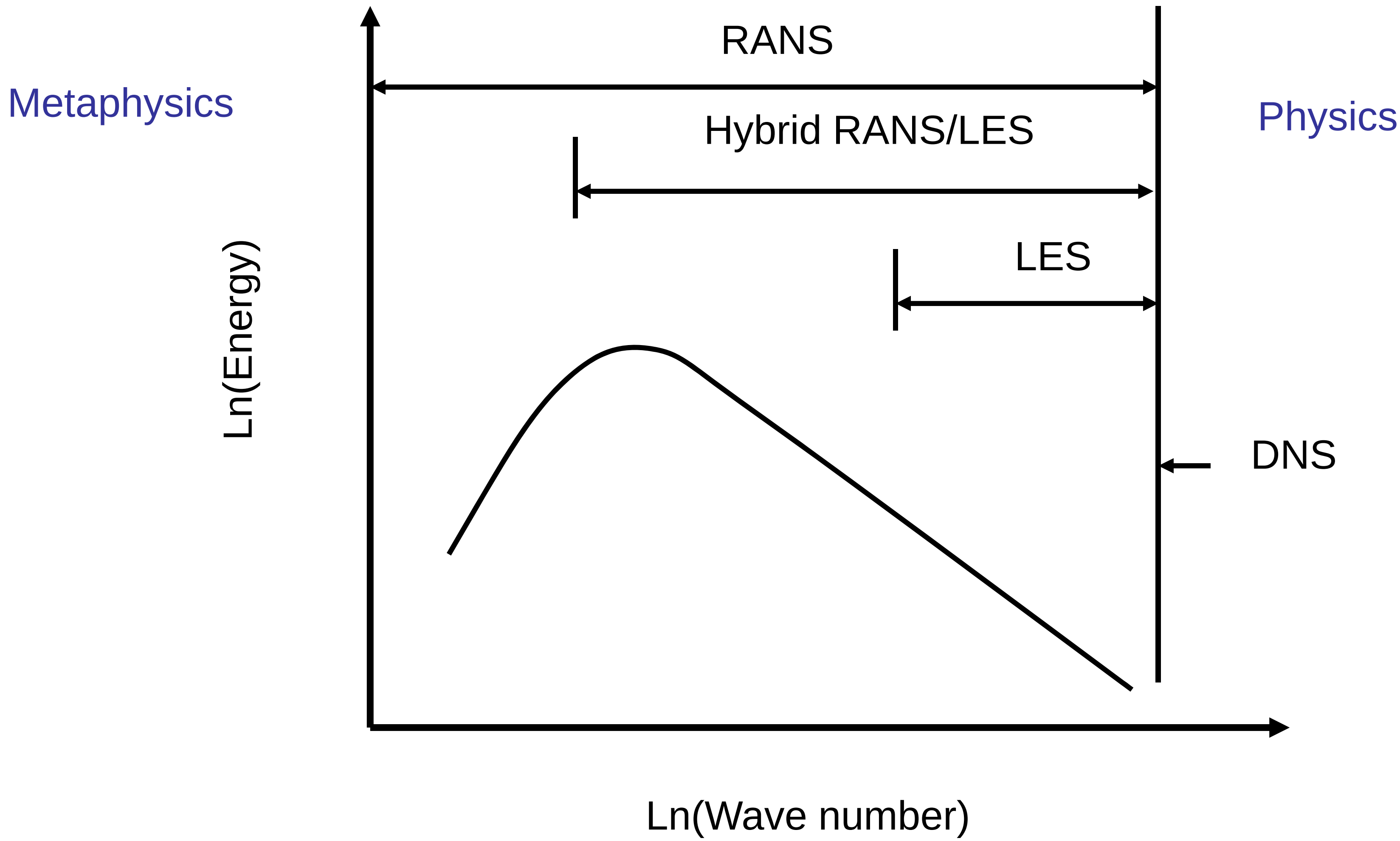
1. Wall functions are available for all the transport turbulence models.
2. The initial wall spacing should be about y+=50.
3. The automatic feature allows you to use wall functions in selected areas of a grid since it is controlled by wall spacing.
4. The force and moment coefficients generated by OVERFLOW 2 include the wall functions for calculating skin friction.
5. The wall functions must be included to accurately post-process skin friction or heat transfer.

Hybrid RANS/LES Unsteady Turbulence Models

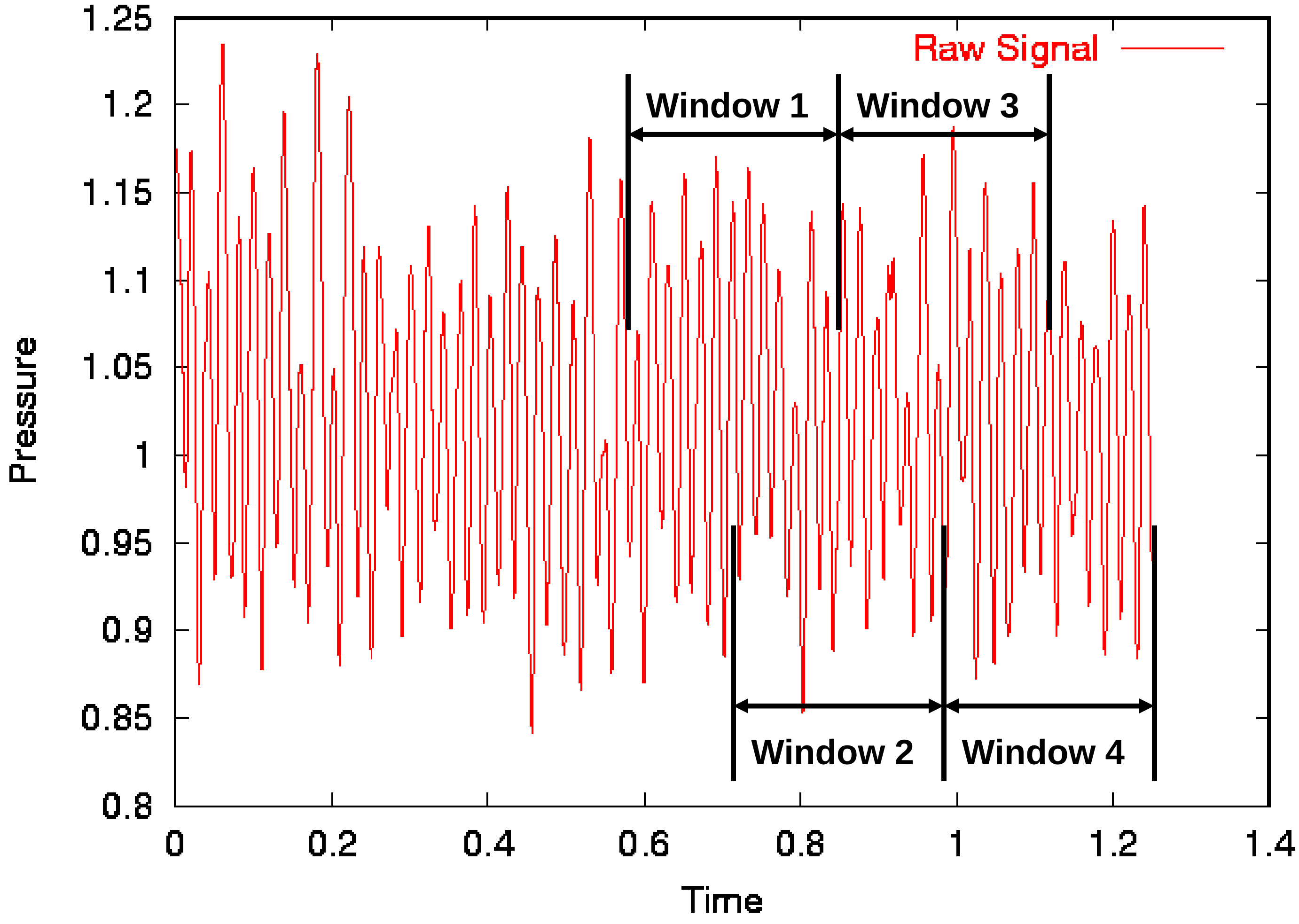
What are Hybrid RANS/LES Models?

- Similar to LES
  - Use a filtered RANS model as an LES subgrid model
  - Require low numerical dissipation in NS flux scheme
  - Only for unsteady applications
- Goal is to get RANS in boundary layer, LES everywhere else
- Applicable to flows with large-scale turbulent structures away from the walls
  - Vortex shedding
  - Weapons bays
  - Shear layers
- Solutions are grid and time step dependent
  - Must use statistical parameters to judge convergence
  - Work best for nearly isotropic grids

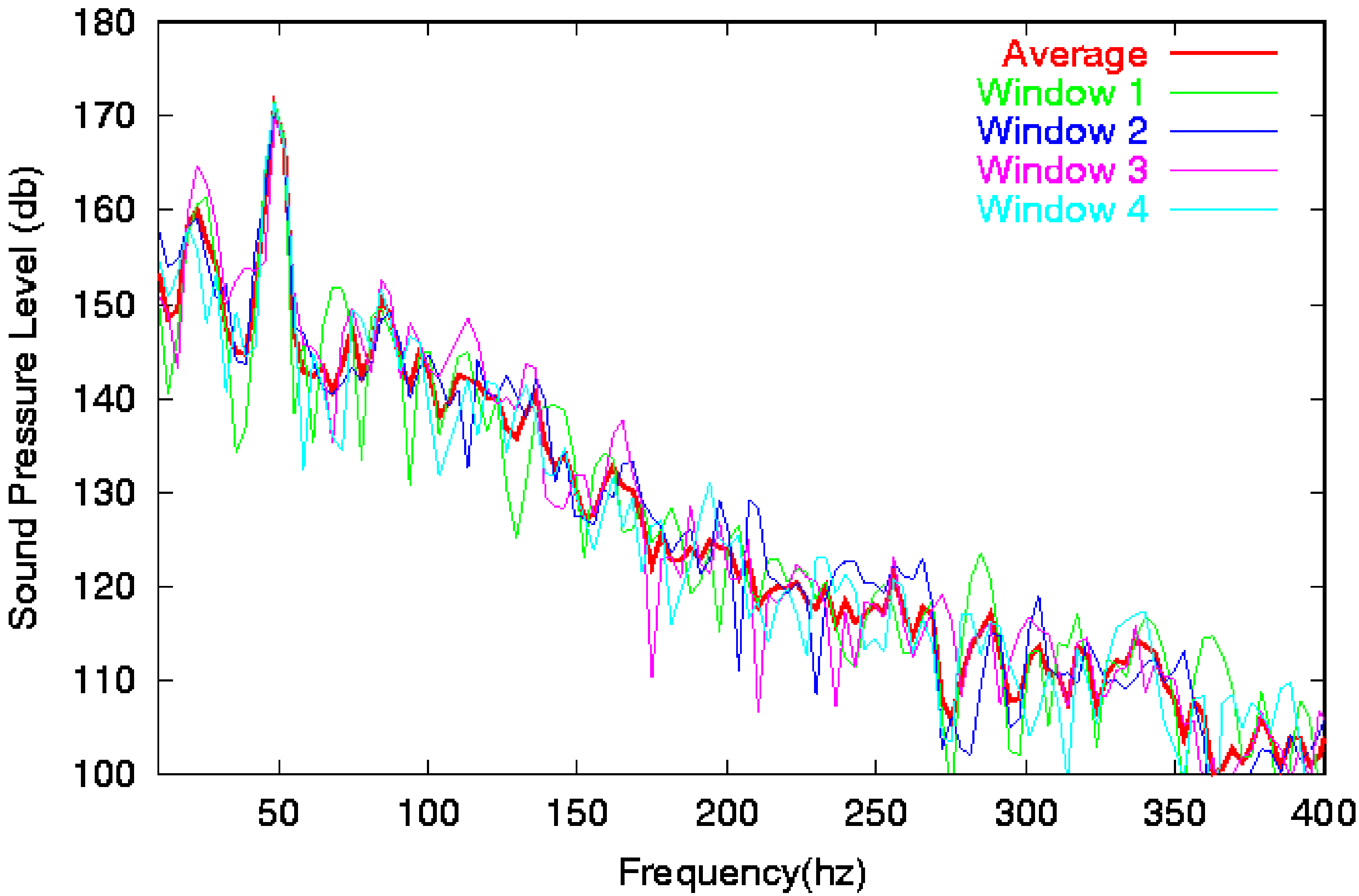
Turbulence Model Partitioning of the Energy Spectrum



Data Sample Windows



Spectral Variation



Statistics Summary

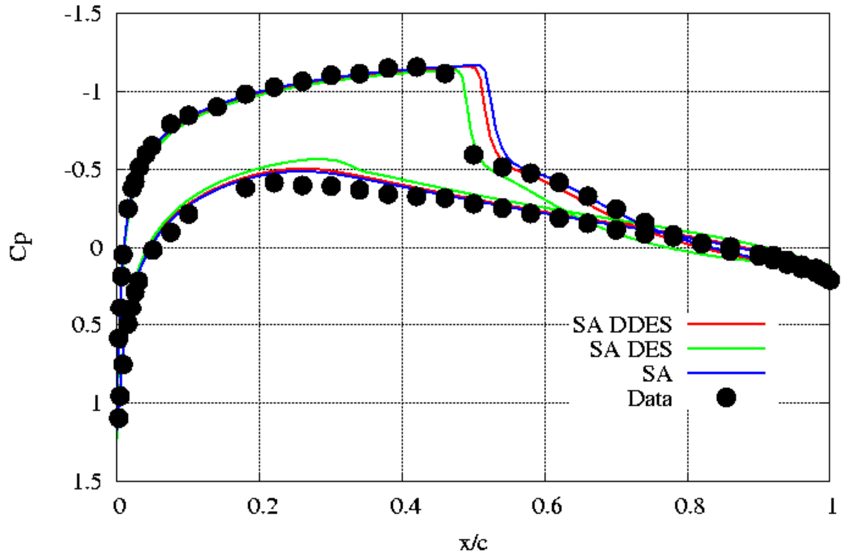
	Average Pressure	OASPL (db)	Peak SPL (db)	Frequency of Peak SPL (hz)
Window 1	1.02447	171	171	48.577
Window 2	1.02412	171	171	48.577
Window 3	1.01888	171	170	48.577
Window 4	1.02000	171	171	48.577
Average	1.02187	171	171	48.577
Max Error	0.29%	0.22%	0.747%	0.0%

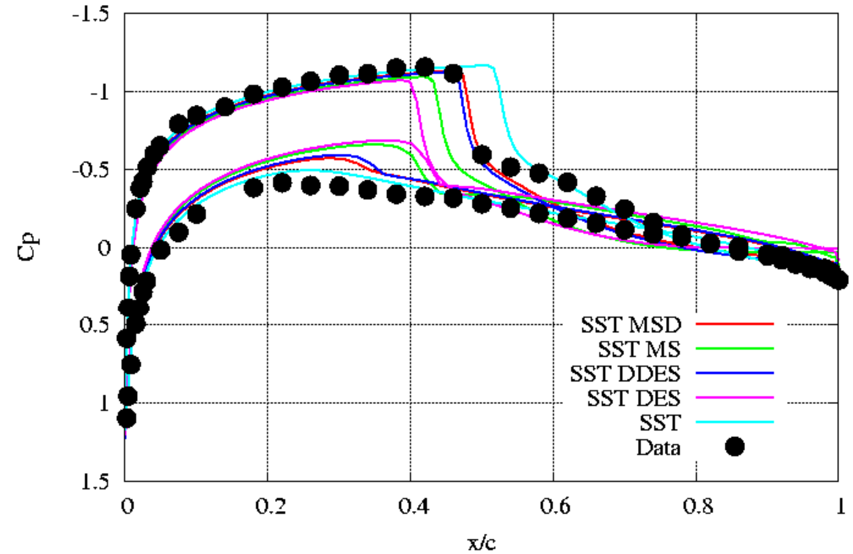
Detached Eddy Simulation (DES) And Multiscale (MS)

- SA – Modify destruction term by replacing distance from wall with local grid scale
  - Increases turbulent destruction as grid is refined
  - Becomes a Smagorinsky model in LES limit
  - Does not include a turbulent length scale
- SST – Modify destruction term in k equation to be a function of the ratio of the turbulent length scale ($k^{3/2}/\varepsilon$) to local grid scale
  - Adds additional turbulent destruction to k equation as grid is refined
- MS – Filters eddy viscosity as a function of the ratio of the turbulent length scale ($k^{3/2}/\varepsilon$) to local grid scale

Delayed Detached Eddy Simulation (DDES)

- DES and MS models tended to transition prematurely to LES in the boundary layer when grid becomes refined
- Can produce solutions that are neither RANS or LES
- DDES slows transition to LES in boundary layer using functions of turbulent length scale to distance from the wall





WICS Bay Experiment

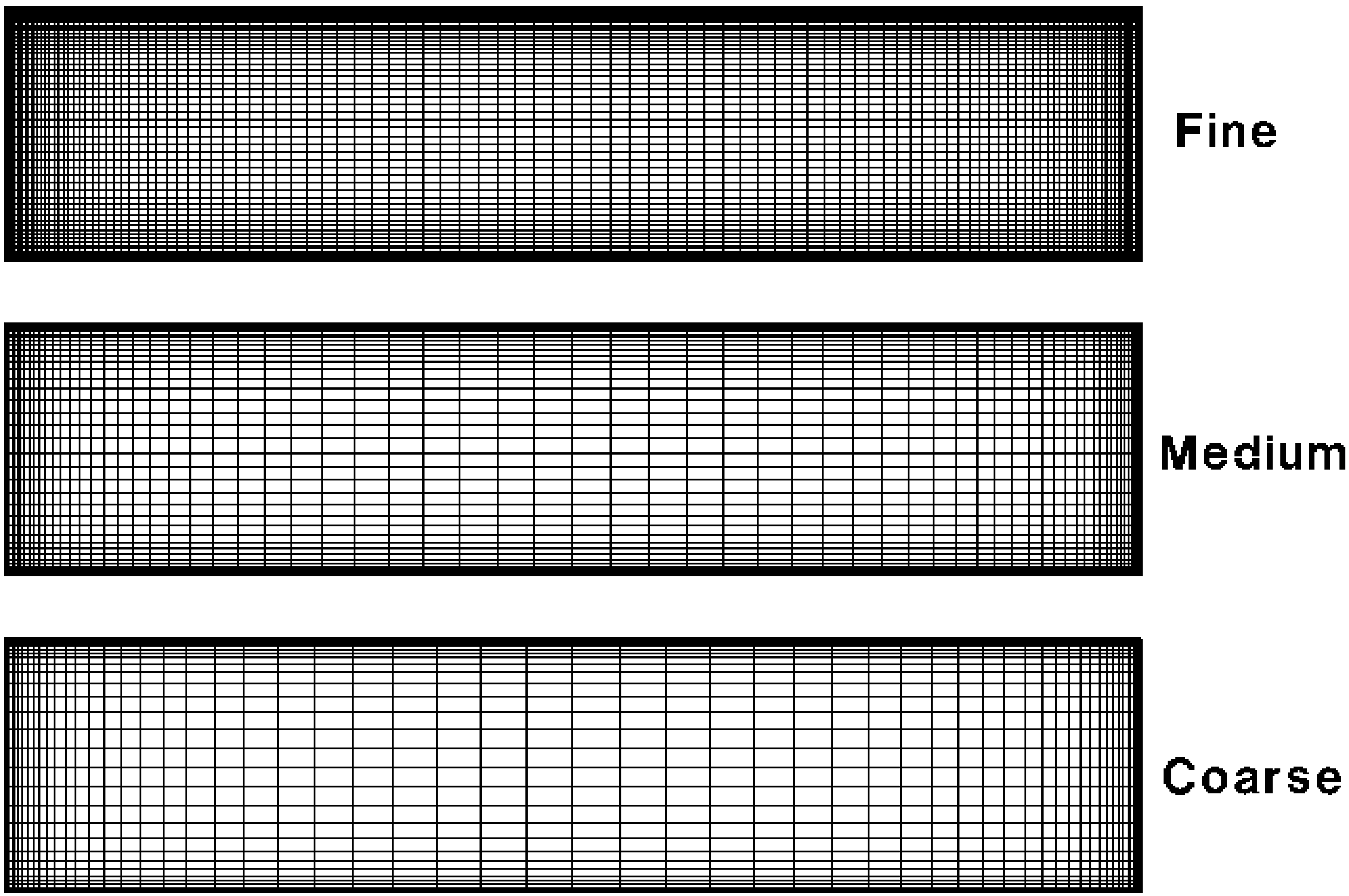
- Geometry
  - –Rectangular bay 18”x4”x4”
  - –15” flat plate in front bay
- Test Conditions
  - –M=0.95
  - –Re=2.5x10^6/ft
- Instrument Locations
  - –K16-bay ceiling centerline 0.275” from back wall
  - –K18-bay back wall centerline 0.725” from opening

WICS Bay Grid Systems

Grid	Total Points	Bay Grid Dimensions	Bay Grid Δx_{max}	Bay Grid Δy_{max}	Bay Grid Δz_{max}
Fine	1.8×10^6	121x61x61	0.3 in.	0.1 in.	0.1 in.
Medium	1.1×10^6	71x41x41	0.6 in.	0.2 in.	0.2 in.
Coarse	7.9×10^5	61x31x31	0.75 in.	0.3 in.	0.3 in.

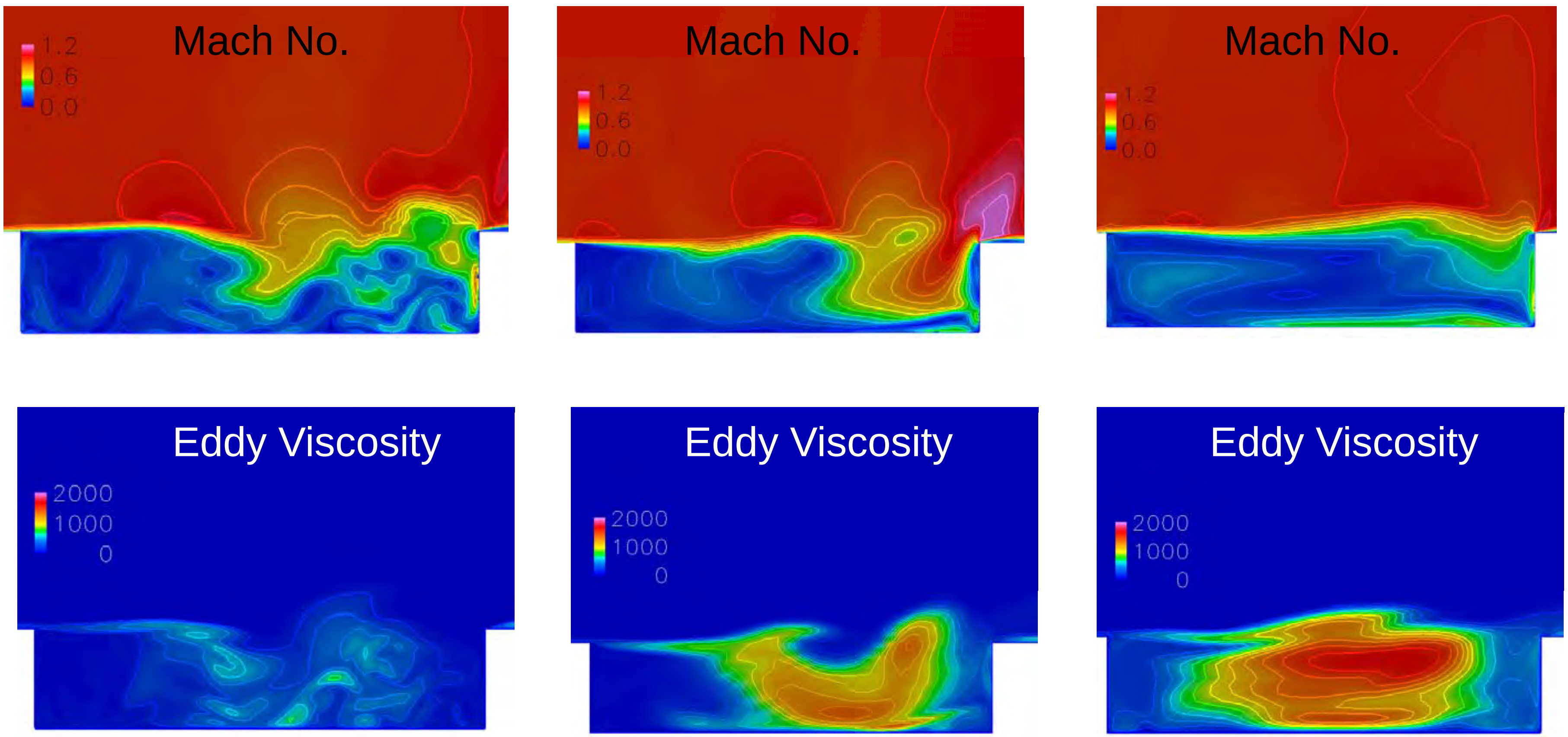
Wall Spacing of $y^+=50$ for all grids

WICS Bay Centerline Grids



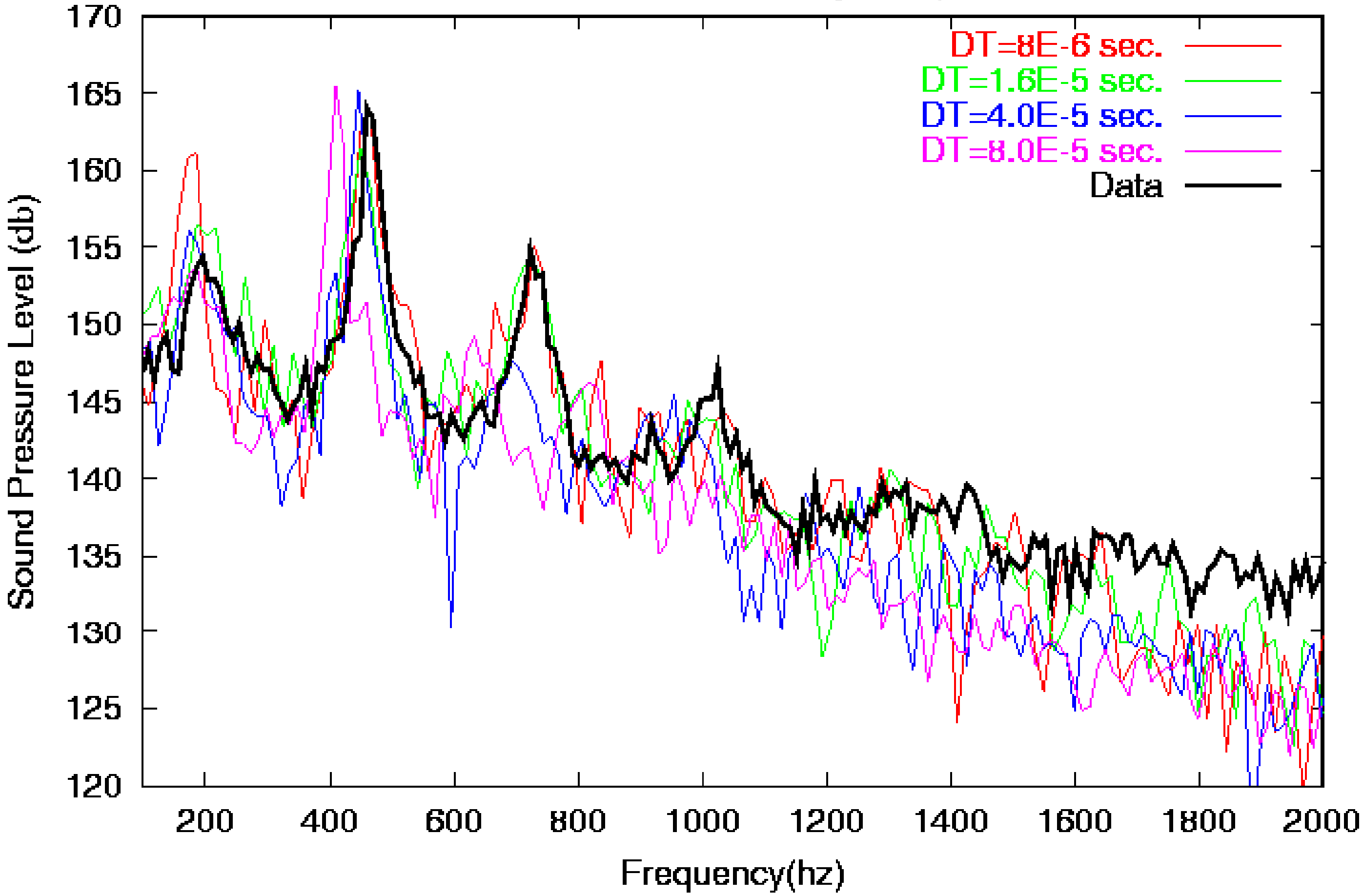
WICS Bay Centerline

FINE | Medium | Coarse

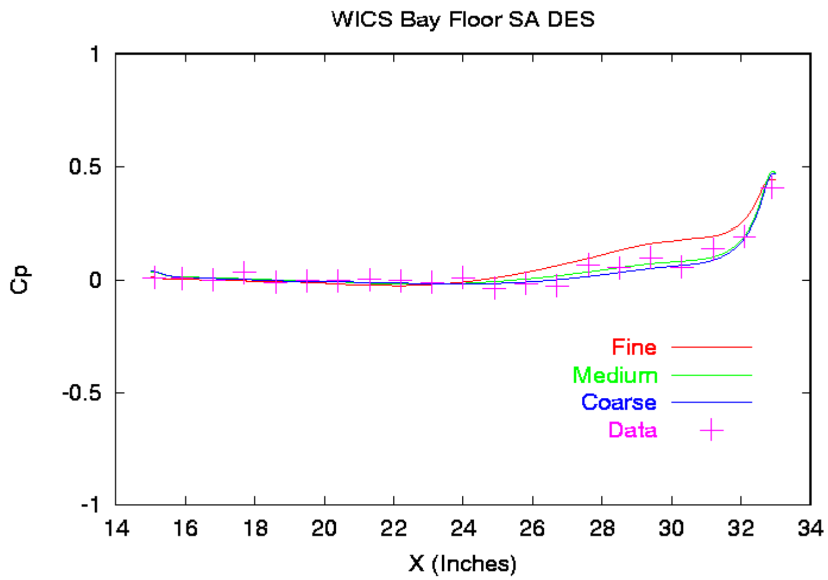


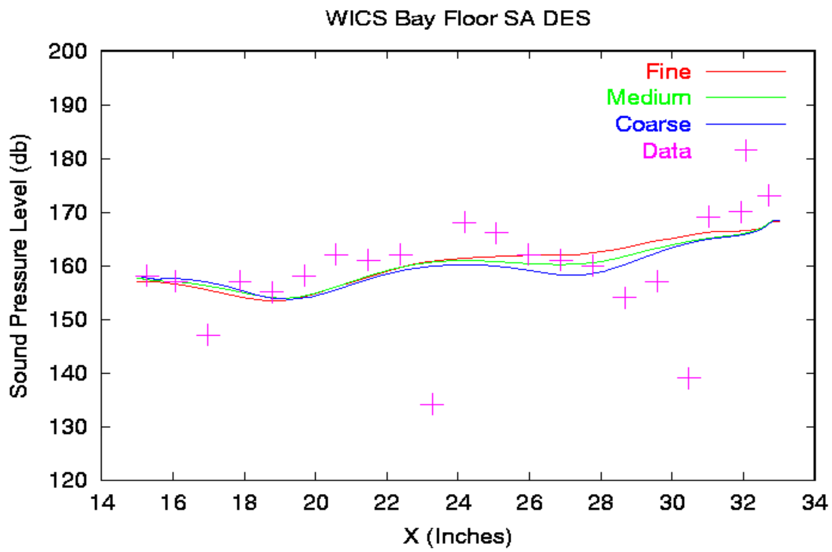
Time Step Study

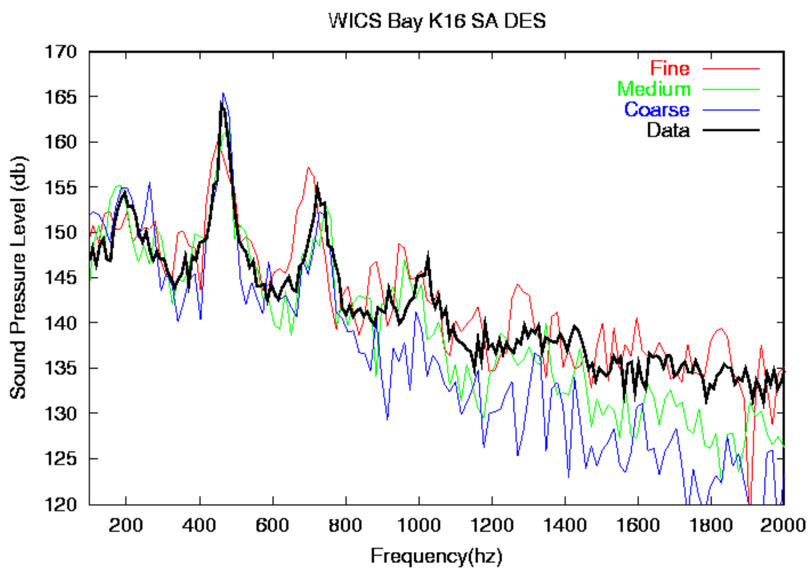
DT=8.0E-6 Sec.=260 Time Steps/Cycle
DT=1.6E-5 Sec.=130 Time Steps/Cycle
DT=4.0E-5 Sec.=53 Time Steps/Cycle
DT=8E-5 Sec.=26 Time Steps/Cycle

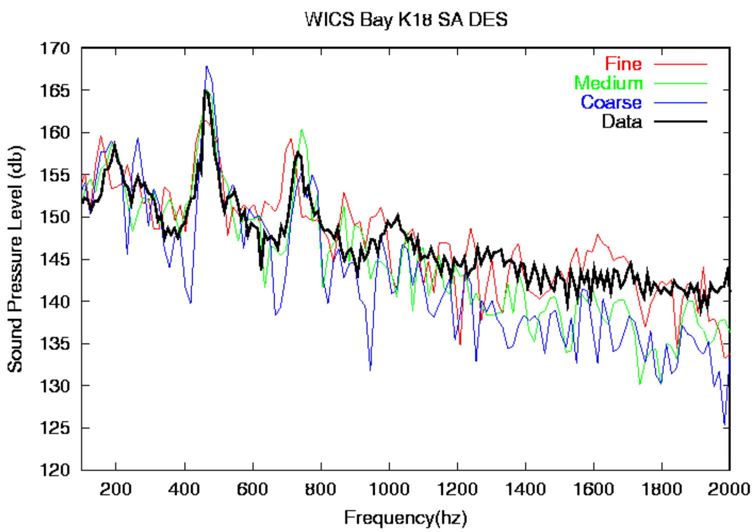


Time Averaged and Spectral Results









Hybrid RANS/LES Application Hints

1. These models are for unsteady applications, and should not be used with local time-stepping or with other non-time accurate algorithms.
2. Turbulent flows are three dimensional, and hence these models should be used only in 3D.
3. Because of the unsteady nature of these models, they may require a large number of time-steps to obtain a statistically stationary solution for analysis.
4. These models may be sensitive to the computational mesh because the filter function is inversely proportional to the grid spacing. A rule-of-thumb is that the ratio of the turbulent scale to grid length scale should be greater than two in the region of interest when using hybrid models.
5. As with all unsteady applications, care should be taken to be sure the time step is small enough to temporally capture the unsteady phenomena of interest.

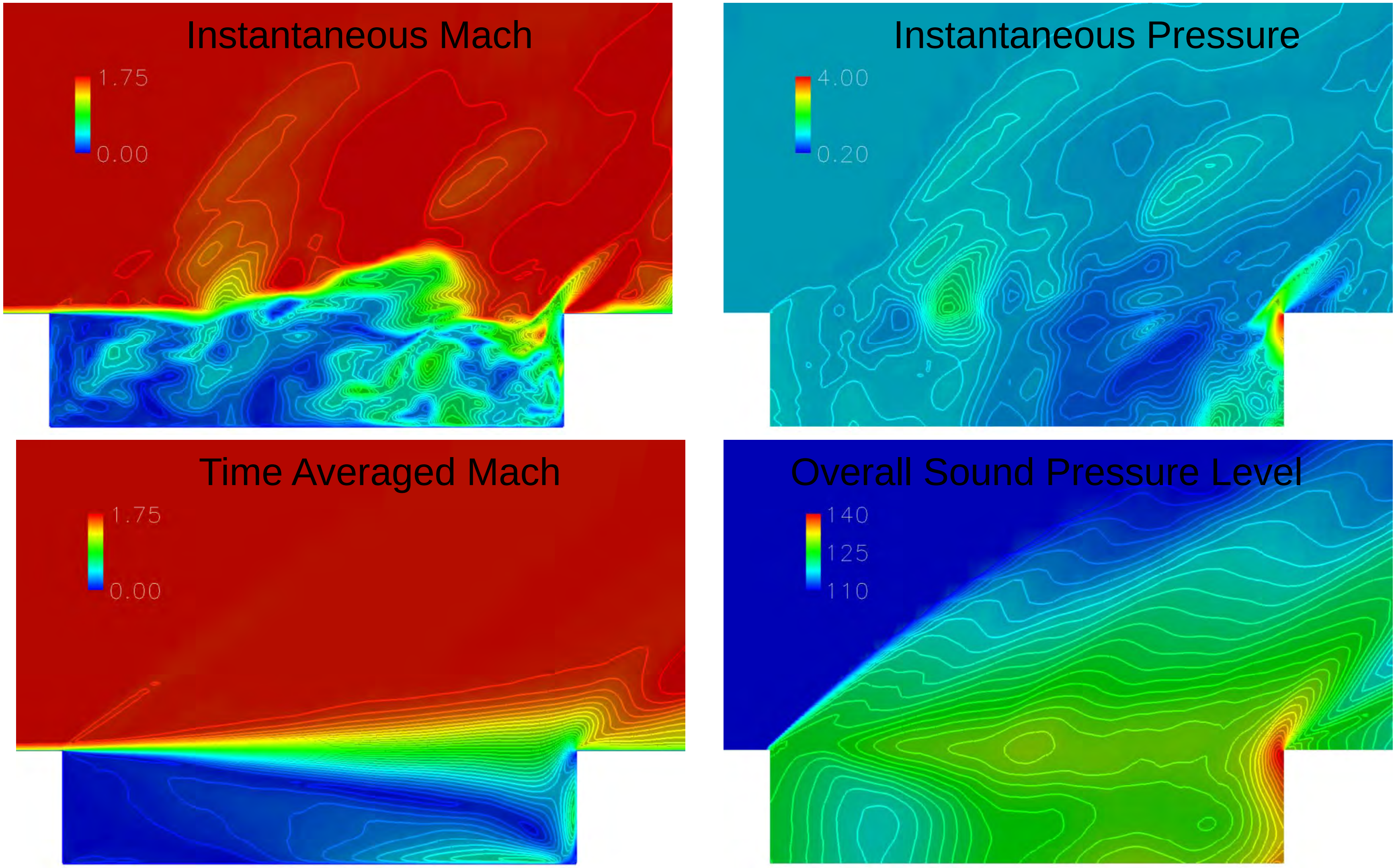
Unsteady Flow Outputs

Time Averaged Flow Output for Entire Grid System

- Code will compute a running average of the q variables and also ρ'^2, u'^2, v'^2, w'^2,and p'^2
- Averaging begins when solution reaches time step ISTART_QAVG
- Results are written to file q.avg
- q.avg file is overwritten each restart

```
$GLOBAL
     NSTEPS = 12000, ISTART_QAVG = 2000,
$END
```

WICS Bay M=1.75

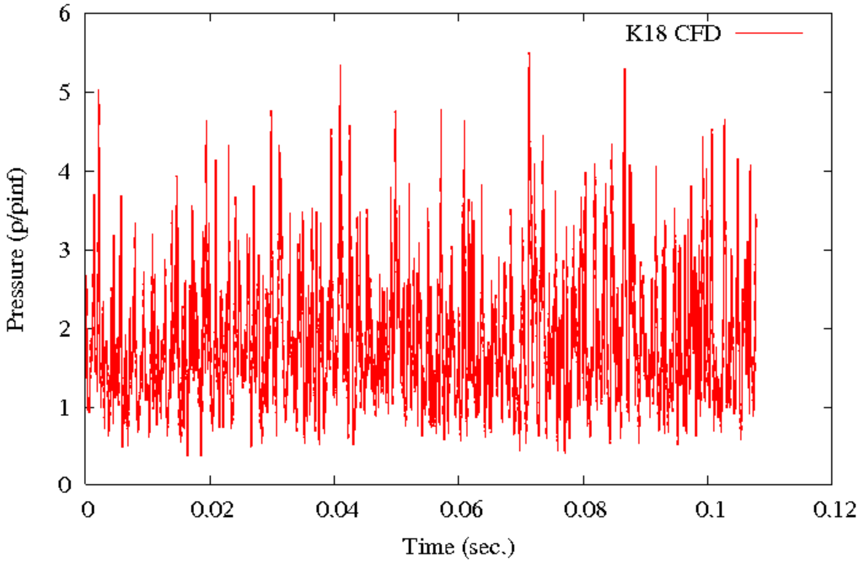


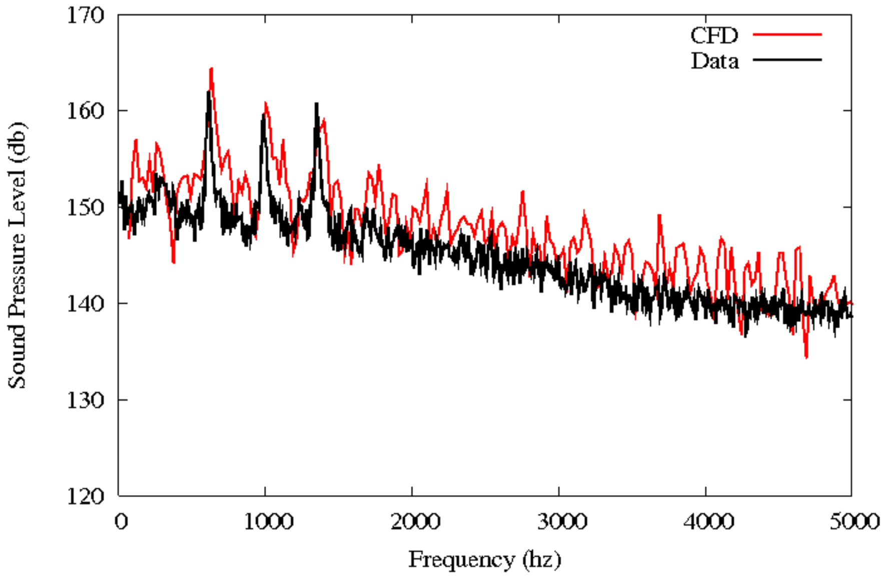
Unsteady Flow Output for Region

- IBTYPE = 201 will write the q values at each iteration for a given region of the flow to a file
- File name can be input using BCFILE
- Default name is BC_201.mesh.bc#

```
$BCINP
    IBTYP = 201,  201,  201,
    IDIR =     1,    1,    1,
    JBCS =     1,    1,    1,
    JBCE =     1,   -1,   -1,
    KBCS =     1,    1,    1,
    KBCE =     1,    1,   -1,
    LBCS =     1,    1,    1,
    LBCE =     1,    1,   -1,
$END
```

WICS Bay M=1.75



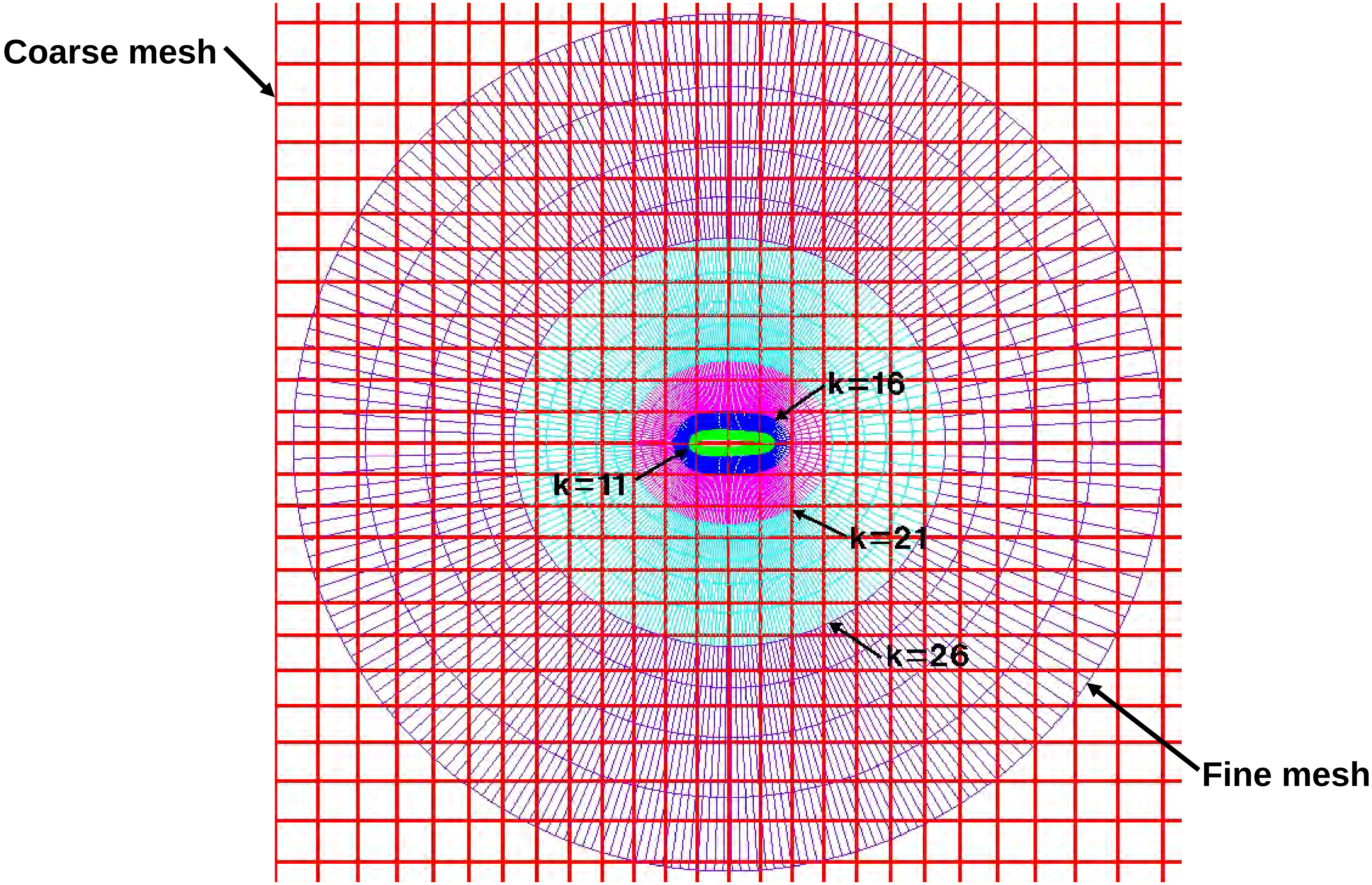


Output Information for Moving Body Simulations

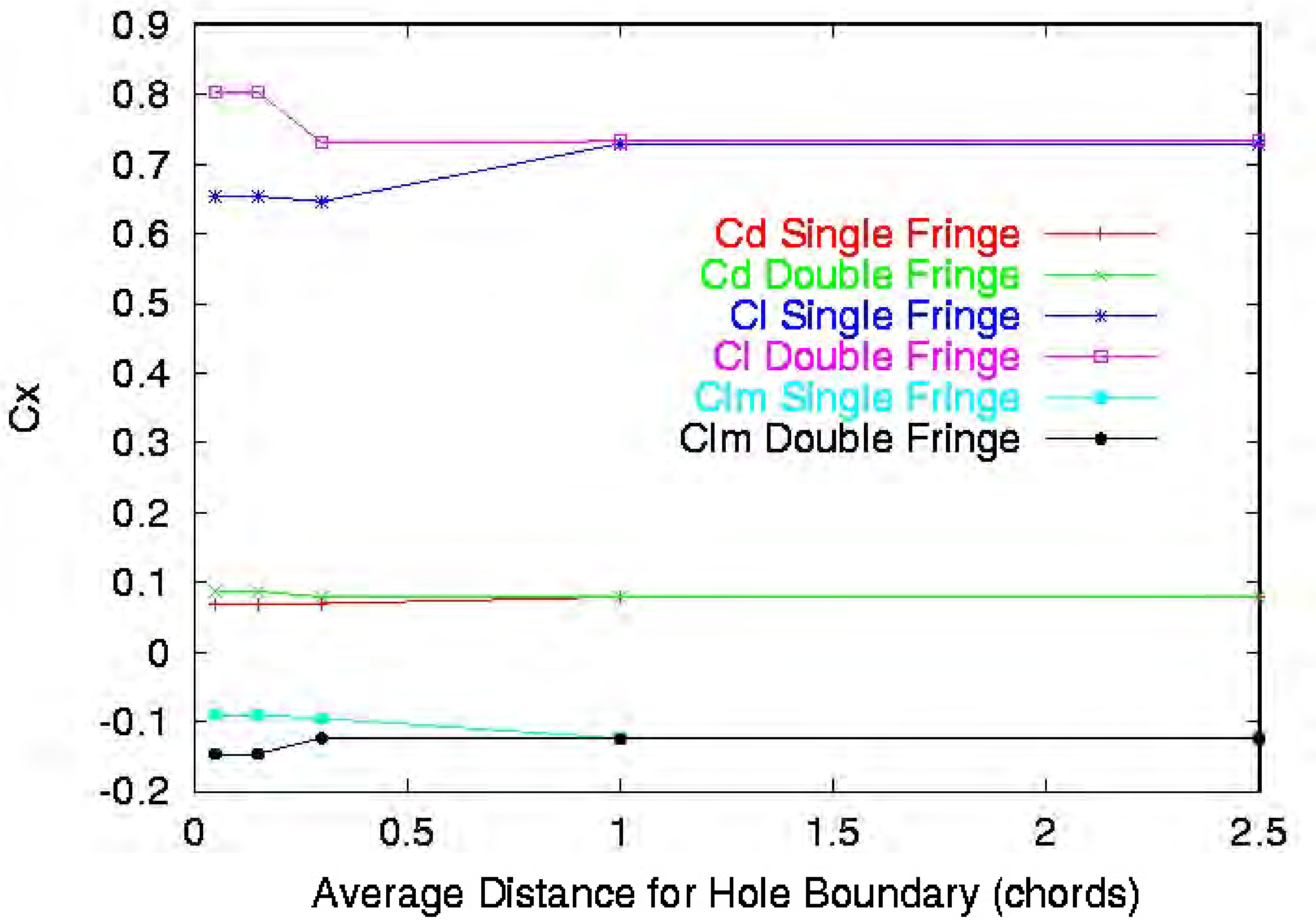
- Input parameters in **$GLOBAL**
  - **NSAVE** – grid system, flow solution, and 6-DOF restart information save every **NSAVE** steps as **x.step#**, **q.step#**, **sixdof.step#**
  - **NFOMO** – force and moment coefficients are written to fomoco.out every **NFOMO** steps (automatically set to 1 for 6-DOF simulations)
- NAMELIST **$SPLITM**: write subsets of grid and solution every nsteps (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# 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(sunset#), 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

Overset Considerations for RANS Turbulence Models

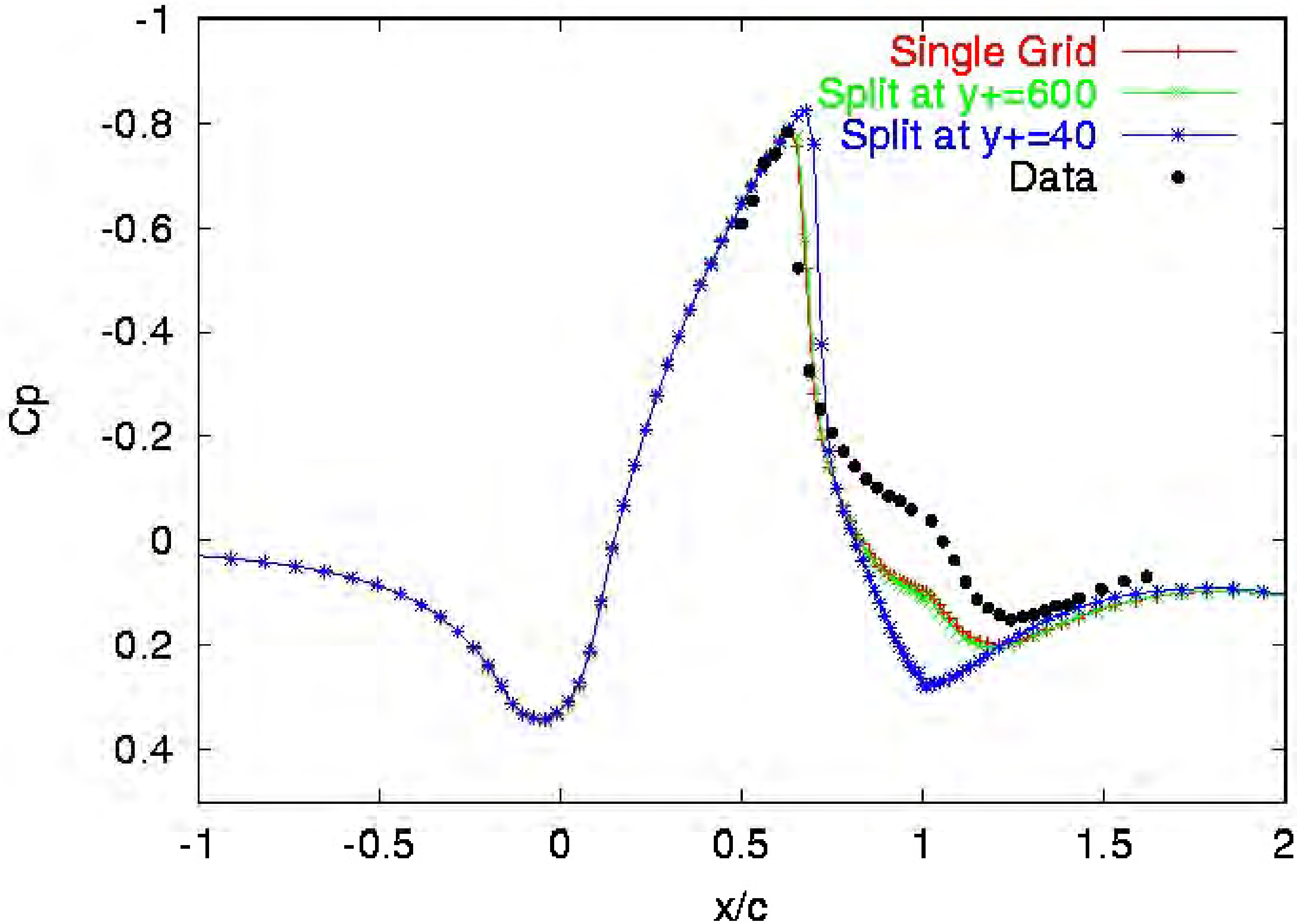
Chimera Domain Decomposition NACA 0012 Example



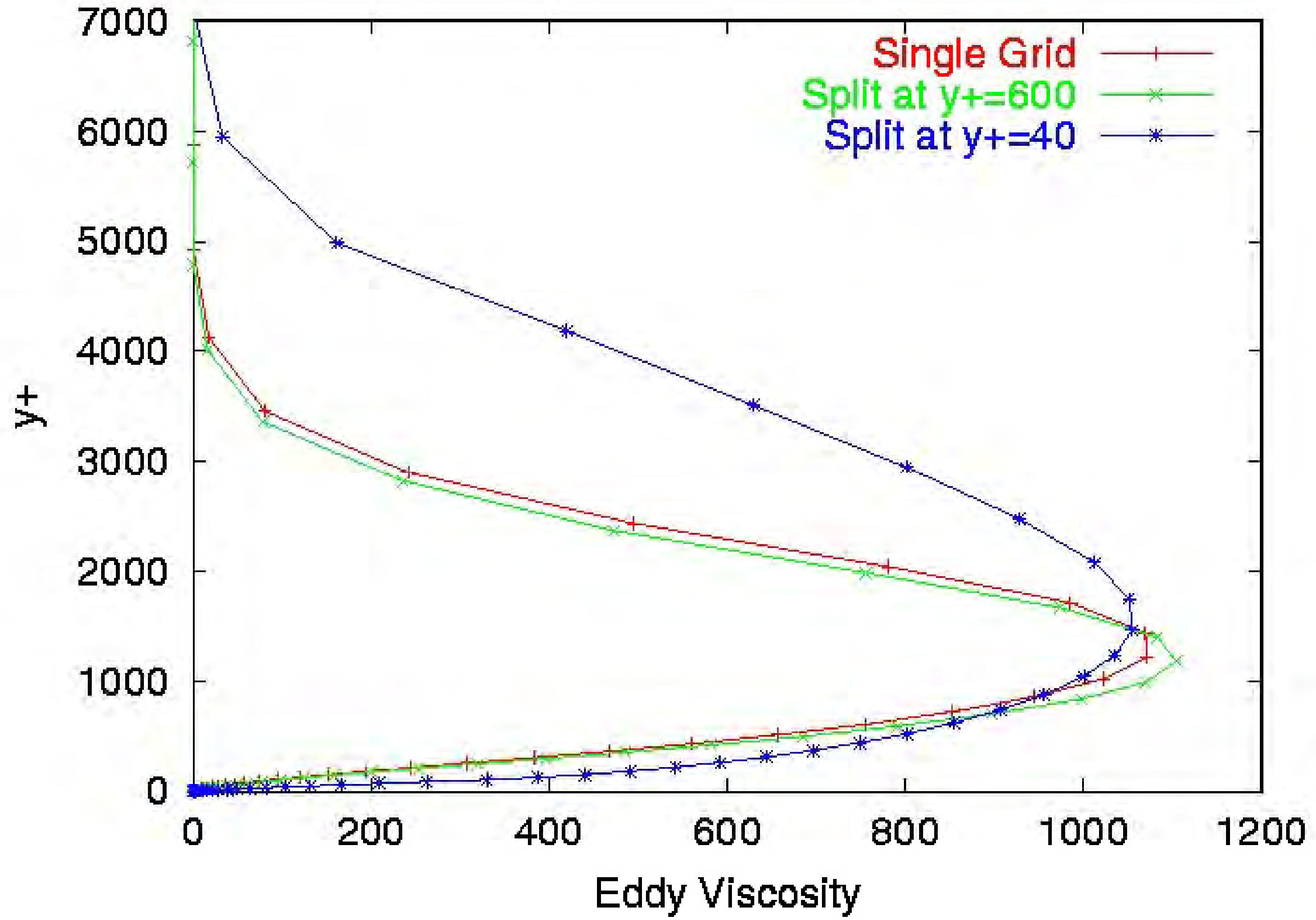
Chimera Domain Decomposition NACA 0012 Example



Chimera Domain Decomposition
Ames Axi Bump Example
WALLDIST = 1



Chimera Domain Decomposition
Ames Axi Bump Example
WALLDIST = 1



OVERSET Application Hints

1. Holes should be cut as far from a body as possible. It is highly desirable to match the cell sizes in the overlap region.
2. Point injected boundaries are preferable if possible since they allow a conservative exchange of information between the computational domains.
3. Double fringe stencils are preferable.
4. Care should be taken to ensure that the turbulence model has all the information it needs within its own domain. Wall distances are required by many turbulence models and the walls that affect a domain should be included in the domain. For algebraic models care should be taken not to split the grid such that the profile from which the eddy viscosity is derived is split.

References

1. Murphy, K.J., Buning, P.G., Pamadi, B.N., Scallion, W.I., and Jones, K.M., “Overview of Transonic to Hypersonic Stage Separation Tool Development for Multi-Stage-to-Orbit Concepts,” AIAA-2004-2595, June 2004.
2. Buning, P.G., Gomez, R.J., and Scallion, W.I., “CFD Approaches for Simulation of Wing-Body Stage Separation,” AIAA-2004-4838, Aug. 2004.
3. Buning, P.G., Chiu, I.T., Obayashi, S., Rizk, Y.M., and Steger, J.L., “Numerical Simulation of the Integrated Space Shuttle Vehicle in Ascent,” AIAA-88-4359, Aug. 1988.
4. Renze, K.J., Buning, P.G., and Rajagopalan, R.G., “A Comparative Study of Turbulence Models for Overset Grids,” AIAA-92-0437, Jan. 1992.
5. Meakin, R.L., “Object X-Rays for Cutting Holes in Composite Overset Structured Grids,” AIAA-2001-2537, June 2001.
6. Meakin, R.L., “Automatic Off-Body Grid Generation for Domains of Arbitrary Size,” AIAA-2001-2536, June 2001.
7. Chan, W.M., Meakin, R.L., and Potsdam, M.A., “CHSSI Software for Geometrically Complex Unsteady Aerodynamic Applications,” AIAA-2001-0539, Jan. 2001.
8. Meakin, R.L., “An Efficient Means of Adaptive Refinement within Systems of Overset Grids,” AIAA-95-1722, June 1995.
9. Meakin, R.L., “On Adaptive Refinement and Overset Structured Grids,” AIAA-97-1858, June 1997.
10. Benek, J.A., Buning, P.G., and Steger, J.L., “A 3-D CHIMERA Grid Embedding Technique,” AIAA-85-1523, July 1985.
11. Pulliam, T.H., and Chaussee, D.S., “A Diagonalized Form of an Implicit Approximate Factorization Algorithm,” *J. Comp. Physics*, Vol. 39, 1981.
12. Kandula, M., and Buning, P.G., “Implementation of LU-SGS Algorithm and Roe Upwinding Scheme in Overflow Thin-Layer Navier-Stokes Code,” AIAA-94-2357, June 1994.
13. Jespersen, D.C., Pulliam, T.H., and Buning, P.G., “Recent Enhancements to OVERFLOW (Navier-Stokes Code),” AIAA-97-0644, Jan. 1997.
14. Jespersen, D.C., “Parallelism and OVERFLOW,” NAS Technical Report NAS-98-013, NASA Ames Research Center, Moffett Field, CA, Oct. 1998.
15. Liou, M.S., and Buning, P.G., “Contribution of the Recent AUSM Schemes to the OVERFLOW Code: Implementation and Validation,” AIAA-2000-4404, June 2000.
16. Pandya, S.A., Venkateswaran, S., and Pulliam, T.H., “Implementation of Preconditioned Dual-Time Procedures in OVERFLOW”, AIAA-2003-0072, Jan. 2003.
17. Nichols, R., Tramel, R., and Buning, P., “Solver and Turbulence Model Upgrades to OVERFLOW 2 for unsteady and High-Speed Applications,” AIAA-2006-2824, June 2006.
18. Beam, R., and Warming, R.F., “An Implicit Finite-Difference Algorithm for Hyperbolic Systems in Conservation Law Form,” *J. Comp. Physics*, Vol. 22, Sep. 1976, pp 87-110.
19. Klopfer, G.H., Hung, C.M., Van der Wijngaart, R.F., and Onufer, J.T., “A Diagonalized Diagonal Dominant Alternating Direction Implicit (D3ADI) Scheme and Subiteration Correction,” AIAA-98-2824, June 1998.
20. Steger, J.L., and Warming, R.F., “Flux-Vector Splitting of the Inviscid Gasdynamics Equations with Application to Finite Difference Methods,” *J. Comp. Physics*, Vol. 40, No. 2, Apr. 1981, pp 263-293.
21. Nichols, R., Tramel, R., and Buning, P., “Evaluation of Two High-Order Weighted Essentially Nonoscillatory Schemes,” *AIAA Journal*, Vol. 46, No. 12, pp 3090-3102.
22. Nichols, R. and Heikkinen, B., “Validation of Implicit Algorithms for Unsteady Flows Including Moving and Deforming Grids,” *Journal of Aircraft*, Vol. 43, No. 5, pp 1341-1351.
23. Nichols, R., “Comparison of Hybrid RANS/LES Turbulence Models for a Circular Cylinder and a Cavity,” *AIAA Journal*, Vol. 44, No. 6, pp 1207-1219.
24. Nichols, R. and Nelson, C., “Wall Function Boundary Conditions Including Heat Transfer and Compressibility,” *AIAA Journal*, Vol. 42, No. 6, pp 1107-1114.
25. Tramel, R., Nichols, R., and Buning, P., “Addition of Improved Shock-Capturing Schemes to OVERFLOW 2.1,”, AIAA-2009-3988., Jul. 2009.
26. Nichols,R., “Algorithm and Turbulence Model Requirements for Simulating Vortical Flows,” AIAA-2008-0337, Jan. 2008.