Getting Started With SU2

Code Basics, Installation, & RANS Analysis

Last Updated: feb 3rd, 2017 for SU2 v5.0.0

Sravya Nimmagadda, Thomas D. Economon

Department of Aeronautics & Astronautics

Stanford University

Downloading the Code

Multiple options…

1. Check out the download portal on the main website to register and download binaries or source for v5.0:

   http://su2.stanford.edu/download.html

2. Download releases from GitHub here:

   https://github.com/su2code/SU2/releases

3. **Recommended**: Clone the open repository directly at the command line to get the latest release (the master branch is always stable):

   ```
   $ git clone https://github.com/su2code/SU2.git
   ```

SU2 v5.0 “Raven” was released on Jan 19, 2017

GETTING STARTED

BUILDING SU2

ANALYSIS EXAMPLE:
ONERA M6 WING

Customizing and Compiling from Source

1. Clone the code, then move into the source directory.

   ```
   $ git clone https://github.com/su2code/SU2.git
   $ cd SU2
   ```

2. Specify various options, compilers, etc. and run configure. The following options are typical for high-performance/parallel operation.

   ```
   $ ./configure --enable-mpi --with-cc=mpicc --with-cxx=mpicxx CXXFLAGS='-O3' —prefix=/home/sravya/SU2
   ```

3. Make and install the code (location specified by the prefix option above). Note that this could be done in two separate steps, as “make” and “make install” consecutively at the command line.

   ```
   $ make -j 8 install
   ```

4. Update your PATH variables appropriately before executing.

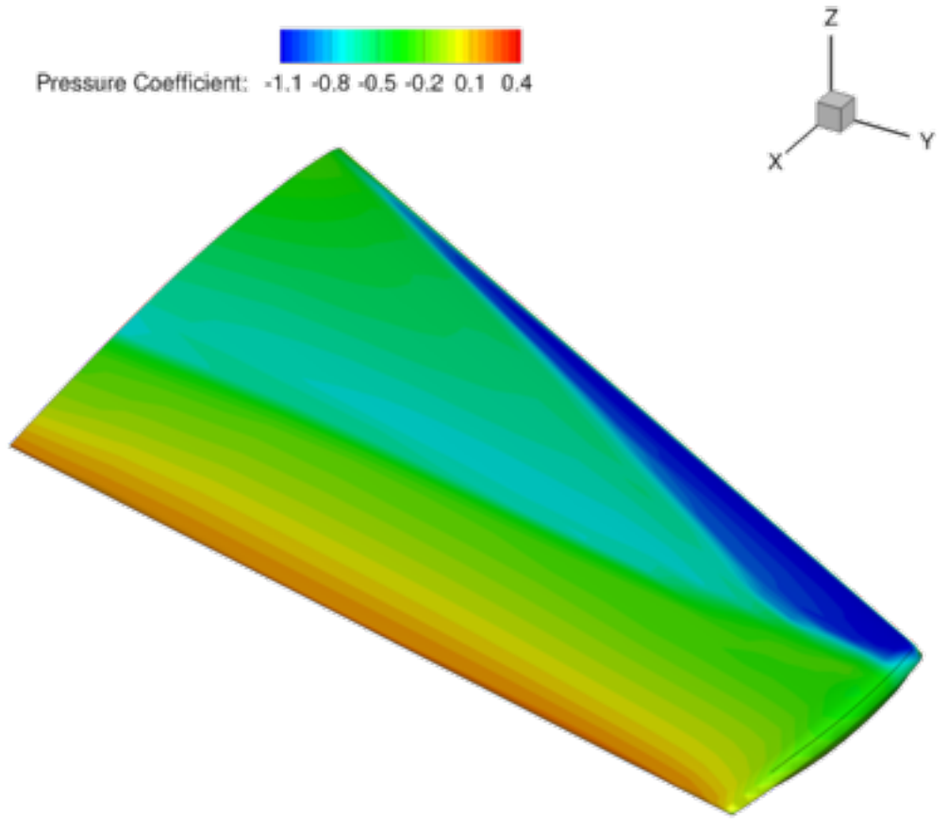
   ```
   $ export SU2_RUN=“/home/sravya/SU2/bin”
   $ export SU2_HOME=“/home/sravya/SU2”
   $ export PATH=$SU2_RUN:$PATH
   $ export PYTHONPATH=$SU2_RUN:$PYTHONPATH
   ```

GETTING STARTED

BUILDING SU2

ANALYSIS EXAMPLE:
ONERA M6 WING

NEXT STEPS



ONERA M6 TestCase

- Config and mesh files - Tutorial1_OneraM6Files.zip

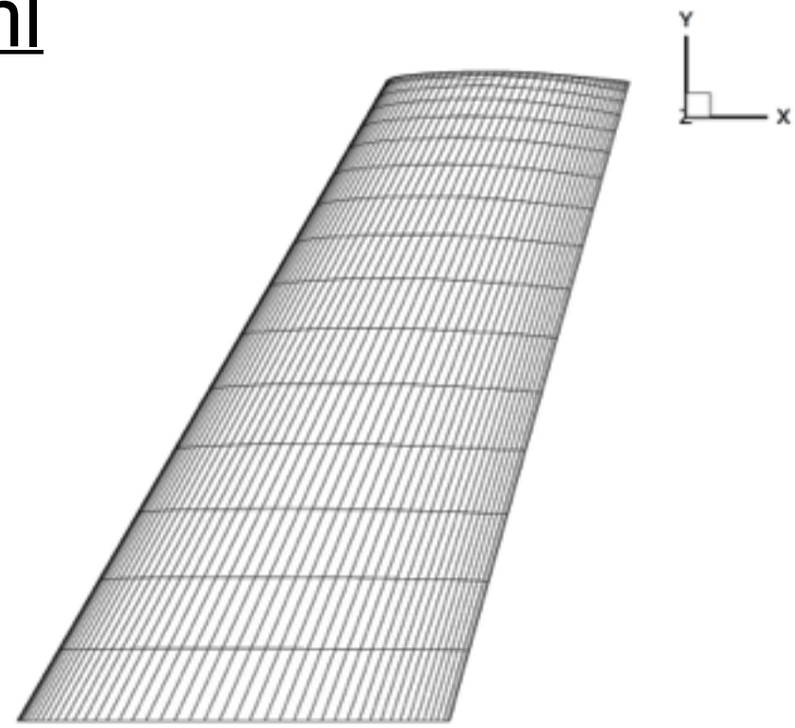
  http://su2.stanford.edu/training.html

The files for this case came with the slides. Run the case in serial with:

$ SU2_CFD turb_ONERAM6.cfg

or in parallel with (can also do manually w/ mpirun):

$ parallel_computation.py -f turb_ONERAM6.cfg -n 12



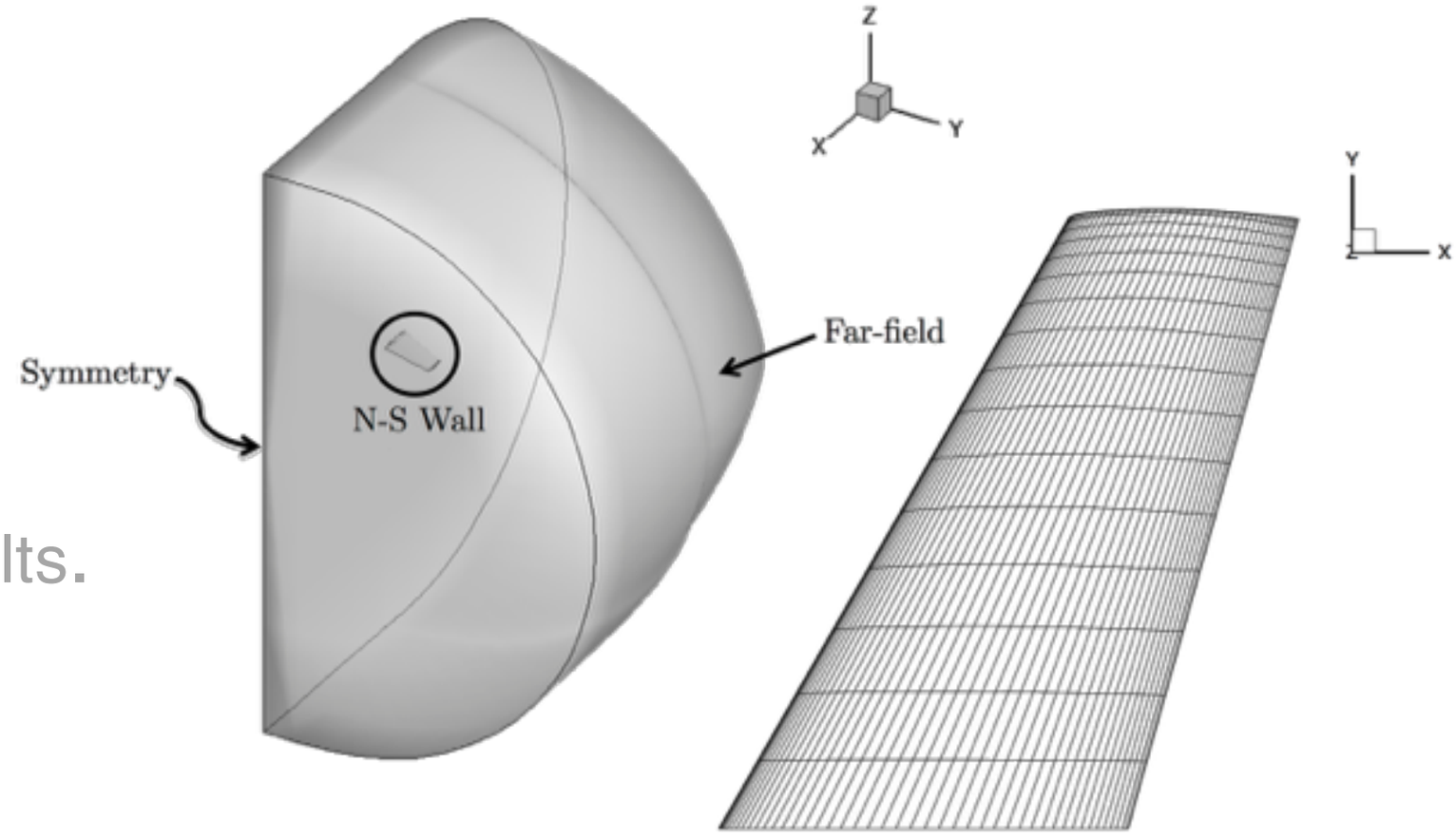
ONERA M6 RANS Analysis

1. Prepare geometry & mesh beforehand.
2. Choose appropriate physics.
3. Set proper conditions for a viscous simulation.
4. Select numerical methods:
   A. Convective terms
   B. Viscous terms
   C. Time Integration
   D. Multi-grid
5. Run the analysis.
6. Post-process the results.

Find the “mesh_ONERAM6_turb_hexa_43008.su2” and “turb_ONERAM6.cfg” files that came with these slides.

```
% Mesh input file
MESH_FILENAME= mesh_ONERAM6_turb_hexa_43008.su2

---------------------- Read grid file information ----------------------
Three dimensional problem.
43008 interior elements. 46417 points, and 0 ghost points.
3 surface markers.
2560 boundary elements in index 0 (Marker = FARFIELD).
1408 boundary elements in index 1 (Marker = WING).
2688 boundary elements in index 2 (Marker = SYMMETRY).
```



ONERA M6 RANS Analysis

1. Prepare geometry & mesh beforehand.
2. Choose appropriate physics.
3. Set proper conditions for a viscous simulation.
4. Select numerical methods:
   A. Convective terms
   B. Viscous terms
   C. Time Integration
   D. Multi-grid
5. Run the analysis.
6. Post-process the results.

```
% ------------ DIRECT, ADJOINT, AND LINEARIZED PROBLEM DEFINITION ------------%
%
% Physical governing equations (EULER, NAVIER_STOKES,
%                               WAVE_EQUATION, HEAT_EQUATION, FEM_ELASTICITY,
%                               POISSON_EQUATION)
PHYSICAL_PROBLEM= NAVIER_STOKES
%
% Specify turbulence model (NONE, SA, SA_NEG, SST)
KIND_TURB_MODEL= SA
%
% Mathematical problem (DIRECT, CONTINUOUS_ADJOINT)
MATH_PROBLEM= DIRECT
%
% Restart solution (NO, YES)
RESTART_SOL= NO
```

ONERA M6 RANS Analysis

1. Prepare geometry & mesh beforehand.
2. Choose appropriate physics.
3. Set proper conditions for a viscous simulation.
4. Select numerical methods:
   A. Convective terms
   B. Viscous terms
   C. Time Integration
   D. Multi-grid
5. Run the analysis.
6. Post-process the results.

```
% ------------------- COMPRESSIBLE FREE-STREAM DEFINITION -------------------%
%
% Mach number (non-dimensional, based on the free-stream values)
MACH_NUMBER= 0.8395
%
% Angle of attack (degrees, only for compressible flows)
AoA= 3.06
%
% Side-slip angle (degrees, only for compressible flows)
SIDESLIP_ANGLE= 0.0
%
% Free-stream temperature (288.15 K by default)
FREESTREAM_TEMPERATURE= 288.15
%
% Reynolds number (non-dimensional, based on the free-stream values)
REYNOLDS_NUMBER= 11.72E6
%
% Reynolds length (1 m by default)
REYNOLDS_LENGTH= 0.64607
```

ONERA M6 RANS Analysis

How are the flow conditions set inside SU2?

a) Store the gas constants and freestream temperature, then calculate the speed of sound.
b) Calculate and store the freestream velocity from the Mach number & AoA/sideslip angles.
c) Compute the freestream viscosity from Sutherland's law and the supplied freestream temperature.
d) Use the definition of the Reynolds number to find the freestream density from the supplied Reynolds information, freestream velocity, and freestream viscosity from step 3.
e) Calculate the freestream pressure using the perfect gas law with the freestream temperature, specific gas constant, and freestream density from step 4.
f) Perform any required non-dim.

Tip: Meshes should be in units of meters for viscous flows (SI system) or inches (US system). If your mesh is not in meters or inches, you can use the SU2_DEF module to scale it.

ONERA M6 RANS Analysis

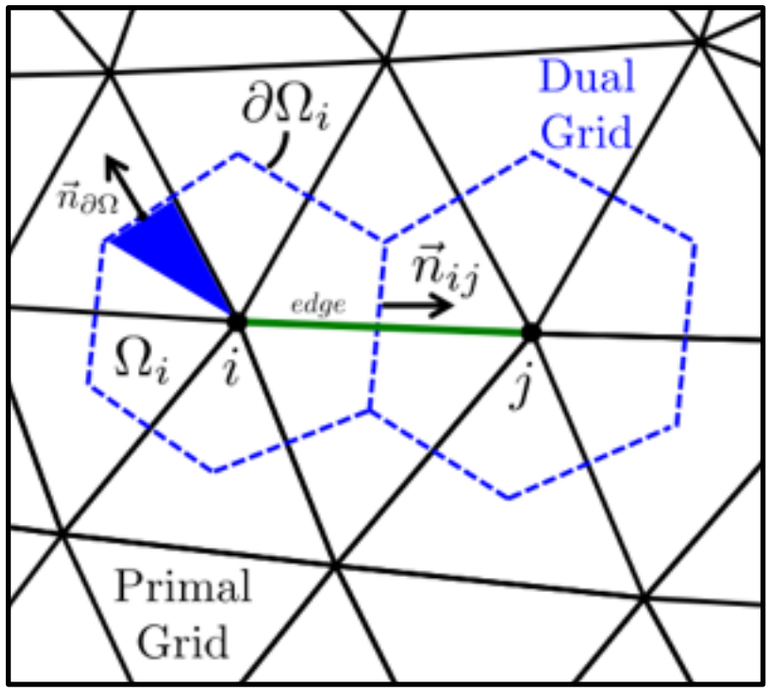
1. Prepare geometry & mesh beforehand.
2. Choose appropriate physics.
3. Set proper conditions for a viscous simulation.
4. Select numerical methods:
   A. Convective terms
   B. Viscous terms
   C. Time Integration
   D. Multi-grid
5. Run the analysis.
6. Post-process the results.

```
% ------------------- FLOW NUMERICAL METHOD DEFINITION -----------------------%
%
% Convective numerical method (JST, LAX-FRIEDRICH, CUSP, ROE, AUSM, HLLC,
%                              TURKEL_PREC, MSW)
CONV_NUM_METHOD_FLOW= ROE
%
% Spatial numerical order integration (1ST_ORDER, 2ND_ORDER, 2ND_ORDER_LIMITER)
SPATIAL_ORDER_FLOW= 2ND_ORDER_LIMITER
%
% Slope limiter (VENKATAKRISHNAN, MINMOD)
SLOPE_LIMITER_FLOW= VENKATAKRISHNAN
%
% 1st, 2nd and 4th order artificial dissipation coefficients
AD_COEFF_FLOW= ( 0.15, 0.5, 0.02 )
%
% Time discretization (RUNGE-KUTTA_EXPLICIT, EULER_IMPLICIT, EULER_EXPLICIT)
TIME_DISCRE_FLOW= EULER_IMPLICIT

% ------------------- TURBULENT NUMERICAL METHOD DEFINITION ------------------%
%
% Convective numerical method (SCALAR_UPWIND)
CONV_NUM_METHOD_TURB= SCALAR_UPWIND
%
% Spatial numerical order integration (1ST_ORDER, 2ND_ORDER, 2ND_ORDER_LIMITER)
SPATIAL_ORDER_TURB= 1ST_ORDER
%
% Slope limiter (VENKATAKRISHNAN, MINMOD)
SLOPE_LIMITER_TURB= VENKATAKRISHNAN
%
% Time discretization (EULER_IMPLICIT)
TIME_DISCRE_TURB= EULER_IMPLICIT
```

ONERA M6 RANS Analysis

1. Prepare geometry & mesh beforehand.
2. Choose appropriate physics.
3. Set proper conditions for a viscous simulation.
4. Select numerical methods:
   A. Convective terms
   B. Viscous terms
   C. Time Integration
   D. Multi-grid
5. Run the analysis.
6. Post-process the results.

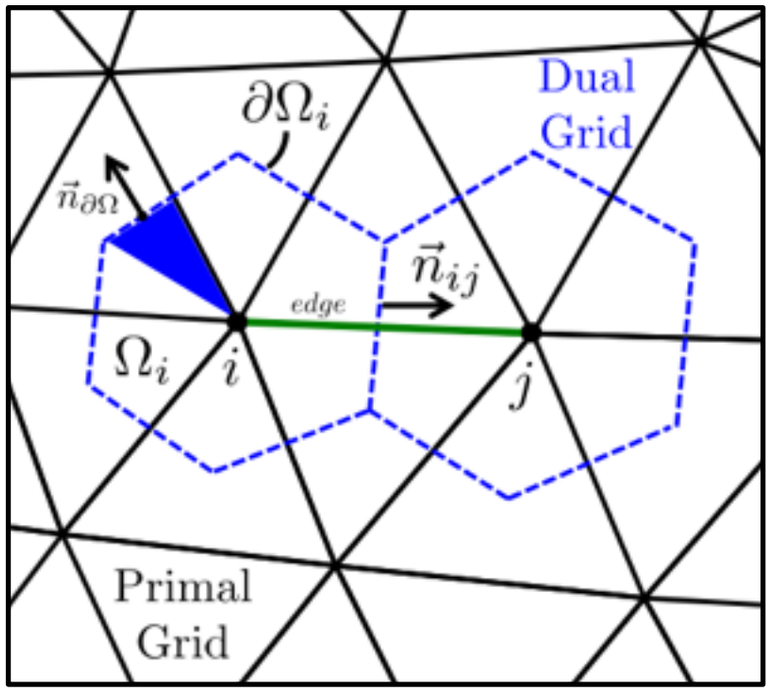



```
% Convective numerical method (JST, LAX-FRIEDRICH, CUSP, ROE, AUSM, HLLC,
%                              TURKEL_PREC, MSW)
CONV_NUM_METHOD_FLOW= ROE
%
% Spatial numerical order integration (1ST_ORDER, 2ND_ORDER, 2ND_ORDER_LIMITER)
SPATIAL_ORDER_FLOW= 2ND_ORDER_LIMITER
%
% Slope limiter (VENKATAKRISHNAN, MINMOD)
SLOPE_LIMITER_FLOW= VENKATAKRISHNAN

% Convective numerical method (SCALAR_UPWIND)
CONV_NUM_METHOD_TURB= SCALAR_UPWIND
%
% Spatial numerical order integration (1ST_ORDER, 2ND_ORDER, 2ND_ORDER_LIMITER)
SPATIAL_ORDER_TURB= 1ST_ORDER
```

ONERA M6 RANS Analysis

1. Prepare geometry & mesh beforehand.
2. Choose appropriate physics.
3. Set proper conditions for a viscous simulation.
4. Select numerical methods:
   A. Convective terms
   B. Viscous terms
   C. Time Integration
   D. Multi-grid
5. Run the analysis.
6. Post-process the results.




```
% Numerical method for spatial gradients (GREEN_GAUSS, WEIGHTED_LEAST_SQUARES)
NUM_METHOD_GRAD= GREEN_GAUSS
```

Viscous fluxes are computed using a corrected average of gradients method by default.

ONERA M6 RANS Analysis

1. Prepare geometry & mesh beforehand.
2. Choose appropriate physics.
3. Set proper conditions for a viscous simulation.
4. Select numerical methods:
   A. Convective terms
   B. Viscous terms
   C. Time Integration
   D. Multi-grid
5. Run the analysis.
6. Post-process the results.

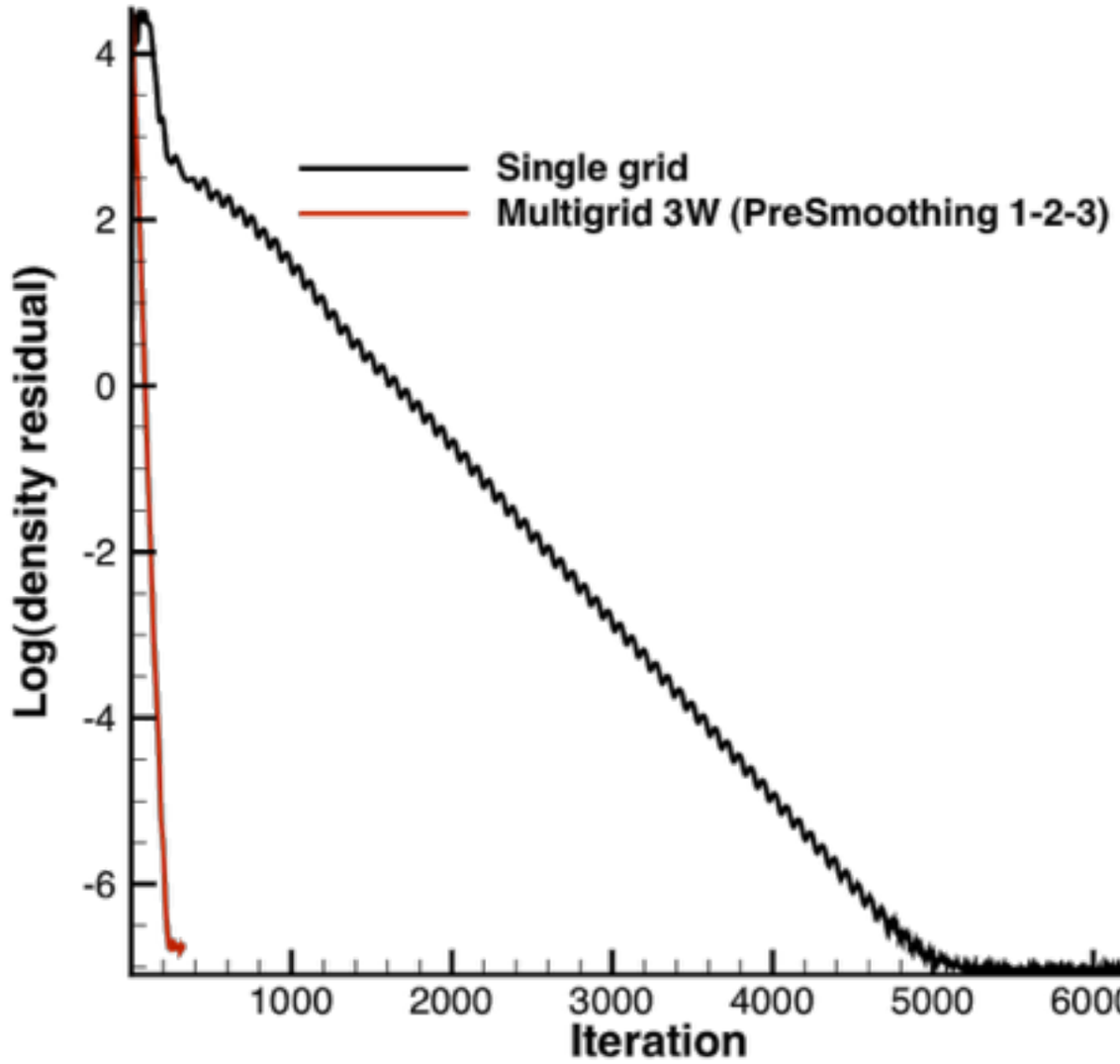
```
% Time discretization (RUNGE-KUTTA_EXPLICIT,
                       EULER_IMPLICIT, EULER_EXPLICIT)
TIME_DISCRE_FLOW= EULER_IMPLICIT

% Time discretization (EULER_IMPLICIT)
TIME_DISCRE_TURB= EULER_IMPLICIT
```

```
% ------------------------ LINEAR SOLVER DEFINITION -------------------------%
%
% Linear solver for the implicit (or discrete adjoint) formulation (BCGSTAB, FGMRES)
LINEAR_SOLVER= FGMRES
%
% Preconditioner of the Krylov linear solver (NONE, JACOBI, LINELET)
LINEAR_SOLVER_PREC= LU_SGS
%
% Min error of the linear solver for the implicit formulation
LINEAR_SOLVER_ERROR= 1E-10
%
% Max number of iterations of the linear solver for the implicit formulation
LINEAR_SOLVER_ITER= 5
```

ONERA M6 RANS Analysis

1. Prepare geometry & mesh beforehand.
2. Choose appropriate physics.
3. Set proper conditions for a viscous simulation.
4. Select numerical methods:
   A. Convective terms
   B. Viscous terms
   C. Time Integration
   D. Multi-grid
5. Run the analysis.
6. Post-process the results.



Multigrid example with the inviscid ONERA M6.

ONERA M6 RANS Analysis

```
% ------------------------- MULTIGRID PARAMETERS -----------------------------%
%
% Multi-Grid Levels (0 = no multi-grid)
MGLEVEL= 0
%
% Multi-grid cycle (V_CYCLE, W_CYCLE, FULLMG_CYCLE)
MGCYCLE= V_CYCLE
%
% Multi-grid pre-smoothing level
MG_PRE_SMOOTH= ( 1, 1, 1, 1 )
%
% Multi-grid post-smoothing level
MG_POST_SMOOTH= ( 1, 1, 1, 1 )
%
% Jacobi implicit smoothing of the correction
MG_CORRECTION_SMOOTH= ( 0, 0, 0, 0 )
%
% Damping factor for the residual restriction
MG_DAMP_RESTRICTION= 0.9
%
% Damping factor for the correction prolongation
MG_DAMP_PROLONGATION= 0.9
```

Tip: There are many knobs available for tuning the agglomeration multigrid in the configuration file. Although, it is turned off for the turb. ONERA M6 case.

ONERA M6 RANS Analysis

1. Prepare geometry & mesh beforehand.
2. Choose appropriate physics.
3. Set proper conditions for a viscous simulation.
4. Select numerical methods:
   A. Convective terms
   B. Viscous terms
   C. Time Integration
   D. Multi-grid
5. Run the analysis.
6. Post-process the results.

The files for this case came with the slides. Run the case in serial with:

$ SU2_CFD turb_ONERAM6.cfg

or in parallel with (can also do manually w/ mpirun):

$ parallel_computation.py -f turb_ONERAM6.cfg -n 12

```
------------------------------ Begin Solver ----------------------------

---------------------- Local Time Stepping Summary ----------------------
MG level: 0 -> Min. DT: 1.95786e-08. Max. DT: 0.00147685. CFL: 4.
-------------------------------------------------------------------------

---------------------- Residual Evolution Summary ----------------------
log10[Maximum residual]: -0.0577369.
Maximum residual point 44935, located at (-4.98478, 4.6362, 0).
Maximum Omega 1.67814e+07, maximum Strain Rate 2.359e+07.
-------------------------------------------------------------------------

 Iter    Time(s)      Res[Rho]       Res[nu]      CL(Total)      CD(Total)
    0   1.037438     -1.906922     -6.575858       0.235583       0.138480
    1   1.025627     -1.922986     -6.723776       0.273875       0.168547
    2   1.018482     -2.087671     -6.710803       0.264640       0.174591
    3   1.015054     -2.242860     -6.621769       0.248622       0.172952
    4   1.026720     -2.358367     -6.577332       0.237411       0.170562
    5   1.066679     -2.396212     -6.559033       0.231783       0.168444
    6   1.094859     -2.399609     -6.554571       0.229946       0.166243
    7   1.110272     -2.390889     -6.556365       0.229763       0.163920
    8   1.124675     -2.375317     -6.559746       0.230027       0.161446
    9   1.129961     -2.353713     -6.562281       0.230292       0.158755
   10   1.121899     -2.327509     -6.563372       0.230438       0.155815
   11   1.120111     -2.305447     -6.563013       0.230451       0.152594
   12   1.127016     -2.284201     -6.561314       0.230352       0.149063
   13   1.122702     -2.259707     -6.558794       0.230055       0.145184
   14   1.122751     -2.242936     -6.556093       0.229536       0.140928
   15   1.124781     -2.225095     -6.552405       0.228819       0.136284
   16   1.136865     -2.209273     -6.549352       0.227924       0.131265
   17   1.146245     -2.196863     -6.545540       0.226872       0.125906
   18   1.143441     -2.188779     -6.542911       0.225663       0.120265
   19   1.154980     -2.175092     -6.538967       0.224373       0.114410
   20   1.165407     -2.173455     -6.535974       0.223009       0.108407
```

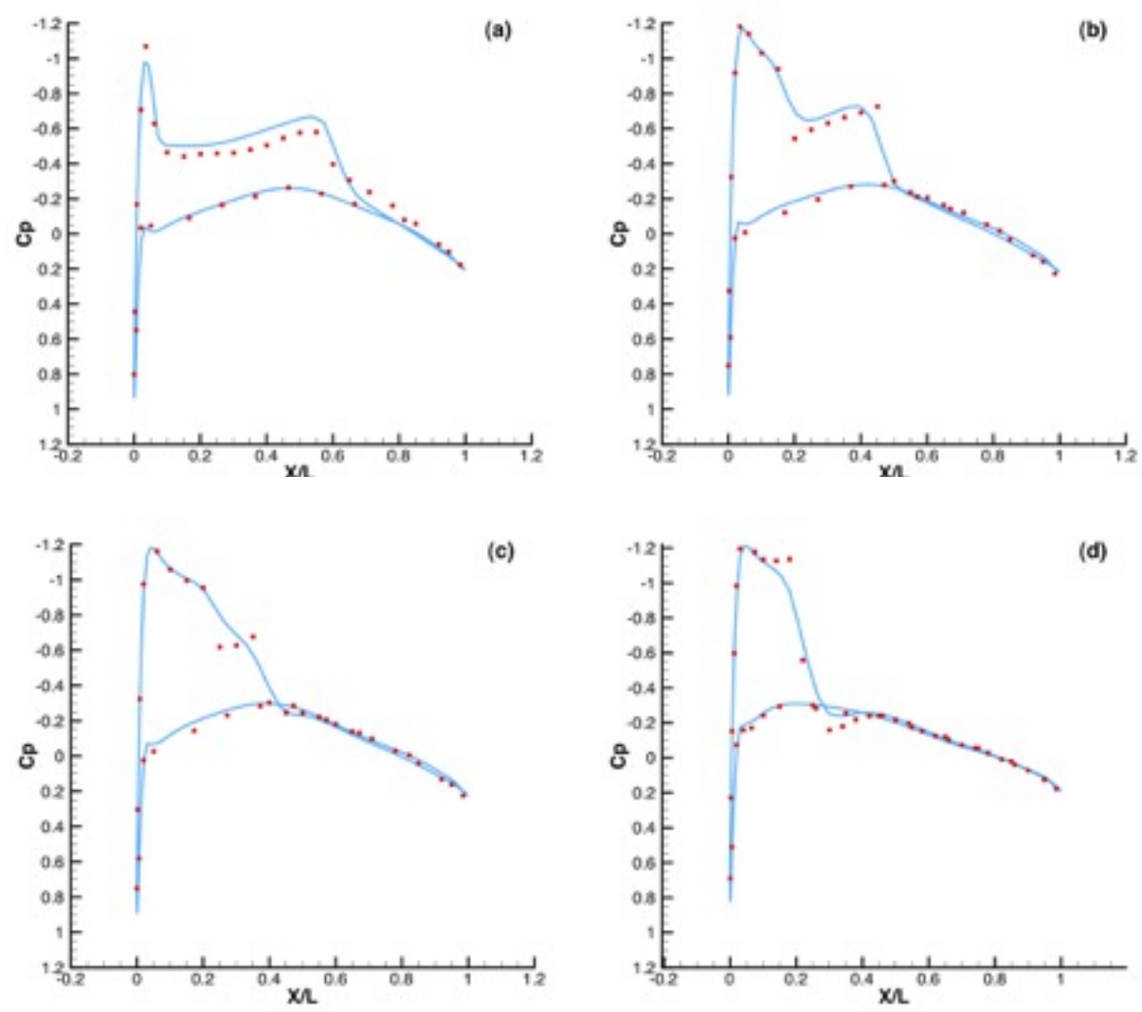
ONERA M6 RANS Analysis

Tip: Convergence can be controlled based on the residual (RESIDUAL) or by monitoring coefficients (CAUCHY).

```
% ------------------------- CONVERGENCE PARAMETERS -------------------------%
%
% Convergence criteria (CAUCHY, RESIDUAL)
%
CONV_CRITERIA= CAUCHY
%
% Residual reduction (order of magnitude with respect to the initial value)
RESIDUAL_REDUCTION= 5
%
% Min value of the residual (log10 of the residual)
RESIDUAL_MINVAL= -10
%
% Start convergence criteria at iteration number
STARTCONV_ITER= 10
%
% Number of elements to apply the criteria
CAUCHY_ELEMS= 100
%
% Epsilon to control the series convergence
CAUCHY_EPS= 1E-6
%
% Function to apply the criteria (LIFT, DRAG, NEARFIELD_PRESS, SENS_GEOMETRY,
%                                 SENS_MACH, DELTA_LIFT, DELTA_DRAG)
CAUCHY_FUNC_FLOW= DRAG
CAUCHY_FUNC_ADJFLOW= SENS_GEOMETRY
```

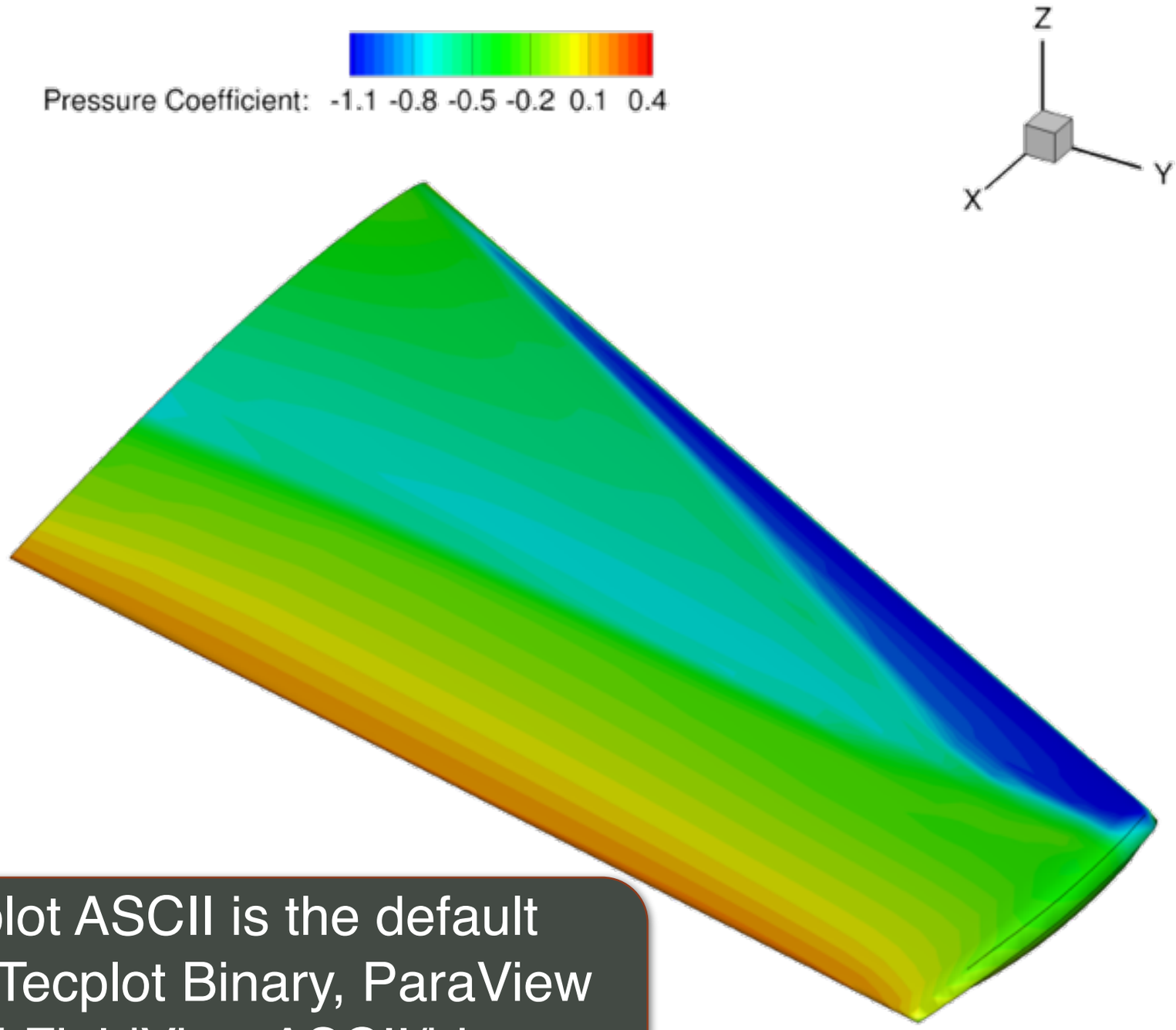
ONERA M6 RANS Analysis

1. Prepare geometry & mesh beforehand.
2. Choose appropriate physics.
3. Set proper conditions for a viscous simulation.
4. Select numerical methods:
   A. Convective terms
   B. Viscous terms
   C. Time Integration
   D. Multi-grid
5. Run the analysis.
6. Post-process the results.



Comparison of Cp profiles of the experimental results of Schmitt and Carpin (red squares) against SU2 computational results (blue line) at different sections along the span of the wing. (a) y/b = 0.2, (b) y/b = 0.65, (c) y/b = 0.8, (d) y/b = 0.95

ONERA M6 RANS Analysis



Tip: Tecplot ASCII is the default output, but Tecplot Binary, ParaView ASCII, and FieldView ASCII/binary formats are available

ONERA M6 RANS Analysis

Tip: SU2_CFD only writes restart files when in parallel mode to save overhead. Use the SU2_SOL module to generate visualization files from the restart. If you run the code in parallel using parallel_computation.py, it will automatically call SU2_SOL to create visualization files.

SU2

The Open-Source CFD Code

http://su2.stanford.edu/
https://github.com/su2code/SU2
Follow us on Twitter: @su2code