Running SU2

Pointwise® and SU2 Joint Workshop
Sept 29th-30th, 2014

Heather Kline, Andrew Wendorff
Aeronautics & Astronautics Department (Stanford University)

I. WHAT DO I NEED TO RUN A SIMULATION?
II. TEST CASE DEFINITION
III. INTERACTIVE SESSION
IV. QUESTIONS?

SU2
The Open-Source CFD Code

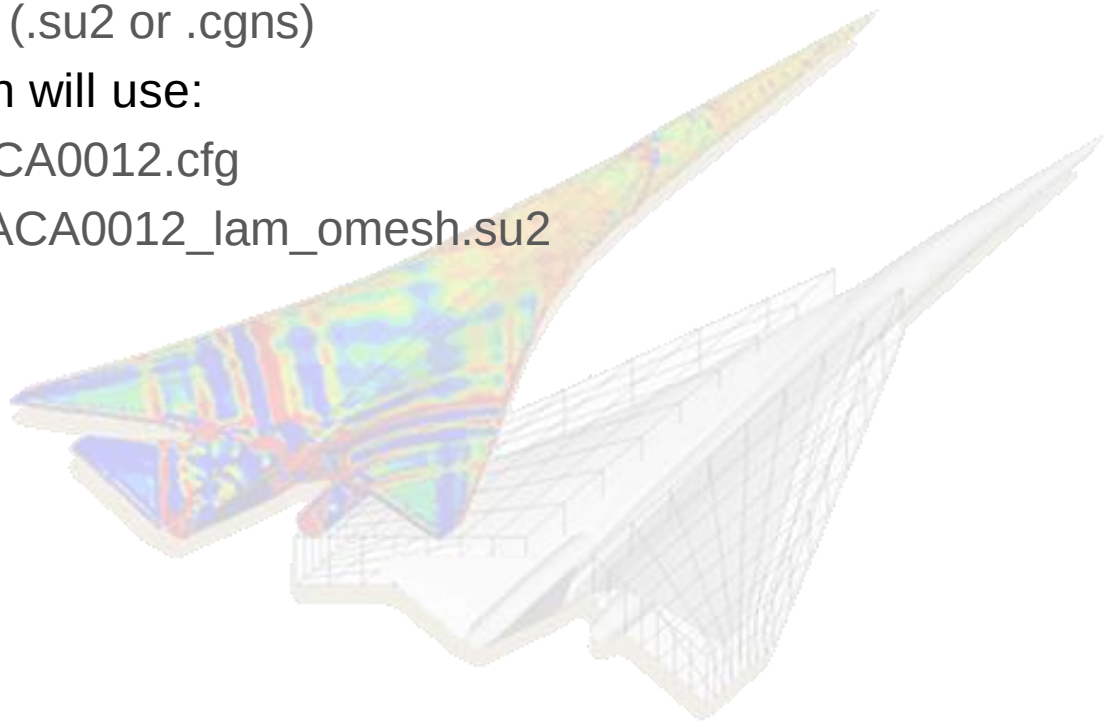
Running Simulations with SU^2

What do I need to run simulations with SU2?

- Configuration file (.cfg)
- Mesh file (.su2 or .cgns)

This session will use:

- Lam_NACA0012.cfg
- Mesh_NACA0012_lam_omesh.su2



Quick Start Tutorial
NACA 0012 Airfoil

Transonic, Euler flow

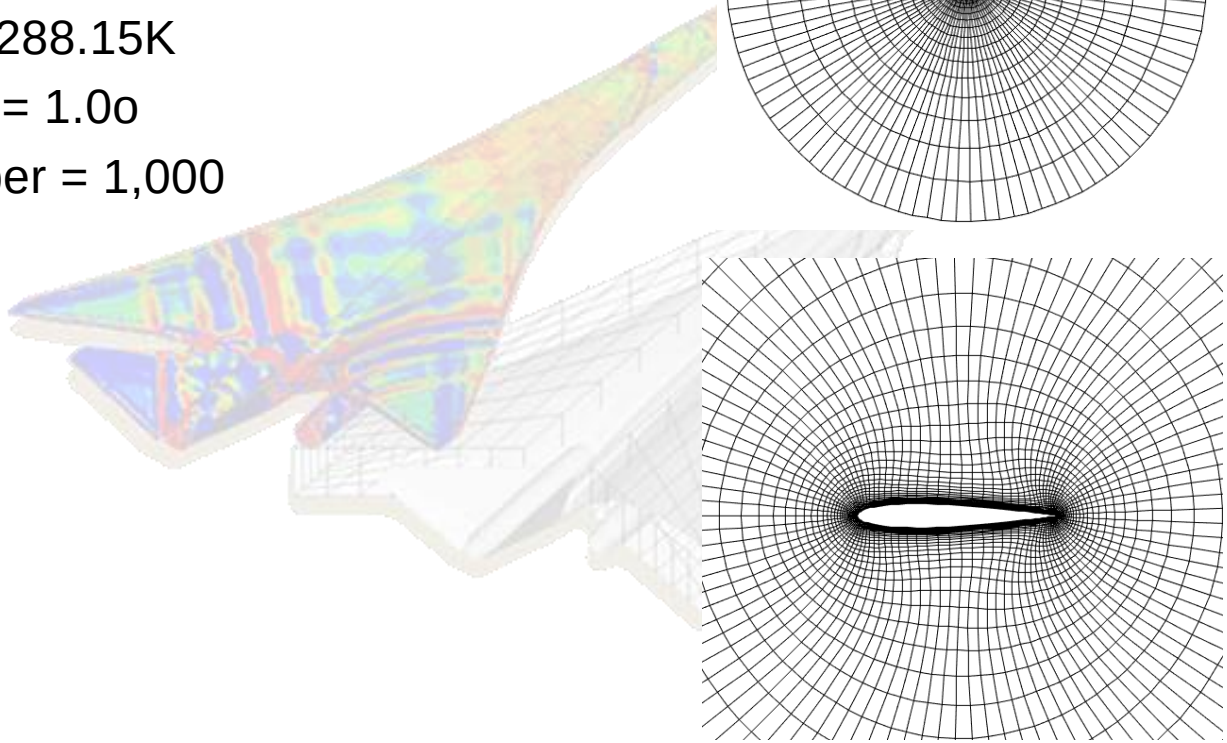
Mach No. = 0.5

Pressure = 101,325 Nm-2

Temperature = 288.15K

Angle of attack = 1.0o

Reynolds number = 1,000



Flow Solution

Config options:

```
PHYSICAL_PROBLEM= NAVIER_STOKES
%
MATH_PROBLEM= DIRECT
%
MACH_NUMBER= 0.5
%
AoA= 1.00
%
REYNOLDS_NUMBER=1000.0
%
FREESTREAM_TEMPERATURE= 288.15
%
MESH_FILENAME= mesh_NACA0012_lam_omesh.su2
```

Most parameters have default values

The order of config options is not important

Restart

Simulations can be restarted from partially converged results

Config options:

```
RESTART_SOL= NO
%
EXT_ITER= 999999
```

```
RESTART_SOL= YES
%
SOLUTION_FLOW_FILENAME= solution_flow.dat
```

Solver Parameters

Among the many options in the config file, various parameters exist to modify the solution method

Config options:

```
RESTART_SOL= NO
%
CFL_NUMBER= 4.0
%
CFL_RAMP= ( 1.1, 10, 10.0 )
%
CONV_NUM_METHOD_FLOW= ROE
%
SPATIAL_ORDER_FLOW= 2ND_ORDER
```

Interactive

Direct Solution

Adjoint Solution

Sensitivity of a functional to changes in the flow

- e.g., How does changing the airfoil shape affect lift?

Additional required file:

- Converged flow solution

Config options:

```
MATH_PROBLEM= ADJOINT
%
RESTART_SOL= NO
%
ADJOINT_TYPE= CONTINUOUS
%
ADJ_OBJFUNC= DRAG
%
SOLUTION_FLOW_FILENAME= solution_flow.dat
```

Interactive

Adjoint Solution

Additional Resources

Online documentation

- http://su2.stanford.edu

Online tutorials

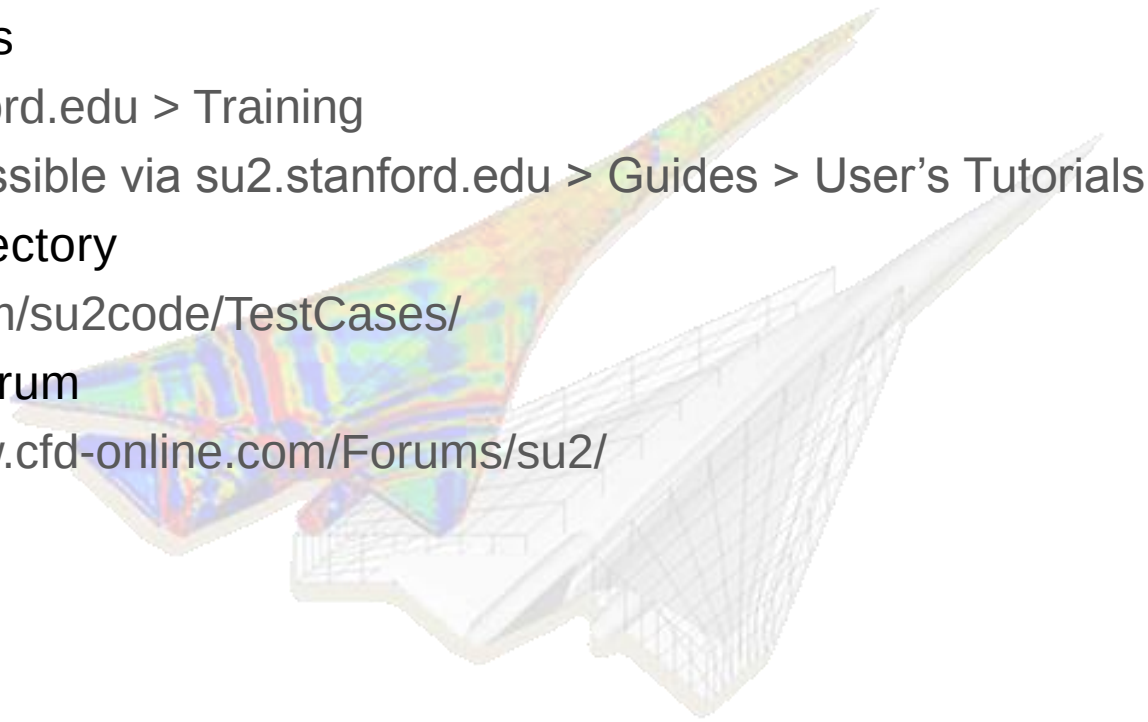
- su2.stanford.edu > Training
- also accessible via su2.stanford.edu > Guides > User's Tutorials

TestCases directory

- github.com/su2code/TestCases/

CFD Online forum

- http://www.cfd-online.com/Forums/su2/



Questions?