Running SU²

SU² Release Version 2.0 Workshop
Stanford University
Tuesday, January 15th, 2013

Sean Copeland & Tom Taylor
Department of Aeronautics & Astronautics
Stanford University
• SU² environment variable
• What do I need to run a simulation?
• Test case definition
• Interactive session
• Questions?
SU² Environment Variables

• Did you set your SU² environment variables during the installation procedure?
• i.e., in .bashrc:

```bash
# Set the SU2 source code home directory
export SU2_HOME=/Users/tomtaylor/SU2

# Identify directory where SU2 executables and python scripts are stored
Export SU2_RUN=/Users/tomtaylor/SU2Py

# Add SU2_RUN to the $PATH
Export PATH=$PATH:$SU2_RUN
```
Running Simulations with SU$^2$

- What do I need to run simulations with SU$^2$?
  - Configuration file (.cfg)
  - Mesh file (.su2 or .cgns)
- This session will use:
  - inv_NACA0012.cfg
  - Mesh_NACA0012_inv.su2
- These are found in
  > User Guide > Quick Start Tutorial
  /SU2/TestCases/inv_NACA0012/
Test Case
NACA 0012 Airfoil

• Transonic, Euler flow
• Mach No. = 0.8
• Pressure = 101,325 Nm$^{-2}$
• Temperature = 273.15K
• Angle of attack = 1.25°
Flow Solution

• Config options:

```plaintext
PHYSICAL_PROBLEM= EULER
%  
MATH_PROBLEM= DIRECT
%  
MACH_NUMBER= 0.8
%  
AoA= 1.25
%  
FREESTREAM_PRESSURE= 101325.0
%  
FREESTREAM_TEMPERATURE= 273.15
%  
MESH_FILENAME= mesh_NACA0012_inv.su2
```

• Most parameters have default values
• Order of config options is not important
• Simulations can be restarted from partially converged results

• Config options:

```plaintext
RESTART_SOL= NO
%  
EXT_ITER= 50

RESTART_SOL= YES
%  
SOLUTION_FLOW_FILENAME= solution_flow.dat
```
• 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-2ND_ORDER
```
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
```
Example:
Separated Flow Over Cylinder
Additional Resources

• Online documentation
  http://su2.stanford.edu

• Online tutorials

• TestCases directory
  /SU2/TestCases/

• CFD Online forum
  http://www.cfd-online.com/Forums/su2/