Running SU²

SU² Release Version 2.0 Workshop

Stanford University

Tuesday, January 15th, 2013

Sean Copeland & Tom Taylor

Department of Aeronautics & Astronautics

Stanford University

Outline

- 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:

```
# 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

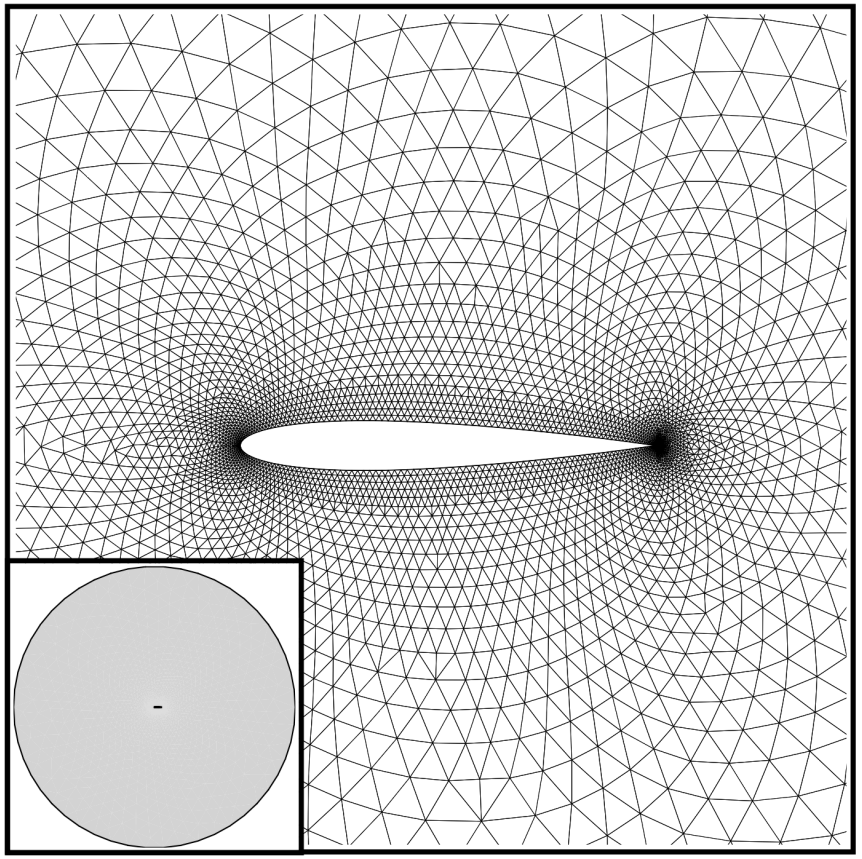
- What do I need to run simulations with SU^2?
  - Configuration file (.cfg)
  - Mesh file (.su2 or .cgns)
- This session will use:
  - inv_NACA0012.cfg
  - Mesh_NACA0012_inv.su2
- These are found in

  > 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:

```
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

Restart

- Simulations can be restarted from partially converged results
- Config options:

```
RESTART_SOL= NO
%
EXT_ITER= 50
```

```
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:

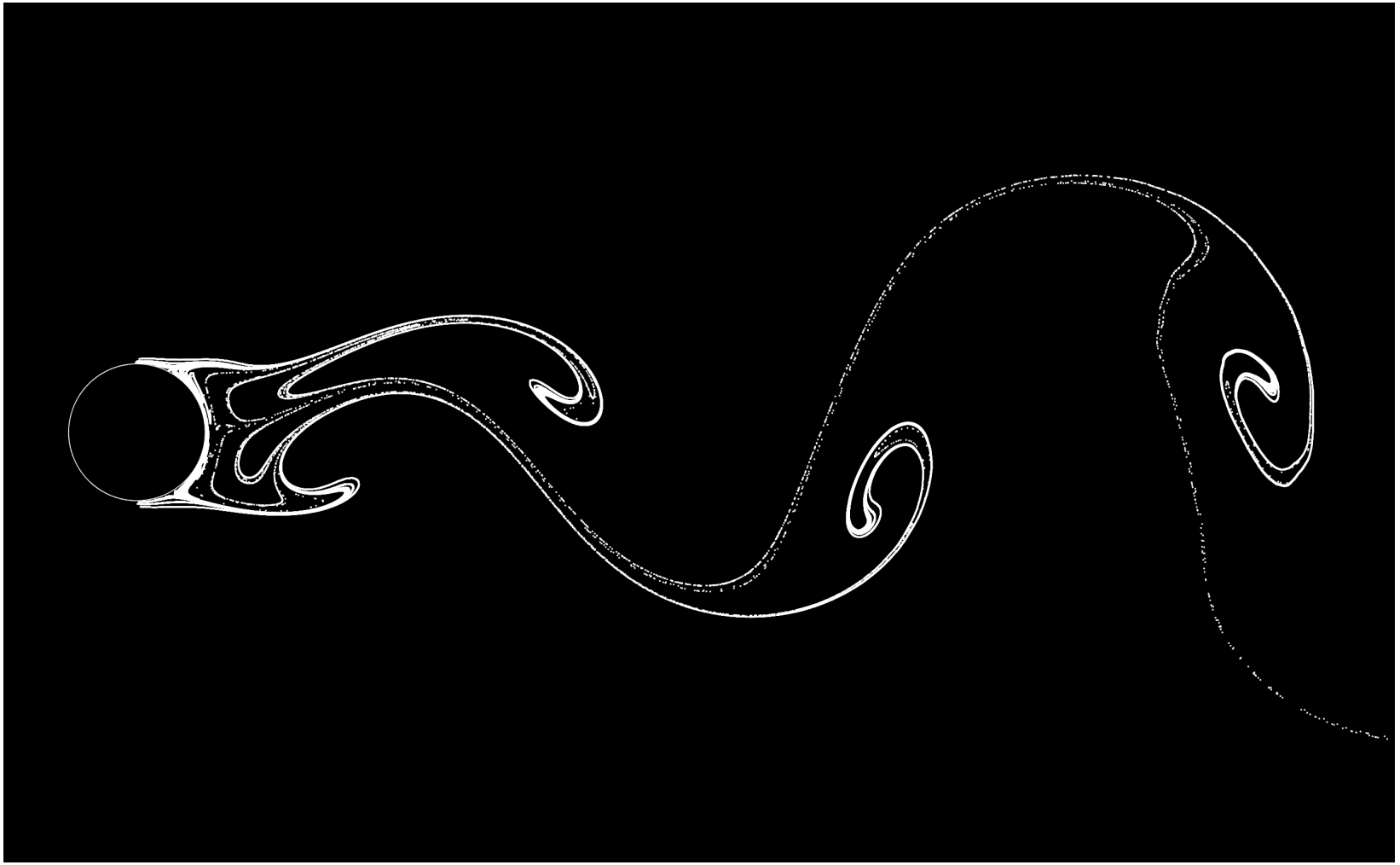
```
RESTART_SOL= NO
%
CFL_NUMBER= 4.0
%
CFL_RAMP= ( 1.1, 10, 10.0 )
%
CONV_NUM_METHOD_FLOW= ROE-2ND_ORDER
```

Adjoint Solution

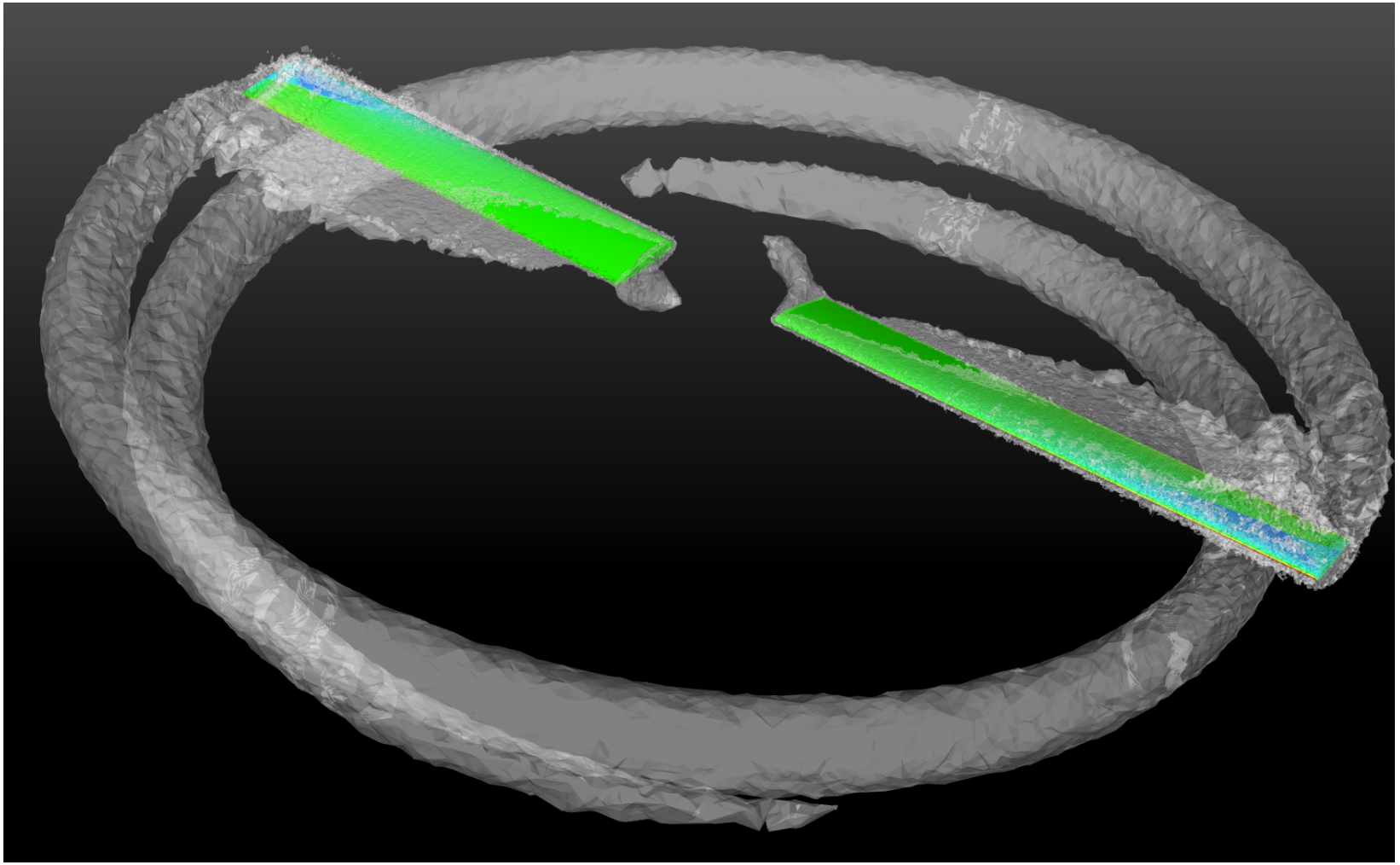
- Sensitivity of a functional to changes in the flow
  - e.g., How does changing the airfoil shape affect lift?
- Additional required file:
  - Converged flow solution
- Config options:

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

Example: Separated Flow Over Cylinder



Example:
Tip Vortices From Rotor



Additional Resources

- Online documentation

  http://su2.stanford.edu

- Online tutorials

  > User Guide > User's Guide > User's Tutorials

- TestCases directory

  /SU2/TestCases/

- CFD Online forum

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