

# Running SU2

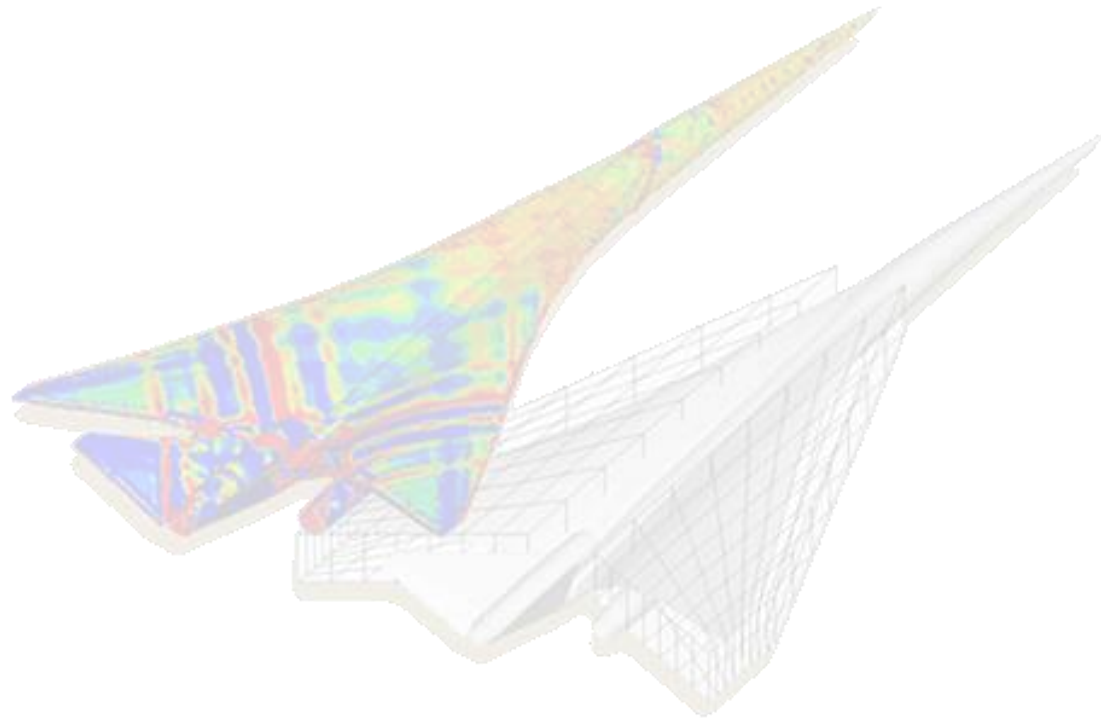
POINTWISE® AND SU2 JOINT WORKSHOP  
SEPT 29<sup>TH</sup>-30<sup>TH</sup>, 2014

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?





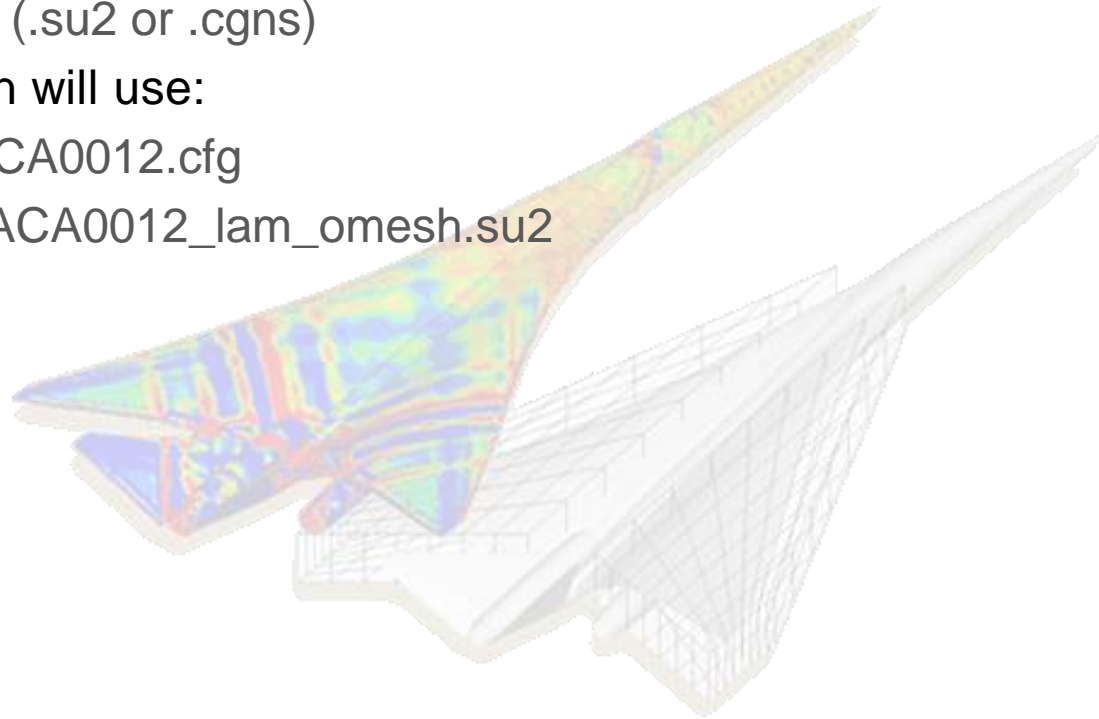
# Running Simulations with SU2

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

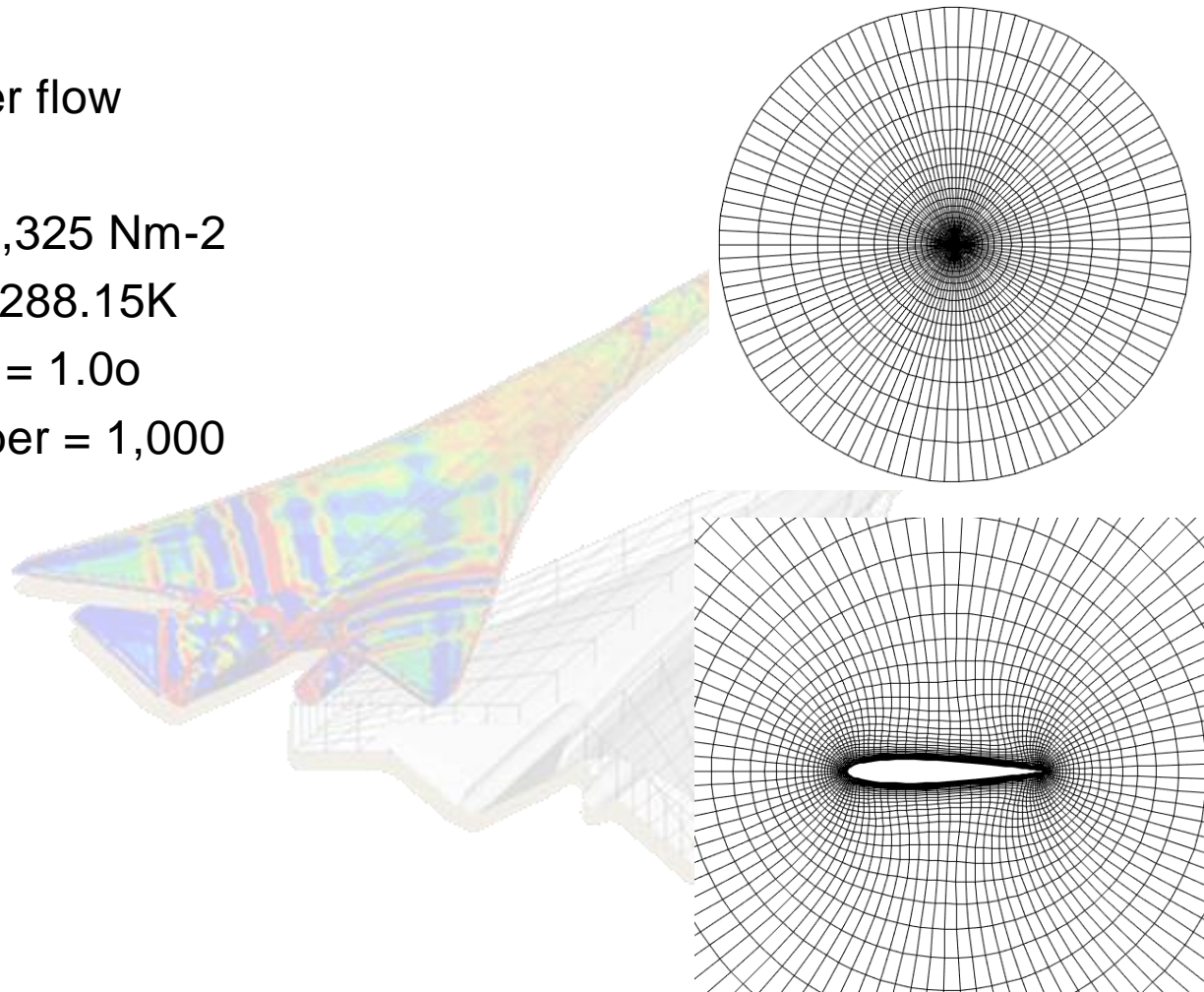
Mach No. = 0.5

Pressure = 101,325 Nm<sup>-2</sup>

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
```

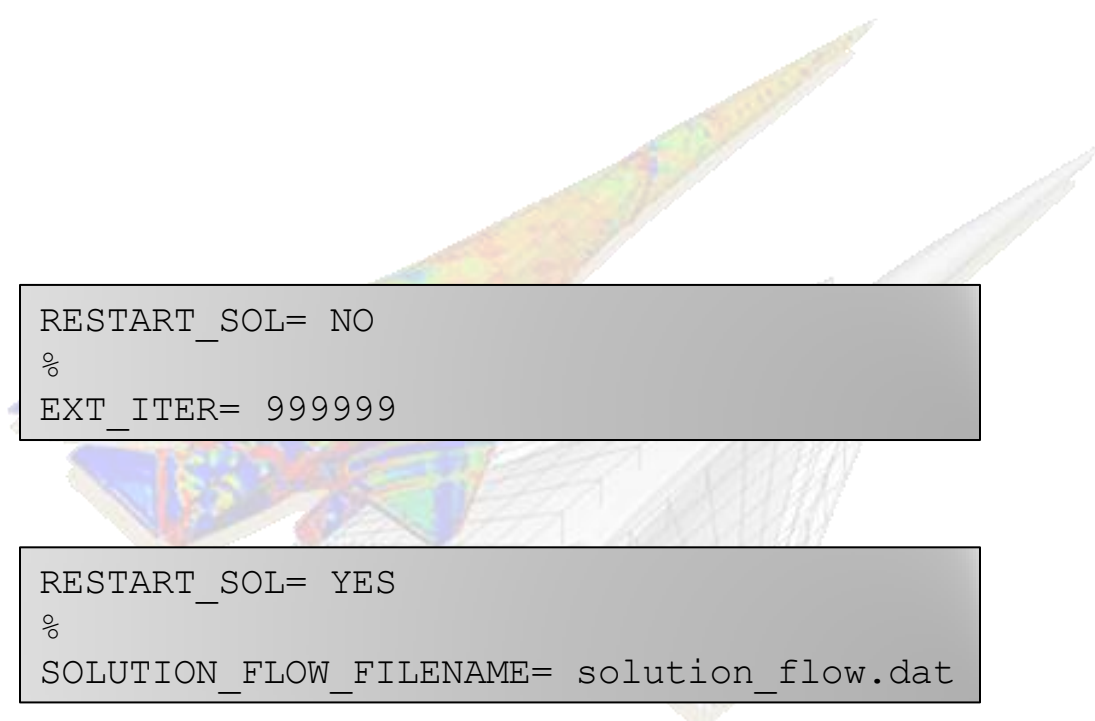
Most parameters have default values

The order of config options is not important

# Restart

Simulations can be restarted from partially converged results

Config options:

A background image showing a 3D mesh of an aircraft wing, rendered with a color gradient from blue to yellow, indicating a simulation result. The mesh is semi-transparent, allowing the text boxes to be overlaid.

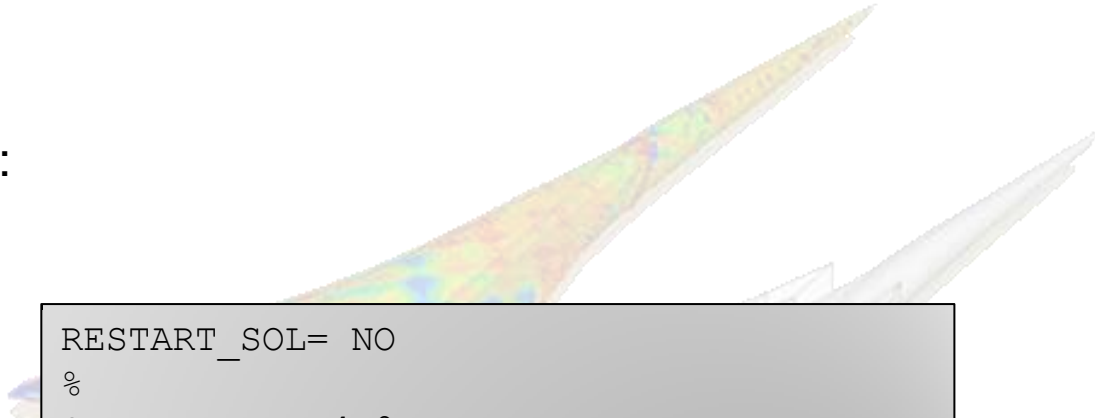
```
RESTART_SOL= NO
%
EXT_ITER= 999999
```

```
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:

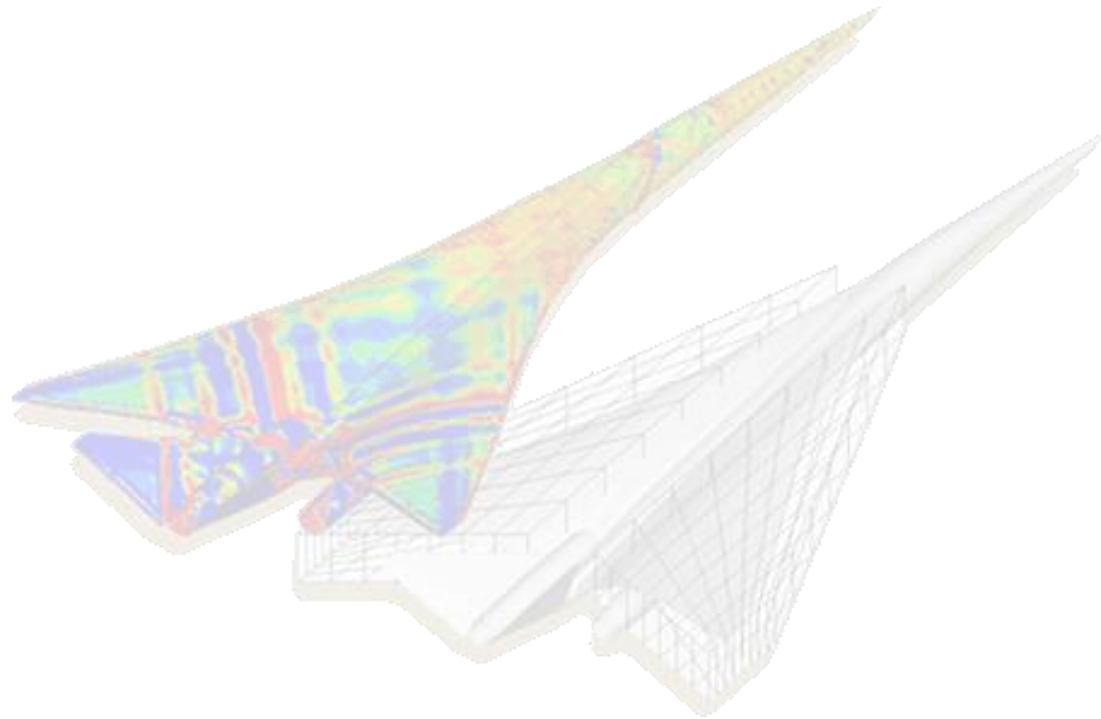
A background image showing a 3D visualization of an airfoil. The airfoil is a curved, wing-like shape. The surface is colored with a gradient from blue to yellow, representing a scalar field like pressure or velocity. The airfoil is shown from a perspective view, with its leading and trailing edges visible.

```
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

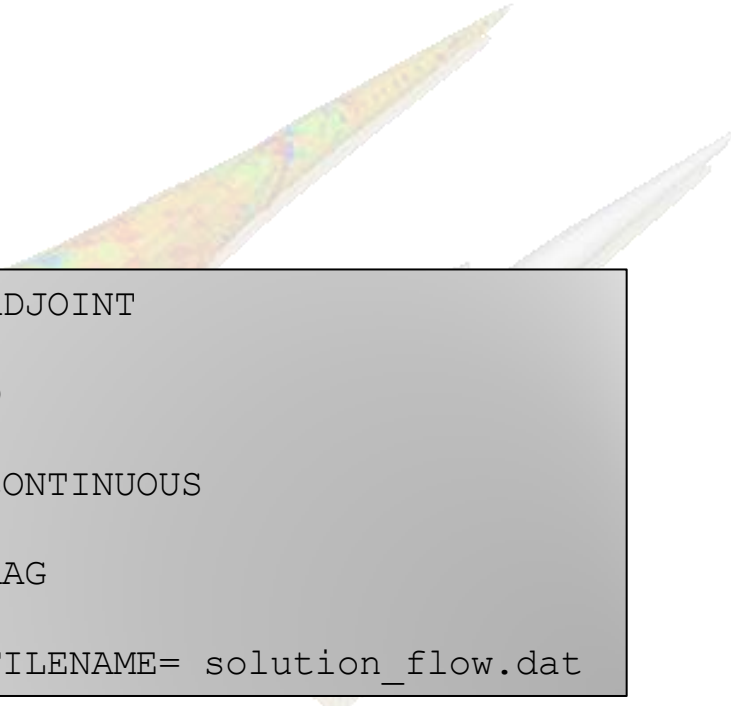
Sensitivity of a functional to changes in the flow

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

Additional required file:

- Converged flow solution

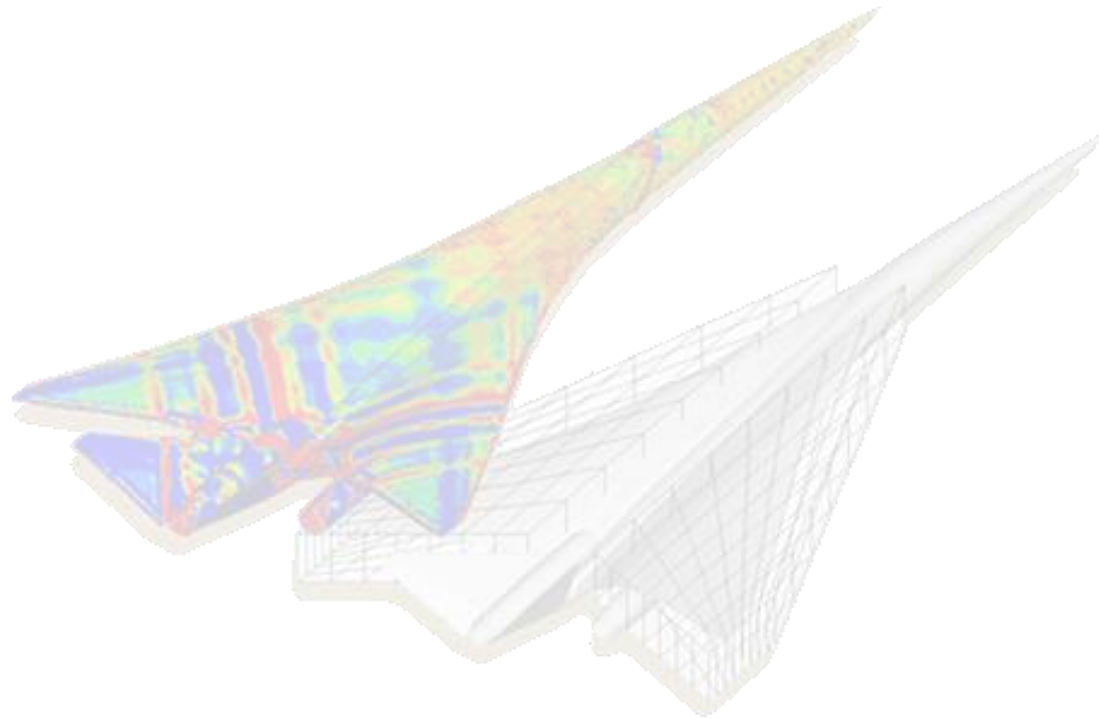
Config options:



```
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](http://su2.stanford.edu) > Training
- also accessible via [su2.stanford.edu](http://su2.stanford.edu) > Guides > User's Tutorials

### TestCases directory

- [github.com/su2code/TestCases/](https://github.com/su2code/TestCases/)

### CFD Online forum

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



Questions?

