

## About SU<sup>2</sup>

SU<sup>2</sup> is an open-source software suite specialized for high-fidelity Partial Differential Equation (PDE) analysis and design of PDE-constrained systems on unstructured grids.

The suite includes C++ analysis modules, linked libraries, and scripts, that:

- Solve the PDE system
- Decompose the domain for parallel computations
- Determine sensitivities of an objective function (e.g. lift, drag)
- Deform the model and grid to perform shape optimization
- Perform adaptive grid refinement

Mac OS X, Linux and Windows binaries and tables can be downloaded from the SU<sup>2</sup> website:  
<http://s2.stanford.edu>

<http://s2.stanford.edu>

SU<sup>2</sup> is a cutting-edge, flexible, open-source tool that can be used for:

- High-fidelity analysis
- Adjoint-based design
- Multi-physics simulations
- Adaptive, goal-oriented mesh refinement

Documentation and a full description of current and upcoming features are available on the SU<sup>2</sup> website:

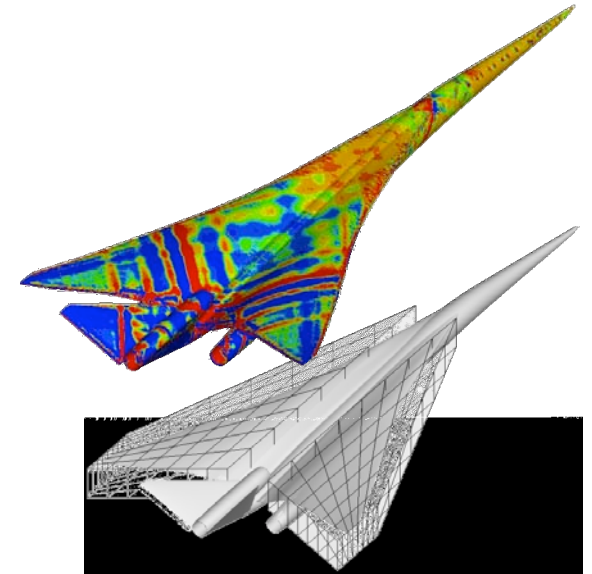
<http://s2.stanford.edu>

Email the development team:  
[ad-de@li.stanford.edu](mailto:ad-de@li.stanford.edu)



SU<sup>2</sup> is under active development by the Aerospace Design Laboratory at Stanford University. Visit the ADL at:

<http://adl.stanford.edu>



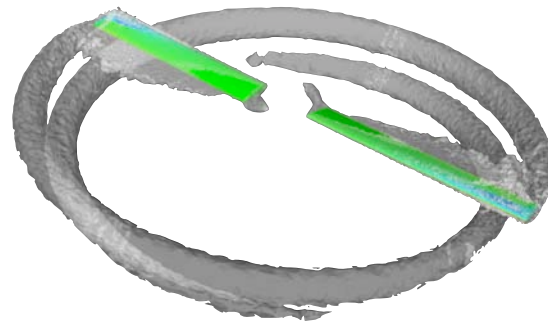
Open-Source Analysis and Design



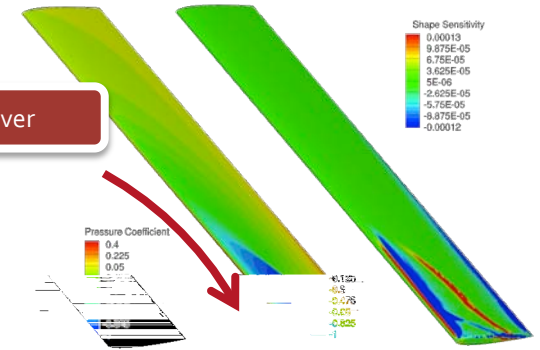
 **aerospacedesignlab**

## High-Fidelity Analysis

- Usable for internal and external aerodynamics
- Handles structured grids and includes a Pointwise plugin for mesh generation
- Euler, Navier-Stokes, RANS, rotating frame, axisymmetric and incompressible equations
- Steady and time-accurate analyses
- Convergence acceleration including agglomeration multi-grid
- Parallelism using MPI



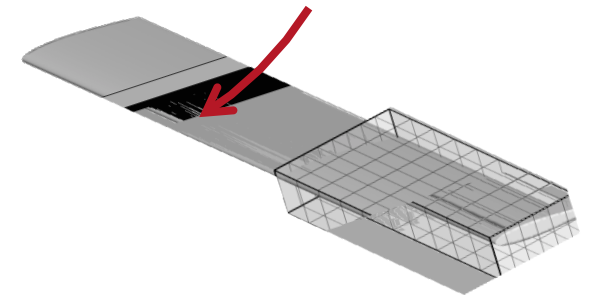
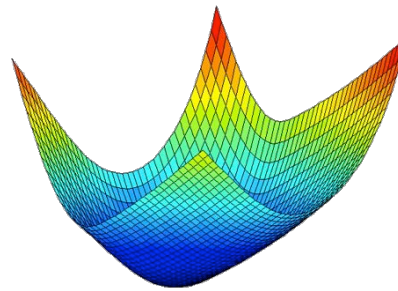
CFD Solver



Adjoint CFD Solver

## Shape Optimization

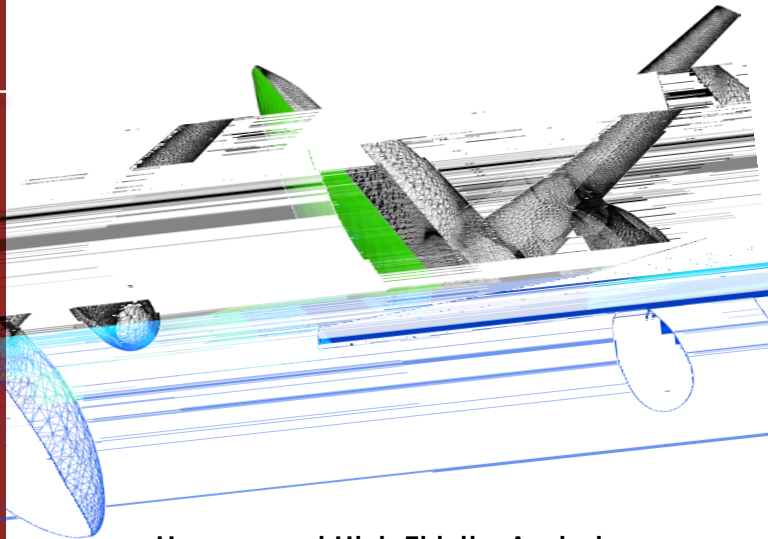
- Self-contained optimization environment using standard python libraries, such as NumPy and SciPy
- Gradient computation using the continuous adjoint approach
- 3D design variable definition using free-form deformation boxes
- Built-in geometric and mesh deformation



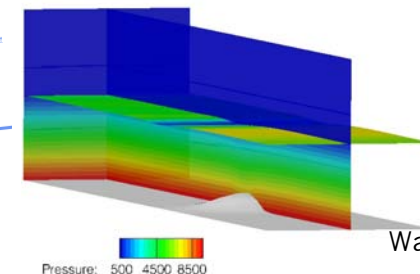
**Reactive Navier-Stokes**  
Electrons generated by arc heater passing over a solid body

## Multi-Physics Simulations

- Flexible C++ based architecture for rapid implementation of new equations and source terms
- Solution containers allow for simultaneous analysis of different equation sets with tight coupling
- Free surface simulations of air-water interfaces using the level-set method
- Multi-species plasma solver for simulating ionized flows in strong electric fields or behind hypersonic shock waves



**Uncoupled High-Fidelity Analysis**  
DLR-F6 pressure contours



**Free Surface Simulation**  
Water flowing over a bump in a channel