A Tool Chain for Aero-elastic Simulations

Mathias Doreille, Thomas Ludwig, **Silvio Merazzi**
SMR SA, CH-2500 Bienne, Switzerland

Luca Cavagna, Peter Eliasson
FOI, SE-164 90 Stockholm, Sweden

Table of Contents

Introduction

Purpose

- The SADE EC project is concerned with flexible wing structures.
- One of the tasks is to create a tool chain which allows to:
  - Compute the static aero-elastic equilibrium of a wing section with a flexible, so-called droop nose for different configurations, flow conditions, and different CSM models.
  - Evaluate lift and drag coefficients and the structural deformation and stresses.
  - The tool chain should be distributed to partners.
- This tool chain will be presented in the following.

Requirements

- Both CFD and CSM problems are nonlinear: Iterative procedure until aero-elastic equilibrium is reached. Robustness has to be adressed!
- CFD and CSM are highly complex and specialized disciplines: Problem of coupling of separate solvers.
- The coupling involves the following components:
  - Pre-processing tools (meshing and set-up of spatial coupling).
  - CFD solver.
  - CSM solver.
  - Spatial coupling.
  - CFD mesh deformation tool.
  - Data conversion tools to/from CFD data format.
- This complexity has to be hidden from the user as much as possible.

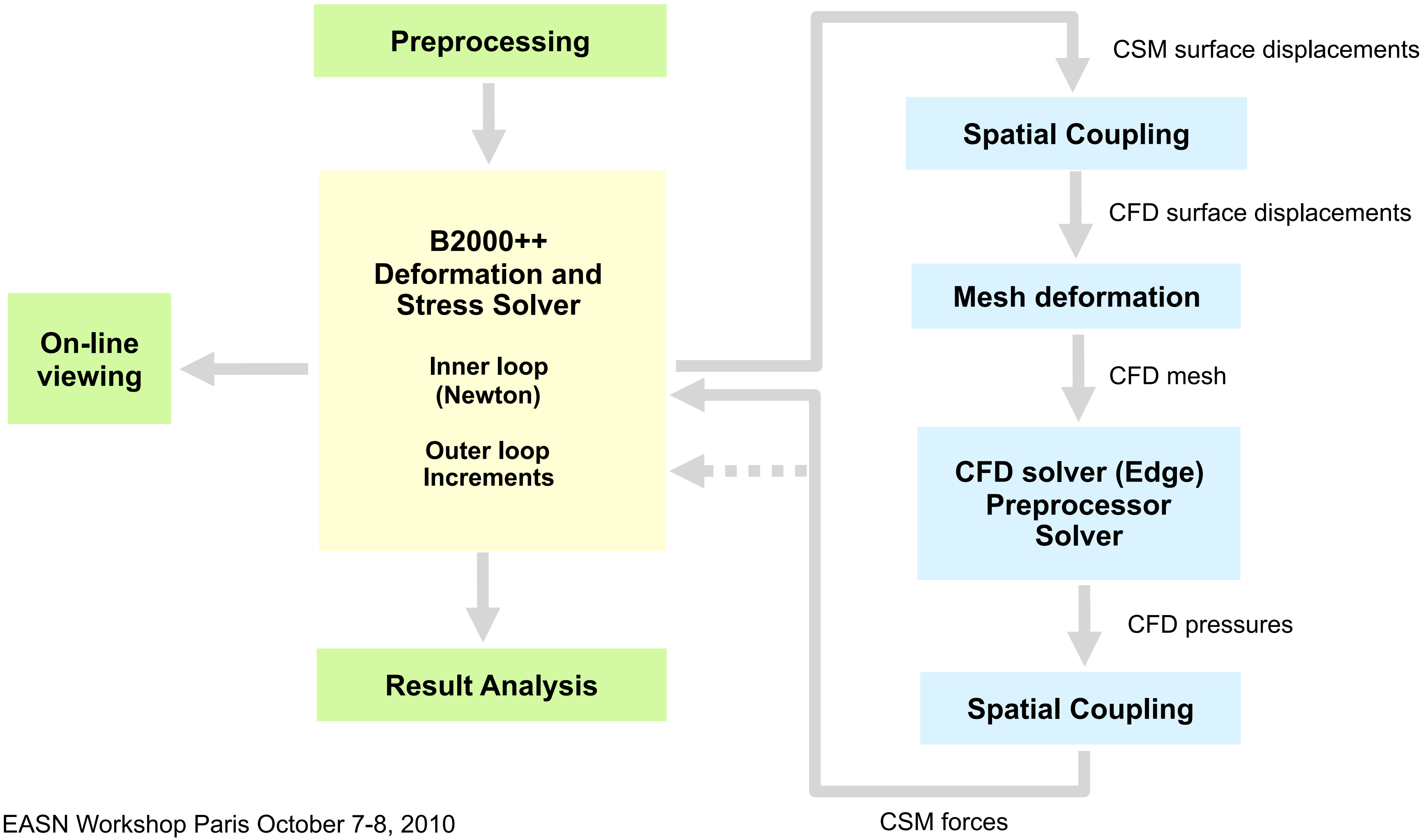
Requirements

- The tool chain is distributed to partners:
  - Requires background information that can be distributed (Edge CFD solver by FOI, B2000++ FE solver by SMR) and for which there is know-how in the consortium.
  - Requires pre- and post-processing tools that can be distributed.
- CFD domain is 2D for the following reasons:
  - 3D CFD is computationally very expensive. Not all partners have the required computing infrastructure.
  - The actuation mechanisms for the droop nose are modelled for wing sections and thus can be coupled to 2D CFD models.

Requirements

- Spatial coupling:
  - Should be conservative.
  - Should not introduce artificial stress concentrations into the CSM model.
  - Should not create a bumpy deformed wetted surface, otherwise, convergence problems occur in the CFD solver.
- Mesh deformation:
  - Should be fast.
  - Should not produce inverted cells for the geometries and deformations that are used.

The Components of the Tool Chain



Input and Output

- Input:
  - CFD profile data.
  - CFD far-field conditions and AoA.
  - CFD numerical parameters.
  - CSM FE mesh and material definitions.
  - CSM boundary conditions imposed by the droop-nose mechanism.
  - CSM numerical parameters.
- Output:
  - Lift and drag coefficients.
  - Pressure, density, temperature, velocity fields in the CFD domain.
  - Deformations and stresses in the CSM domain.

Pre-Processing

Preprocessing

Preprocessing

- **Import structural FE model**
- **Import CFD profile data**
- **Generate spatial coupling info**

Analysis

- From
  - Structure FE model and droop-nose deformation information from NASTRAN BDF file.
  - CFD profile data and flight conditions.
- Try to build, as automatically as possible
  - The CFD mesh (for Euler or RANS).
  - The mesh interpolation information.
- How ?
  - Re-mesh CFD profile data for the required mesh density.
  - Mesh CFD for boundary layer and “volume”.
  - Augment CFD and CSM models with definitions of spatial coupling surfaces.

Preprocessing (Structure)

Preprocessing

Import structural FE model

↓

Import CFD profile data

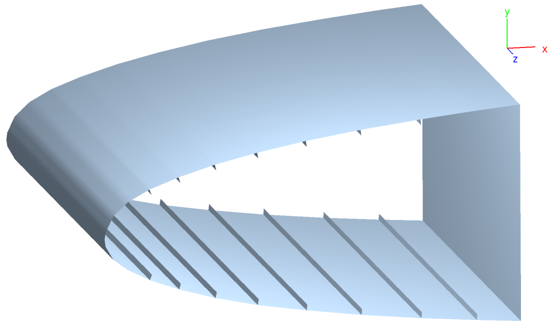
↓

Generate spatial coupling info

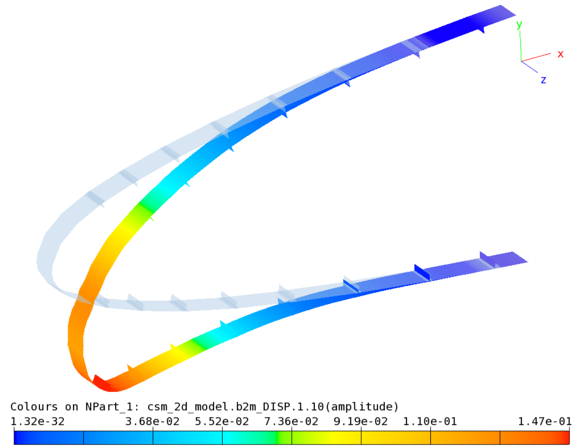
↓

Analysis

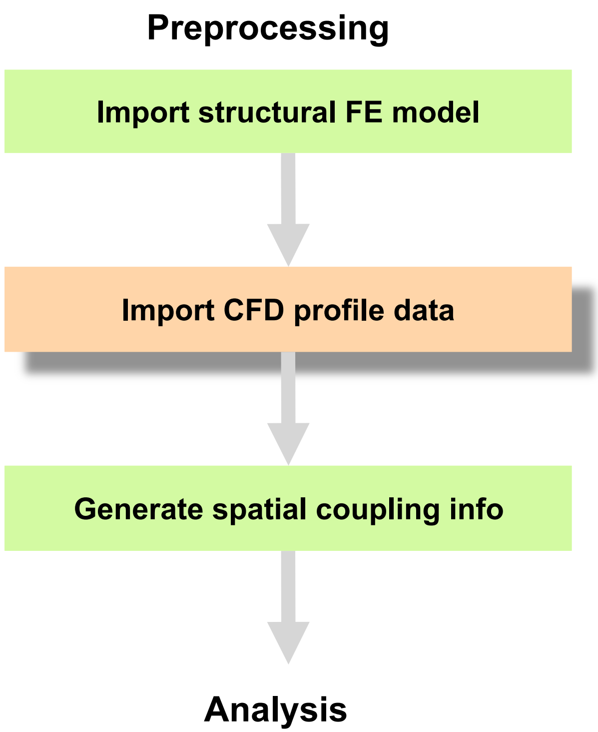
- Import structure FE model from Nastran BDF.



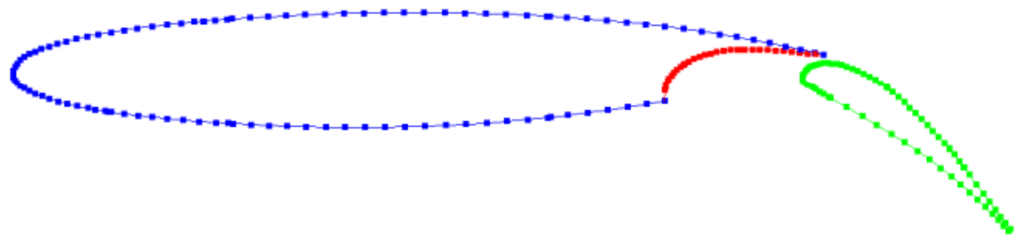
- Check and modify.
- Make 'dry' run if possible.



Preprocessing (CFD)



- Profile data are given as a sequence of points (example: F15).



- This profile data is not directly suited for meshing. Solution: Approximate the profile data by a spline of order > 3 and re-mesh that spline.
  - Take curvature and distances between walls into account.
- Run the mesh4flow tool which creates 2D CFD Euler and hybrid meshes from profile data.

Spatial Coupling Information

Preprocessing

Import structural FE model

↓

Import CFD profile data

↓

Generate spatial coupling info

↓

Analysis

- The spatial coupling requires for both CFD and CSM the definition of a set of coupling surfaces.
  - E.g. A wing with flap has two coupling surfaces, one for the wing, and one for the flap.
- The CFD and CSM models must be augmented with this information. It will be used by the spatial coupling method during the analysis.
- SMR has developed the graphical/batch tool FSCON which can be used to generate the spatial coupling information.

Analysis

3

The Edge CFD Solver

- Flow solver for unstructured grids made by FOI (Swedish Defense Research Agency).
  - Industrial strength.
  - Runs on small Linux platforms as well as on large clusters.
- Features:
  - 2D and 3D.
  - Euler, RANS, DES, LES.
  - Steady and unsteady flows.
  - Node-centered Finite Volume scheme.
  - Convergence acceleration: Multi-grid, residual smoothing.
- For the tool chain: 2D Euler and RANS steady calculations.

The B2000++ CSM FE Solver

- B2000++ is:
  - A nonlinear FE research code for CSM and CHT.
  - Extensible (via C++).
  - http://www.smr.ch/b2000
- Implementation of the coupling:
  - The CSM FE solver considers the CFD calculation to be a nonlinear boundary condition ('natural boundary condition').
  - The aero-elastic components (CFD solver, spatial coupling, mesh deformation) are integrated into this boundary condition.
  - This boundary condition is evaluated several times per load increment to reach the aero-elastic equilibrium.
  - This method can be used for linear and nonlinear CSM models.

Common Mesh Refinement

- **Common mesh refinement and L_2 minimization** (X Jiao, M. T. Heath, Common-refinement-based data transfer between non-matching meshes in multiphysics simulations, Int. J. Numer. Meth. Engng. 2004; 61.)
- Couple two polynomial surface fields from similar but non-matching meshes, such as displacements and pressures/tractions.
- Method based on minimization of difference of surface pressure and surface displacement fields between the coupled surfaces (i.e. lines in 2D) with L2 norm.
- Requires accurate integration over the surfaces, hence common refinement of the 2 surface meshes.
- Gives very accurate results, which is important for the type of structures studied within SADE.
- Guarantees conservation of quantities.

Common Mesh Refinement: Method

- Couple two polynomial surface fields, such as displacement fields or pressures/ tractions fields.
- Method: L_2 minimization:
  - Given polynomial functions s_i , f_i defined piecewise in discretized form

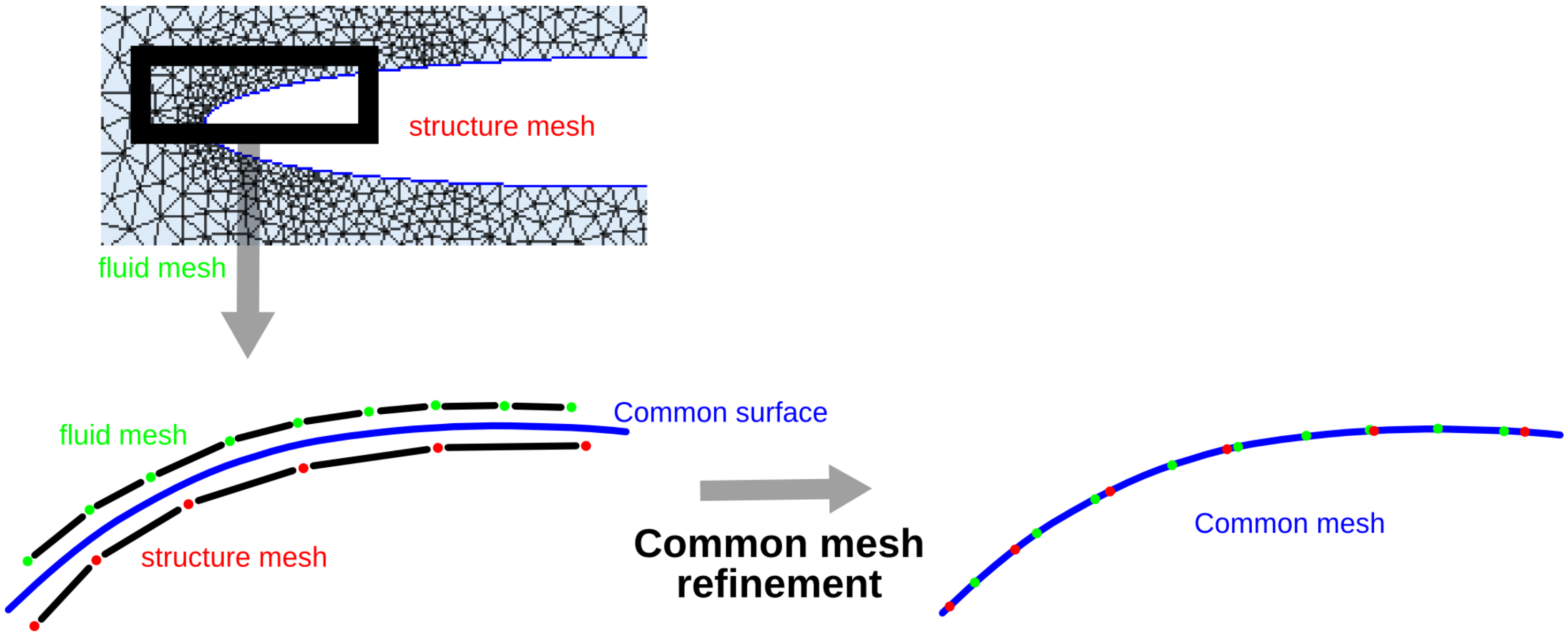
    $$s = \sum s_i \varphi_i \quad f = \sum f_i \eta_i$$

  - Couple s (structure mesh) and f (fluid mesh) fields.
  - L_2 minimization

    $$\frac{\int (s - f)\, dA}{s_i} \rightarrow \min$$

    over a common mesh of area A. Sparse symmetric positive definite coupling operator!
  - Solve a system of non-linear constraints together with FE solver. Method: Augmented Lagrange multipliers.

Common Mesh Refinement: Method



Combine fluid mesh and structure mesh...

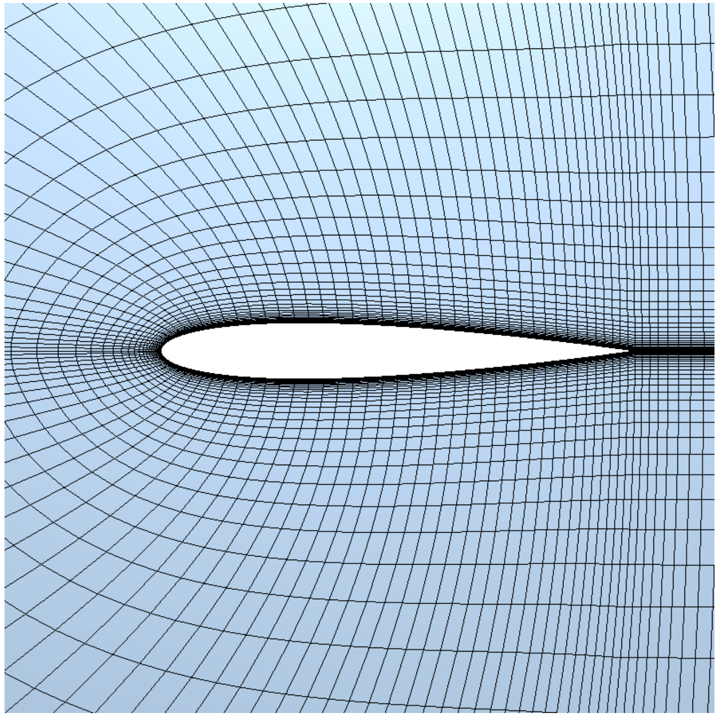
To one common mesh. Pressure/tractions (CFD) and displacements (structure) become integral DOFs of the structural (FE) model.

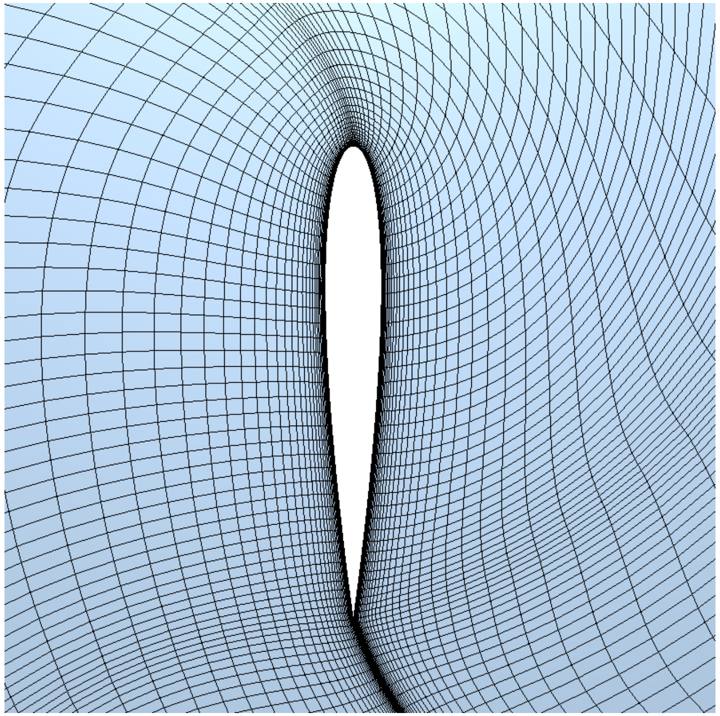
CFD Mesh Deformation

- The method chosen is well suited for 2D grids with large deformations. It is better than interpolation methods.
- Procedure:
  - From the CFD mesh, create a CSM FEM mesh with identical coordinates and connectivities and with isotropic elastic material.
  - The Poisson number depends on the distance to the wall. Hence, cells with large aspect-ratio are almost incompressible. This prevents inverted cells.
  - Lock the far field nodes.
  - Impose the mapped displacements at the wall nodes.
  - Run a deformation analysis using the B2000++ FEA solver.
  - Modify the coordinates in the CFD mesh accordingly.
- For 3D grids this method is not effective.

CFD Mesh Deformation

- Example: NACA0012 profile (linear analysis in a single step):





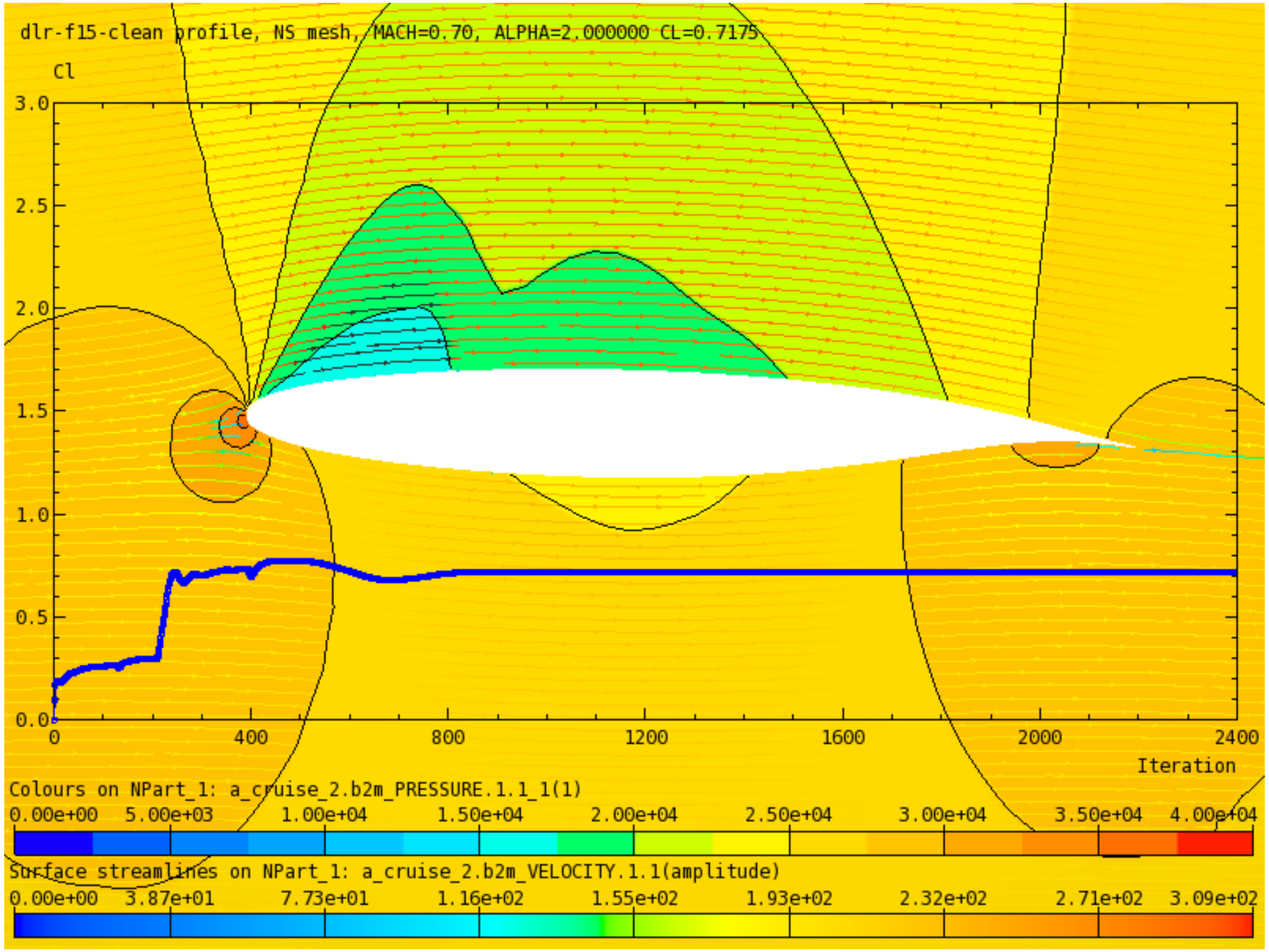
Tool Chain: Post-Processing

4

Baspl++ Post-Processor

- Can be steered by graphical user interface and via Python scripts.
- Useful for both CFD and CSM.
  - Both models can be visualized simultaneously.
- Many of the typical methods for the CFD domain are available or can be implemented with Python modules. Examples:
  - Plotting of Cl vs. AoA.
  - Velocity field: Surface streamlines and 3D streamlines.
  - Other fields: Colouring, iso-lines, etc.
- For CSM:
  - Extraction and plotting of material directions, principal stress directions.
  - Deformation plot.

The baspl++ Post-Processor



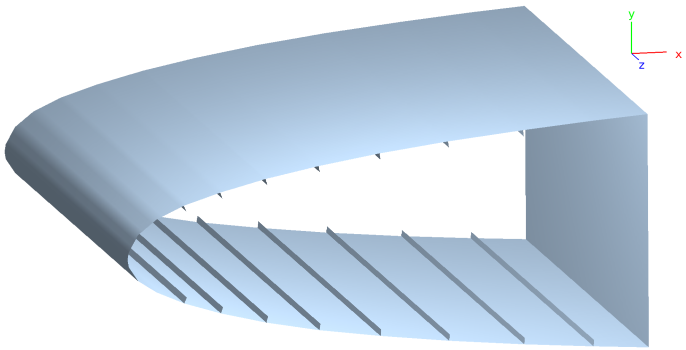
Analysis results for cruise configuration ('2D' N-S calculation)

A Closer Look at the SADE Droop Nose

5

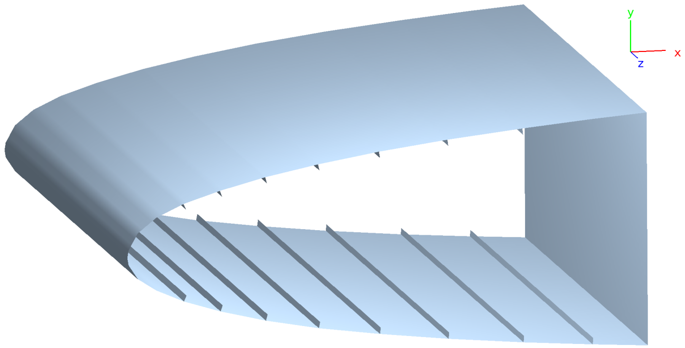
SADE droop nose

- Example reflects current status of tool:
  - Automatic 2D and '2.5D' fluid mesh generation (Euler, Navier-Stokes) available, starting from profile data.
  - B2000 structural solver (deformations, stresses, mesh deformation, mesh interpolation) and Edge CFD solver coupled.
- Test use case: *Cranfield FE model* with simple deformation pattern, DLR F15 profile without and with flap extended.

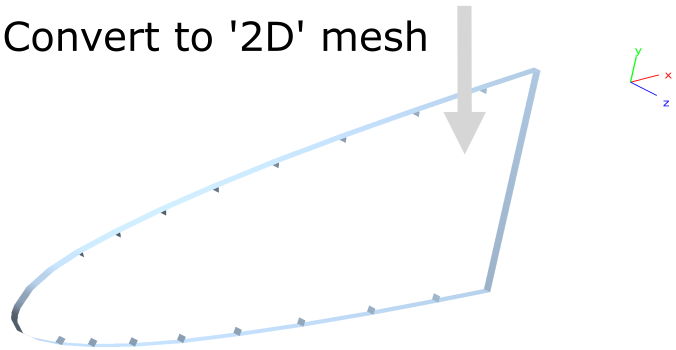


SADE droop nose: Structure model

Full 3D shell mesh

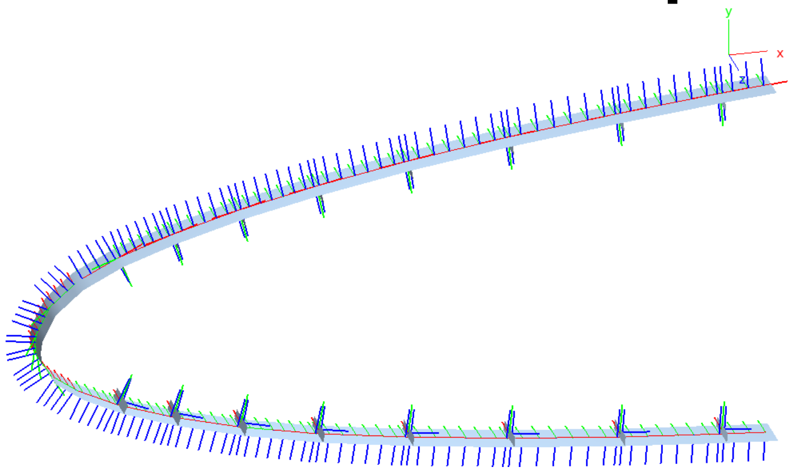


Convert to '2D' mesh

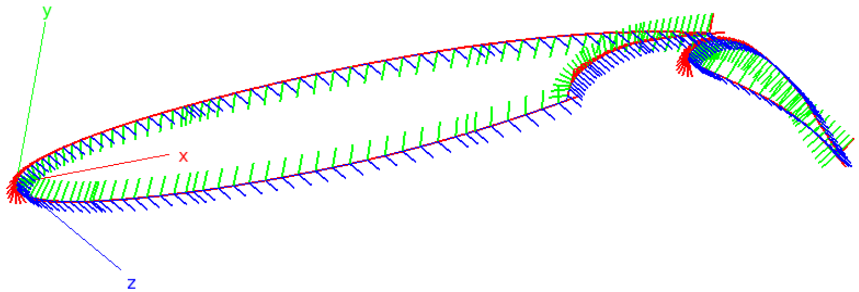


- Original Nastran model in BDF format.
- Automatic conversion to B2000++ format.
- Rotation to same reference frame as CFD mesh (x-y plane).
- Visual checks.
- Automatic application of coupling boundary conditions for mesh interpolation finite elements.
- Structural model 'steers' analysis: Apply load step.

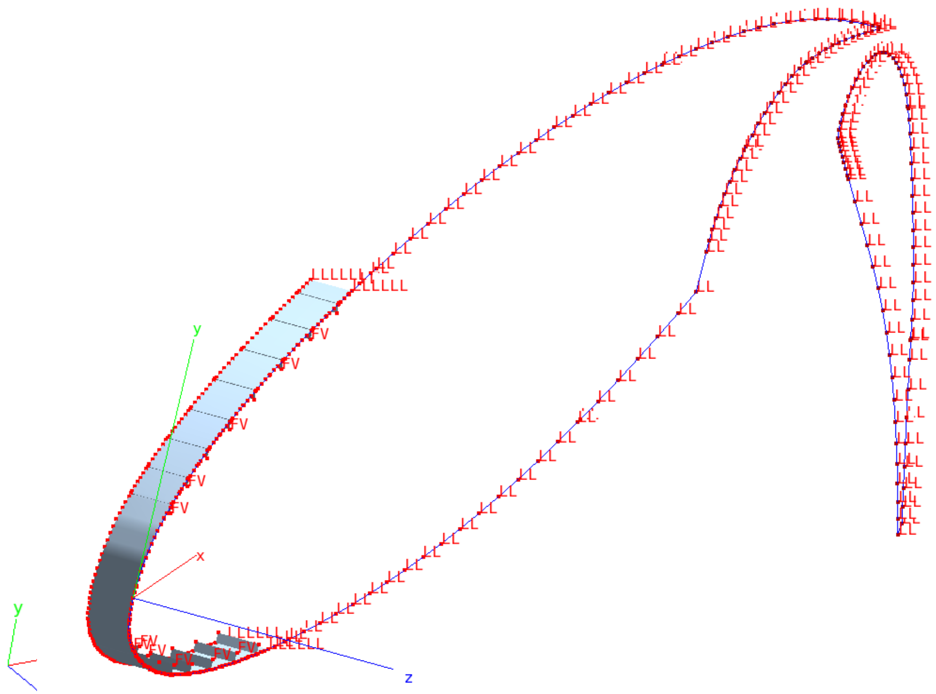
SADE droop nose: Structure model



Check structural model surface normals:
Must point into the fluid (blue lines).

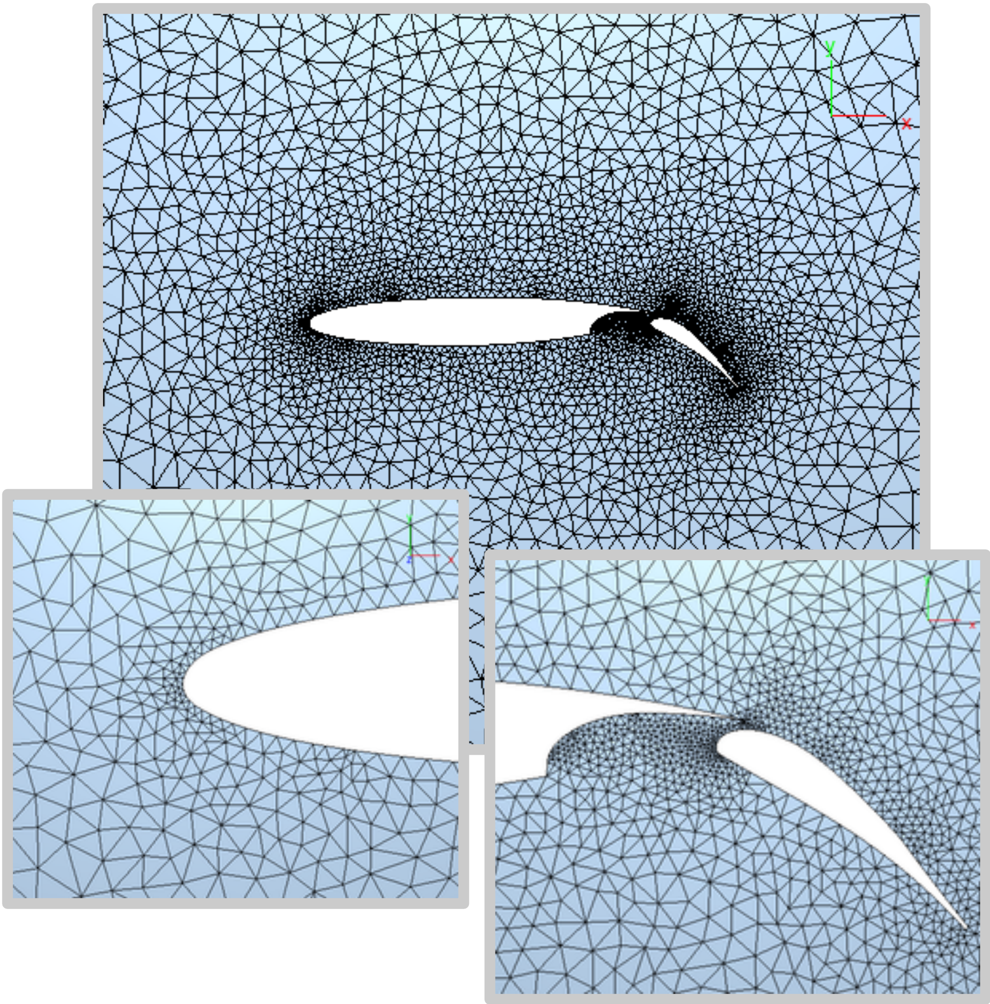


Check fluid model surface normals:
Must point into the structure (blue lines).

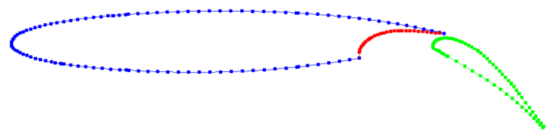


Check problem boundary conditions: **Structure mesh nodes** must be locked where clamped (L code). **Fluid mesh nodes** not coupled to structure must be locked (L code)

SADE droop nose: CFD Isotropic Meshes

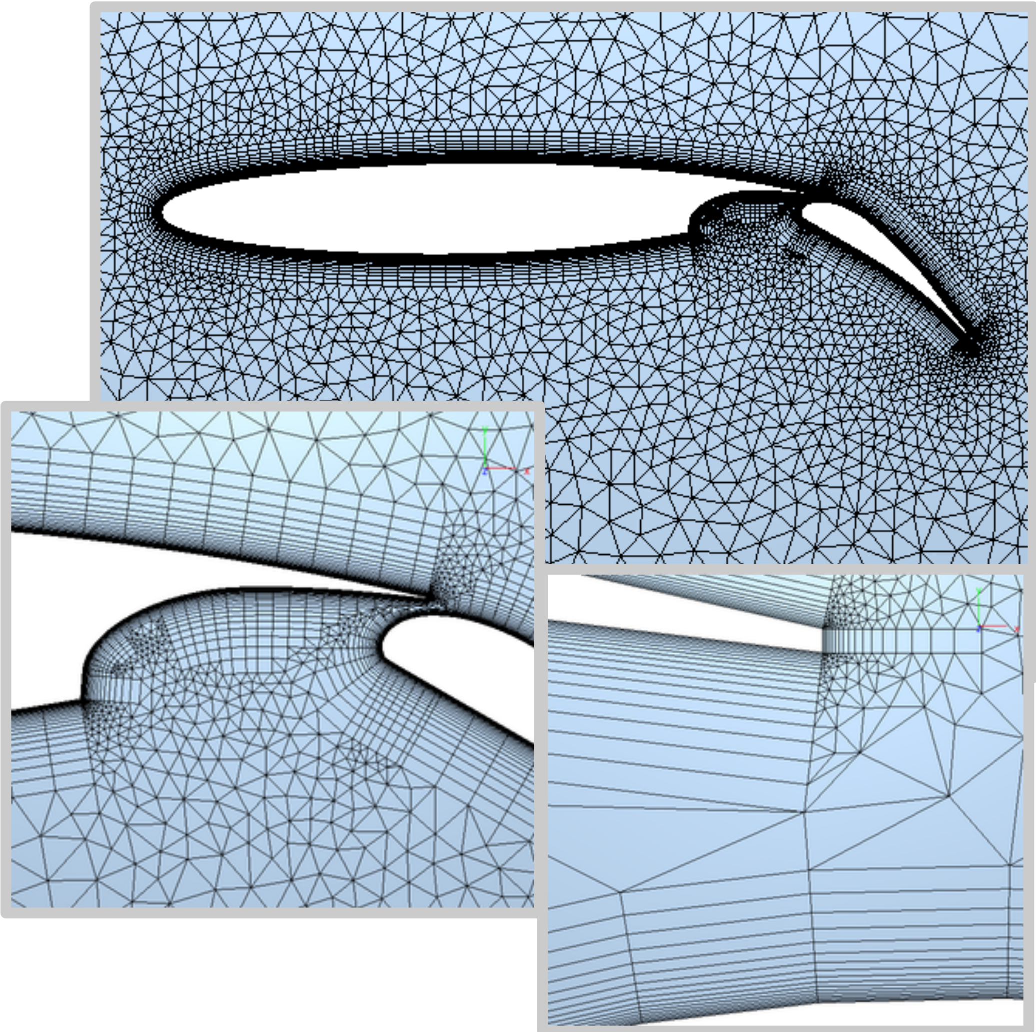


- Targeted for Euler simulations solely.
- User specification:
  - One outer bound for far field, such as circle or 'C' boundary.
  - Arbitrary number of inner bounds for wall boundary.



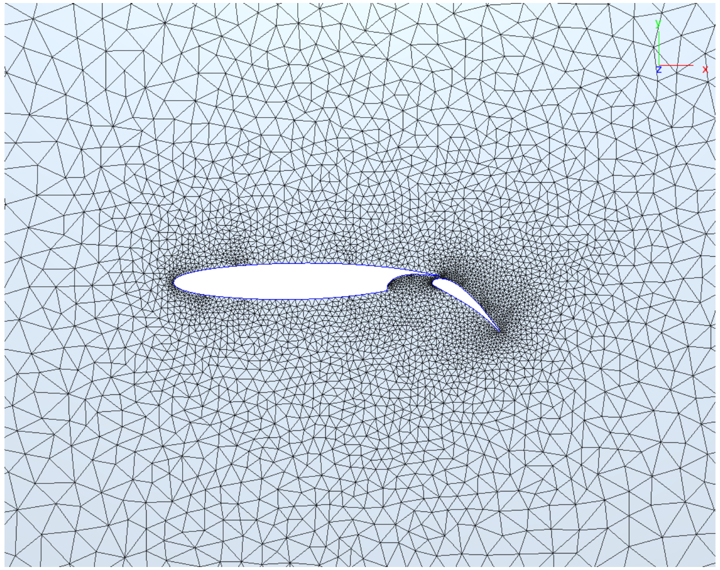
- Algorithmic features:
  - Constraint Delaunay triangulation.
  - Discrete boundary defines mesh resolution/density.

SADE droop nose: High Aspect Hybrid Meshes

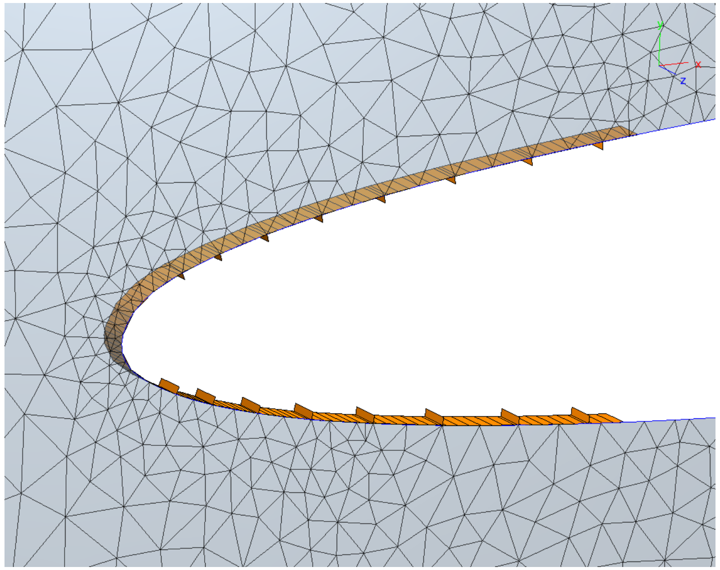


- Targeted for Euler and Navier-Stokes simulations.
- User specification:
  - Isotropic model definition.
  - In addition: Re, No of growth layers in boundary layer, etc.
- Algorithmic features:
  - Estimation of thickness of boundary layer from Re, (Re=1 --> Euler).
  - Non overlapping fronts of quadrilateral elements.
  - Delaunay for remaining domain.

SADE droop nose: Coupled meshes

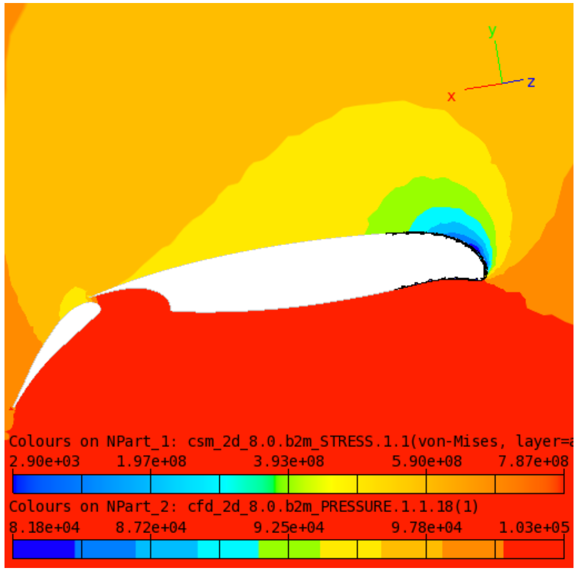


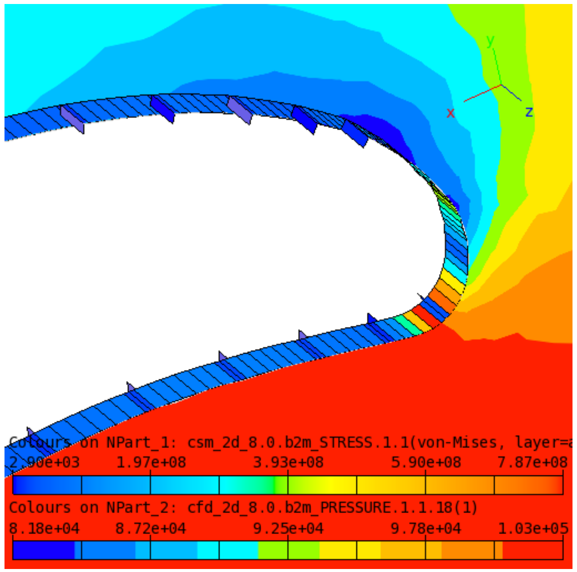
Fluid mesh (Euler mesh)
generated with mesher tool



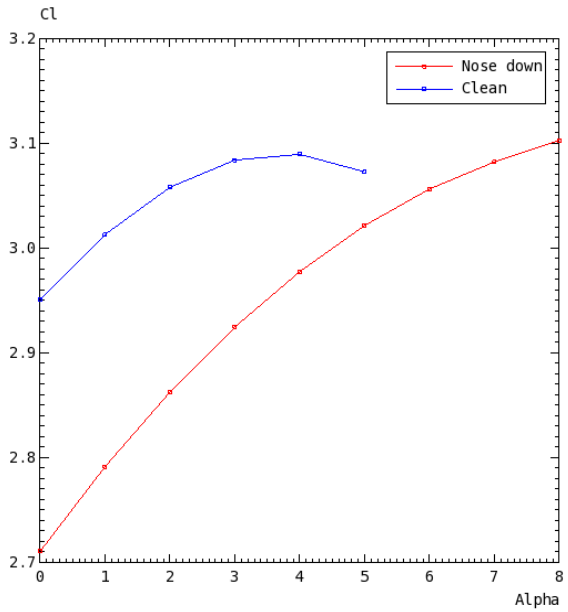
Fluid mesh (grey) and
structure mesh (red)

SADE droop nose: Static deformation





- Sequence of test computations with increasing AoA.
- Left: Deformed shape with final converged CFD-CSM solution.
- Left below: Detail. Observe 'wobbles'.
- Below: C_l vs. AoA for clean and deformed configuration.



Summary

Summary

- The aero-elastic tool chain as presented:
  - Is capable of computing the static nonlinear equilibrium of
    - 2D and '2.5D' CFD models.
    - Handles complex CSM models of wing sections containing actuator mechanisms.
  - Allows for evaluation of:
    - Lift and drag coefficients and CFD variable fields.
    - Deformation and stresses in the CSM model.
  - Features conservative spatial coupling, and with the common mesh-refinement method, precise introduction of aerodynamic loads.
  - Complements frequency-domain aero-elastic tools.

Financial support from the EU under FP7 contract No. 213442
“SADE Smart High Lift Devices for Next Generation Wings ”
is acknowledged.