

**A comparison of ANSYS Fluent and STAR-CCM+ simulations for a
tangent ogive slender body
with a structured mesh at incompressible flow conditions**

Janine Schoombie

CSIR



ANSYS[®]
FLUENT[®]

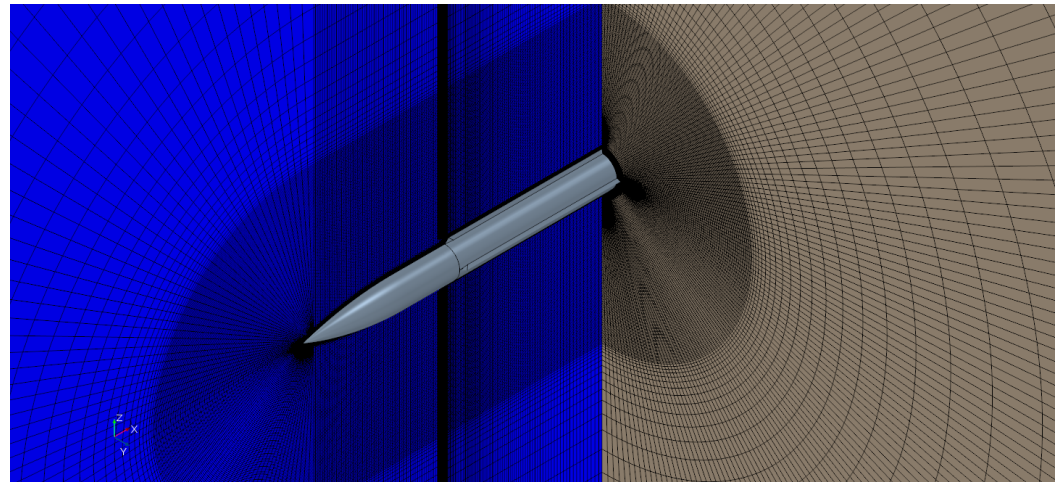
VS

STAR-CCM+[®]

Background

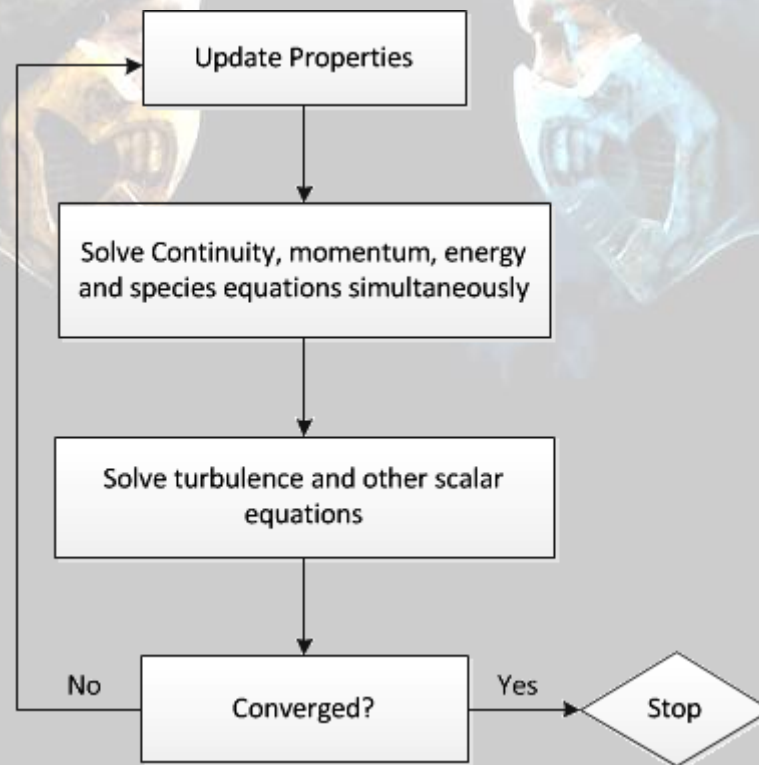
- First round of simulations in 2013 in ANSYS Fluent v15.0
- Validated experimentally
- Three Missile configurations with very low aspect ratio wings
- GAMBIT structured mesh
- Mesh convergence at 22million cells
- $y^+ \approx 1$

Span-to-body diameter ratio (s/D)	Aspect Ratio
1.25	0.011
1.5	0.022
1.75	0.033



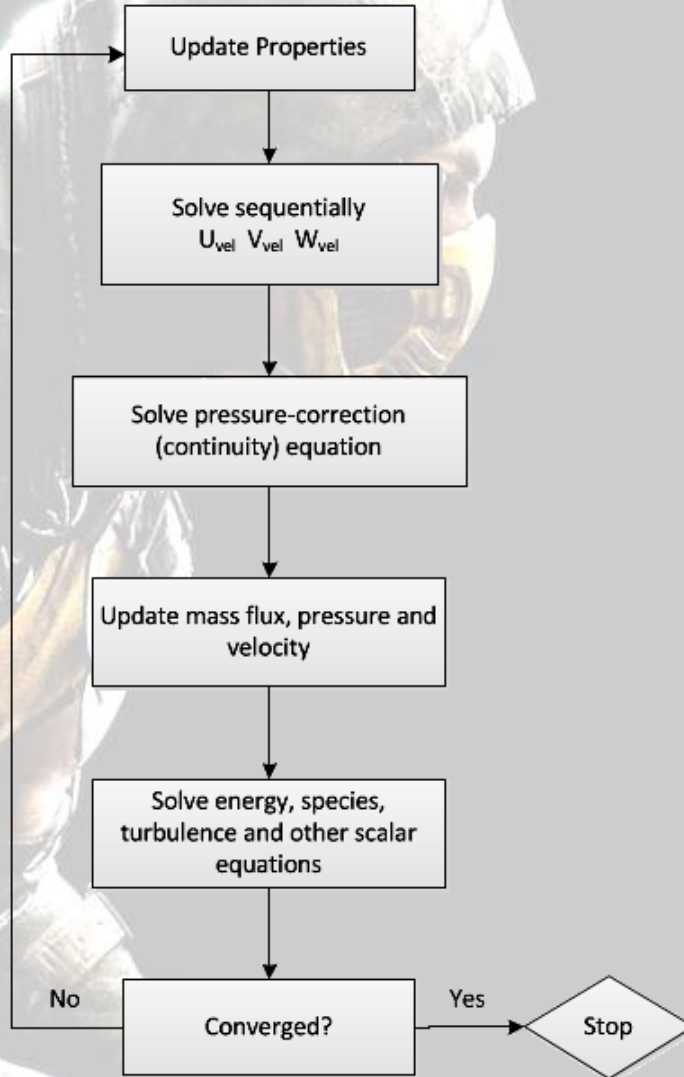
Solver Settings

Density-based (fully coupled) Algorithm

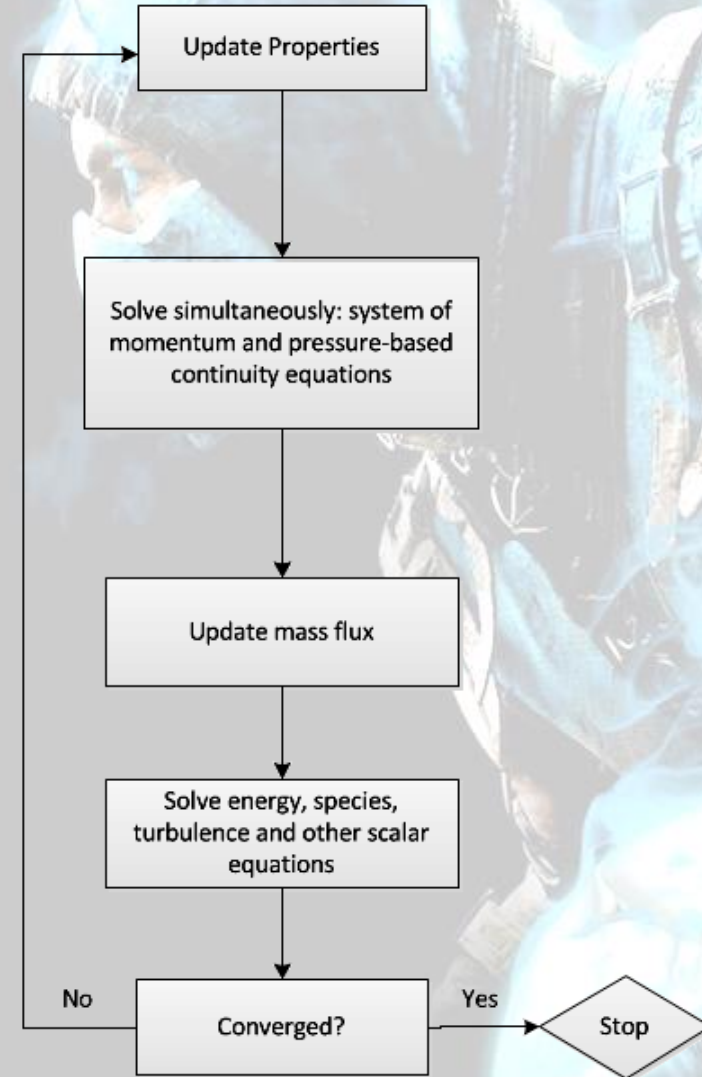


Solver Settings

Pressure-Based Segregated Algorithm



Pressure-Based Coupled Algorithm



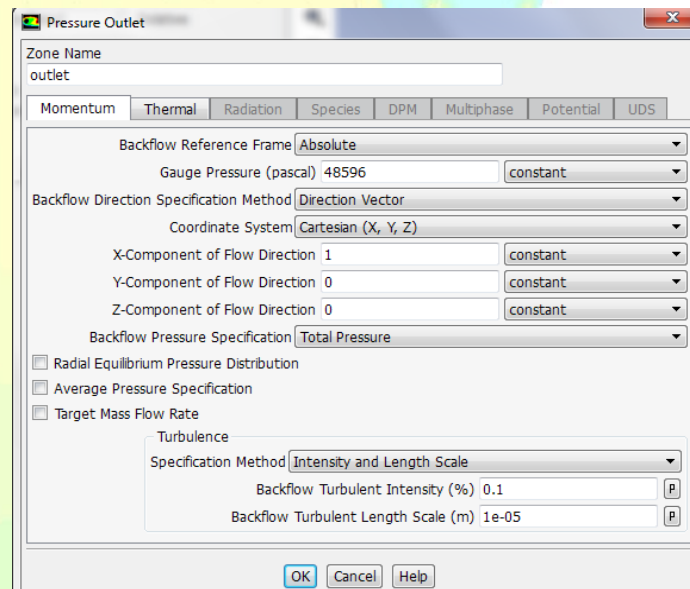
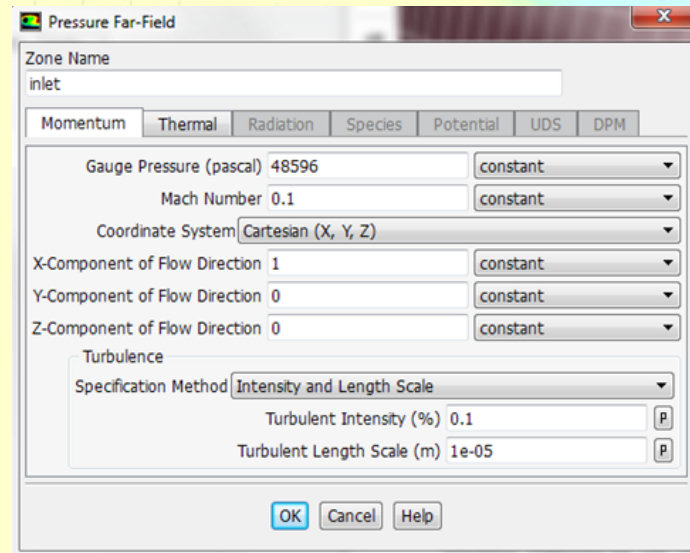
SIMPLE

~~SIMPLEC~~

PISO

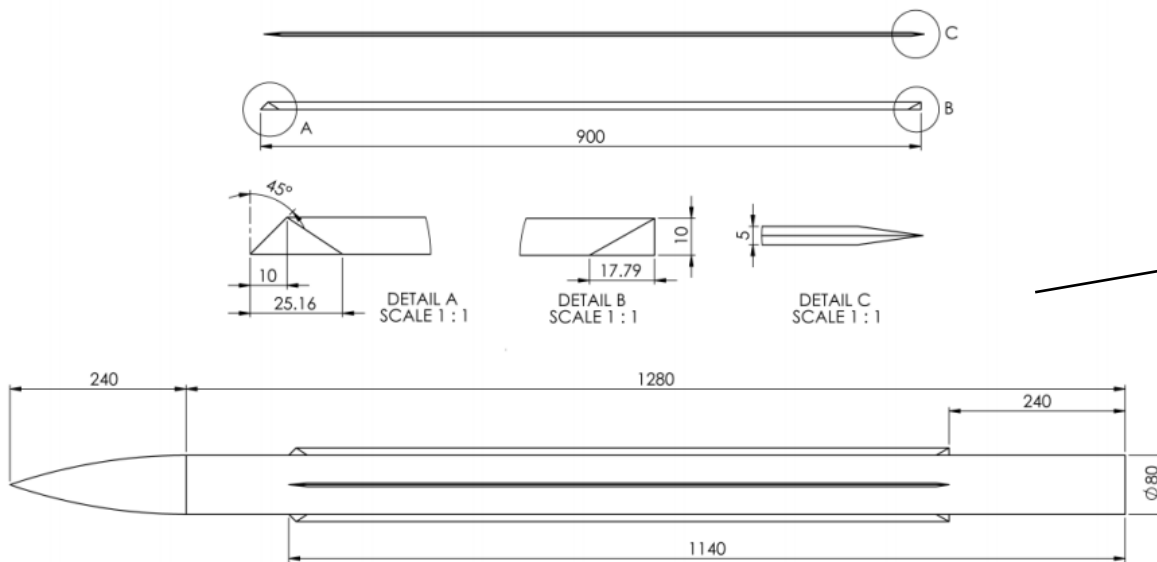
Fluent settings

- Pressure-Based Coupled Algorithm
- RANS
 - Spalart-Allmaras
- Velocity range
 - $0.1 < \text{Mach} < 0.3$
 - Mach independence shown for all three configurations
- Angle of attack (α) range
 - $0^\circ \leq \alpha \leq 25^\circ$
- Solutions converged after $\approx 10\,000$ iterations

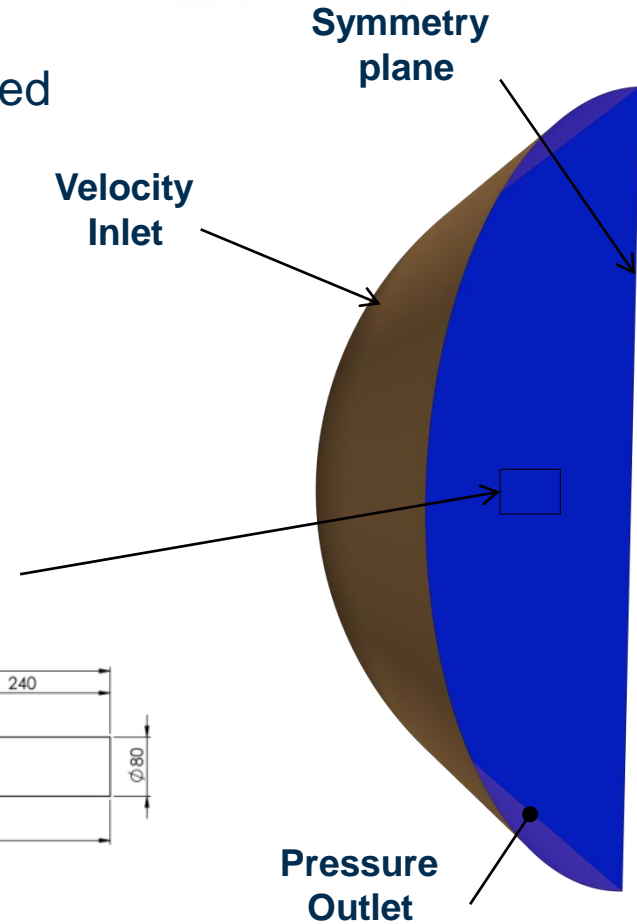


STAR-CCM+ Simulated Geometry

- Configuration $s/D=1.25$ successfully simulated to convergence



Body geometry

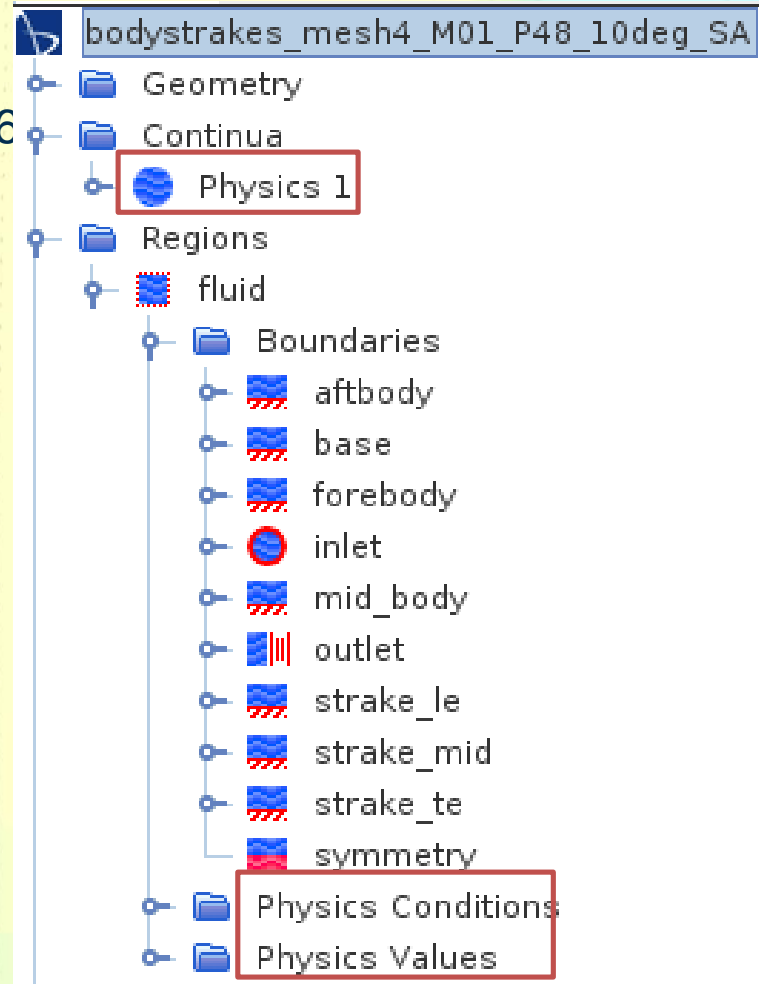


Import Fluent case files into STAR-CCM+ (1)

- Why STAR-CCM+?
 - In 2017 only a STAR-CCM+ v11.06 commercial licence was available
 - Didn't want to start from scratch
 - Test solver with identical mesh

STEP 1:

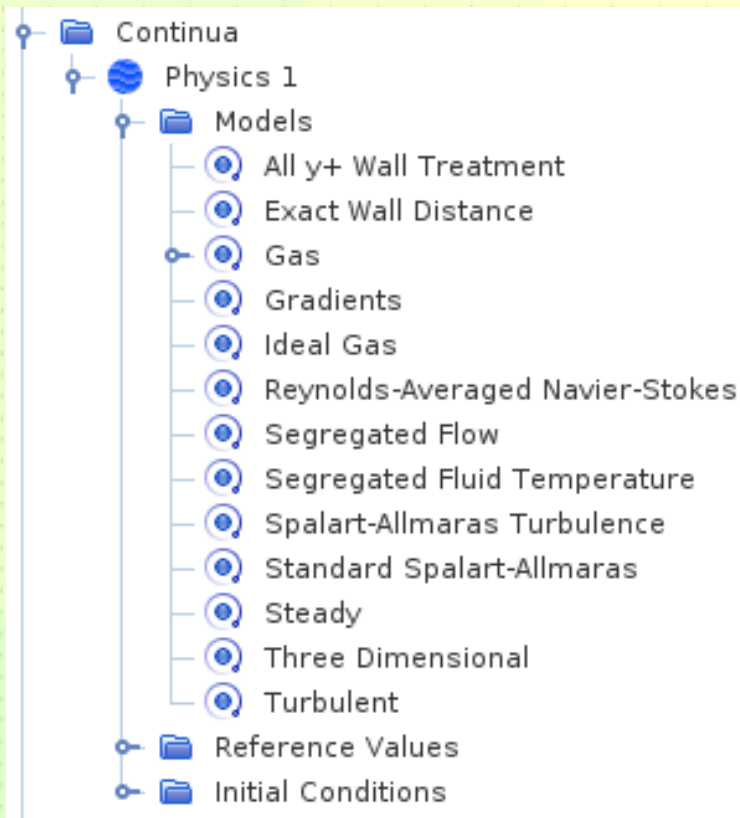
- Import Fluent .cas files
 - Imported as a volume mesh
 - No mesh continuum is created



Import Fluent case files into STAR-CCM+ (2)

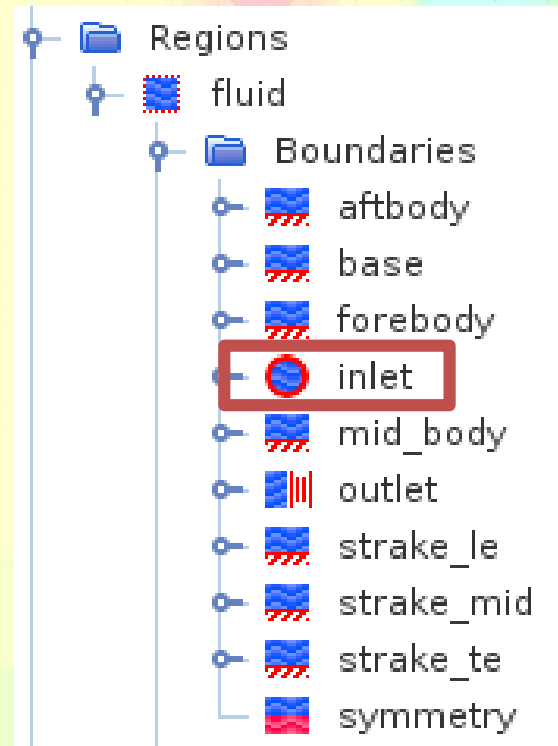
STEP 2:

- Select physics models



STEP 3:

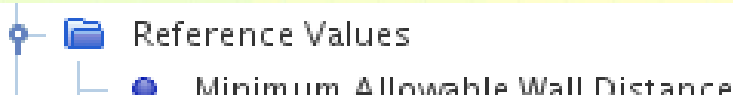
- Set up boundary conditions
 - Boundary conditions recognised by volume import action



Import Fluent case files into STAR-CCM+ (3)

STEP 4:

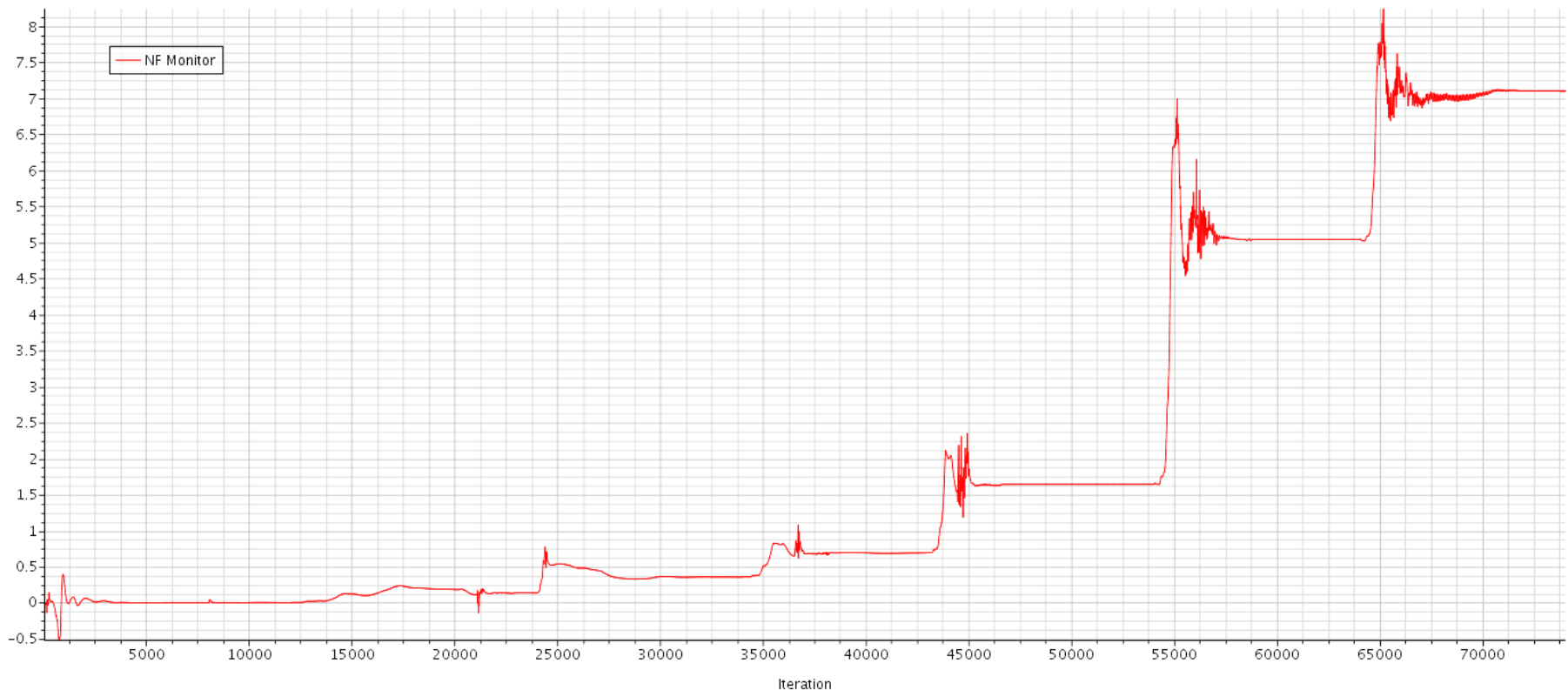
- Enter reference pressure



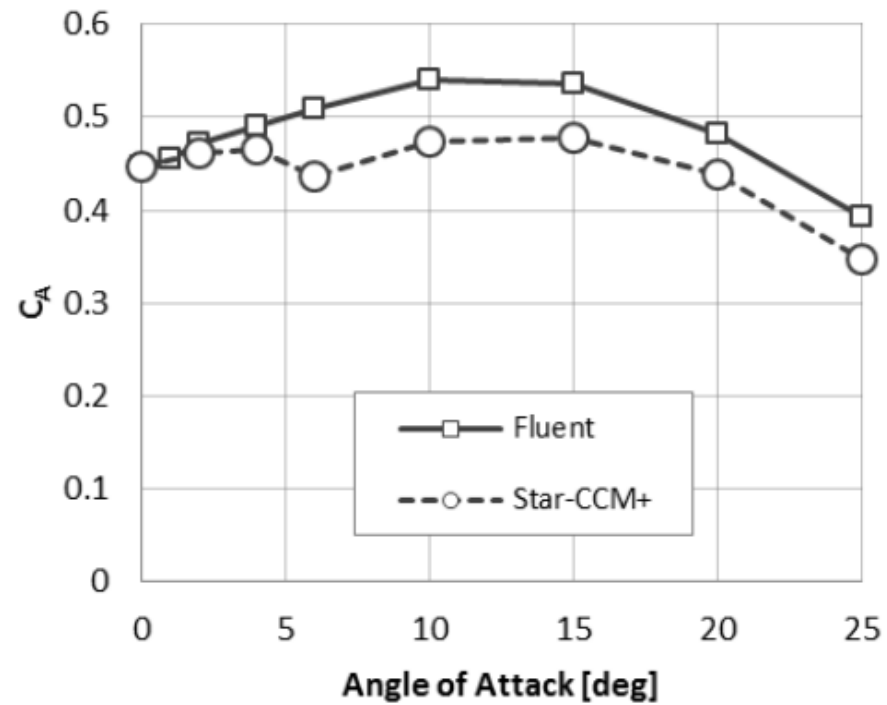
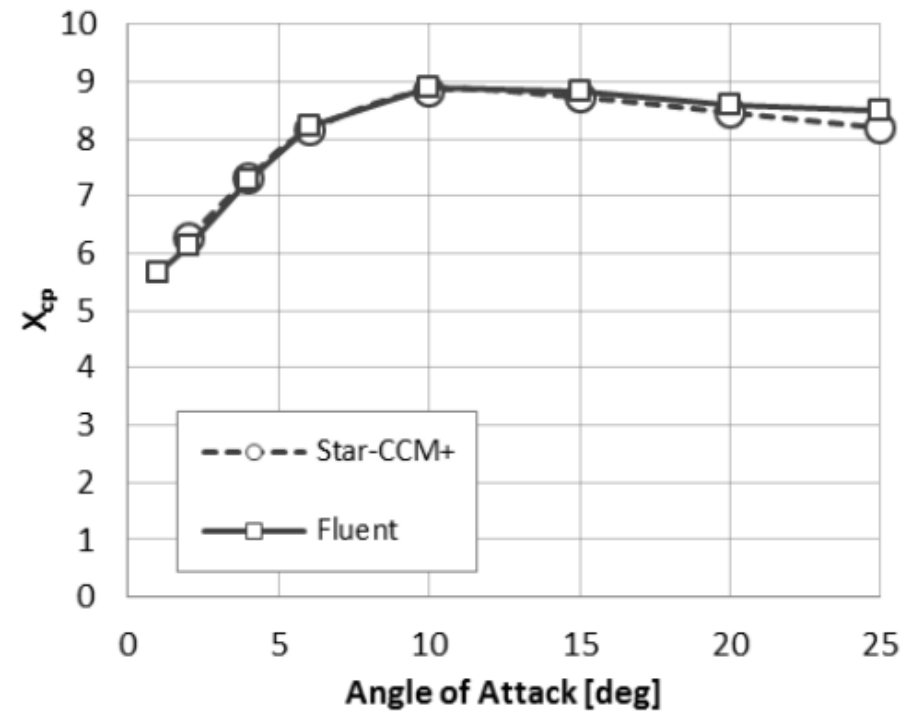
STEP 5:

- Enter initial conditions
 - Simulation started at 0° angle of attack – increased angle of attack and used previous solution as initial condition

NF Monitor Plot

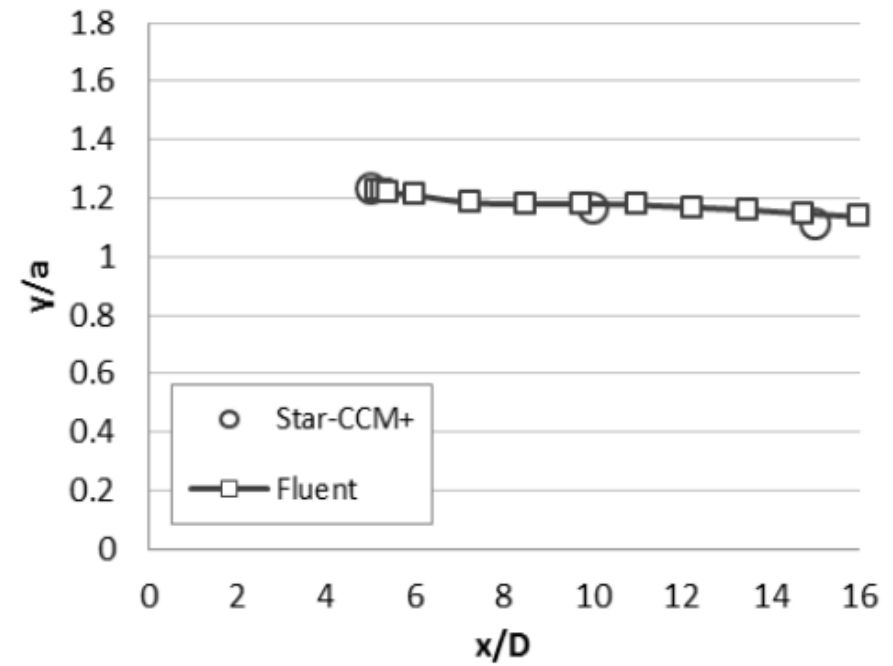
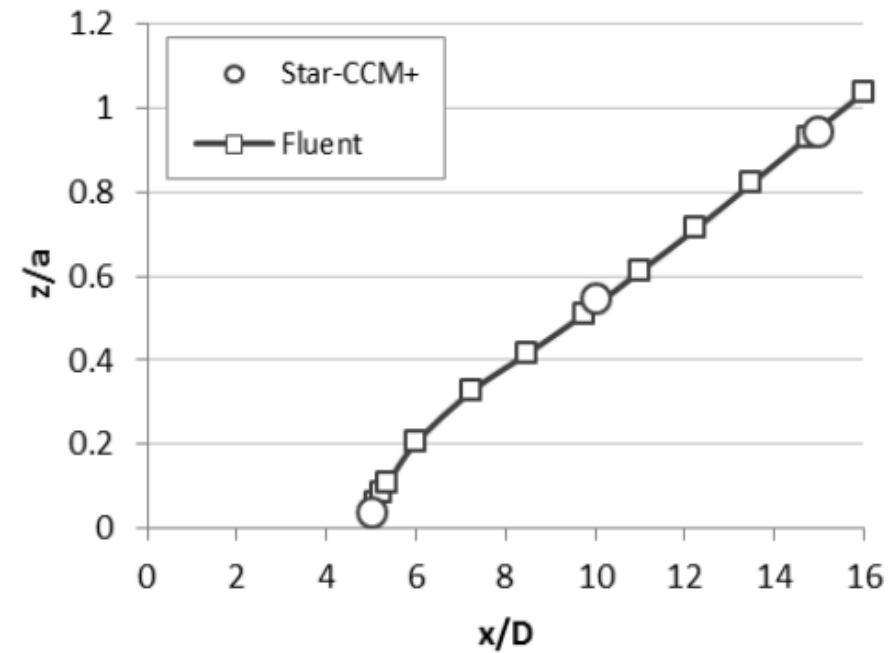


Results (1)



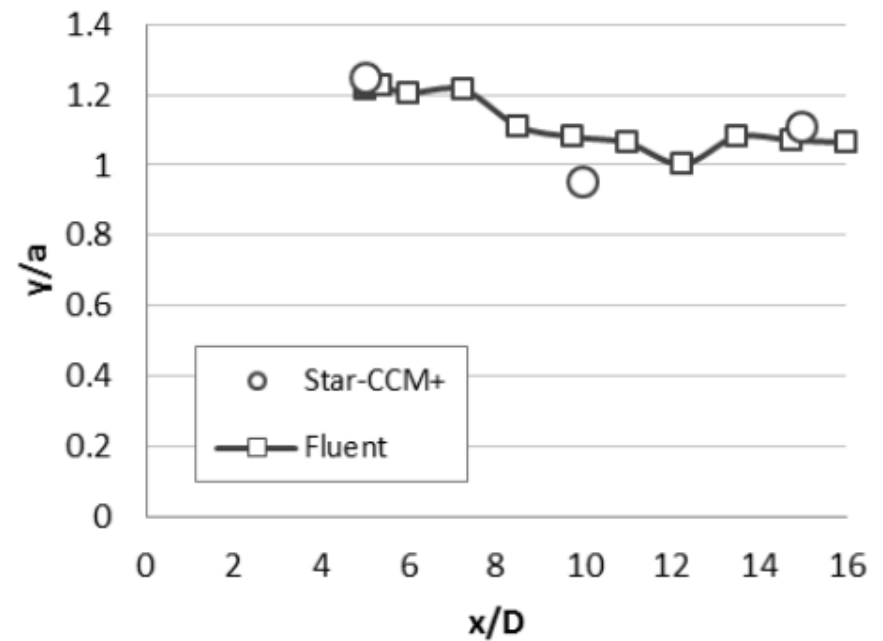
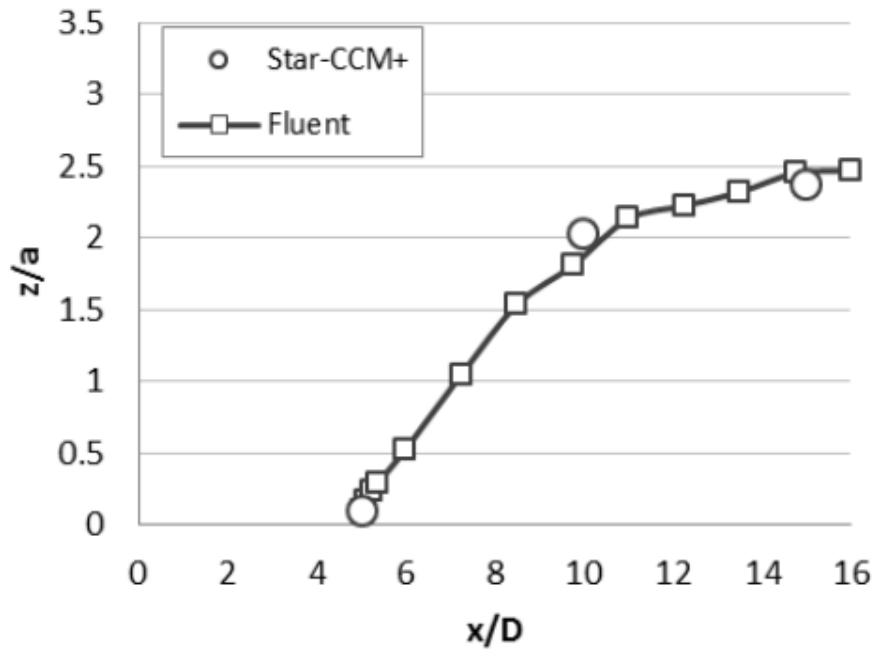
Results (2)

Vortex positions at **6 degrees** angle of attack



Results (3)

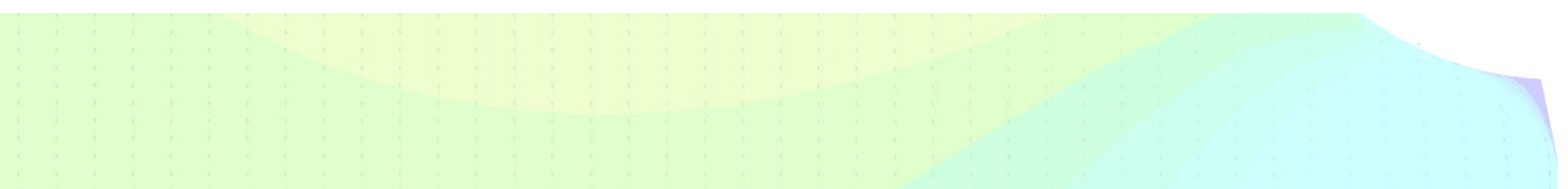
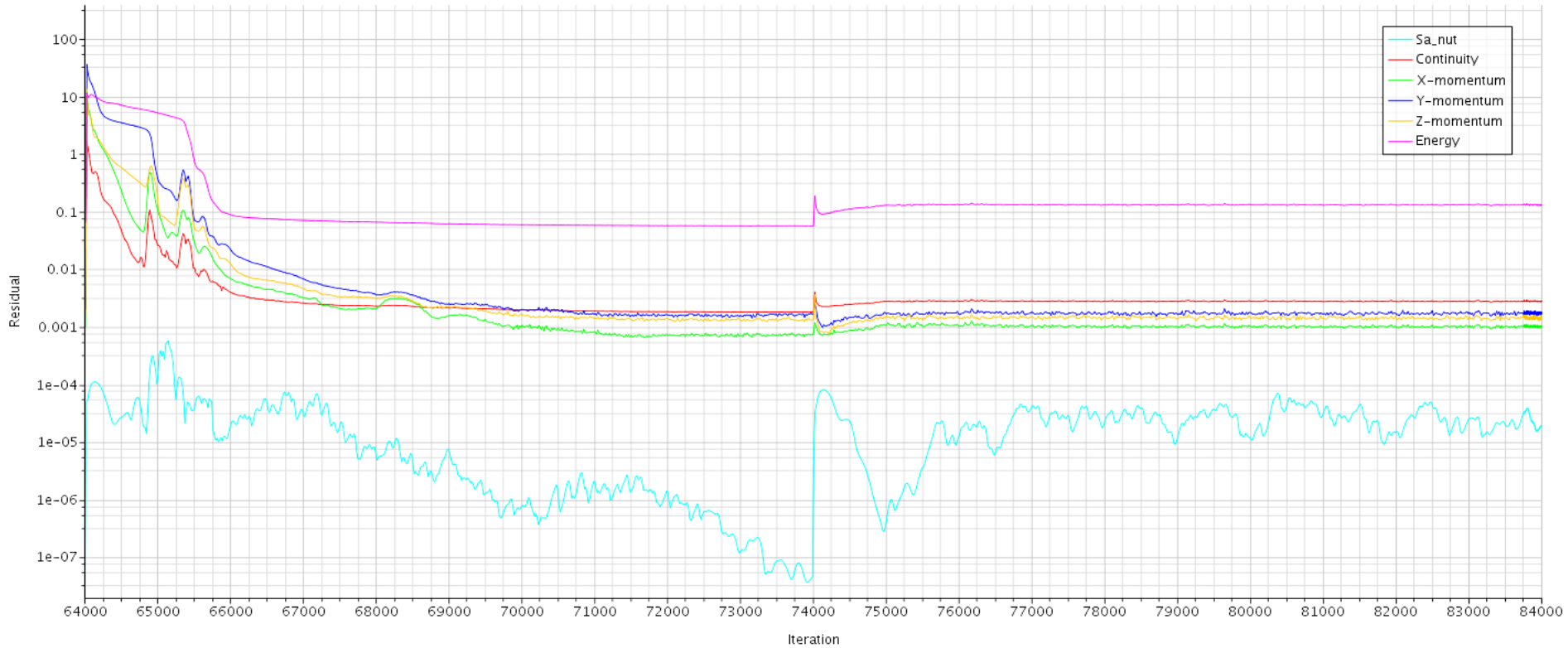
Vortex positions at **25 degrees** angle of attack



Segregated vs Coupled



Residuals



Summary

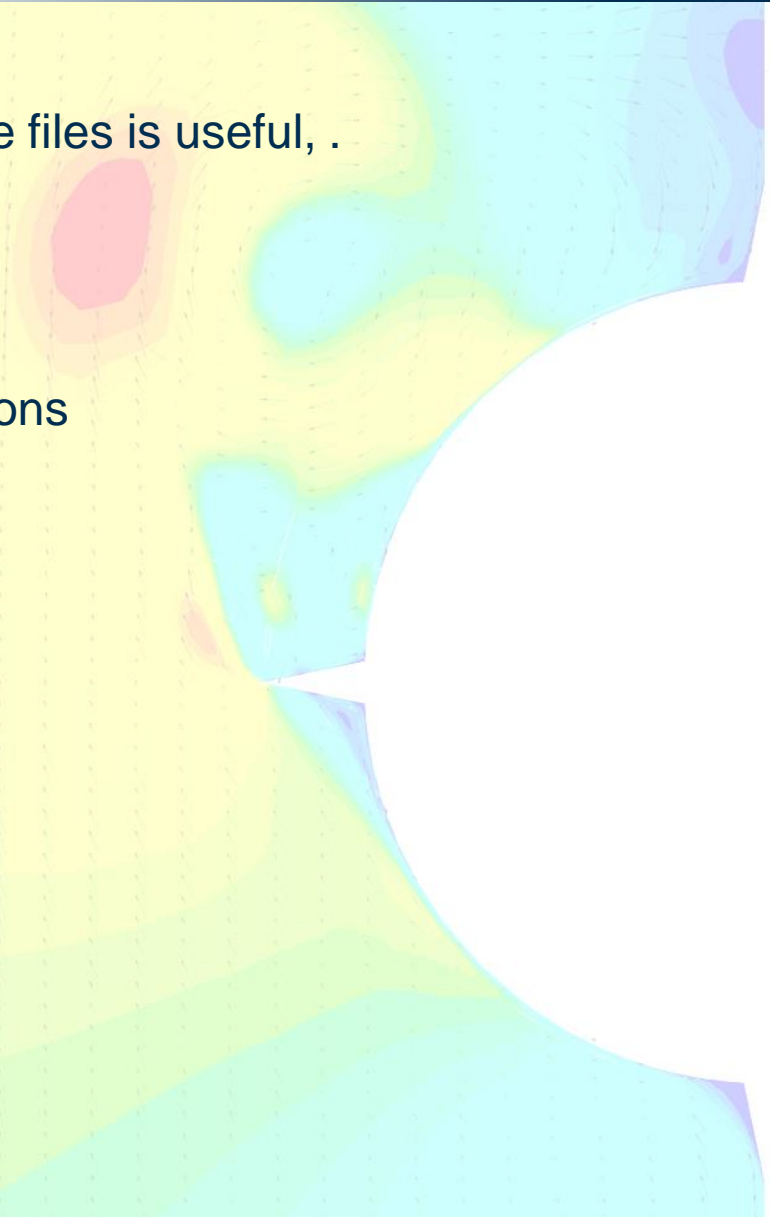
- The segregated solver results in STAR-CCM+ correlated well with the Fluent pressure-based coupled algorithm results for **normal force, pitching moment and centre-of-pressure position**.
- **Reasonable correlation** was observed between the **predicted wing vortex locations** with some minor discrepancies at higher angles of attack.
- A comparison of **the axial force results shows significant discrepancies** between the two CFD codes, particularly at angles of attack of 6 degrees and higher, where flow separation is expected.
- Further study is required to determine the possible **differences turbulence modelling and in predicting boundary layer flow**, which may affect the axial force and lee-side flow features.
- The **difficulties experienced in implement the coupled solver and other configurations** in the STAR-CCM+ simulations also require additional investigation.

Additional comments

- The ability of STAR-CCM+ to import fluent case files is useful, .

BUT.

- No mesh continuum means no mesh modifications



Acknowledgements



Danie de Kock &
Dawie Marais



Johan Heyns
Jan-Hendrik Grobler

Thank you

