Drag-Prediction Workshop

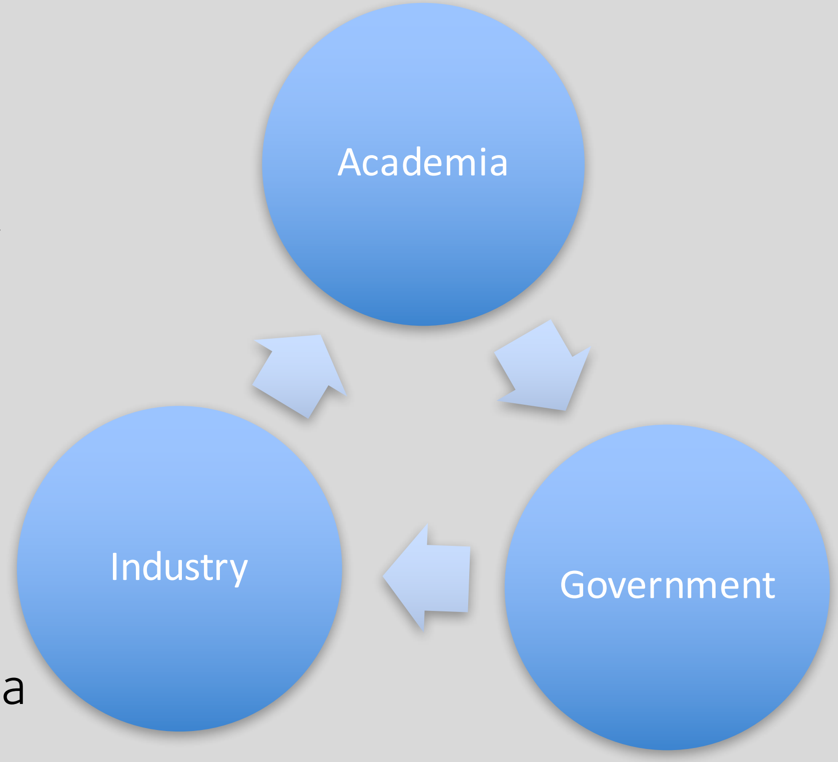
Neil Ashton* – University of Oxford
Vangelis Skaperdas - BETA-CAE Systems
Fred Mendonca– ESI Group

*neil.ashton@oerc.ox.ac.uk

- Motivation
- Partners
- OpenFOAM V&V
  - ZPG Flat Plate
  - NACA 0012 (Case 1)
- DPW Case 2
  - Meshing
  - Preliminary results
- Conclusions and future work

Motivation

- Open-source software is the basis for the majority of open collaborations.
- Few open-source software has all the features of a commercial or government code
- Typically academic codes are built for a specific purposes and set of cases
- OpenFOAM is arguably one of the most complete open-source finite-volume CFD codes available.



Motivation

- Mature code with many features
    - RANS (SA,SST, k-e, k-w, RSM (SSG,LRR) + more)
    - LES (Smag, Dynamic, WALE) + variety of filters
    - Hybrid RANS-LES (SA/SST – DES,DDES,IDDES + SAS)
- Automatic Mesh Refinement (AMR)
- Pressure and density based solvers
    - SIMPLE(C), PISO

- Poor I/O structure (one folder + one file per variable (and mesh) for each processor) e.g 10,000 cores = 150,000 per checkpoint
- No density-based implcit compressible solver e.g the bedrock of all NASA codes
- Published Verification and validation?

Aims

- We want to assess the current capability of OpenFOAM for transonic complex airframe geometries
- What is the current performance and where can we go from here?

Partners

- Whilst OpenFOAM has its own internal mesher – SnappyHexMesh (cartesian/prismatic cut-cell mesher) we want to focus on solver, turbulence models
- BETA-CAE systems created high-quality unstructured meshes using their ANSA software
- ESI Group have participated in an observatory role but are working with the University of Oxford to improve the code

OpenFOAM Turbulence Model verification

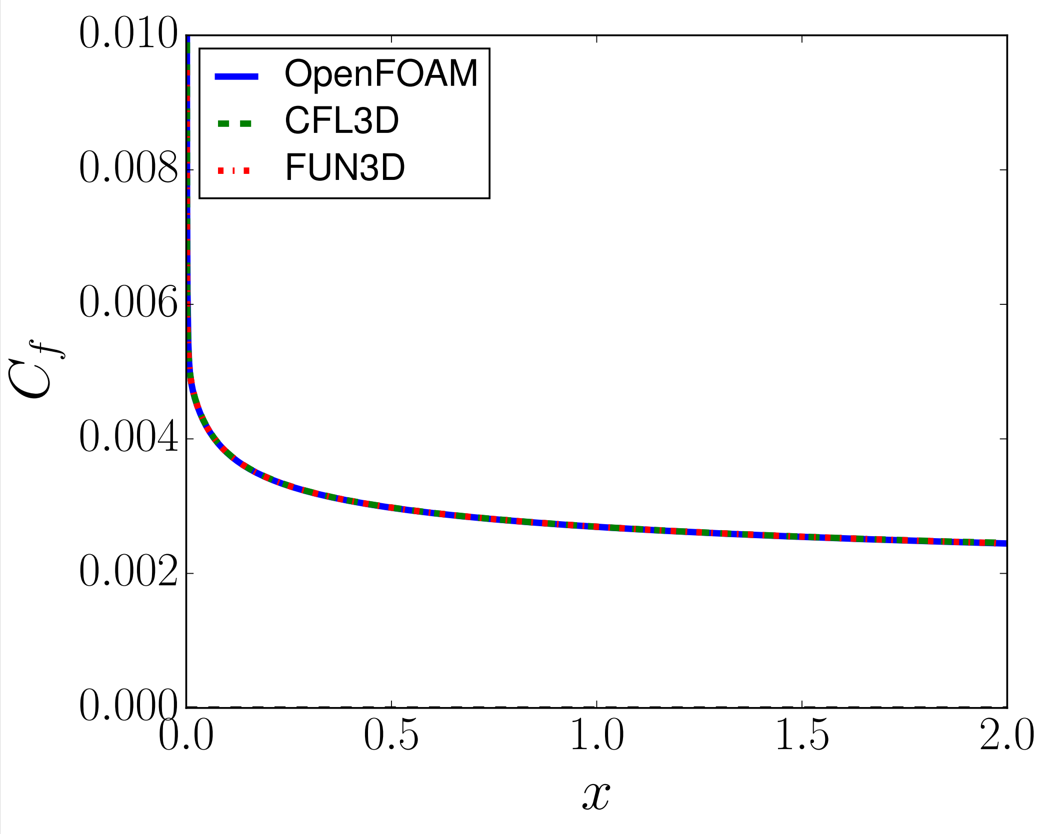
- Spalart-Allmaras – as per original reference

- Required a number of changes to bring it in-line with the original publication – as per the NASA TMR site

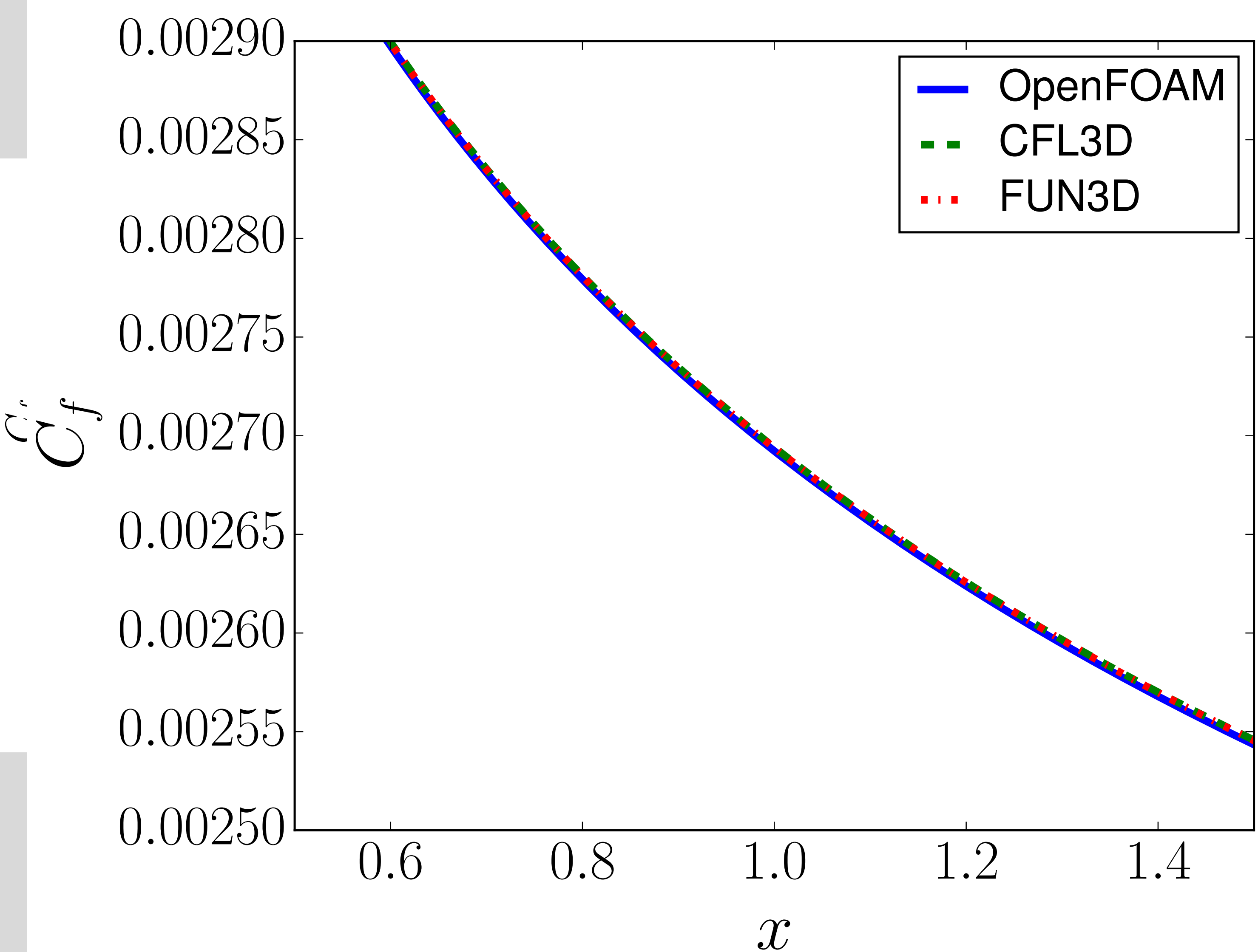
Important to establish that the SA/SST are correctly implemented in OpenFOAM and of the same form as other codes

- ZPG Flat Plate
- NACA0012
- Bump in a channel (not shown here)
- 2D hump (not shown here)

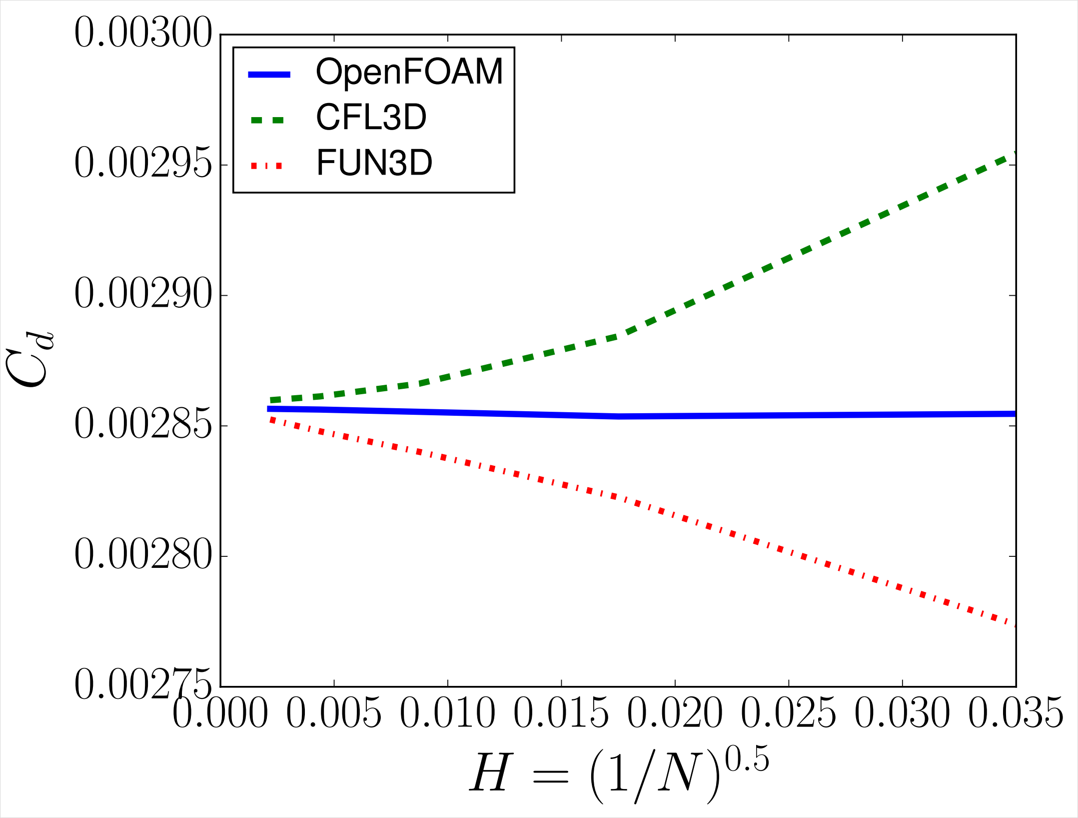
Flat Plate – compressible - SA



- 545x385 mesh from NASA TMR
- M=0.2
- Re=5x10^6
- Compressible steady-state
- Second order upwind for momentum + 1st order upwind for turbulence (to match CFL3D)
- Inlet: Turbulent viscosity ratio = 3

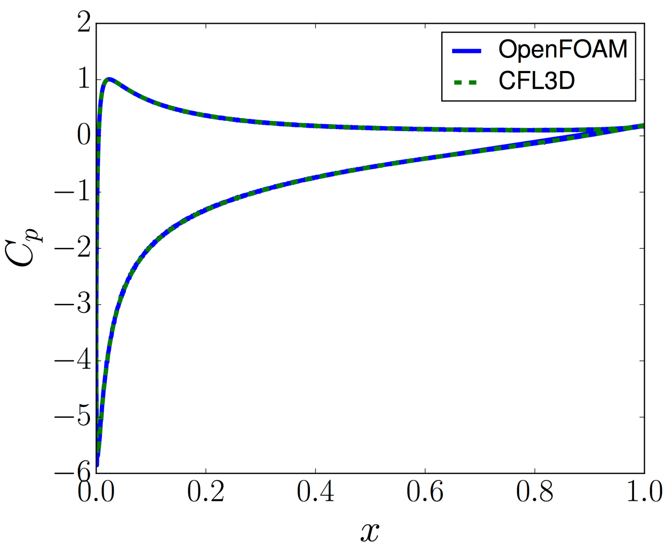

OpenFOAM
CFL3D
FUN3D
C_f
x
0.00290
0.00285
0.00280
0.00275
0.00270
0.00265
0.00260
0.00255
0.00250
0.6
0.8
1.0
1.2
1.4

Flat Plate – compressible - SA



NACA0012 – 10deg - SA

	Cl	Cd
OpenFOAM	1.08986	0.0121
CFL3D	1.0909	0.0123
FUN3D	1.0983	
NTS	1.0891	



- 897x257 mesh from NASA TMR
- M=0.15
- Re=6x10^6
- Compressible steady-state
- Second order upwind for momentum + 1st order upwind for turbulence (to match CFL3D)
- Inlet: Turbulent viscosity ratio = 3

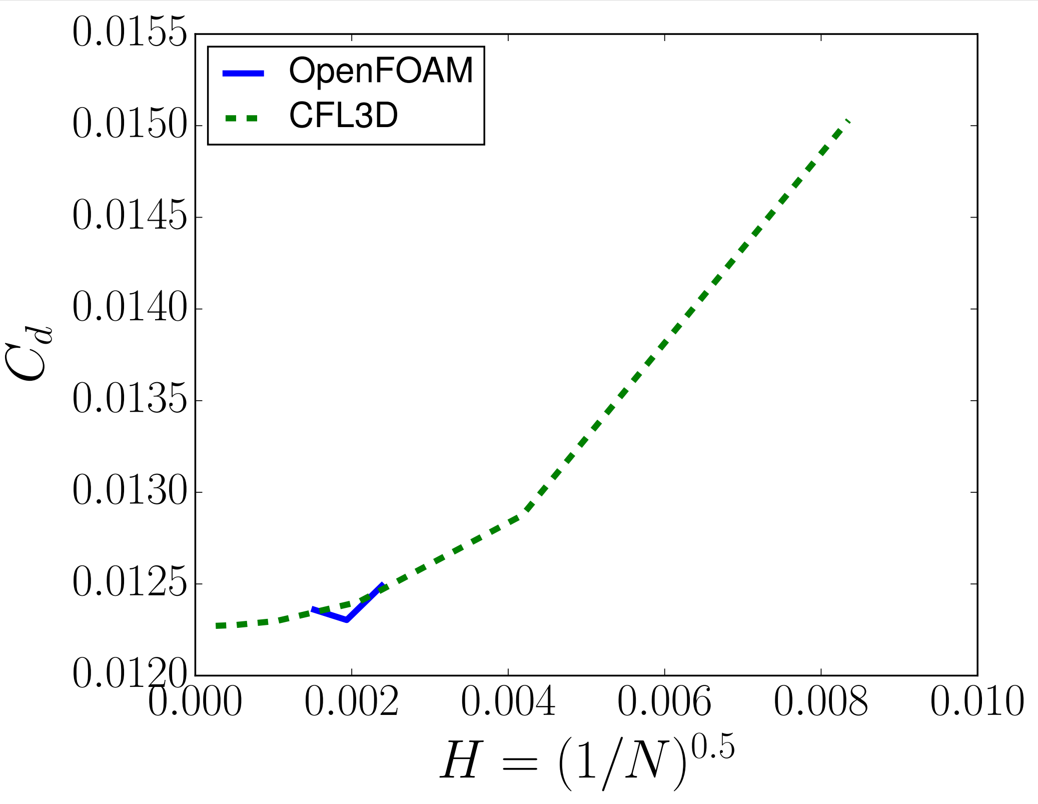
Case 1 – NACA0012

- We have worked to validate the code against the NASA Turbulence Modelling Benchmark page

- Three ANSA generated unstructured meshes

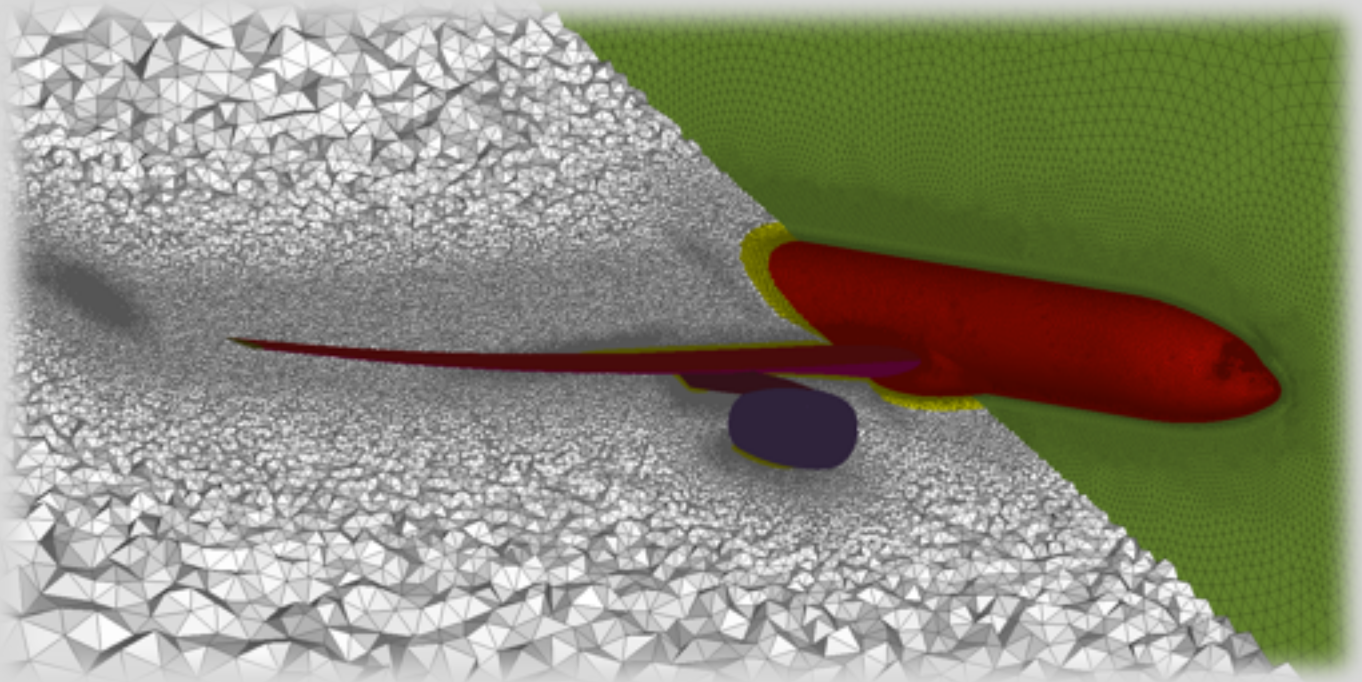
- Improved convergence

- Blimp on medium-grid needs to be investigated



Case 2 – CRM Model

ANSA models for DPW6



BETA CAE Systems S.A.
Kato Scholari, Thessaloniki,
GR- 57500, Epanomi,
Greece
Tel : +30-392-021420
Fax : +30-392-021417
Email : ansa@beta-cae.com
URL : http://www.beta-cae.com

Project overview

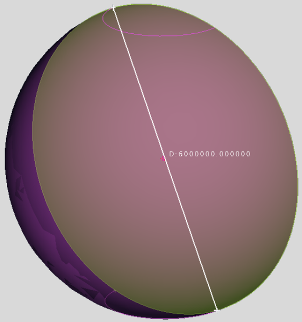
- All pre-processing work was performed in ANSA v16.2.0

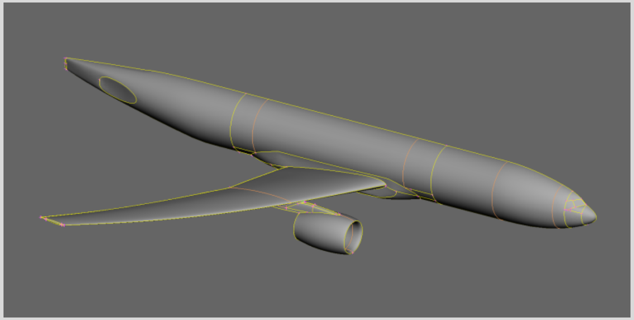
- The DP6_CRM_wbnpt_ih+0_v09_2016-01-28_ae2.75deg_cf model was used as input in STEP format

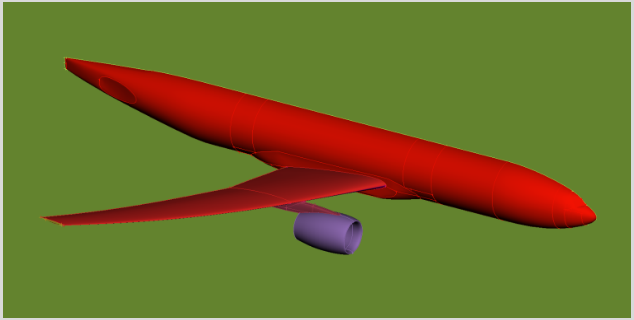
- Model units were converted to mm (CRM model length = 62748 mm)

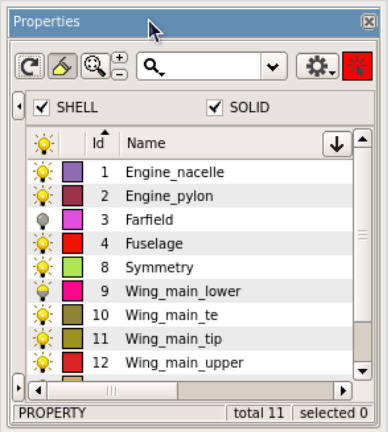
- A spherical domain was created with a diameter of $6x10^6$ mm

- Model was separated in twelve different properties to facilitate meshing and post-processing



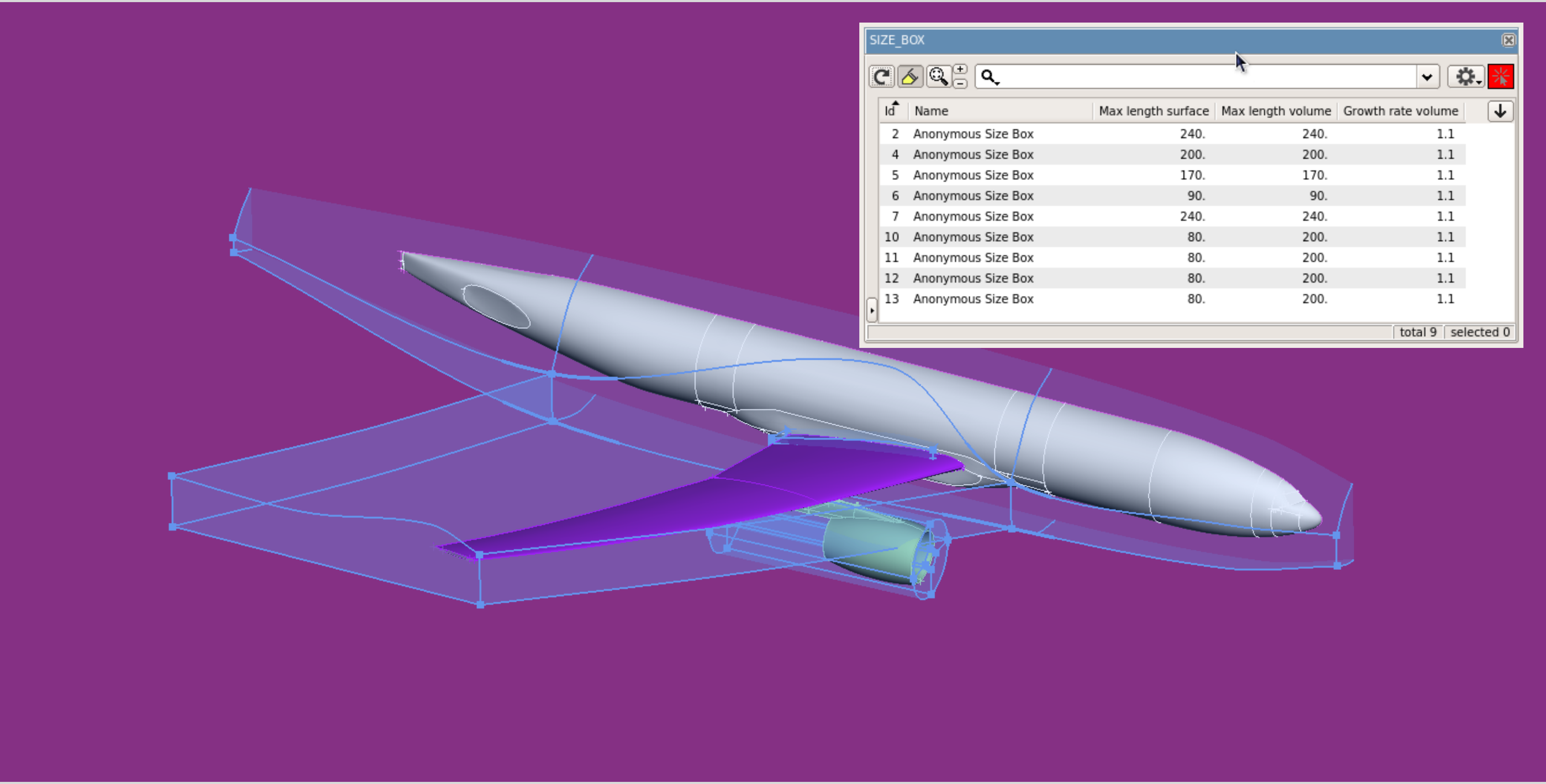






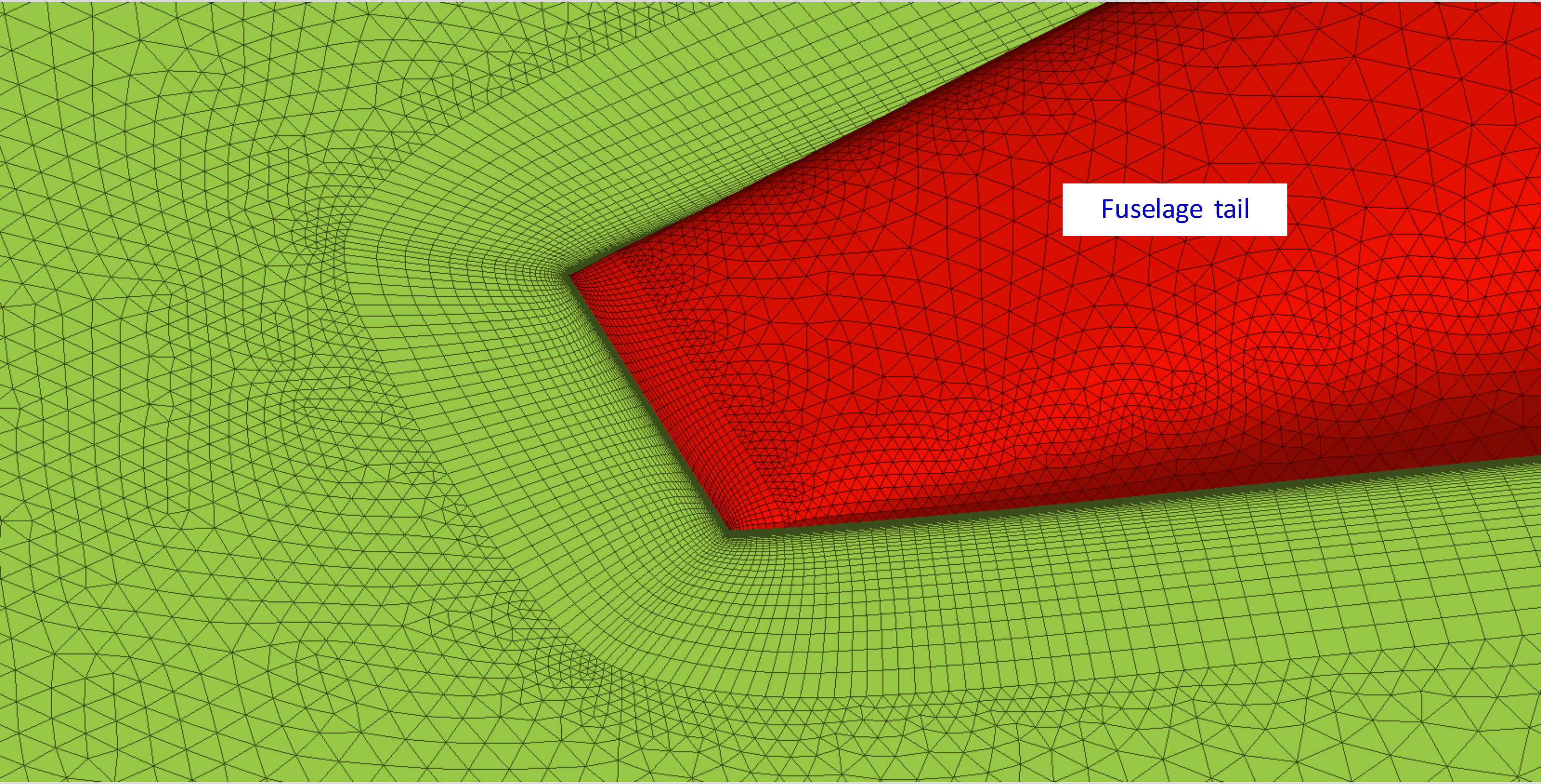
Batch meshing

Batch meshing tool was used to automate the three meshing steps: surface, layers and volume



Surface mesh – coarse model

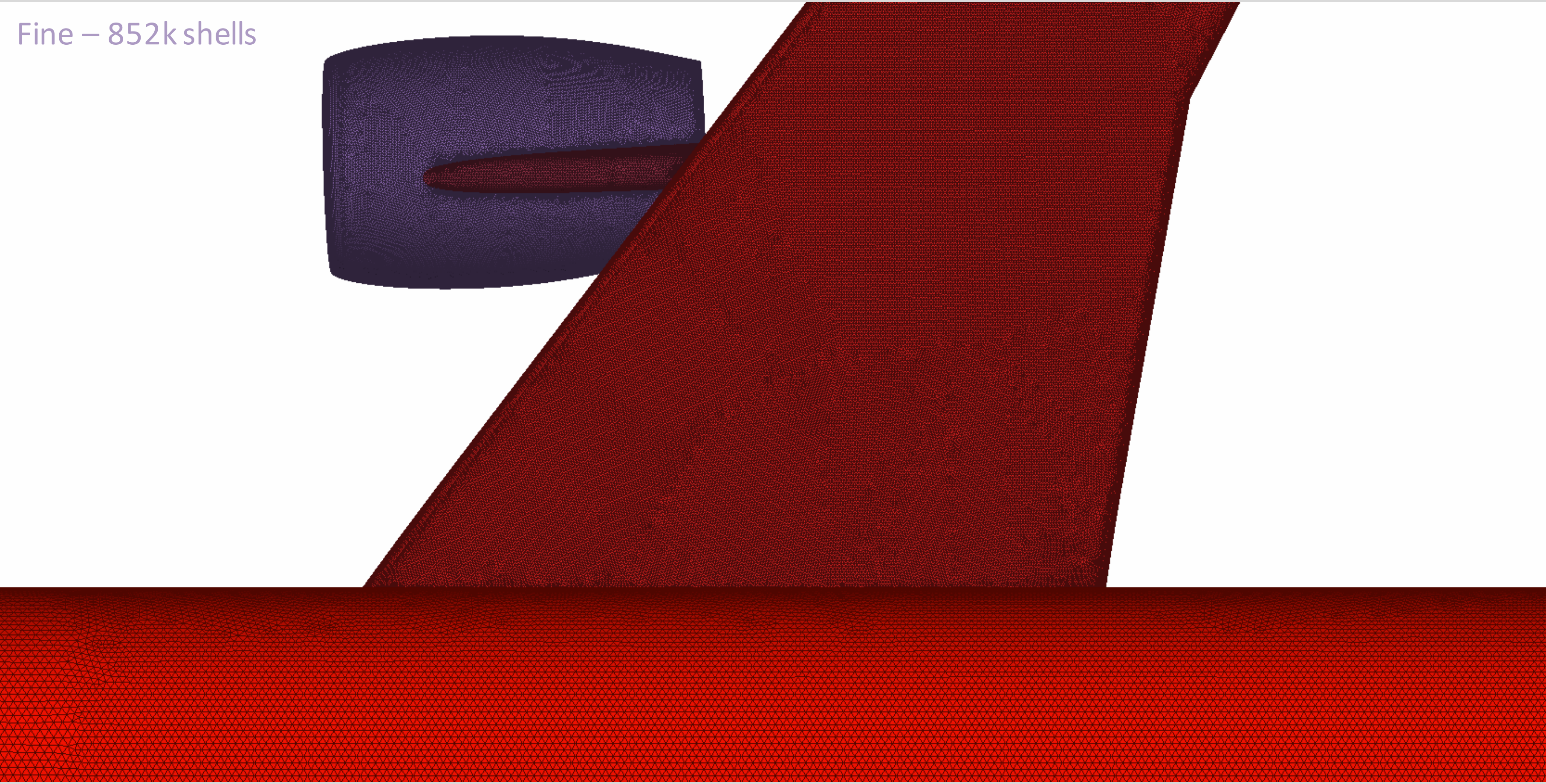
Variable tria mesh with anisotropic map quad patterns for better capturing of details without increasing mesh size, and for improved mesh quality for Open FOAM



Surface mesh – Mesh refinement study

Using batch mesh tool 3 mesh refinement models were automatically created for both variants

Fine – 852k shells

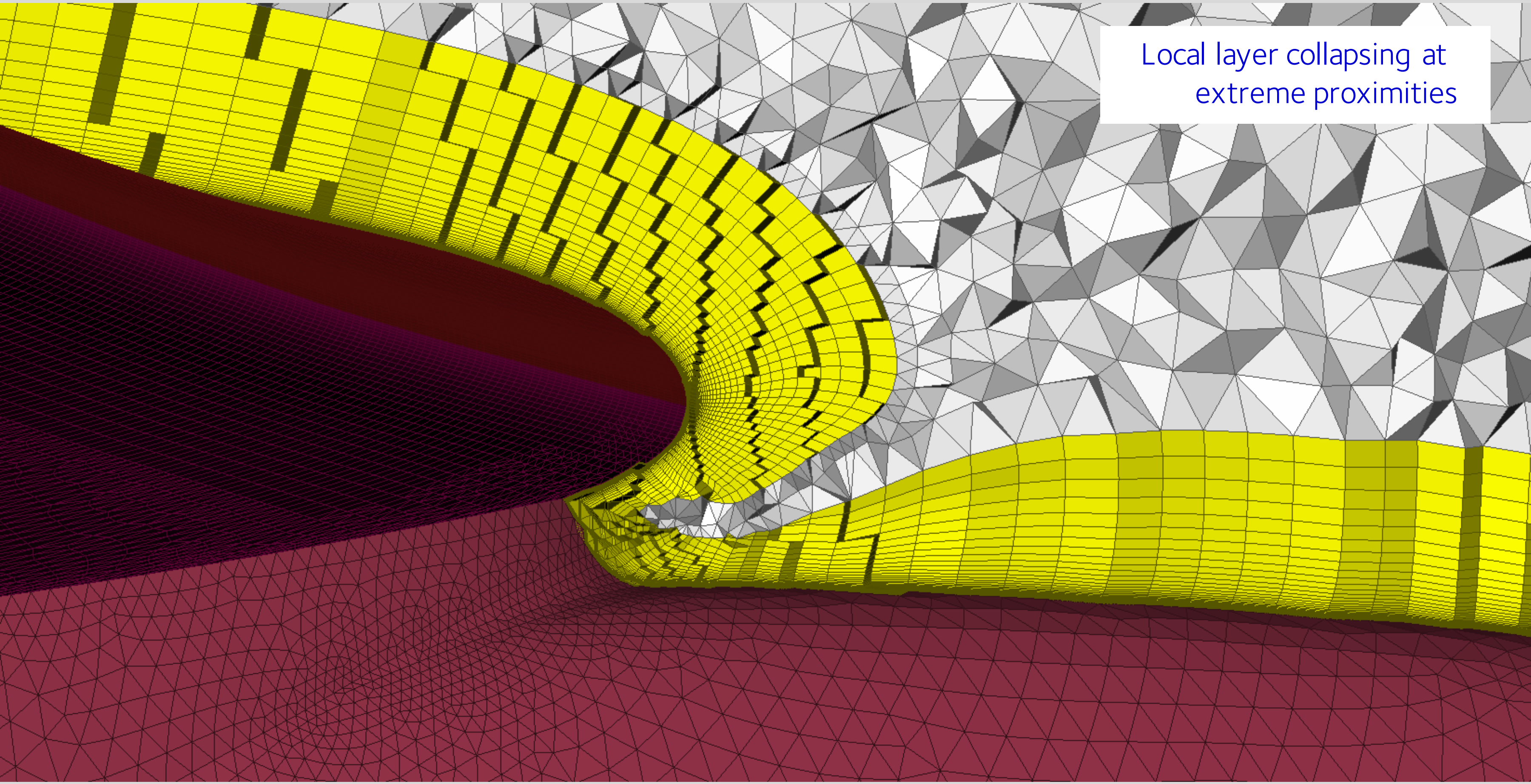


Based on initial CFD runs the path of the tip vortex was meshed with a pure hexa mesh

Tip vortex treatment

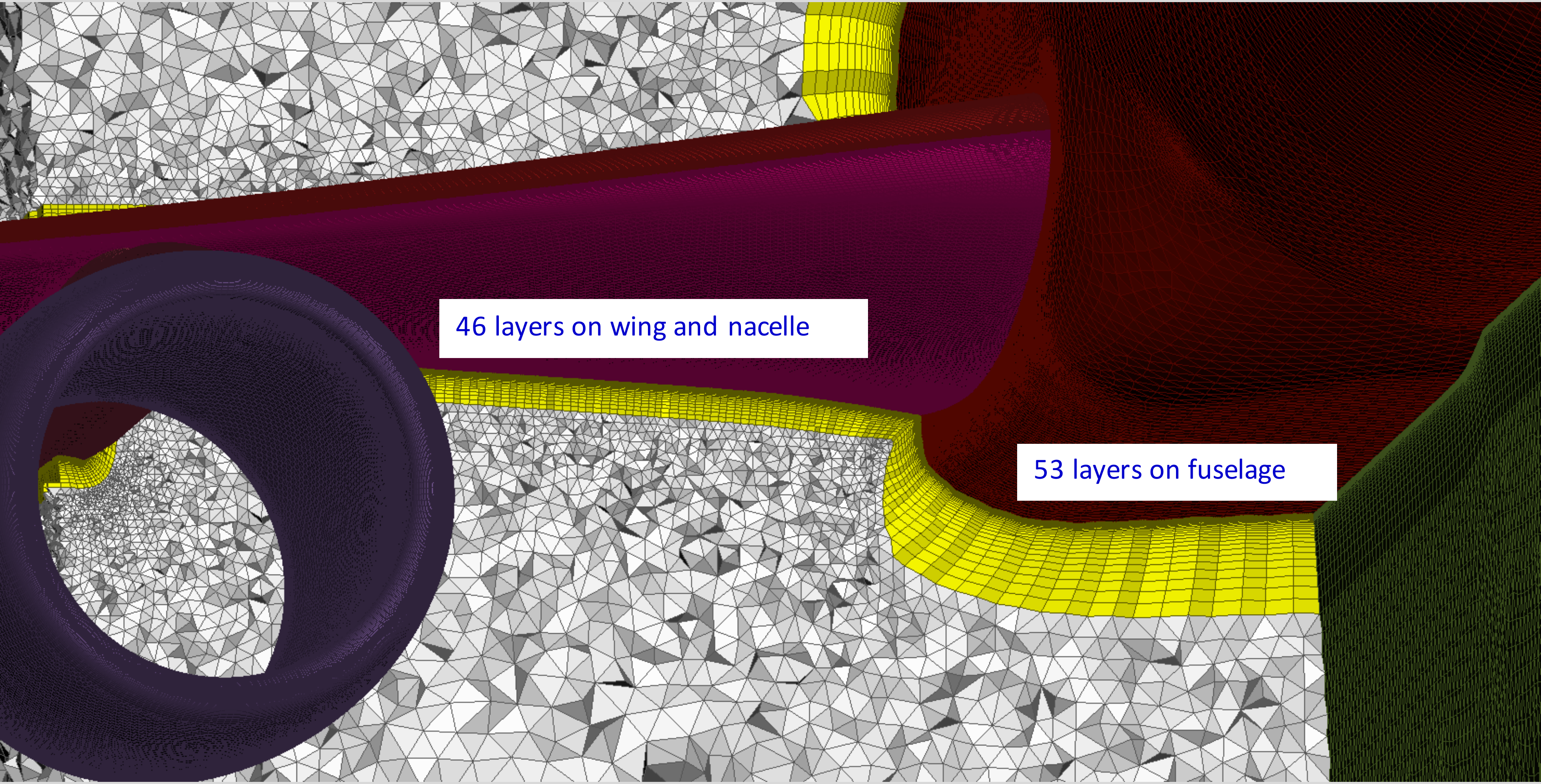
First layer height=0.0326mm Growth Rate=1.17

Volume mesh – Coarse model



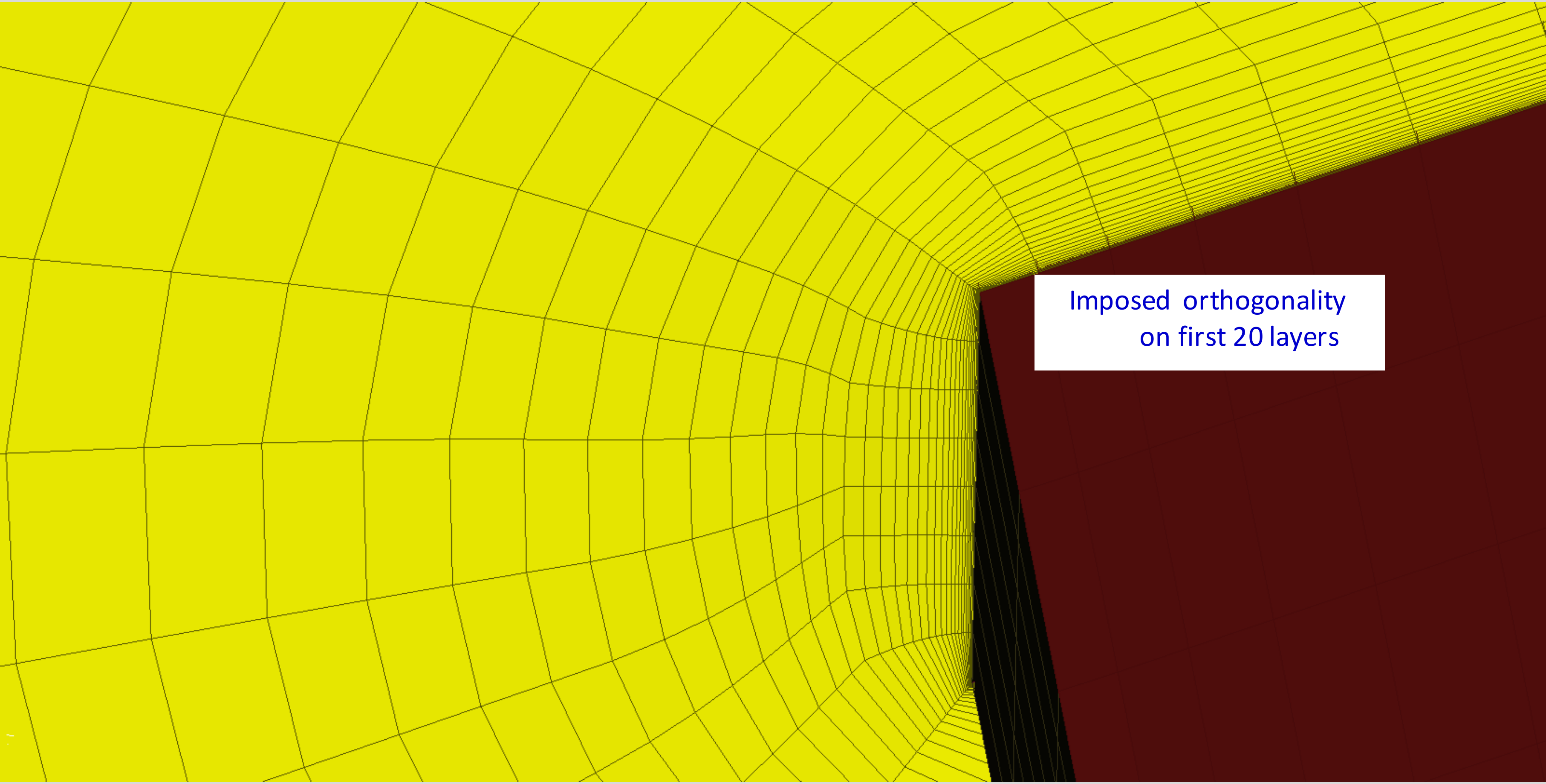
Volume mesh – Different number of layers per regions

First layer height=0.0326mm Growth Rate=1.17



Volume mesh – Layers at trailing edge

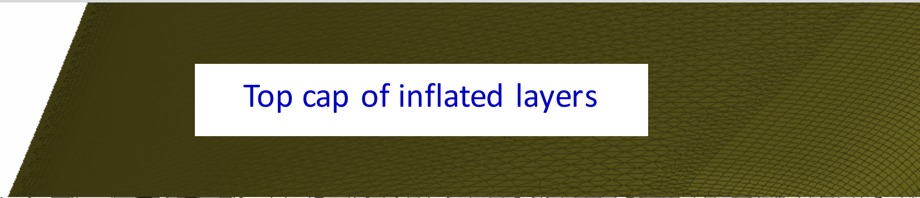
Trailing edge has 8 to 10 rows of quads depending on the refinement level

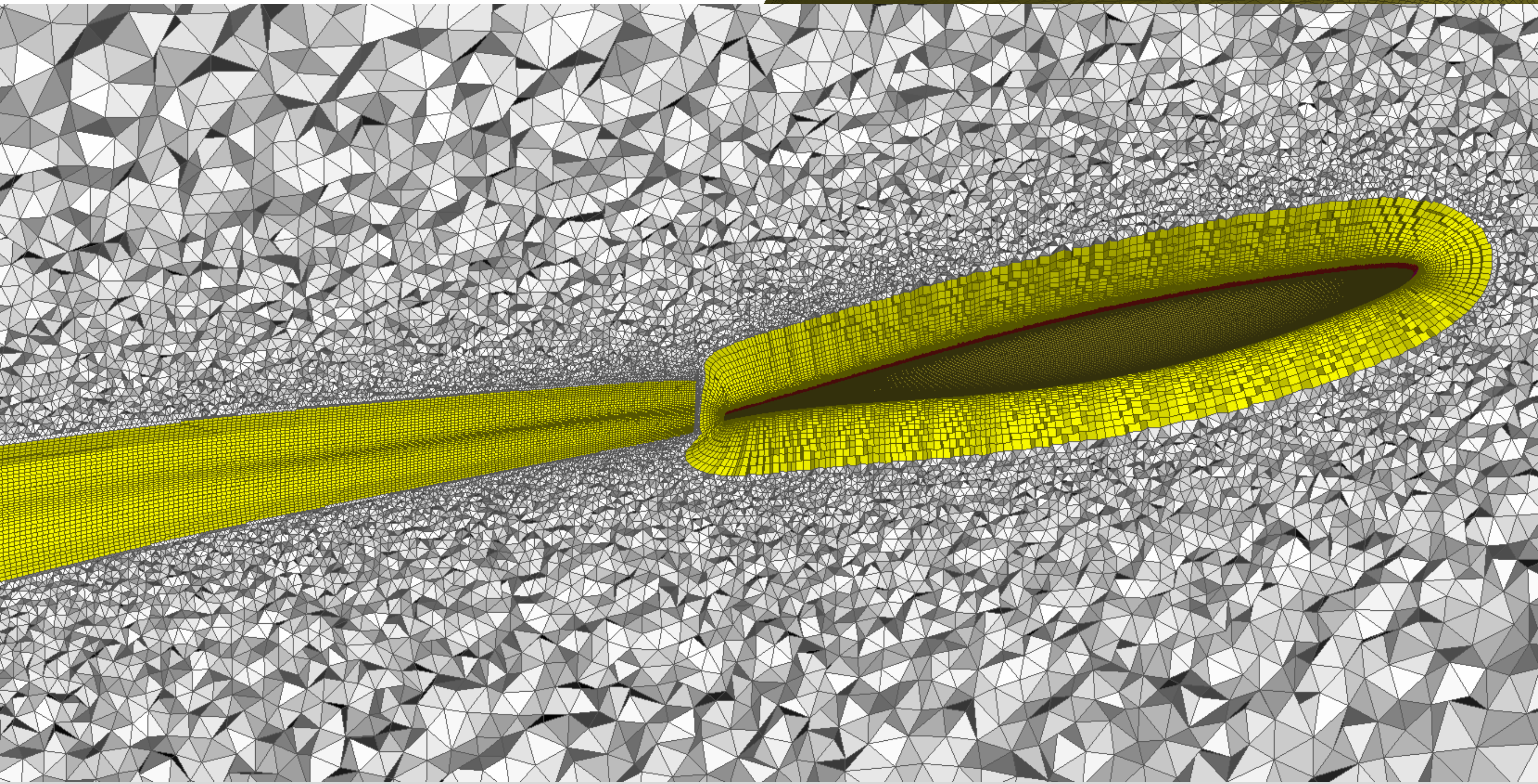


UNIVERSITY OF OXFORD

Volume mesh – Tip vortex area

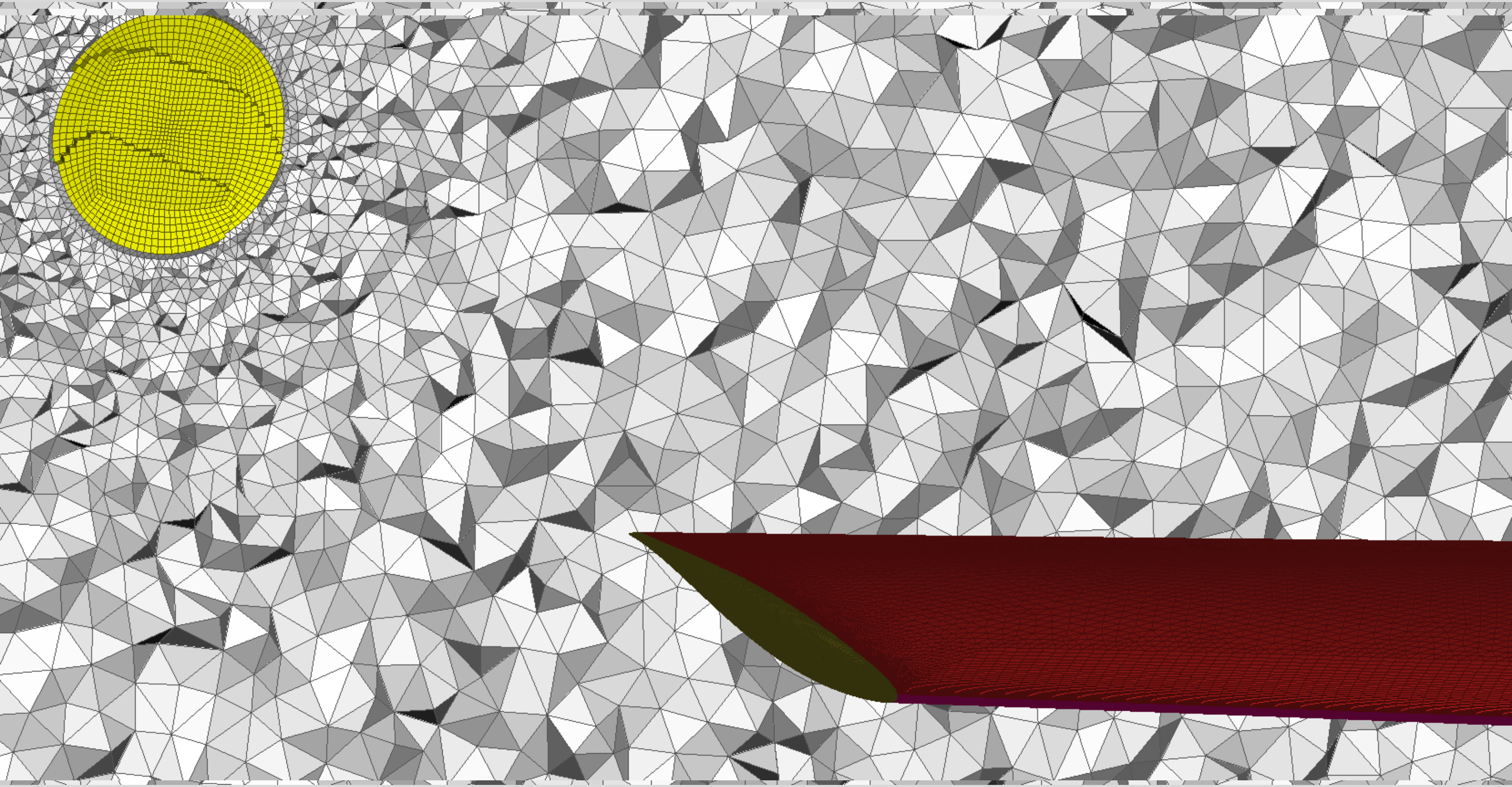
First layer height=0.0326mm Growth Rate=1.17





Volume mesh – Tip vortex area

Using batch mesh tool 3 mesh refinement models were created for both variants



Millions of cells

	WB	WBNP	Δy_1 (mm)
Coarse	38	45	0.0326
Medium	55	66	0.0284
Fine	83	99	0.0247

- Both WB and WBNP models follow the 1.5 factor for number of elements
- WBNP models are within the recommended size ranges
- WB models exceed the recommended size ranges by approximately 20%

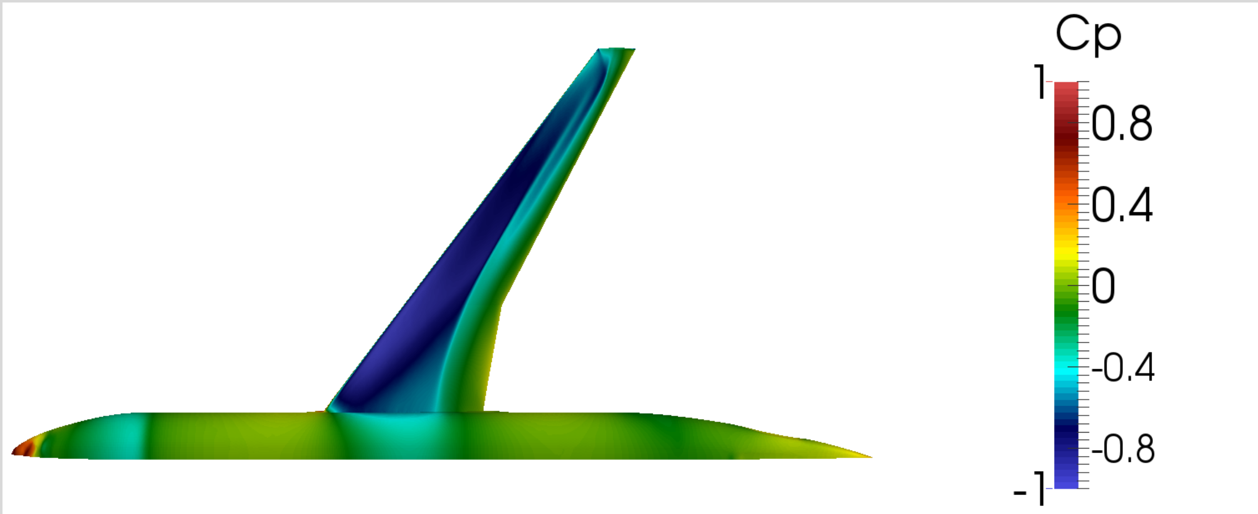
Case Setup

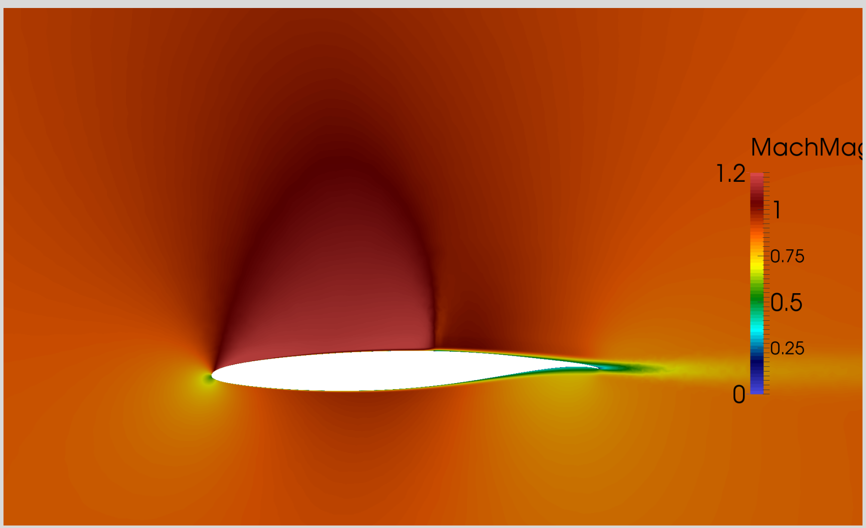
- Pressure based compressible local-time-stepping (LTS) scheme (rhoLTSPimpleFoam)
- VanAlbada shock capturing scheme for momentum and turbulence.
- Spalart-Allmaras model
- $M=0.85$, $Re=5\times10^6$, $T=310.9K$
- Adjusted AoA to reach $Cl=0.500$

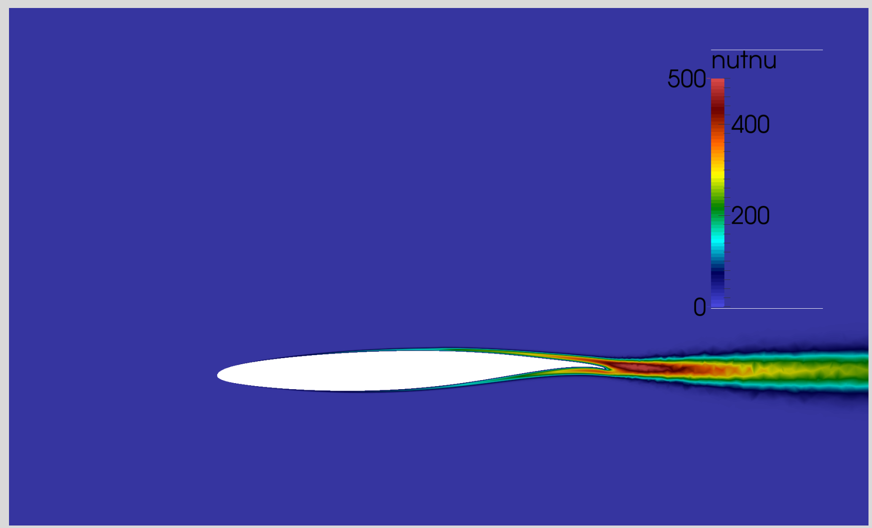
Preliminary results

Wing-Body – Coarse**

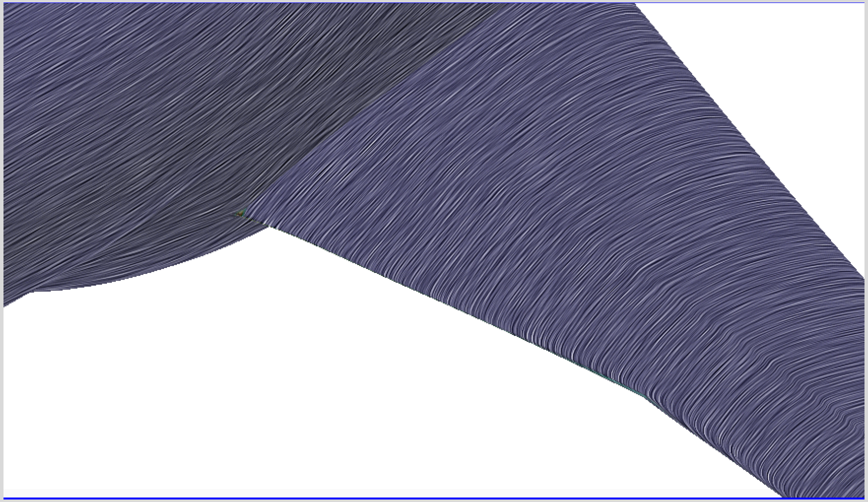
**Not fully converged



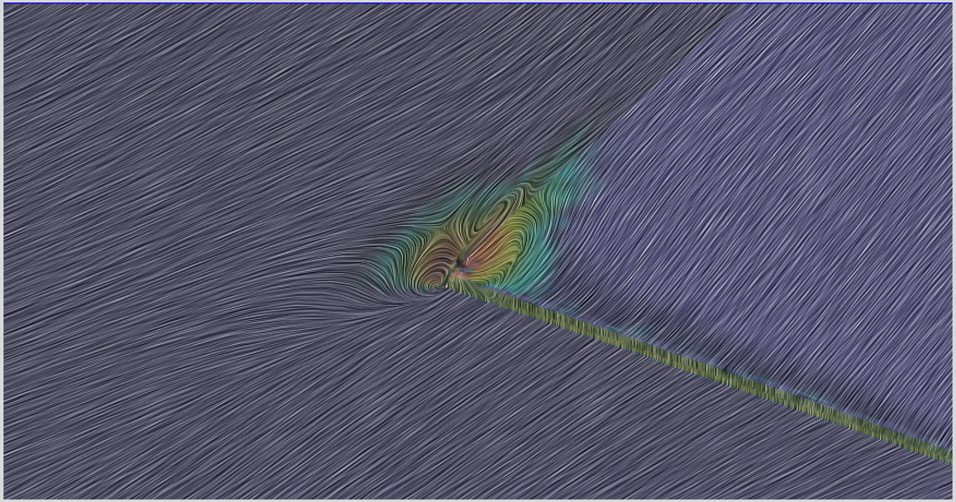




Wing-Body – Coarse**



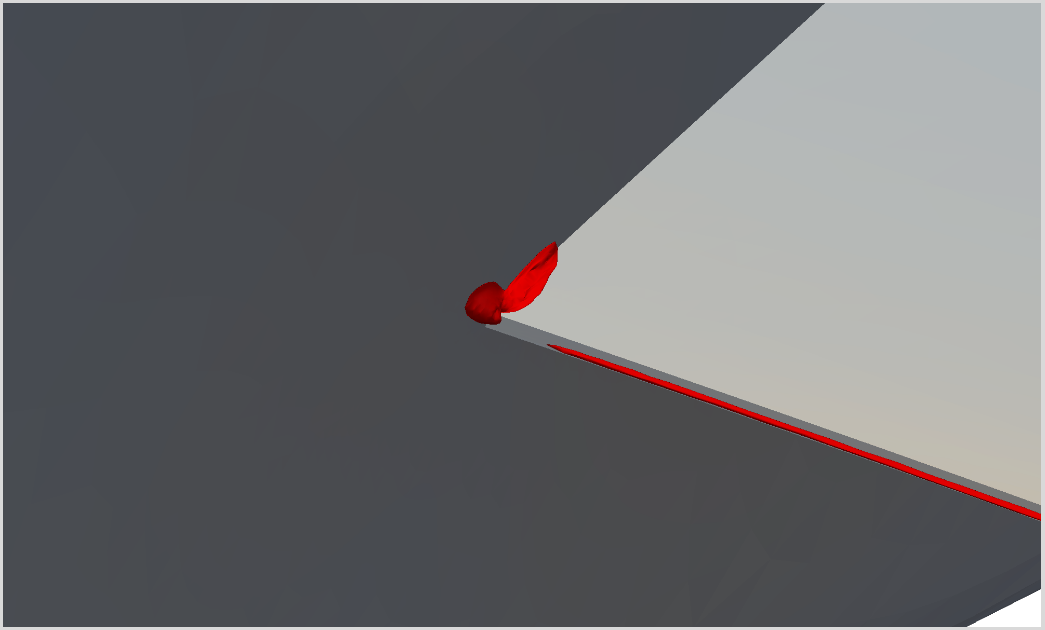
Rest of the plane is clean with the only expected separation in the junction region



SOB separation is present although medium/fine meshes are needed to assess its trend with mesh refinement

**Not fully converged

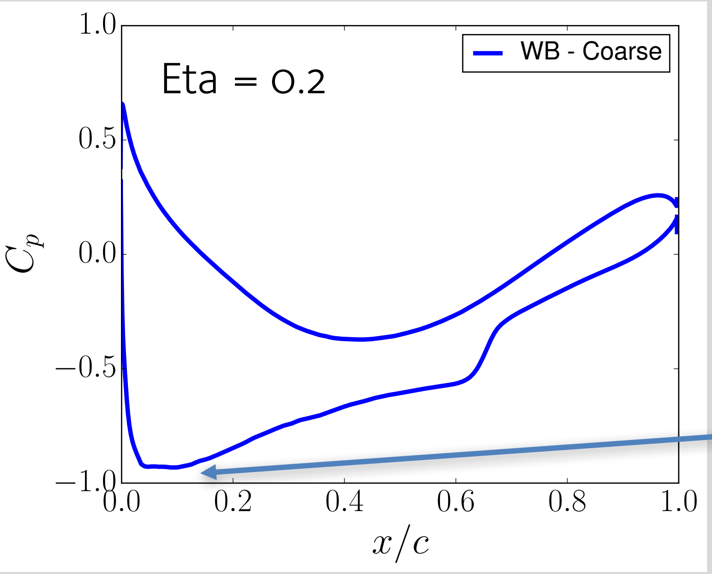
Wing-Body – Coarse**



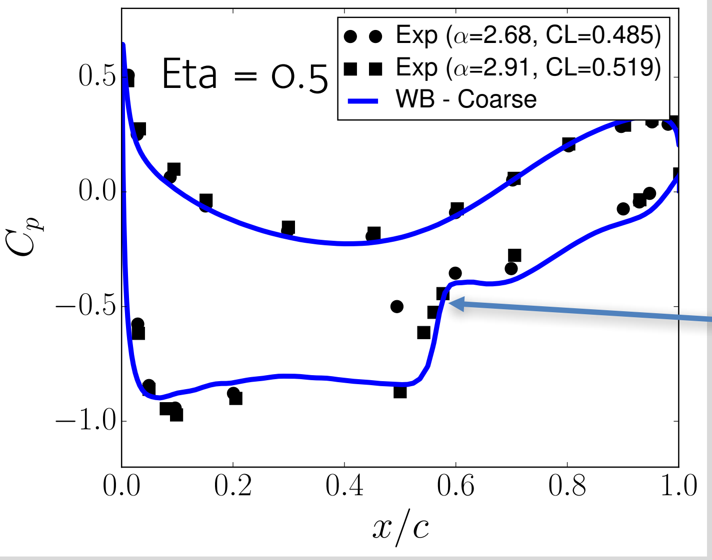
Showing Iso-contour of Ux=-10. Wing-body separation as expected.

Slight separation from trailing edge, may be a mesh refinement issue

**Not fully converged



Signs of requiring better convergence but largely smooth



Reasonably sharp shocks (not too dissipated)

**Not fully converged

- Simulations on the remaining meshes are underway – no automatic Cl driver takes time to hit 0.5000
- Currently investigating turbulence model stability – limiting of SA S_tilda term + SA-NEG for coarse grid
- Robustness is still an issue – actively looking into implementing an implicit-density based compressible solver in OpenFOAM
- Learning experience as first DPW participation but useful experience to assess the current capabilities of OpenFOAM

Conclusions

Successful V&V

Coarse-Fine mesh generation complete

Still assessing optimum solver settings for convergence + turbulence model stability

Initial results are in the right direction but need further work

Thank you

Dr Neil Ashton
neil.ashton@oerc.ox.ac.uk