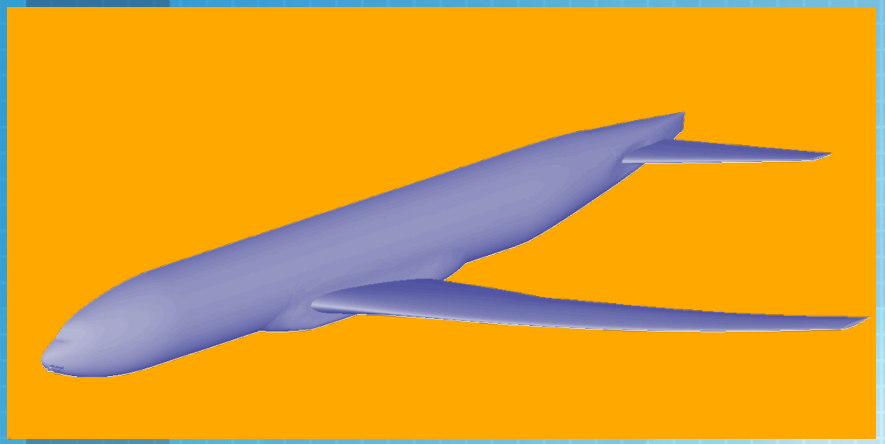
Presentation of Results for DPW4

4th AIAA CFD Drag Prediction Workshop
20th – 21st June 2009, San Antonio, TX, USA



Basant Kumar Gupta
bkgupta@zeusnumerix.com

Venkatesh G.
venkatesh@zeusnumerix.com

ZEUSNUMERIX
Diverse Needs, Converged Solution

Contents

- Background
- Grid Generation Details
- Simulation Setup
- Description of Results
- Future Work

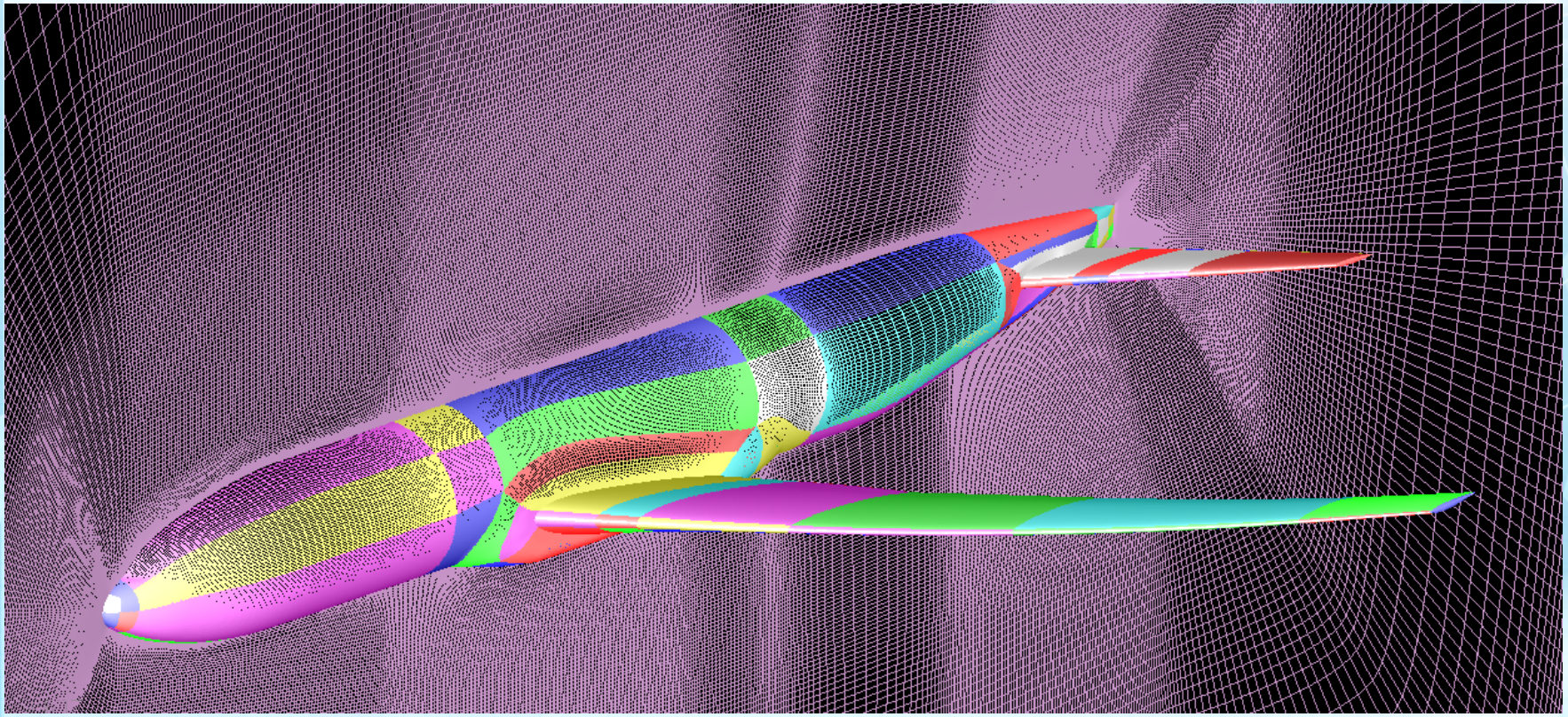
Background

- Zeus Numerix was incorporated in August 2004, India
- Develops proprietary CFD simulation packages covering mesh generation, flow solver, visualization and parallelization modules
  - GridZ – Structured Multi-block Mesh Generator
  - FlowZ – Compressible NS solver; Pressure correction based solver for Incompressible flow
  - ViewZ – Scientific Visualizer
- All modules connected through CGNS File. Compatible with other third party packages through this file format
- We deliver solutions and customized application specific design softwares (CFD, FEA) to our clients

Mesh Generation

- Mesh is generated on Body Wing Tail (iH = 0) configuration
- Block-structured mesh generated using GridZ.
- Initial mesh took approximately 10 days
- O grid generated around the body, wing and tail, which is much more efficient than C-H topology
- Requirement of highly clustered near wall cells near wing & tail tip posed mesh quality issues
- 3 levels of grid were generated using same block topology
- Number of blocks in each are 326

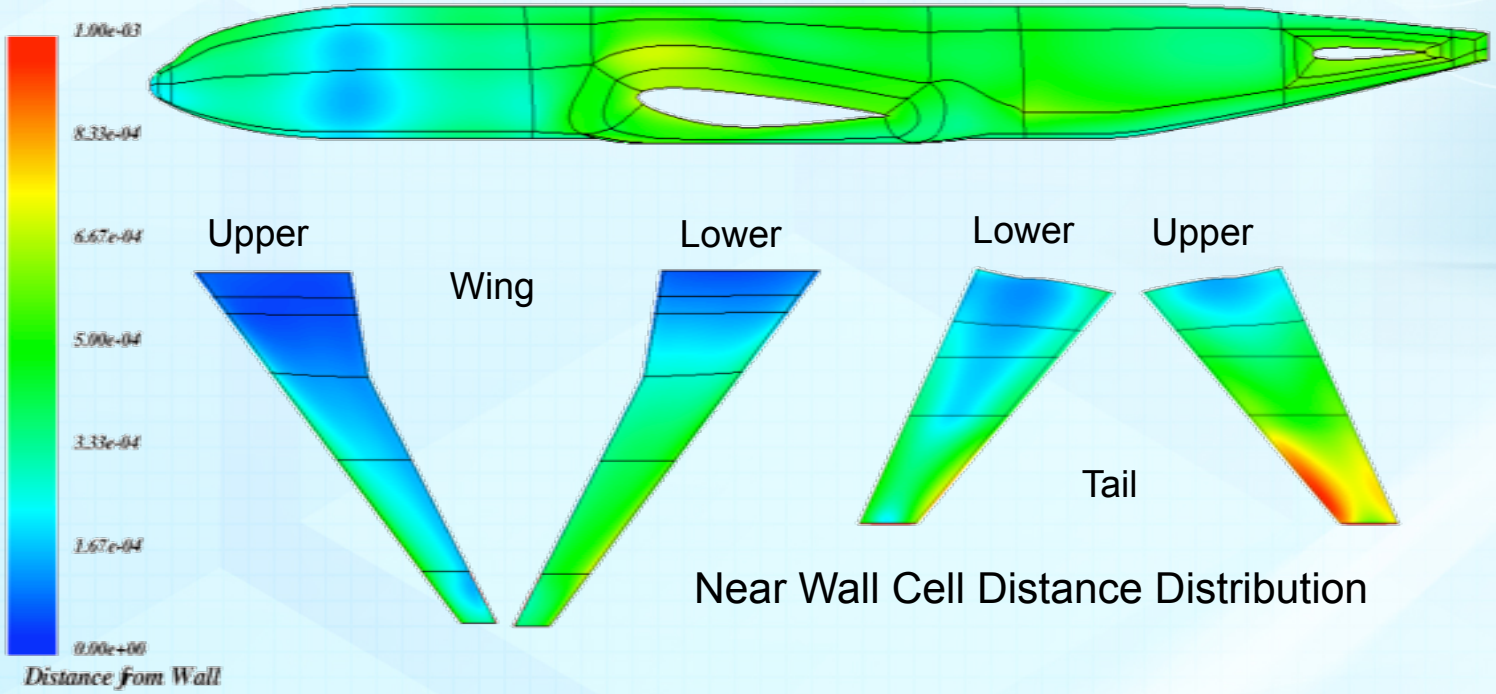
Mesh Generation



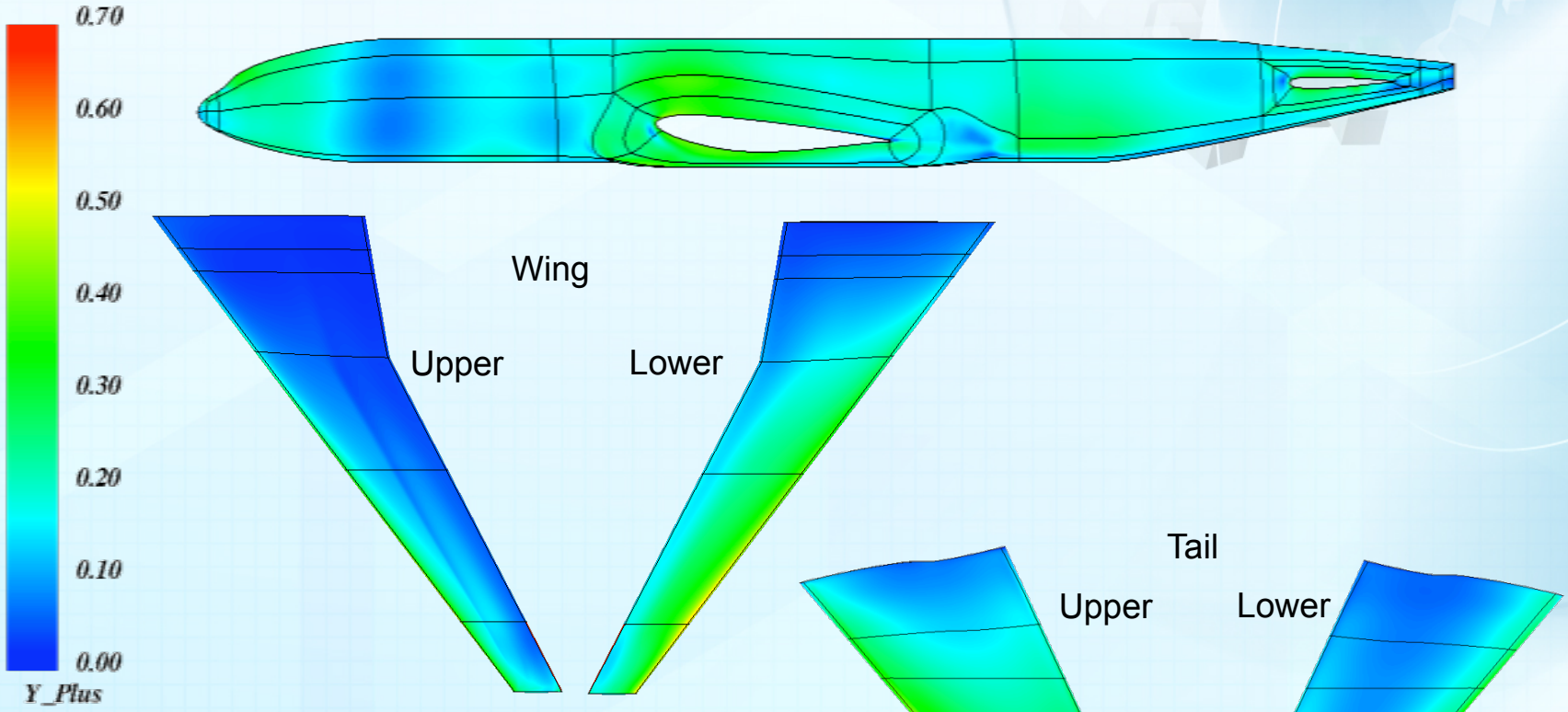
The distribution of blocks on the surface and the volume mesh on symmetry plane

Mesh Generation

Mesh	Volume Cells	Surface Mesh		
		Fuselage	Wing	Tail
Coarse	4,369,091	28,849	18,941	8,338
Medium	15,365,720	80,672	63,800	32,120
Fine	42,520,772	183,302	136,798	67,640



Computational Grids, y^+



Mesh satisfies wall clustering for low Re turbulence model

Method

- **FlowZ** – Developed at Zeus Numerix & IIT Bombay
- Finite volume formulation
- Multi-block with 1-1 mapping
- HLLC scheme
- Spalart Allmaras turbulence model, full N-S
- Explicit time marching
- Mesh sequencing for acceleration
- Runs on parallel machines through MPI

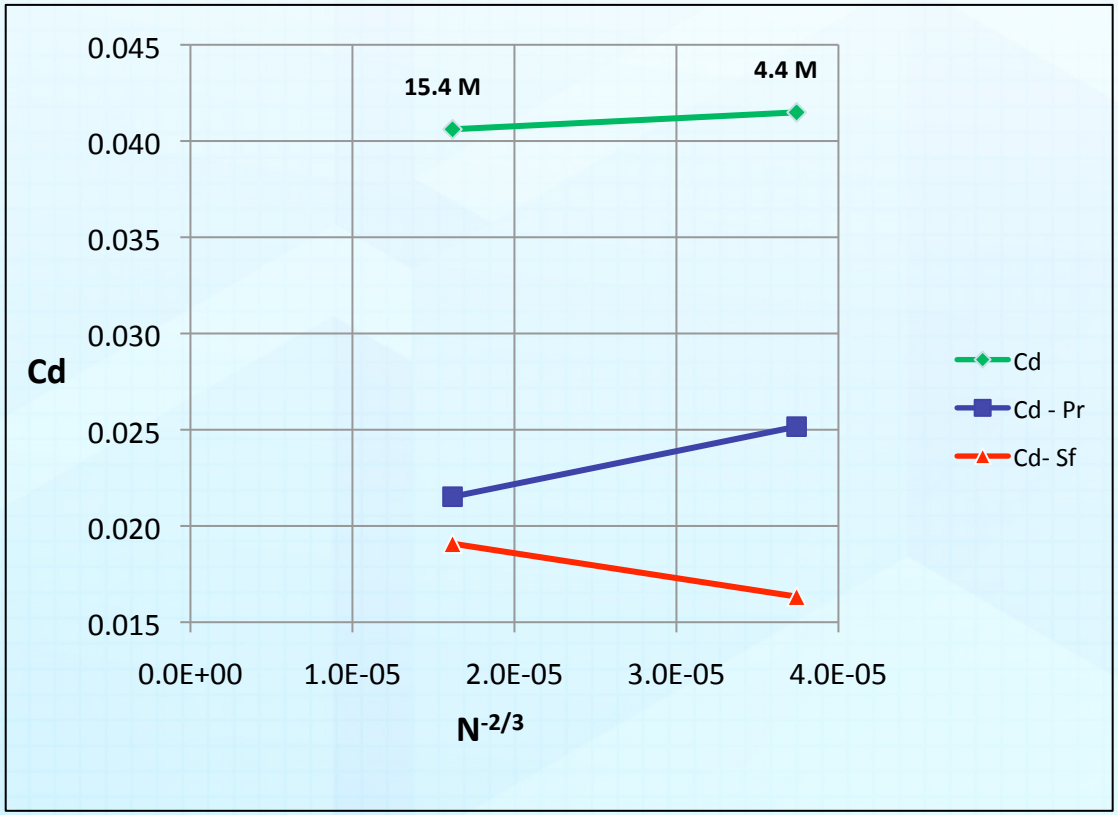
Computational Platform

- Parallel Processing Done on **CDAC**'s gg-blr (Grid Computing Resource at Bangalore) (http://www.cdac.in)
- Operating system: Redhat Enterprise Version
- 3.16 GHz Xeon processor

- NASA CRM medium grid run on 96 processors (12 nodes)
- Convergence reached after 100,000 explicit iterations
- Roughly 128 hours of wall clock time needed
- Would have been faster if mesh sequencing was utilized

Grid Convergence

NASA CRM, C_L = 0.5, Re = 5 Million, iH = 0°

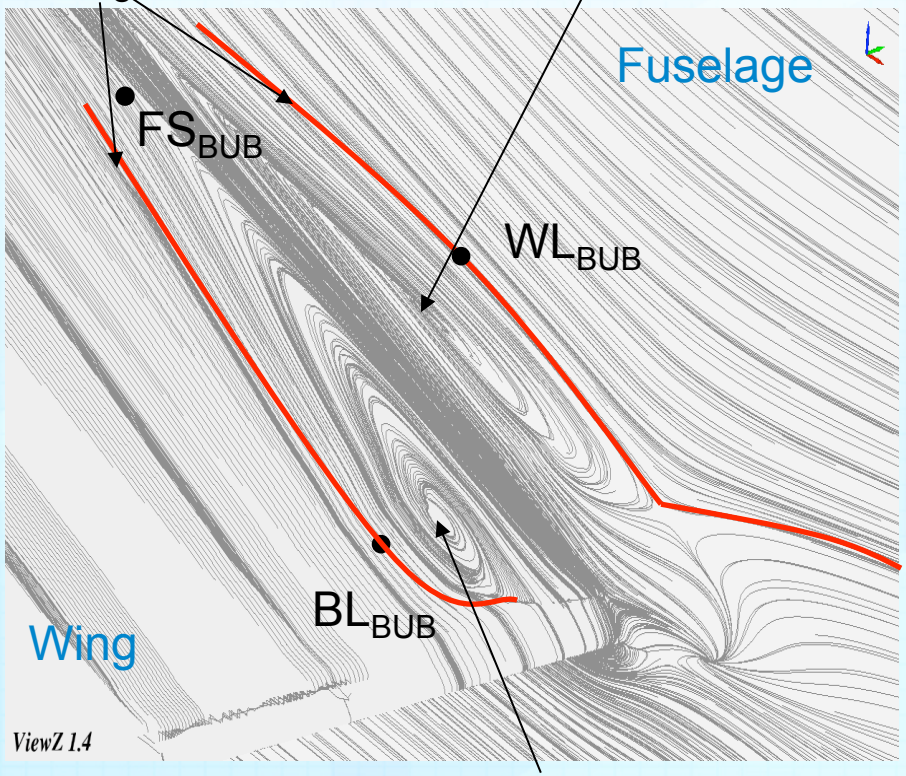


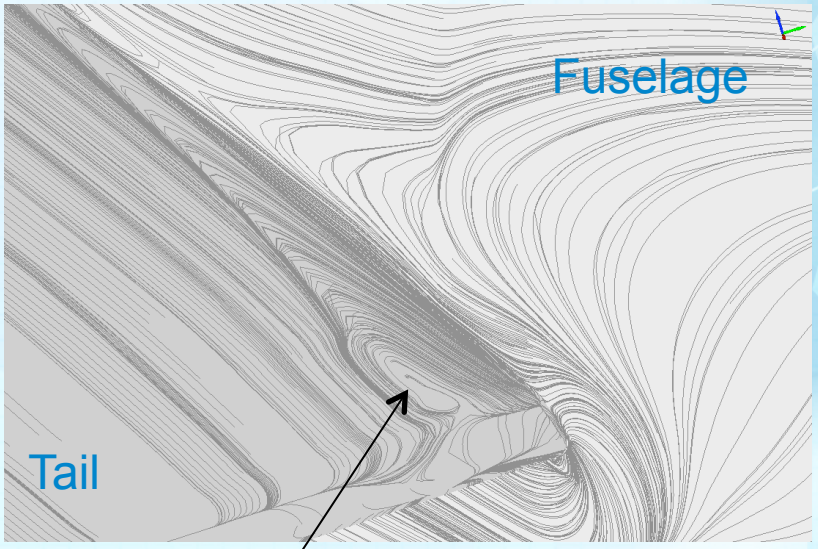
	Coarse	Medium
AoA	3.1°	3.0°
C_l	0.498	0.496
Cd	0.0415	0.0406
Cm	-2.376	-2.386

Difficult to judge quality of results in absence of any reference

PS: Simulations on fine mesh is in progress

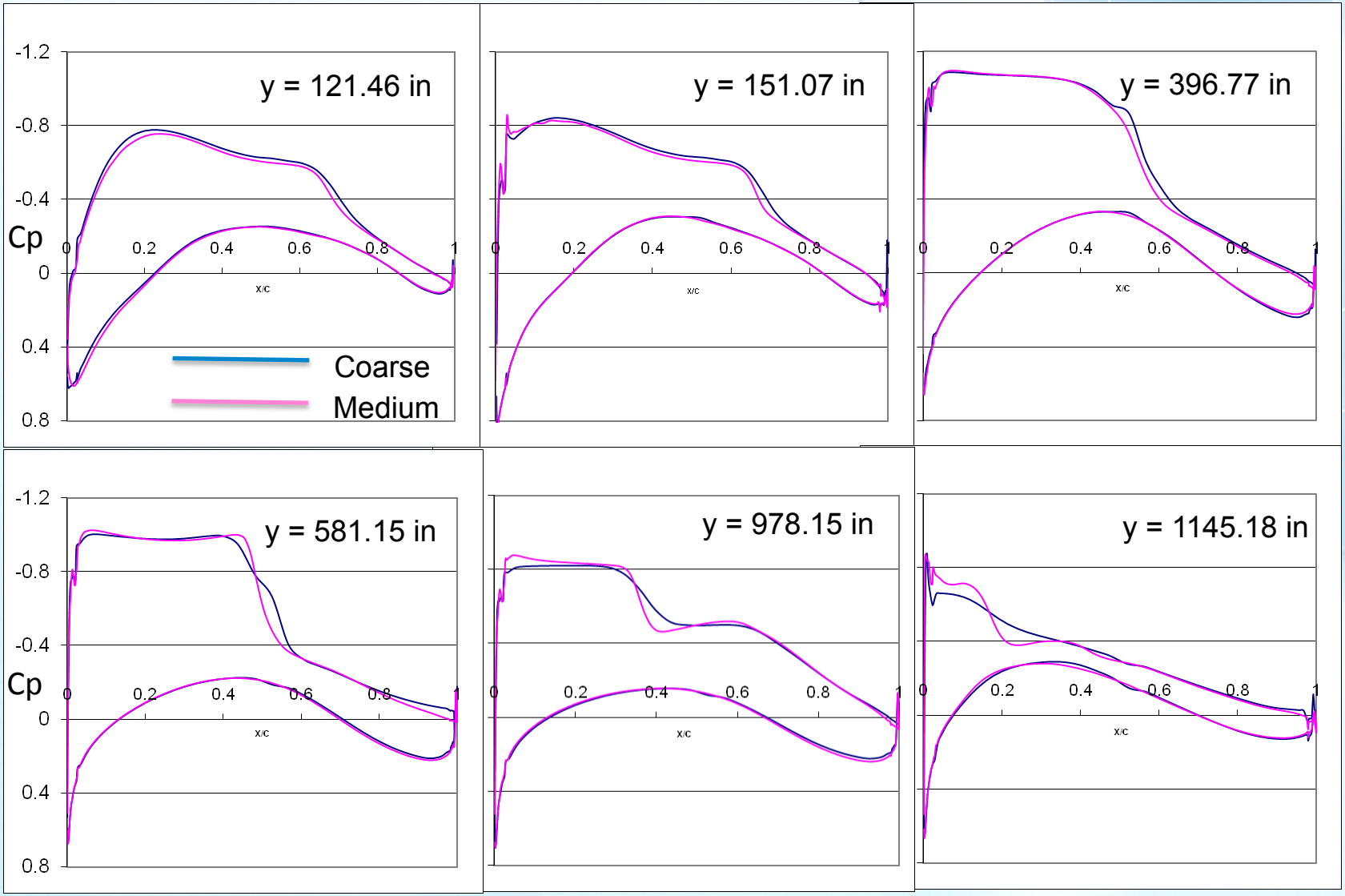
Surface Streamline



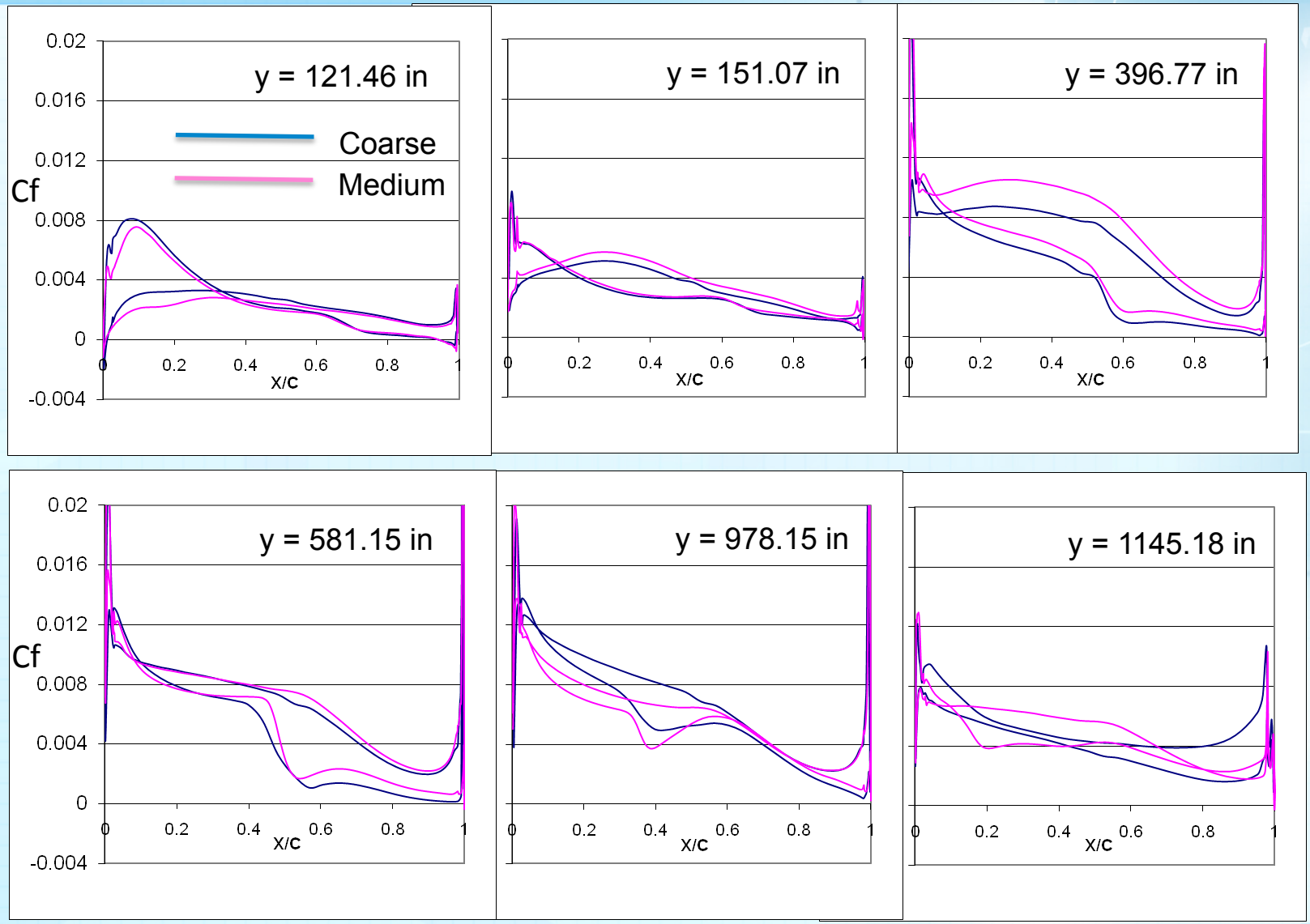


Except for near fuselage region, separation absent at rest of trailing edge, both wing & tail

Grid Density effect on Cp, Wing Sections



Grid Density effect on Cf, Wing Sections



Discussion & Future Work

- C_L converged to 0.5 +/- 0.005
- In absence on fine mesh results, not sure if medium mesh achieves asymptotic convergence
- Keen to get a reference based on results from other participants

- Deriving experience from earlier DPW, mesh quality / topology may be required to look into
- Perform the fine grid computation
- Compute drag polar & cases with deflection to tail
- Explore SST turbulence model

Thank You