HyperFlux Documentation

Release 0.1

hyperflux

May 18, 2015

Contents:

CHAPTER 1

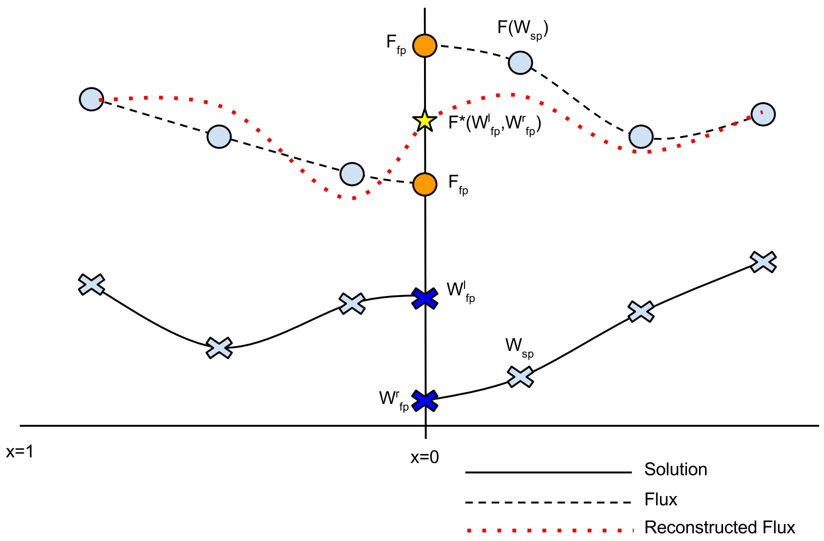
Introduction

Computational Fluid Dynamics (CFD) is used in a wide range of engineering sectors for the analysis and design of products – from aircraft to racing cars and jet turbines. Current CFD technology is lacking in accuracy and efficiency for unsteady flow problems (including acoustics) and for resolving shed vortices and wakes. This is a real problem for industry: expensive physical prototypes are still required since computational tools are unreliable for these unsteady flow phenomena. High-order methods (HOMs) provide a potential solution, providing higher fidelity solutions than currently achievable with low-order schemes on unstructured grids. Dr. Peter Vincent and his group at Imperial College are experts in this field - underpinned by an EPSRC Early Career Fellowship and DTA PhD Studentships. The group will work with technical specialists at the Centre for Modelling and Simulation (CFMS) and cloud high performance computing (HPC) specialist Zenotech to create a prototype software base for industrial evaluation. HOMs has the potential to speed up the design cycle, reduce costs and improve products.

Airbus, ARA, BAE Systems, Airbus Group Innovation, Rolls-Royce, DSTL, Arup, Williams F1 and the newly formed UK Aerospace Technology Institute will be non-financial partners / provide evaluation. Each of the industrial primes will contribute a test case to evaluate the prototype software – validating and verifying it against existing data and processes, and providing feedback on the impact on the need for physical prototyping. The KTN will be directly involved to provide additional dissemination. CFMS will lead the integration and dissemination activities from its supercomputer facility at the Bristol and Bath Science Park. The software prototype will embody the latest in high-performance computing, particularly heterogeneous configurations of conventional and many-core processors for speed and energy efficiency. NVIDIA will support the project via its technical team in Bristol. Via remote (cloud) access to its virtual engineering hub, CFMS will make the prototype software available to other sectors (civil engineering, automotive and renewable energy) and support its uptake with local specialists.

Hyper Flux will be a shared UK-based software tool, underpinned by expertise within the UK. This is in line with government strategies for HVM and ICT, and forms a cornerstone for the new UK aerodynamics ATI. This will further establish a center of expertise in the application of the new models to on-ramp new users – particularly SMEs.

Algorithm



F(W_sp)
F_fp
F*(W^l_fp,W^r_fp)
F_fp
W^l_fp
W_sp
W^r_fp
x=1
x=0
Solution
Flux
Reconstructed Flux

Test Cases

3.1 Unit Test Cases

Name	Status	Comment
Taylor Green vortex decay		
Decaying Isotropic Turbulence		
3D Cylinder static		
2D vortex across sliding interface		
3D Cylinder rotating		

3.2 Open Test Cases

Name	Status	Comment
NASA High Lift Prediction Workshop		
NASA Drag Prediction Workshop		
NATO RTO AVT VFE 2		
SMC006 Serrated Nozzle		
DARPA SUBOFF		
DARPA HIREP		
NASA Rotor 37		
Civil Aircraft Landing Gear		

3.3 Closed Test Cases

From ARUP, ARA, Willams F1

High Lift Prediction Workshop - Validation

Authors: A. Cimpoeru (CFMS), J. Appa (Zenotech) and D. Standingford (Zenotech)

November 2014

4.1 Abstract

This document summarizes some initial results obtained within the second order benchmarking of a new CFD software. The zCFD code was used for the DLR F11 high lift configuration in order to compute the flow field using the k-ω SST turbulence model. The results consist of high Reynolds number computations for the simplified (Case 1) and complex (Case 2B) configurations at 7° angle of attack. The results were validated against the numerical solutions obtained using CFD++ and ANSYS FLUENT and showed a good agreement with the experimental data.

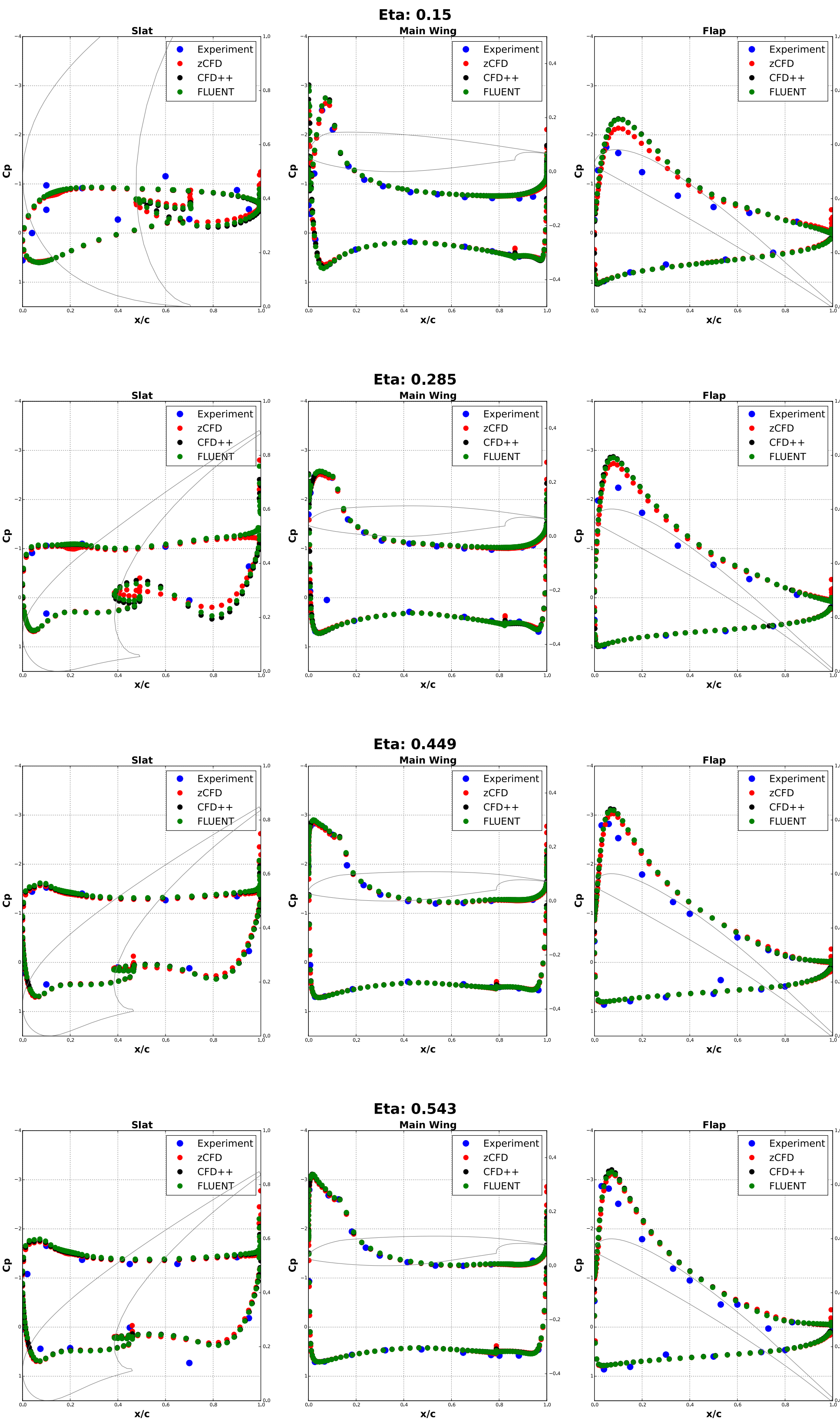
4.2 Introduction

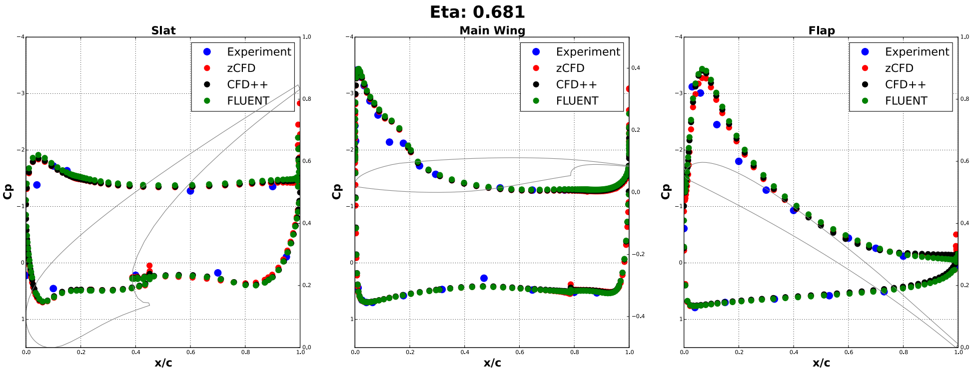
The zCFD code is a GPU accelerated high performance computational fluid dynamic software. The zCFD solver is an unstructured cell centered finite volume code which solves explicitly the compressible Navier-Stokes equations. The solution is preconditioned and the convergence is accelerated using the Multigrid technique along with a dual time stepping method.

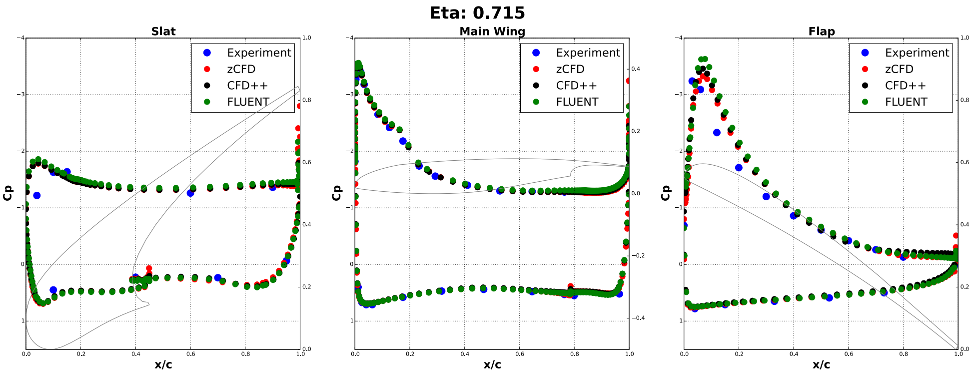
The results consist in high Reynolds number computations using k-ω SST turbulence model for the simplified (Case 1) and complex (Case 2B) configurations at 7° angle of attack. The zCFD solutions were compared with wind tunnel data and other CFD codes such as Metacomp CFD++ and Ansys FLUENT. These codes were selected since they use the same cell-centered approach in combination with k-ω SST model.

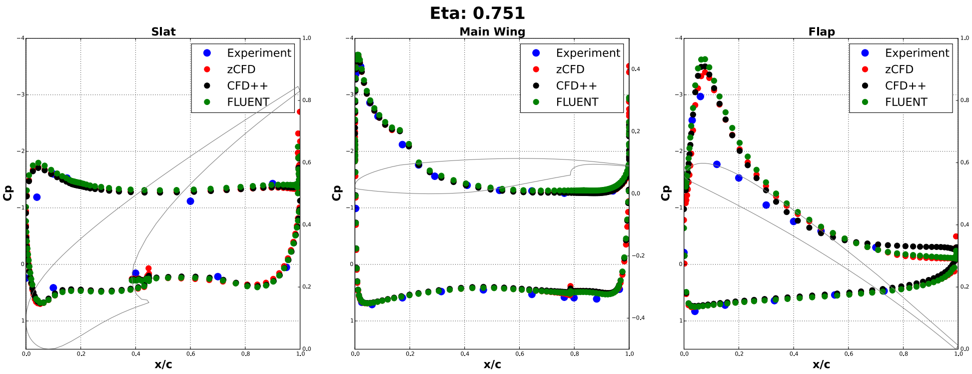
4.3 Results

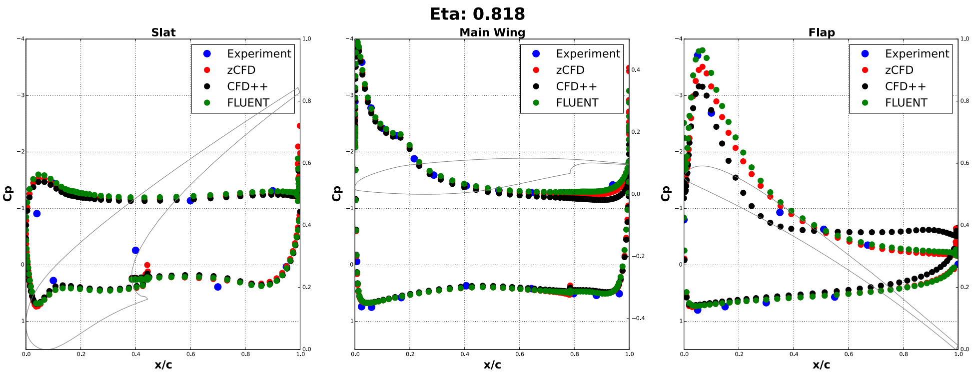
Case 1, Coarse mesh (9.5M cells), $Mach = 0.175$, $Re = 15e6$, $\alpha = 7^\circ$

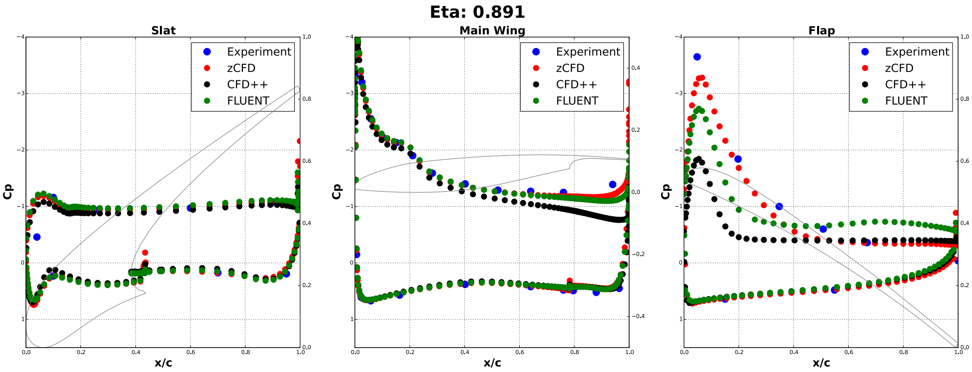

Eta: 0.15
Slat
Main Wing
Flap
Experiment
zCFD
CFD++
FLUENT
Cp
x/c
Eta: 0.285
Slat
Main Wing
Flap
Experiment
zCFD
CFD++
FLUENT
Cp
x/c
Eta: 0.449
Slat
Main Wing
Flap
Experiment
zCFD
CFD++
FLUENT
Cp
x/c
Eta: 0.543
Slat
Main Wing
Flap
Experiment
zCFD
CFD++
FLUENT
Cp
x/c

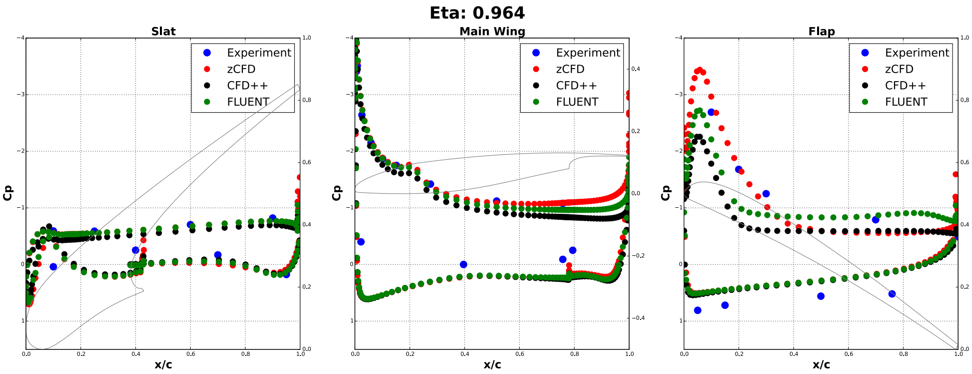

Eta: 0.681
Slat
Main Wing
Flap
Experiment
zCFD
CFD++
FLUENT
Cp
x/c


Eta: 0.715
Slat
Main Wing
Flap
Experiment
zCFD
CFD++
FLUENT
Cp
x/c

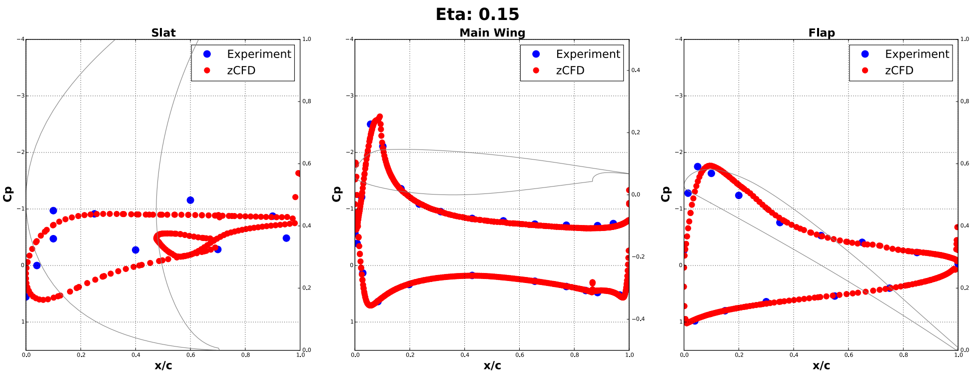

Eta: 0.751
Slat
Main Wing
Flap
Experiment
zCFD
CFD++
FLUENT
Cp
x/c

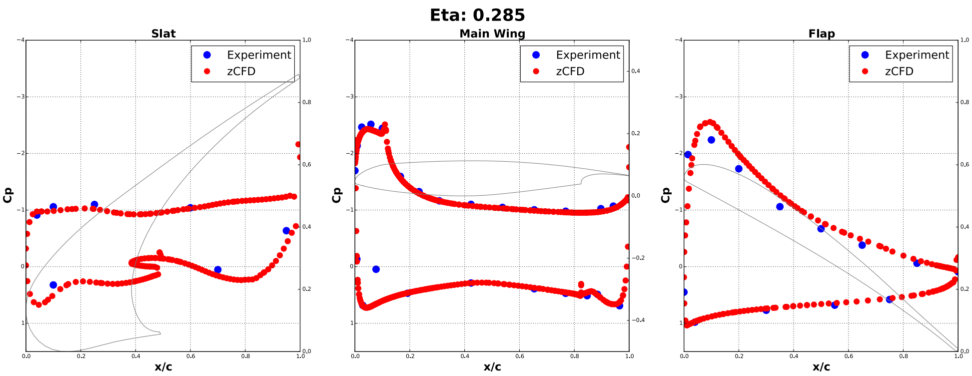

Eta: 0.818
Slat
Main Wing
Flap
Experiment
zCFD
CFD++
FLUENT
Cp
x/c

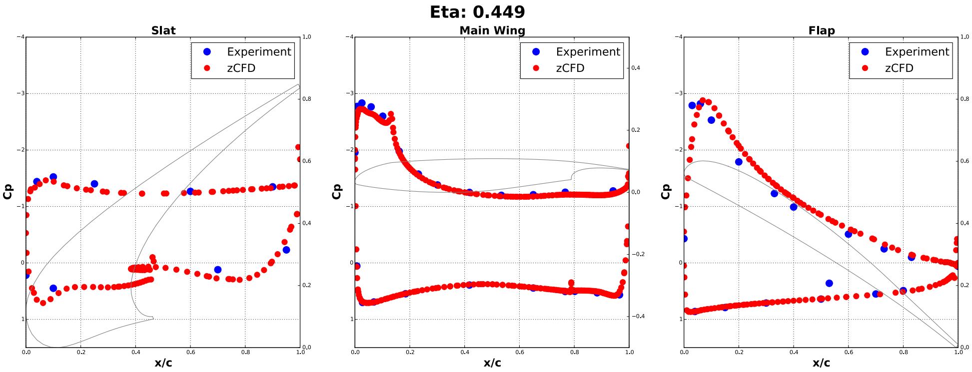


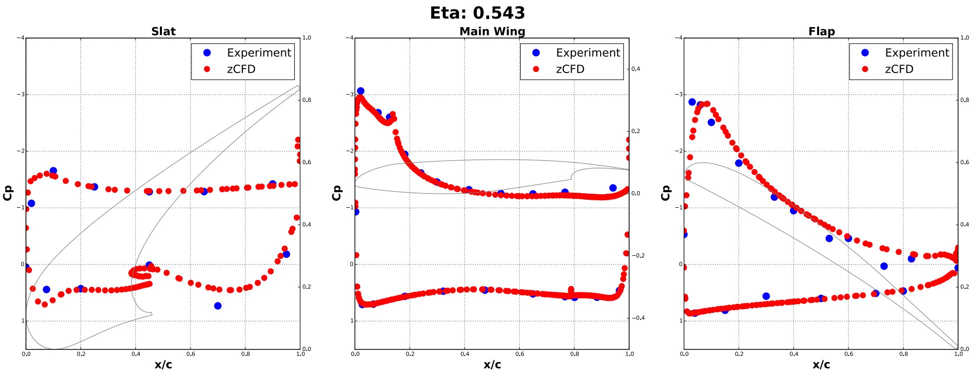


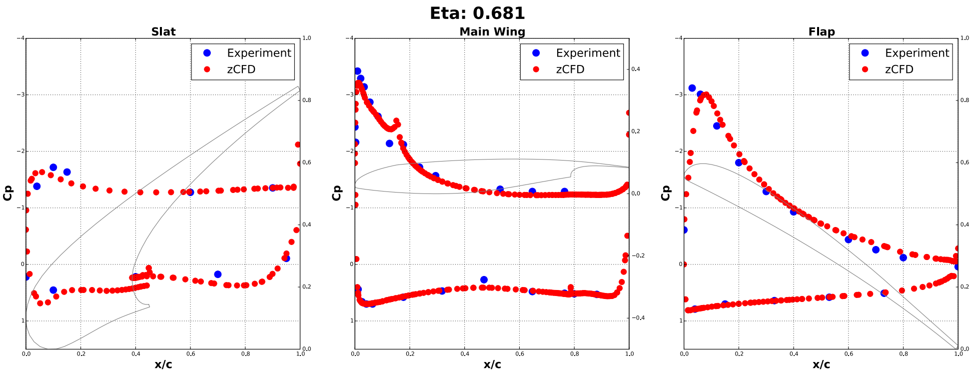
Case 2b, Solar mesh (50M cells), $Mach = 0.175$, $Re = 15e6$, $\alpha = 7°$

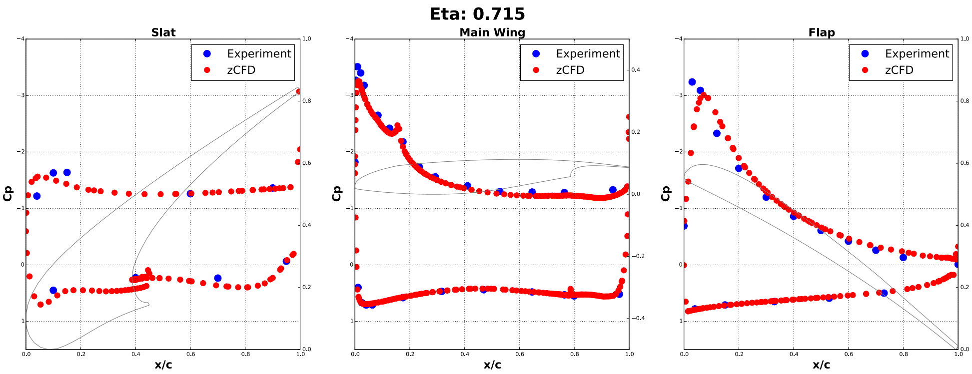


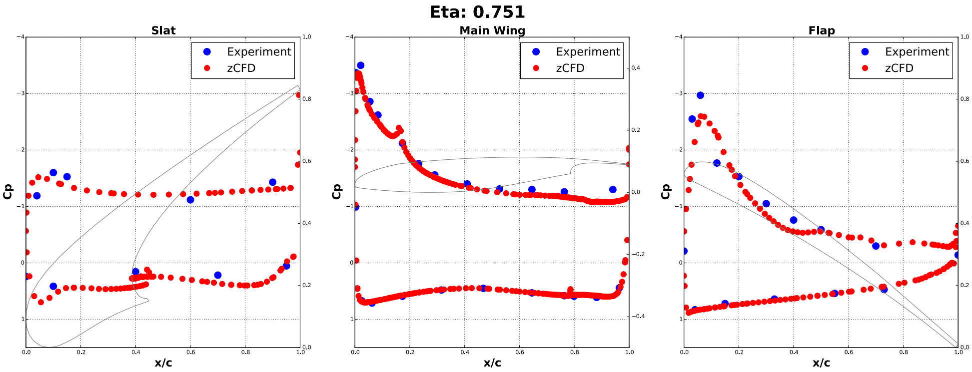


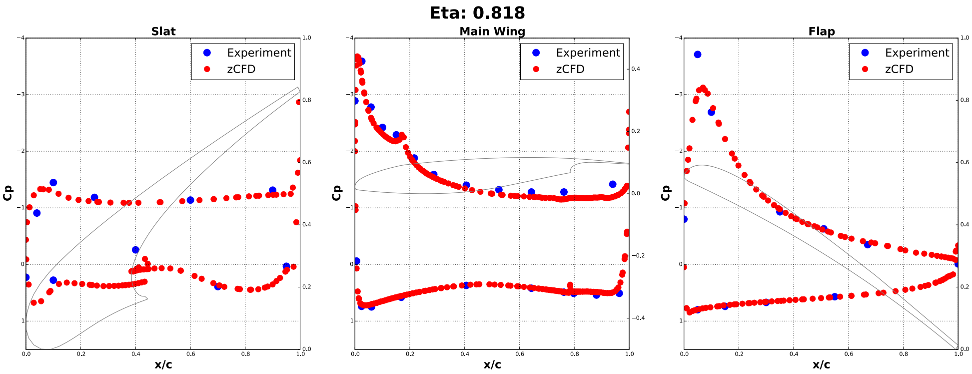

Eta: 0.449
Slat
Main Wing
Flap
Experiment
zCFD
Cp
x/c

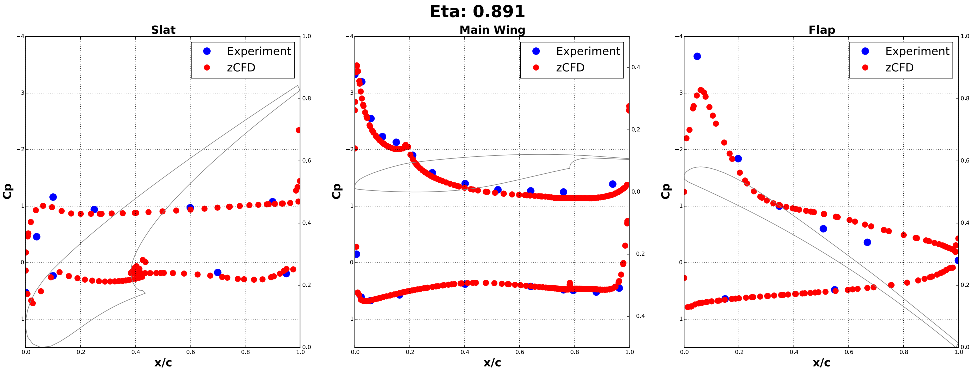

Eta: 0.543
Slat
Main Wing
Flap
Experiment
zCFD
Cp
x/c

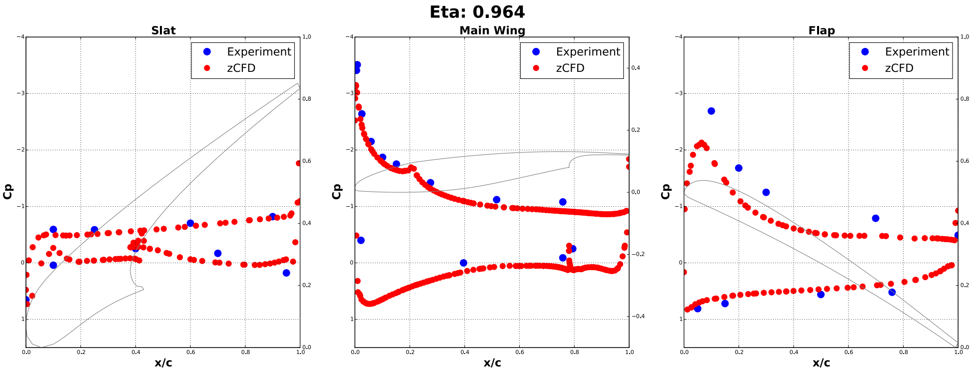

Eta: 0.681
Slat
Main Wing
Flap
Experiment
zCFD
Cp
x/c

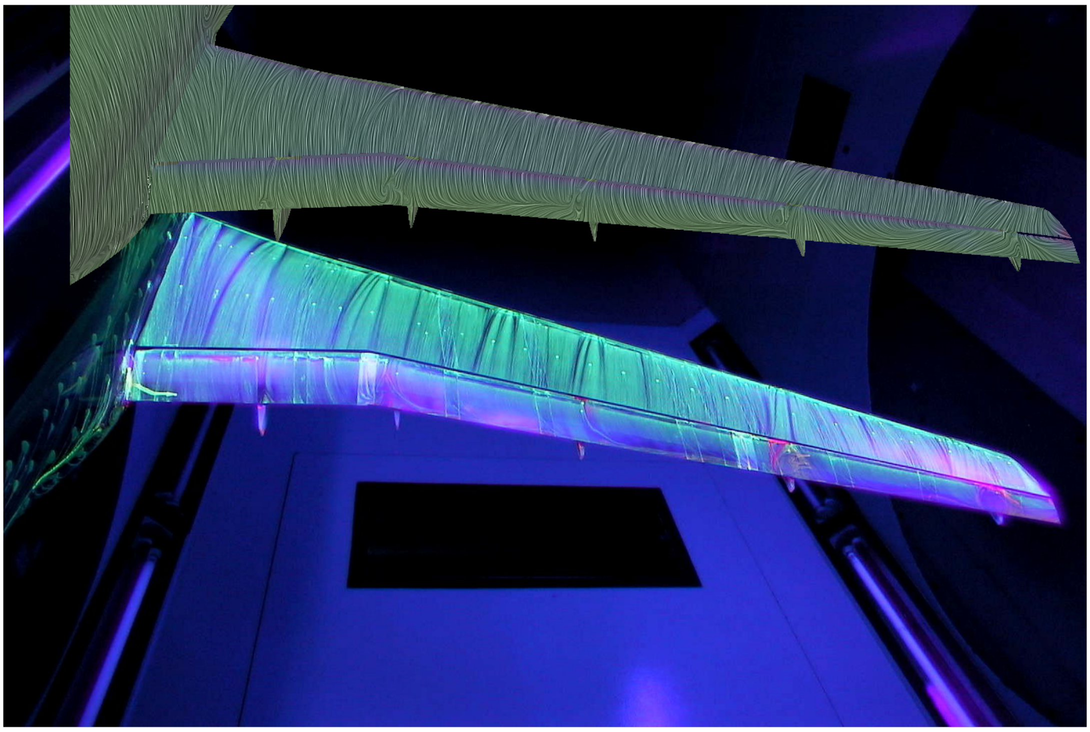

Eta: 0.715
Slat
Main Wing
Flap
Experiment
zCFD
Cp
x/c


Eta: 0.751
Slat
Main Wing
Flap
Experiment
zCFD
Cp
x/c


Eta: 0.818
Slat
Main Wing
Flap
Experiment
zCFD
Cp
x/c


Eta: 0.891
Slat
Main Wing
Flap
Experiment
zCFD
Cp
x/c


Eta: 0.964
Slat
Main Wing
Flap
Experiment
zCFD
Cp
x/c



DLR F11 High Lift Configuration; Locations of Pressure Sections. For the full definition of cutting planes refer to workshop website.

See also:

High Lift Workshop

Model dimensions

Case 1 Notebook

Case 2 Notebook

NASA Drag Prediction Workshop - NASA Common Research Model - Validation

Authors: A. Cimpoeru (CFMS), J. Appa (Zenotech) and D. Standingford (Zenotech)

December 2014

5.1 Configuration (Wing-Body no Tail) , L3 mesh unstructured hexahedral - 5.1 x 106 cells - Match CL = 0.5

Fig. 5.1: NASA Common Research Model

5.2 Conditions

Reynolds number = 5.0 x e06 based on MAC

Mach = 0.85

Angle of attack = 2.217

Reference static pressure = 101325 Pa

Reference static temperature = 310.928 K

RANS steady state

k-W SST model

Preconditioned

MUSCL scheme for

5.3 Results

○ Isosurfaces of Mach number

○ CL match for L3 mesh unstructured hexahedral mesh

zCFD	Experiment	Error
CL = 0.4884	CL = 0.508	-0.46 %

○ Code to code validation

See also:

Drag Prediction Workshop

NASA Common Research Model Notebook

Numerical investigation of the flow field around 65° delta wing using the unstructured code zCFD

Authors: A. Cimpoeru (CFMS), J. Appa (Zenotech) and D. Standingford (Zenotech)

December 2014

6.1 Abstract

In the present study the flow field around 65° delta wing is investigated using the unstructured density based solver zCFD. The simulations were carried out by solving the steady state RANS equations using the k-ω SST turbulence model at :math:'13.3^circ' flow angle for :math: 'Re=6.0e06' (NASA) and :math:'Re=3.0e06' based on MAC. Within this analysis the numerical solutions are compared against wind tunnel results performed at NASA and DLR.

6.2 Geometry Specification

The geometry has been generated using the analytical definition available in the reference (Chu and Luckring). For the present study two sets of simulations were performed: (1) Comparison against the experiment performed at NASA and (2) Comparison against the experiment performed at DLR as part of the VFE 2 project.

Fig. 6.1: Delta wing geometry

6.3 Results

Steady state RANS, SOLAR - octree mesh (30M cells), $Mach = 0.4$, $Re = 6.0e06$ (based on MAC = 0.3268 m)

○ Large Radius - Validation against NASA Wind Tunnel Experiment

○ Sharp Radius - Validation against NASA Wind Tunnel Experiment

○ Large (right) and Sharp radius (left)

See also:

Experimental data and geometry specification (4 volumes available on NASA Technical Reports Server)

DLR Project VFE-2

J Chu and J M Luckring. Experimental Surface Pressure Data Obtained on 65 degrees Delta Wing Across Reynolds Number and Mach Number Ranges. NASA TM 4645, 1996

R M Cummings and A Schute. Detached-Eddy Simulation od the vortical flow field about the VFE-2 delta wing. Aerospace Science and Technology 24 (2013) 66-76

R Konrath, C Klein and A Schroder. PSP and PIV investigations on the VFE-2 configuration in sub- and transonic flow. Aerospace Science and Technology 24 (2013) 22-31

Delta Wing Sharp Radius Notebook

Validation of the flow field around DARPA SUBOFF model

Authors: A. Cimpoeru (CFMS), J. Appa (Zenotech) and D. Standingford (Zenotech)

December 2014

7.1 Abstract

In this section the flow field around DARPA SUBOFF geometry is computed using the unstructured cell centered finite volume density based solver zCFD. For this study the AFF8 configuration was employed due to complex flow features such as boundary layer - vortex and vortex-vortex interactions. The Reynolds number for this test case is $Re = 1.2e07$ and the employed turbulence model is k-ω SST. The steady state solutions are validated against wind tunnel data (see Reference) and have shown agreement.

7.2 Configuration (Axisymmetric hull + Fairwater + Stern appendages)

Fig. 7.1: SUBOFF Configuration

7.3 Results

Steady state RANS, SOLAR - octree mesh (14 and 30 M cells), $Mach = 0.0179$, $Re = 1.2e7$ (model length based)

Fig. 7.2: Coefficient of pressure in the symmetry plane

Fig. 7.3: Coefficient of pressure contours

Wake development (Left – Coarse Mesh ; Right – Fine Mesh)

Comparison against zCFD, experimental data and Bhushan et al (2013) (FLUENT)

See also:

Summary of DARPA Suboff Experimental program data

Geometric Characteristics of DARPA SUBOFF models

Fig. 7.4: x/L = 0.3

Fig. 7.5: x/L = 0.4

'S. Bhushan, M. F. Alam and D. K. Walters. Evaluation of hybrid RANS/LES models for prediction of flow around surface combatant and Suboff geometries. Computer and Fluids 88 (2013) 834-849'

DARPA SUBOFF Notebook

Fig. 7.6: x/L = 0.5

Fig. 7.7: x/L = 0.978 (Propeller Plane)

Fig. 7.8: x/L = 0.978 (Propeller Plane)

CHAPTER 8

RANS simulations of cold jet flows from a serrated nozzle

Authors: A. Cimpoeru (CFMS), J. Appa (Zenotech) and D. Standingford (Zenotech)

February 2015

8.1 Abstract

8.2 Geometry and Mesh Generation (SMC-006 Nozzle)

Fig. 8.1: SMC006 chevron type nozzle. Left (Model) and Right (Domain Topology) (Xia et. al. (2009))

Fig. 8.2: SMC006 mesh on x-y plane. Multiblock Stuctured Mesh (19M cells) (Xia H. and Tucker P. (2009))

8.3 Initial Conditions (Steady-State RANS)

Ambient Conditions

Variable	Value	Unit
P_amb	97000	Pa
T_amb	280.2	K
Rho_amb	1.225	Kg/m^3
mu	1.79e-5	Pa.s
speed_of_sound	335.549	m/s
gas_constant	287.0	KJ/KgK

Jet Conditions

Variable	Value	
NPR	1.83	Nozzle Pressure Ratio
TPR	1.022	Nozzle Temperature Ratio
Mjet	0.9	Jet Mach number
Ujet	300	Reference Velocity [m/s]
Reynolds	1.03e06	Reynolds Number (See Note)

8.4 Results

Fig. 8.3: Jet velocity profiles in the Tip-to-Tip plane at different stations

Fig. 8.4: Jet velocity profiles in the Notch plane at different stations

Fig. 8.5: Comparison between LES study of Xia et. al. (2009) (top) and present RANS study (bottom) at different stations

- Note

The Reynolds number is based on the jet diameter and reference velocity

See also:

'Hao Xia, Paul G. Tucker and Simon Eastwood (2009). Large-eddy simulations of chevron jet flows with noise predictions. International Journal of Heat and Fluid Flow 30 (2009) 1067-1079.'_

'Hao Xia and Paul G. Tucker (2011). Numerical Simulation of a Single-Stream Jets from a Serrated Nozzle . Flow Turbulence Combust 2011.'_

Serrated Nozzle Notebook

Fig. 8.6: Isosurfaces of Q criterion (present study)

NASA Rotor 37 zCFD Code Validation - Flow Field in a Transonic Axial Compressor

Authors: A. Cimpoeru (CFMS), J. Appa (Zenotech) and D. Standingford (Zenotech)

May 2015

9.1 Abstract

9.2 Geometry and Mesh Generation (NASA ROTOR37)

9.3 Parameters

9.4 Initial Conditions (Steady-State RANS)

Ambient Conditions

Variable	Value	Unit
P_amb	101523	Pa
T_amb	288.15	K
Rho_amb	1.225	Kg/m^3
mu	1.79e-5	Pa.s
gas_constant	287.0	KJ/KgK

Inflow Total Conditions.

Variable	Ratio
Ptotal/P_amb	1.0
T_total/T_amb	1.0

9.5 Results

Note that the map has been initially generated by varying the static pressure ratio on the outflow from 0.7 to 1.8 in order to determine the chocked and stalled conditions.

More results will be publised in the following weeks.

Fig. 9.1: Cp distribution on the rotor blades

Fig. 9.2: Mach number in the flow field and pressure contours.

9.6 References

Fig. 9.3: Turbulent eddy viscosity and pressure contours