2nd AIAA Aeroelastic Prediction Workshop – Loci/CHEM Results

Eric Blades
Azariah Cornish

Prepared for:

Second Aeroelastic Prediction Workshop

Date:

January 2-3, 2016

13290 Evening Creek Drive, Suite 250, San Diego CA 92128
(858) 480-2000
www.ata-e.com
ata-engineering
@ATAEngineering

Agenda

- Methodology and Tools
  - CFD solver
  - CSD solver
  - Wetted surface transfer
  - Mesh deformation
- Coupling Approach
- Case 1 Results (3 mesh levels)
- Case 2 Results (3 mesh levels)

FSI Methodology

- Domain-decomposition or partitioned approach
  - Each solver is optimized for the numerics of each domain
  - Fluid domain: Loci/CHEM finite volume solver
    - Developed at Mississippi State University (Prof. Ed Luke)
    - Open source
    - Production solver at NASA/MSFC
  - Structural domain: FEM solver
    - Linear: Native CSD solver module for CHEM
    - Non-linear: Couple to Abaqus NLCSD solver
- Each domain suitably discretized for domain-specific analysis
  - Wetted surface will likely be discretized differently

CFD Flow Solver: Loci/CHEM

Key Features

- Unsteady Reynolds-Averaged Navier-Stokes solver
  - Cell-centered
  - Supports unstructured mesh topologies including polyhedral meshes
- Parallel, highly scalable
- Finite-rate chemically reacting flows
- Implicit and explicit time integration methods
- Multiple turbulence models and hybrid RANS-LES
- Moving meshes (ALE, GCL)
  - Mesh deformation
  - Overset mesh capability

CFD Flow Solver: Loci/CHEM

Important Parameters

- Unsteady Reynolds-Averaged Navier-Stokes solver
  - Case1: Used SA and SST turbulence models (SA for unsteady)
  - Case 2: SST turbulence model
- Inviscid flux construction: Roe scheme
  - Spatially second-order accurate
  - Venkatakrishnan flux limiter used
- Temporal evolution: Newton-relaxation
- Temporal discretization
  - 3-pt backward Euler
  - 2nd order accurate
- Used provided unstructured meshes

Linear CSD Solver

- Developed as module for CHEM
- Import mass/stiffness matrices or modal model from Nastran and Abaqus
  - Allows use of existing structural models
- Utilizes PETSc parallel linear algebra library
- Newmark-beta time integration scheme
  - Implicit, 2nd order, backwards finite difference
  - Hilbur-Hughes-Taylor similar, adds numerical damping
- No structural damping used
- Initial *rotational velocity* applied to perturb structure

Wetted Surface Information Transfer

- Algorithm ensures *global* conservation of loads and displacements
  - Principal of virtual work and is used to transform the aerodynamic forces to the finite element nodal forces → Conserves forces/moments
  - Structural displacements at the FE nodes are transformed back to the CFD wetted surface grid points through the reciprocal theorem → Ensures conservation of work
- Requires wetted surface definition for CFD and CSD domains
- Robust
  - Allows for gaps between CFD and FEM wetted surfaces
  - Allows disjoint wetted surfaces

Wetted Surface Algorithm

- For each CFD wetted surface node
  - Find nearest FEM node
  - Find host FEM element
    - Search performed using element neighbor search
    - Only performed once
- Project all CFD forces to FEM wetted surface
  - Distribute force to nodes of host element using isoparametric shape functions

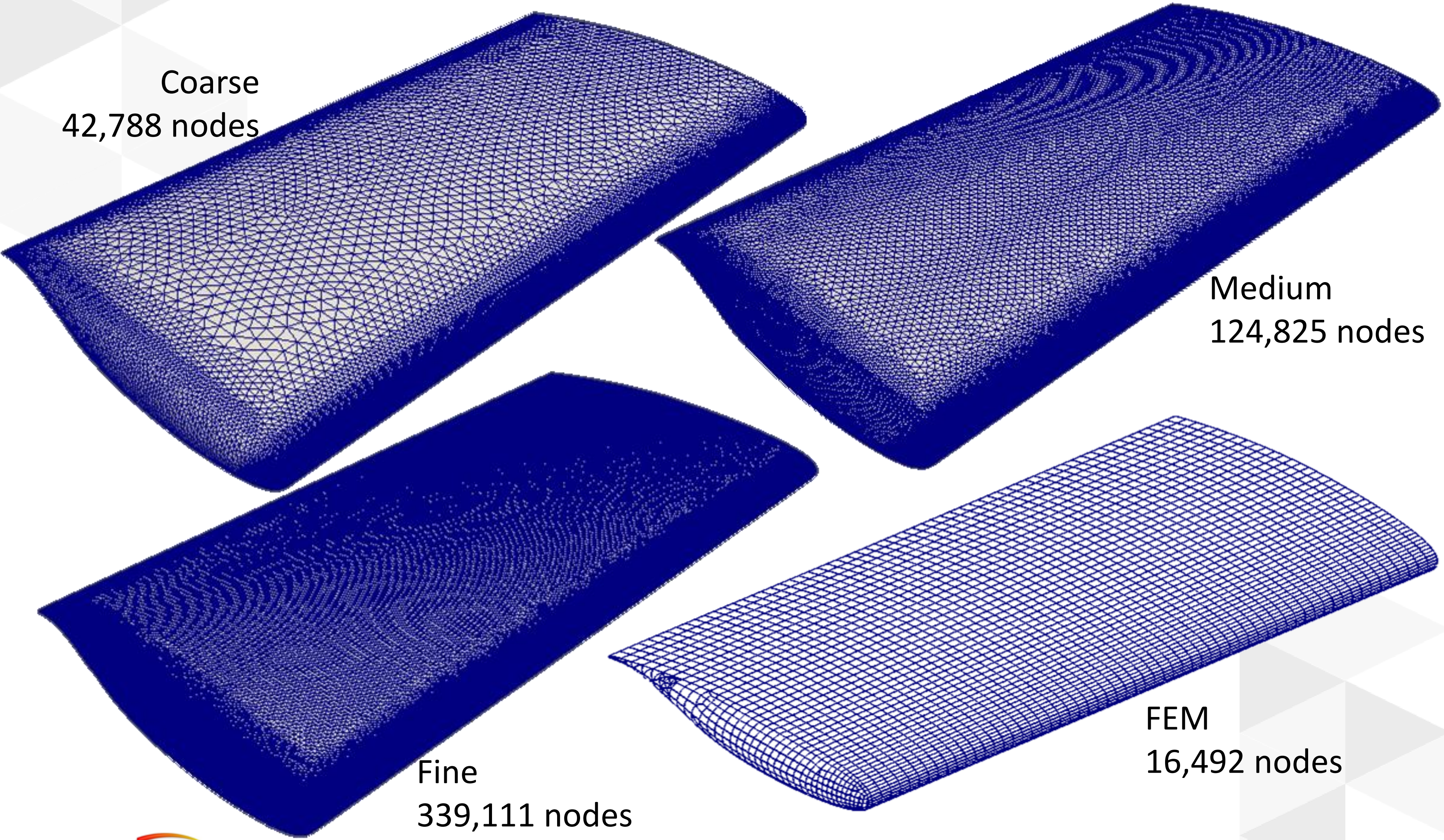
$$F_i^{CSD} = \sum_{j=1}^{m} \varphi_j f_i^{CFD} \qquad M_i^{CSD} = \sum_{j=1}^{m} \varphi_j \left(\vec{f}^{\,CFD} \times \vec{d} \right)$$

  - Use of same shape functions for displacement transfer ensure conservation of work

$$\delta_i^{CFD} = \sum_{j=1}^{m} \varphi_j \left(\delta_i^{CSD} + \vec{d} \times \vec{\delta}^{\,CSD} \right)$$

ATA ENGINEERING, INC.

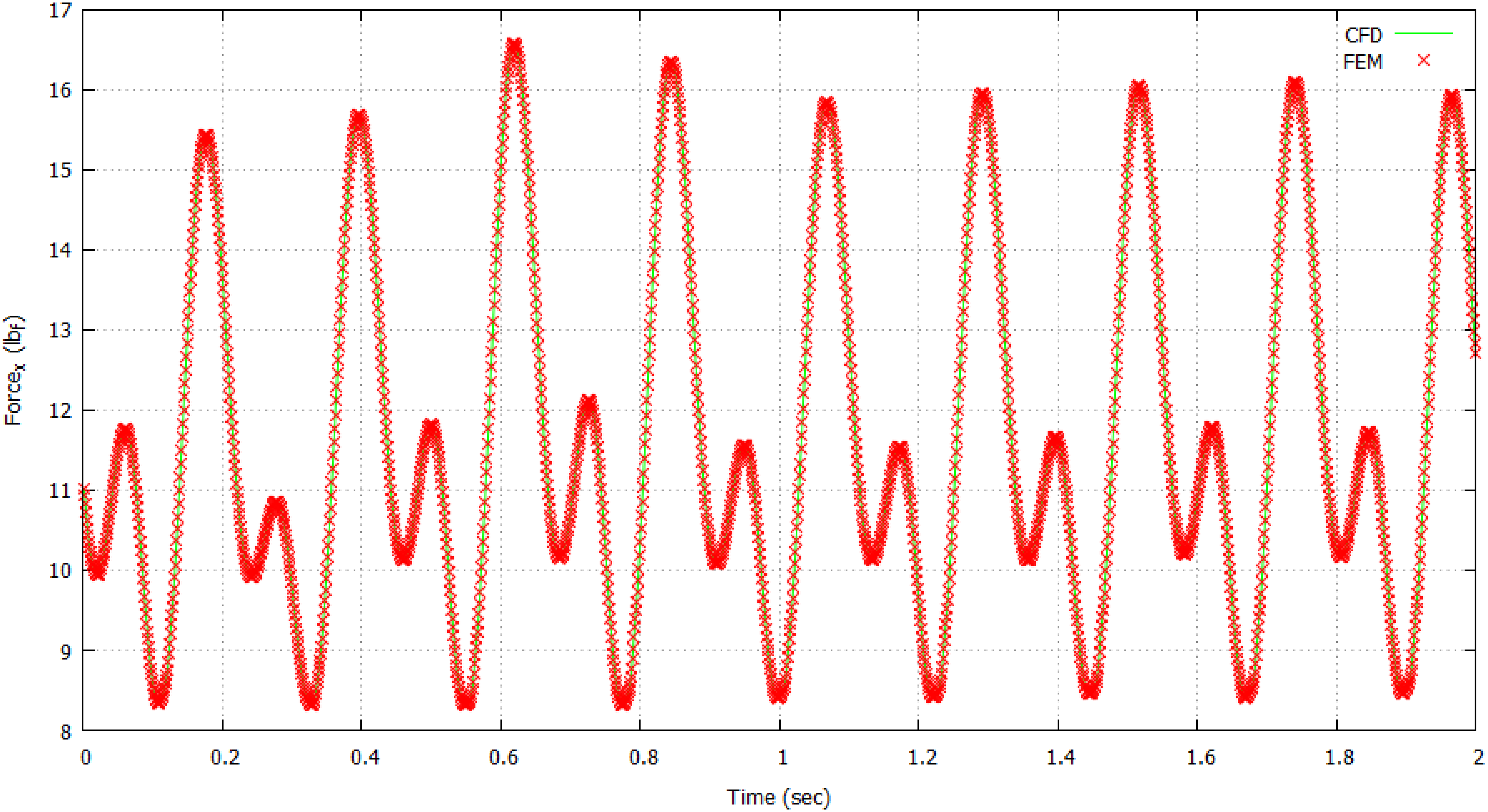
Wetted Surface Comparison



Wetted Surface Force Comparison: Fx

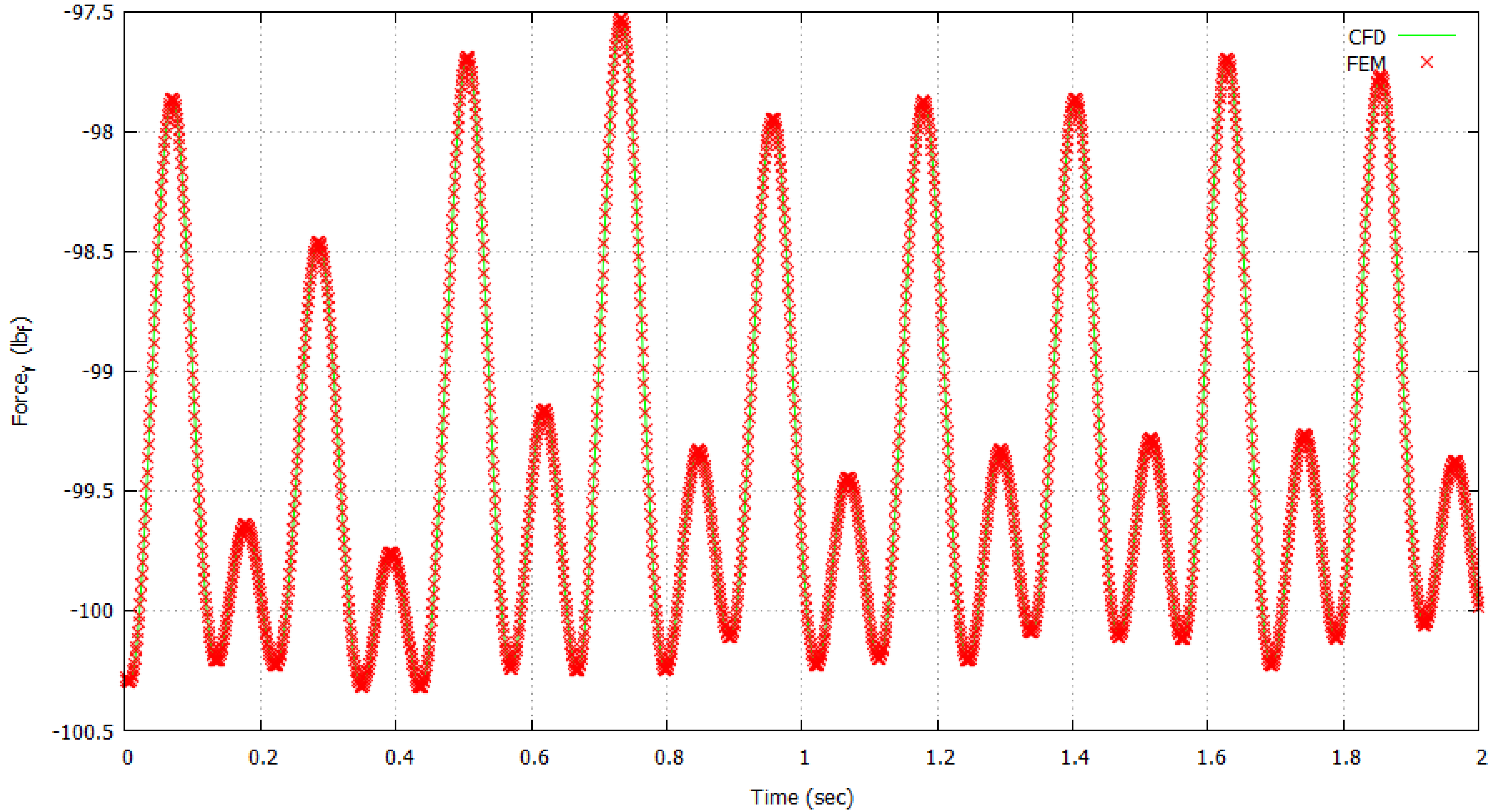
Medium CFD grid at the predicted flutter condition

Differences to within machine precision



Wetted Surface Force Comparison: Fy

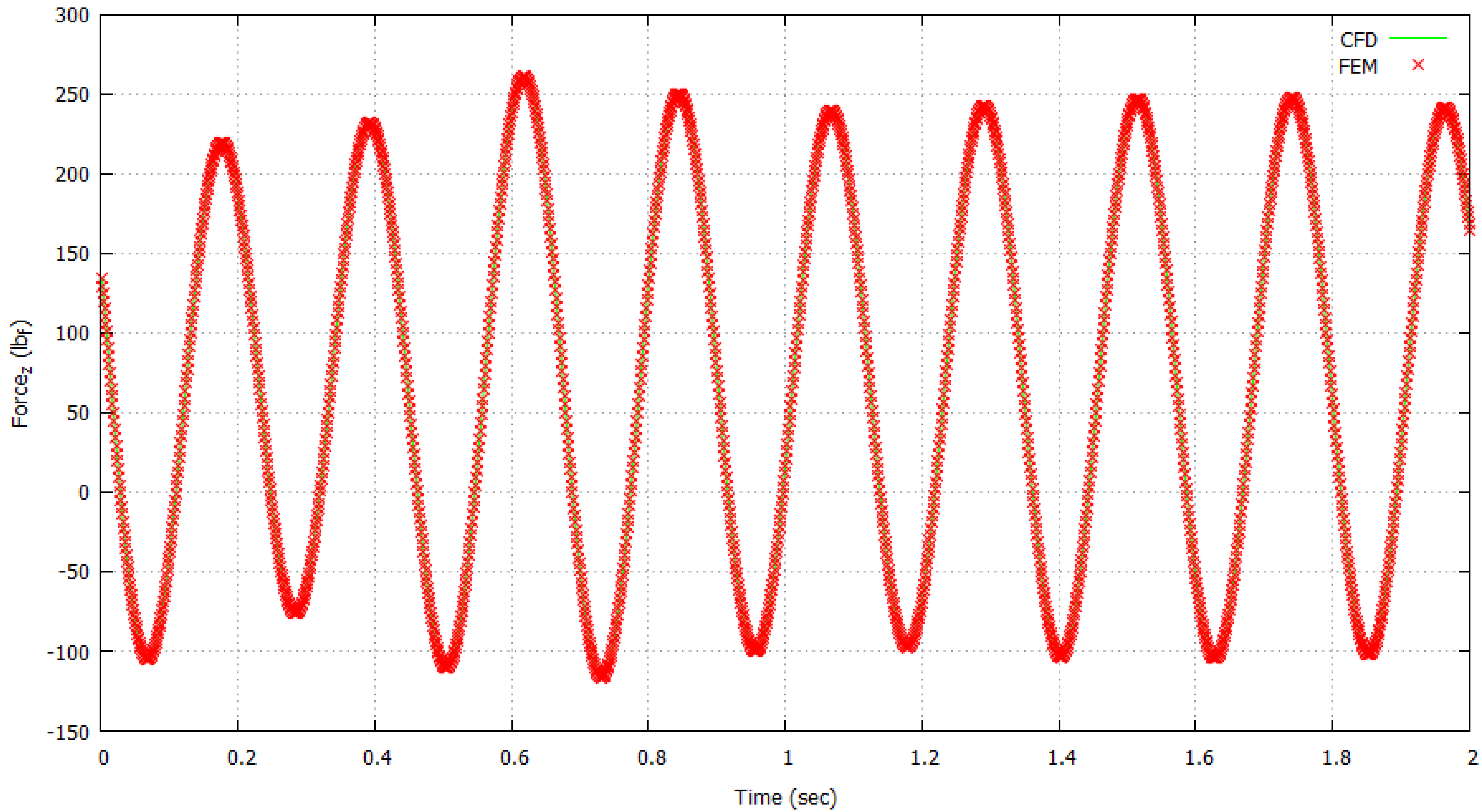
Medium CFD grid at the predicted flutter condition



ATA ENGINEERING, INC.

Wetted Surface Force Comparison: Fz

Medium CFD grid at the predicted flutter condition



Mesh Deformation

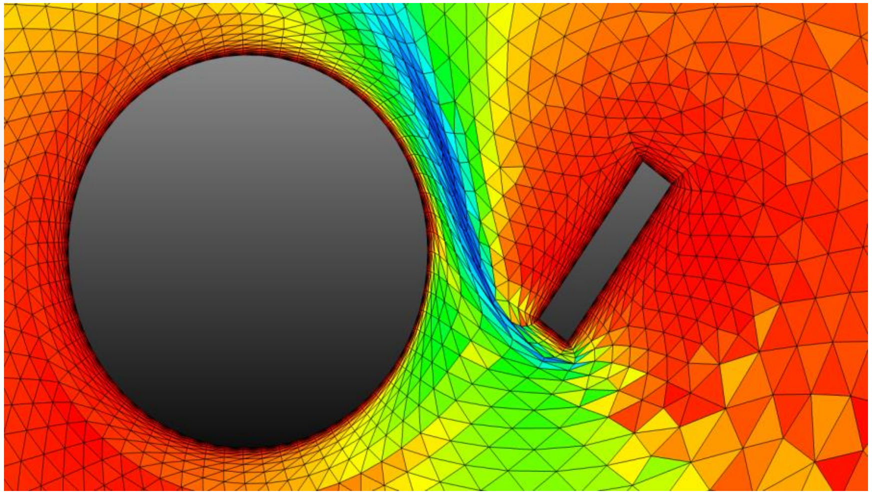
- Interpolation based approach†
  - Uses reciprocal distance-weighed averages of rotation and displacements of surface nodes
  - Local deformation field is modeled as a rigid body transformation involving both a rotation and a translation
  - Rotations of the surface about local nodes are computed using a nonlinear least squares method to find the closest rotation about the node that best matches the displacements of all edges and normals from surface facets that reference the given node

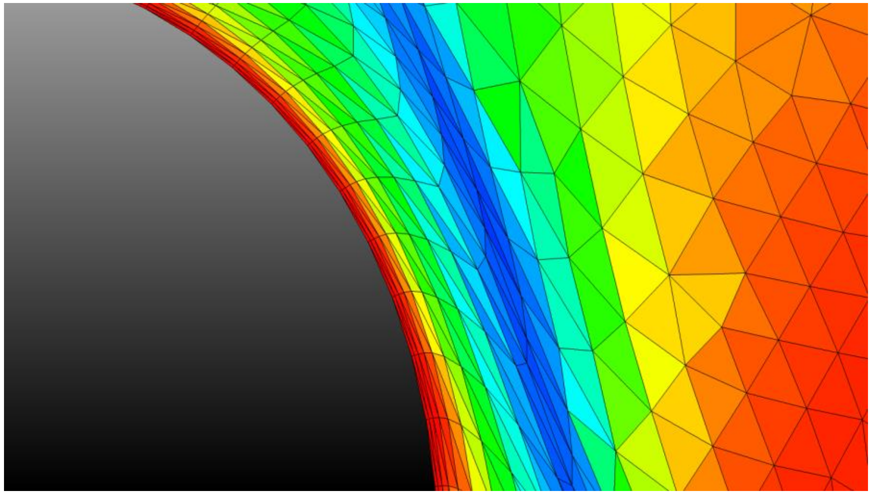
†E. Luke , E. Collins, and E. Blades, "A fast mesh deformation method using explicit interpolation," Journal of Computational Physics:231(2), pp. 586-601

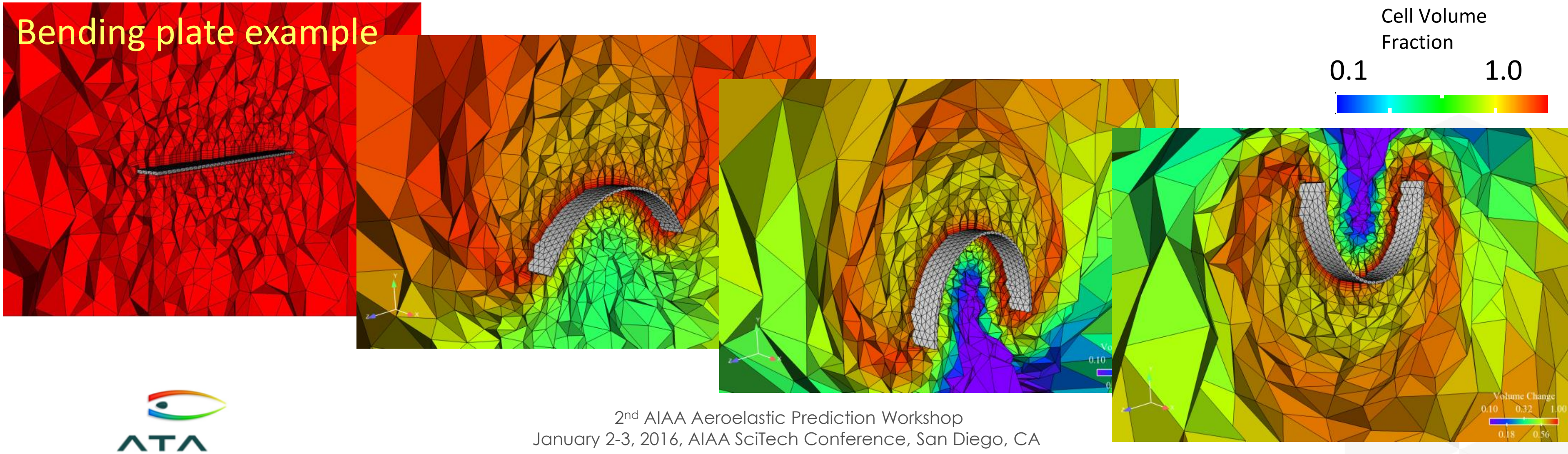
Mesh Deformation

- Utilize a two-exponent form of the weighting function
  - Preserve near-boundary deformations while providing a smooth transition in the interpolation from a near-body region of strong boundary influence into the bulk of the volume mesh
  - Works well on arbitrary mesh types including viscous BL meshes and hanging node adapted meshes
- Deformation applied in single step so mesh quality doesn't degrade for periodic motions

Mesh Deformation Examples





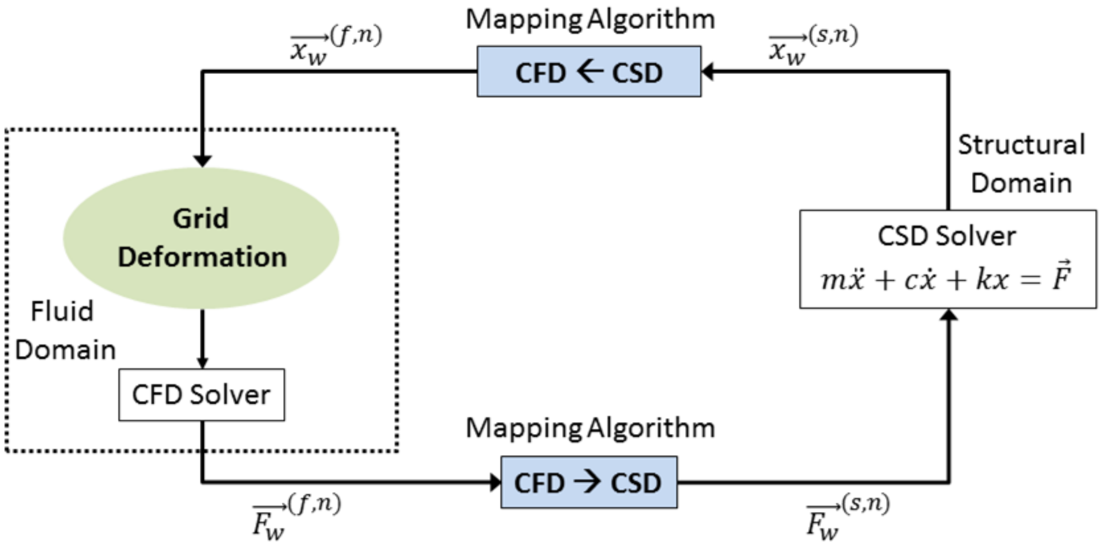


ATA ENGINEERING, INC.

Coupling Approach

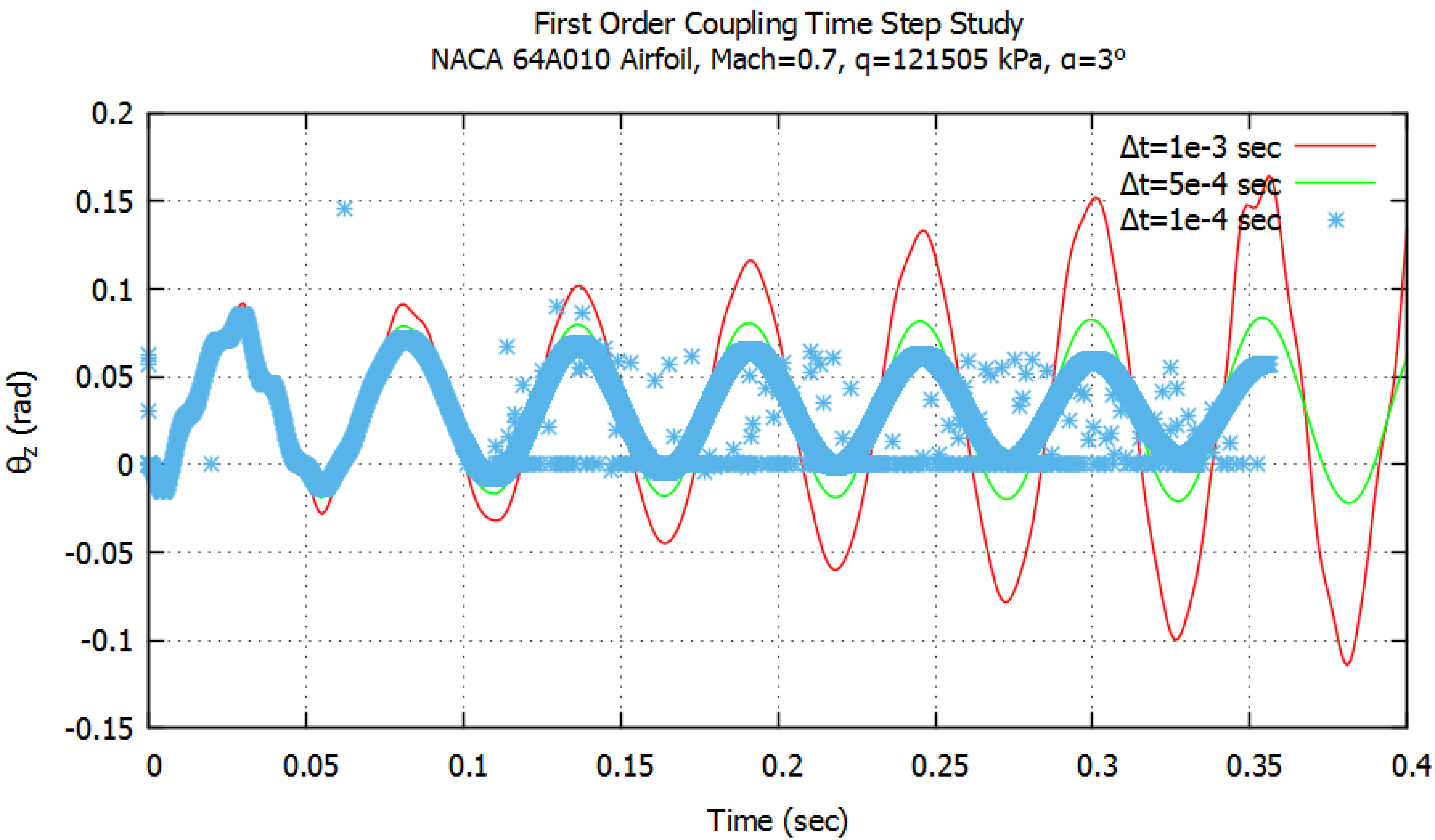
When to couple/exchange information?

- Weak/loose coupling: Information exchange at end of each solver time step → coupled solution is temporally 1st order

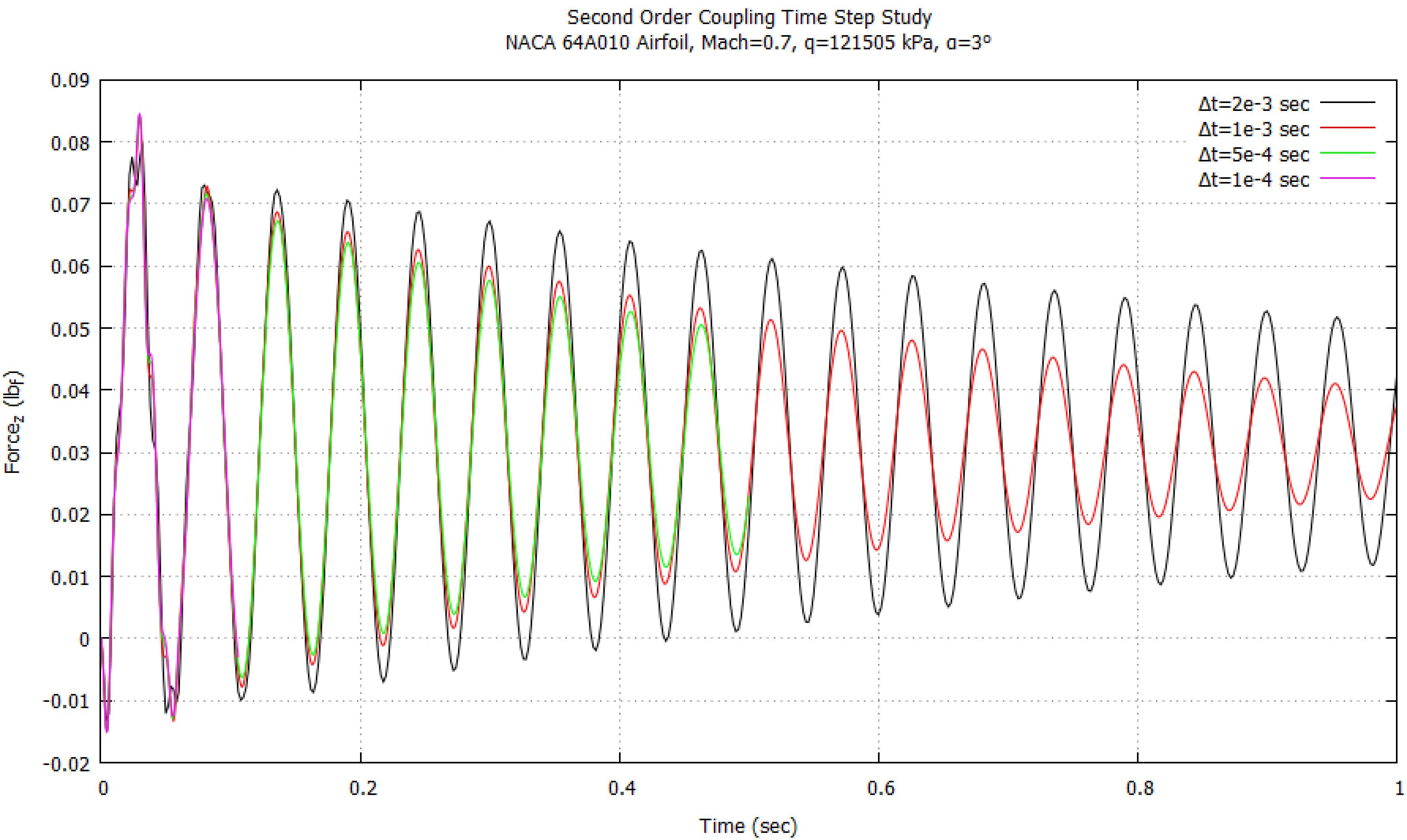


- Strong/tight coupling: Exchange information within time step at sub-iteration level → coupled solution is temporally 2nd order

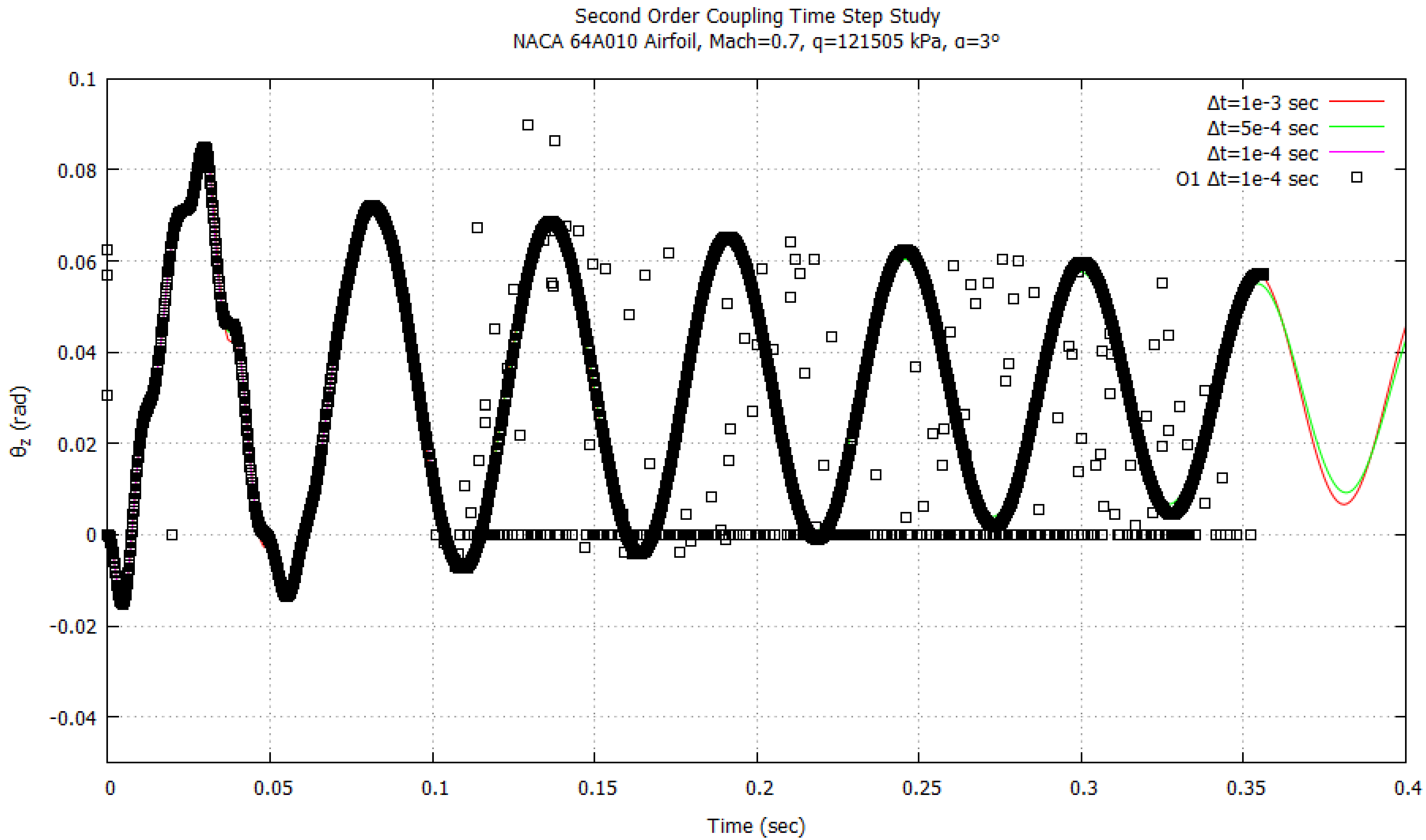
Coupling Scheme: First Order



Coupling Scheme: Second Order

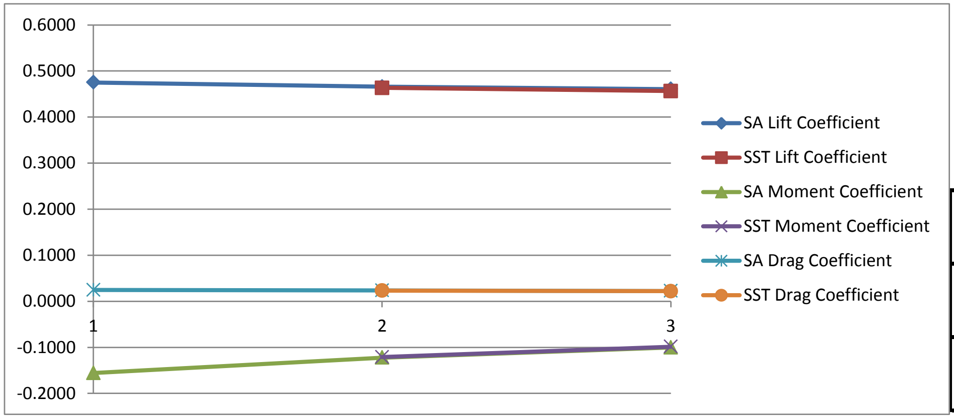


Coupling Scheme Comparison



Case 1: Steady Results

- All grids were run steady-state before beginning oscillation and 60% Span coefficients calculated
- Minimal difference between turbulence models (SA used for dynamic runs)
- Lift and drag coefficients changed <5% between grids
- Moment coefficient changed ~36% from coarse to fine grid



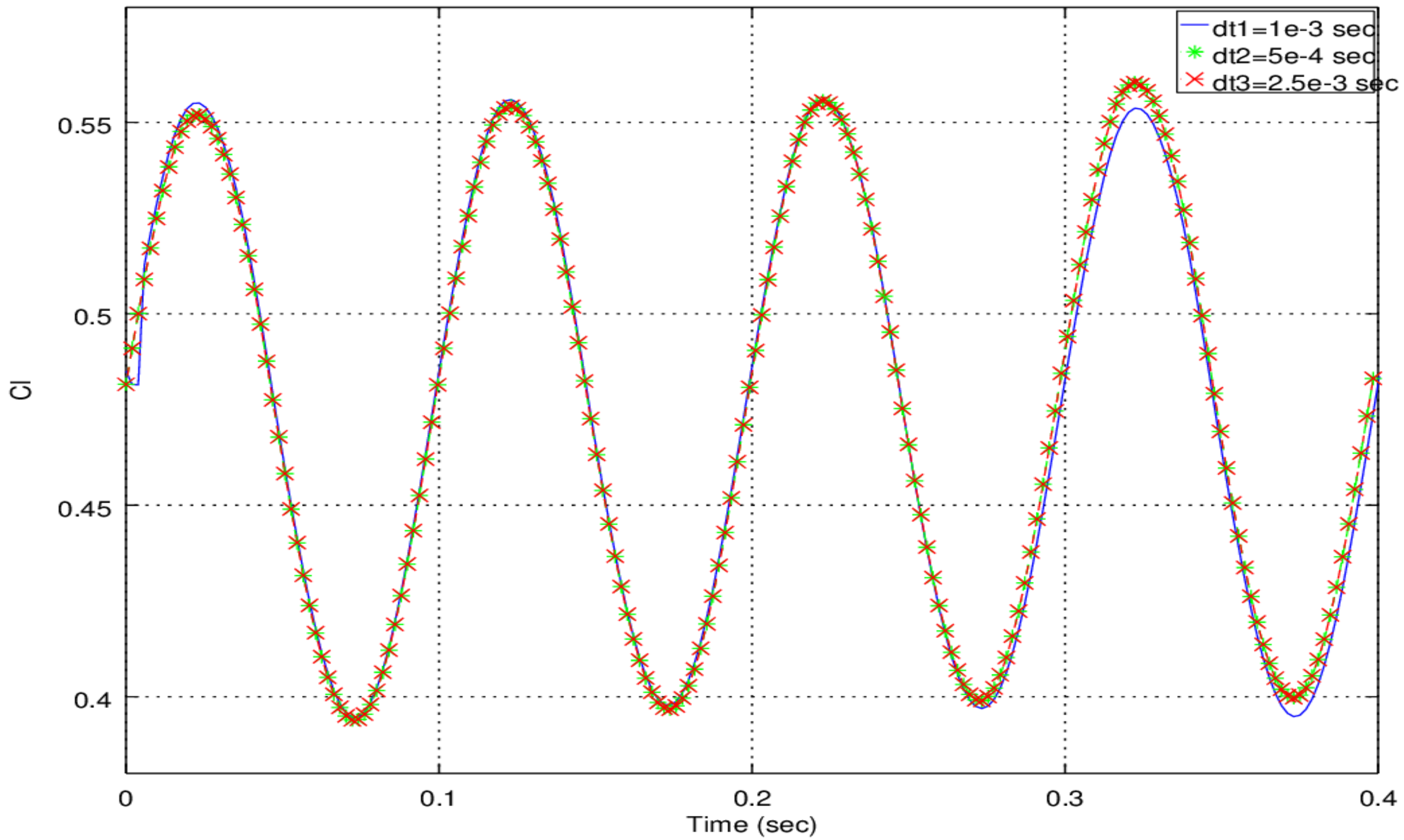
		Coarse		Med		Fine	
		SA	SST	SA	SST	SA	SST
C_L	Avg.	0.4750		0.4660	0.4635	0.4607	0.4565
	Std.	0.23%		0.30%	0.30%	0.06%	0.06%
C_m	Avg.	-0.1557		-0.1223	-0.1212	-0.1003	-0.0985
	Std.	0.11%		0.11%	0.11%	0.08%	0.08%
C_D	Avg.	0.0246		0.0236	0.0233	0.0224	0.0220
	Std.	0.34%		0.37%	0.37%	0.38%	0.39%

Case 1 Pertinent Unsteady Parameters

- Temporal resolution study
  - Δt_1 = 1e-3 sec → 100 pts per cycle
  - Δt_2= 5e-4 sec → 200 pts per cycle
  - Δt_3= 2.5e-4 sec → 400 pts per cycle
  - Solution using Δt_2 and Δt_3 nearly identical
- Used 5 Newton sub-iterations
- Ran for 1 sec of physical time → 10 cycles
- Grid was rigidly rotated, not deformed

Case 1 Temporal Resolution Study

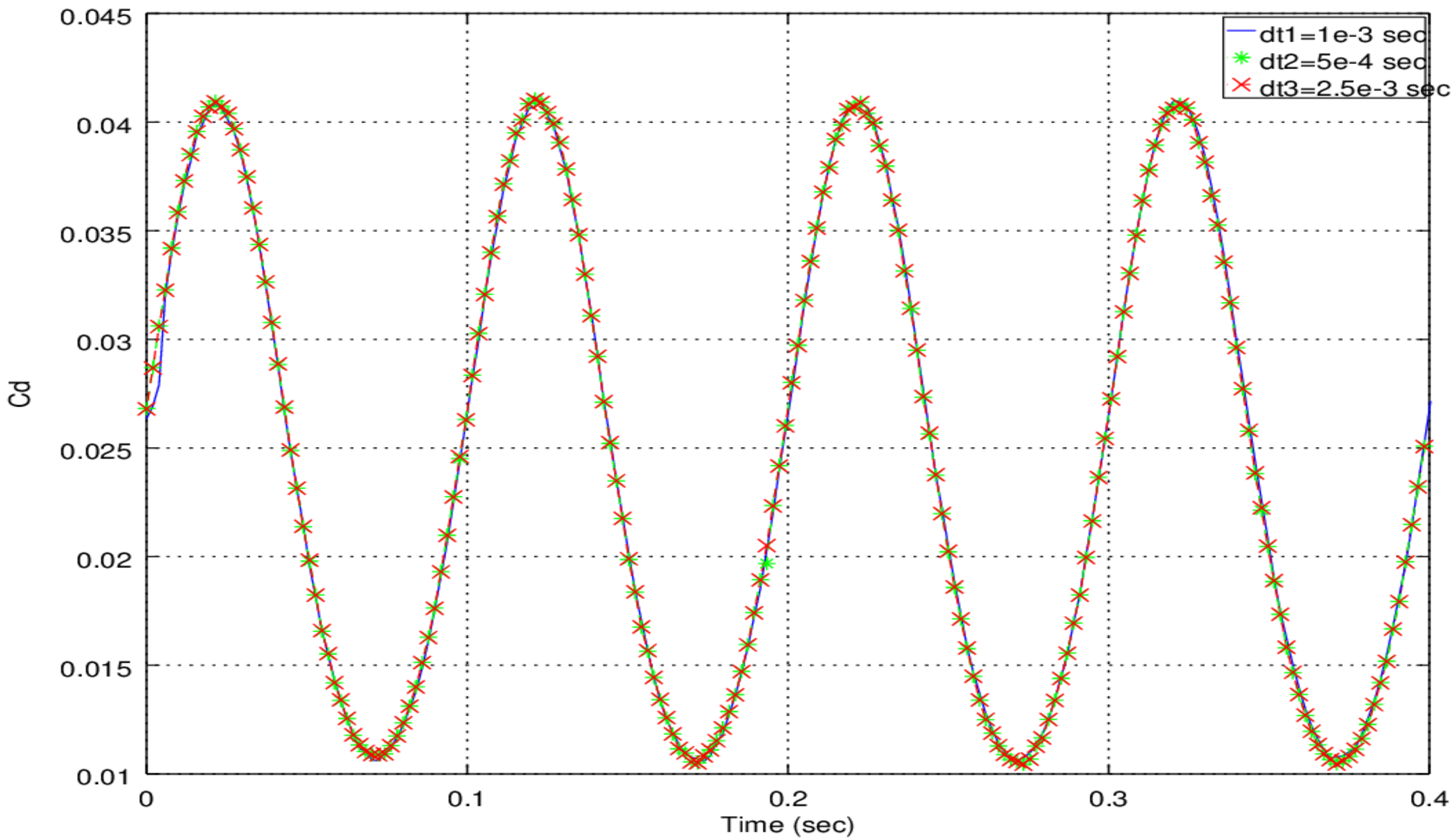
Coarse Grid: Lift Coefficient



ATA ENGINEERING, INC.

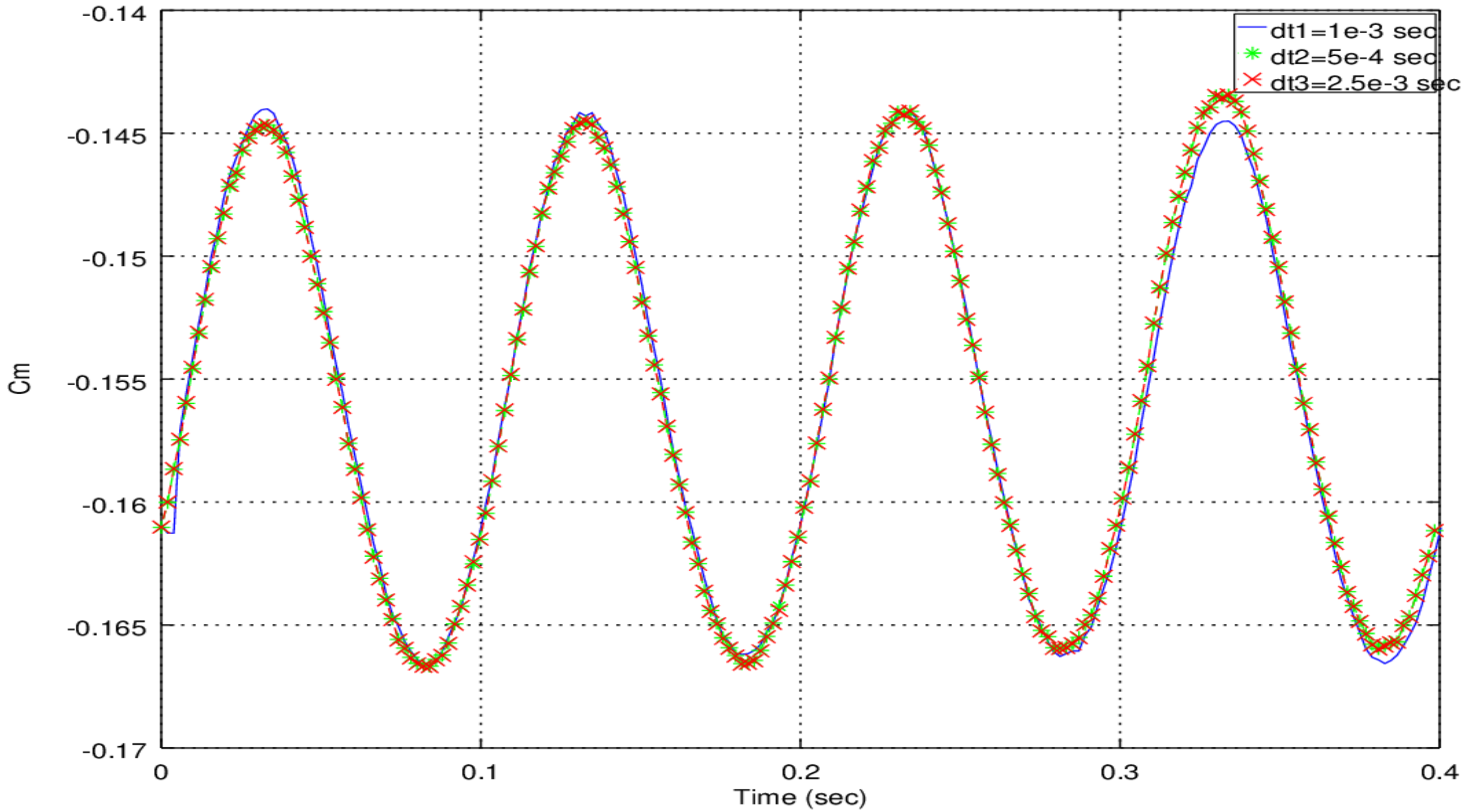
Case 1 Temporal Resolution Study

Coarse Grid: Drag Coefficient



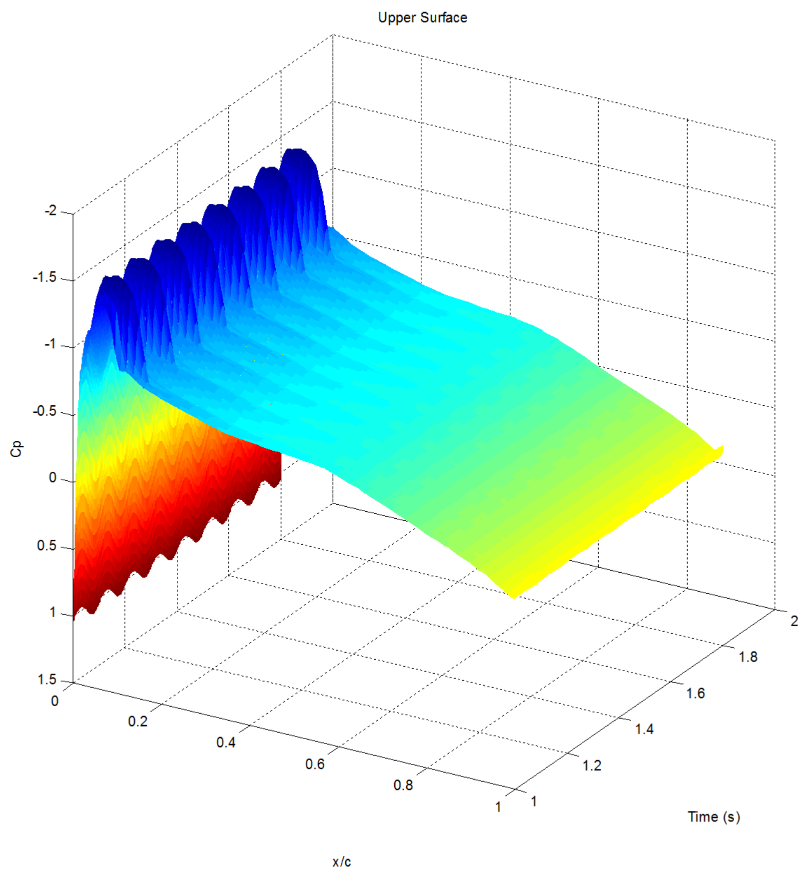
Case 1 Temporal Resolution Study

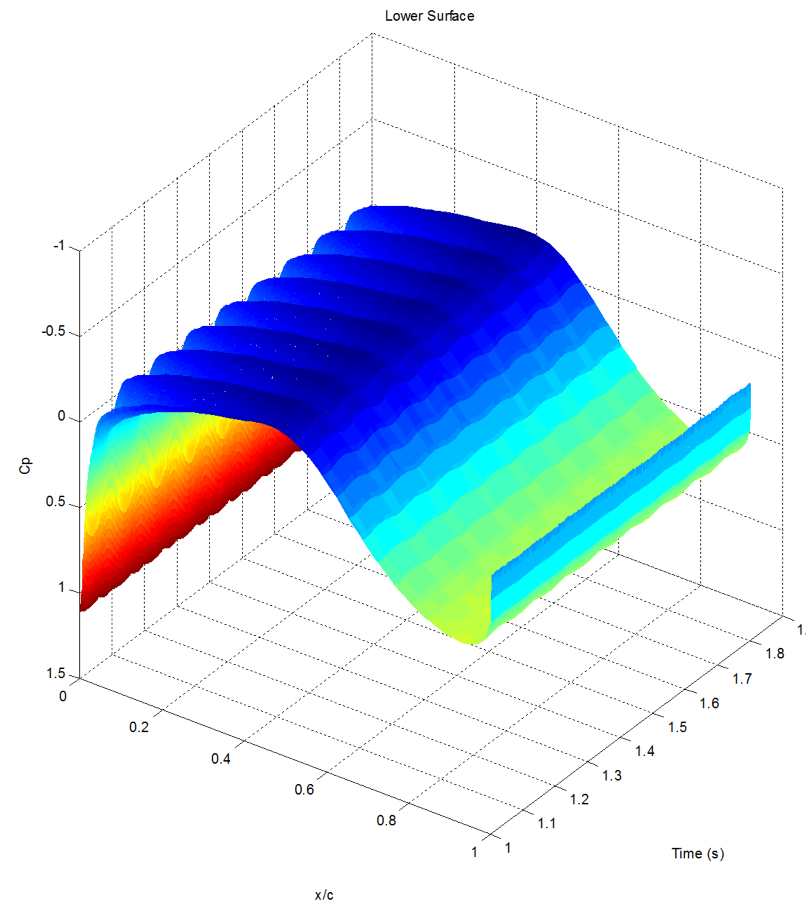
Coarse Grid: Pitching Moment Coefficient



Case 1 Results

Fine Grid

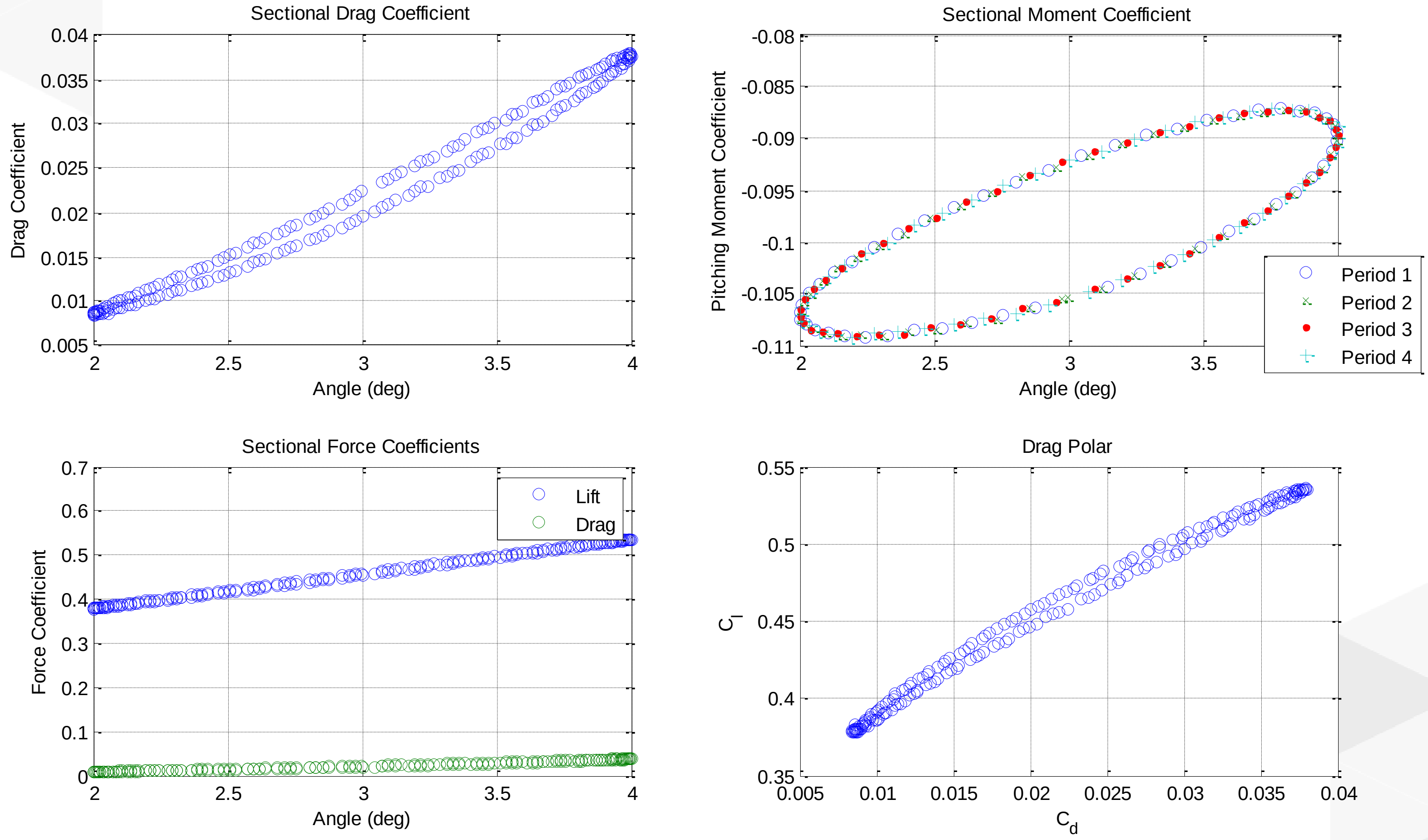




ATA ENGINEERING, INC.

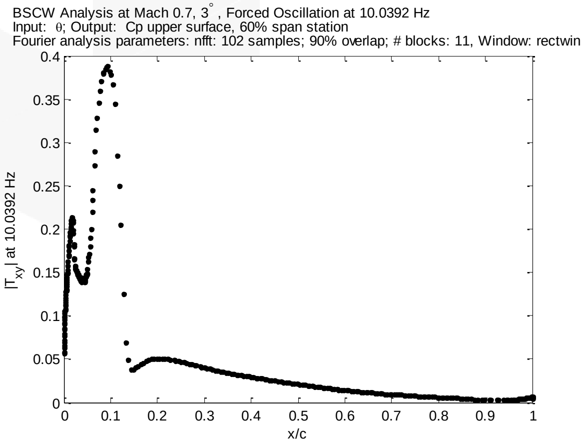
Case 1: Coefficients vs Angle of Attack

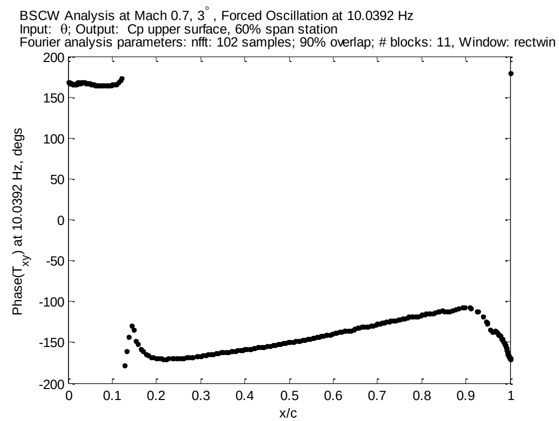
Fine Grid, 60 % Span

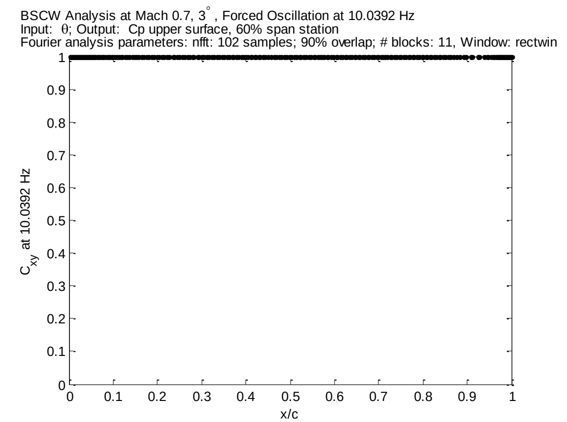


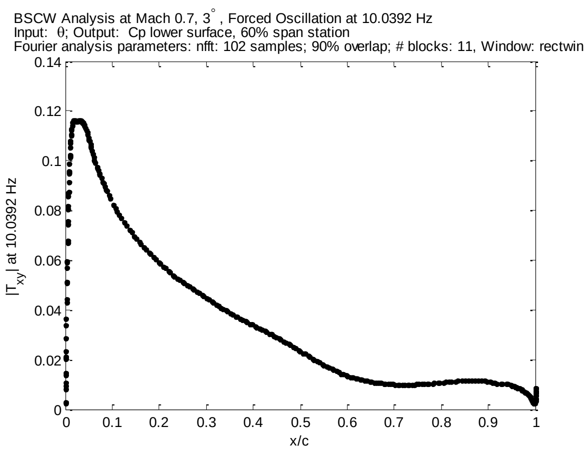
Case 1: FRFs

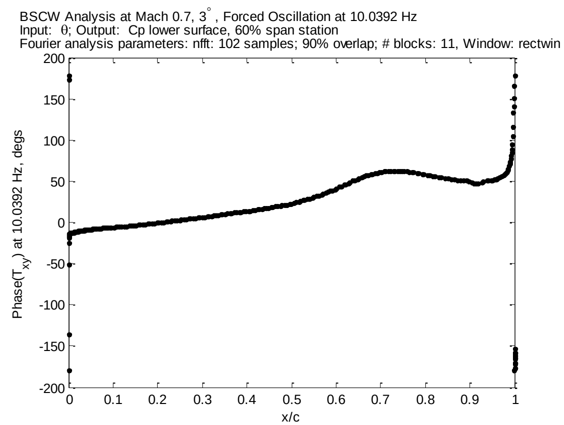
Fine Grid, 60 % Span, dt_1=1e-3

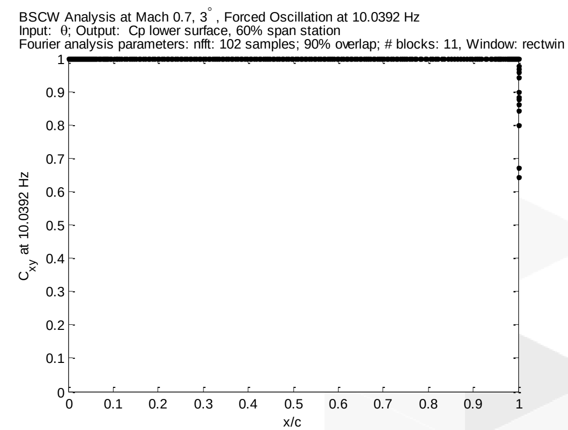






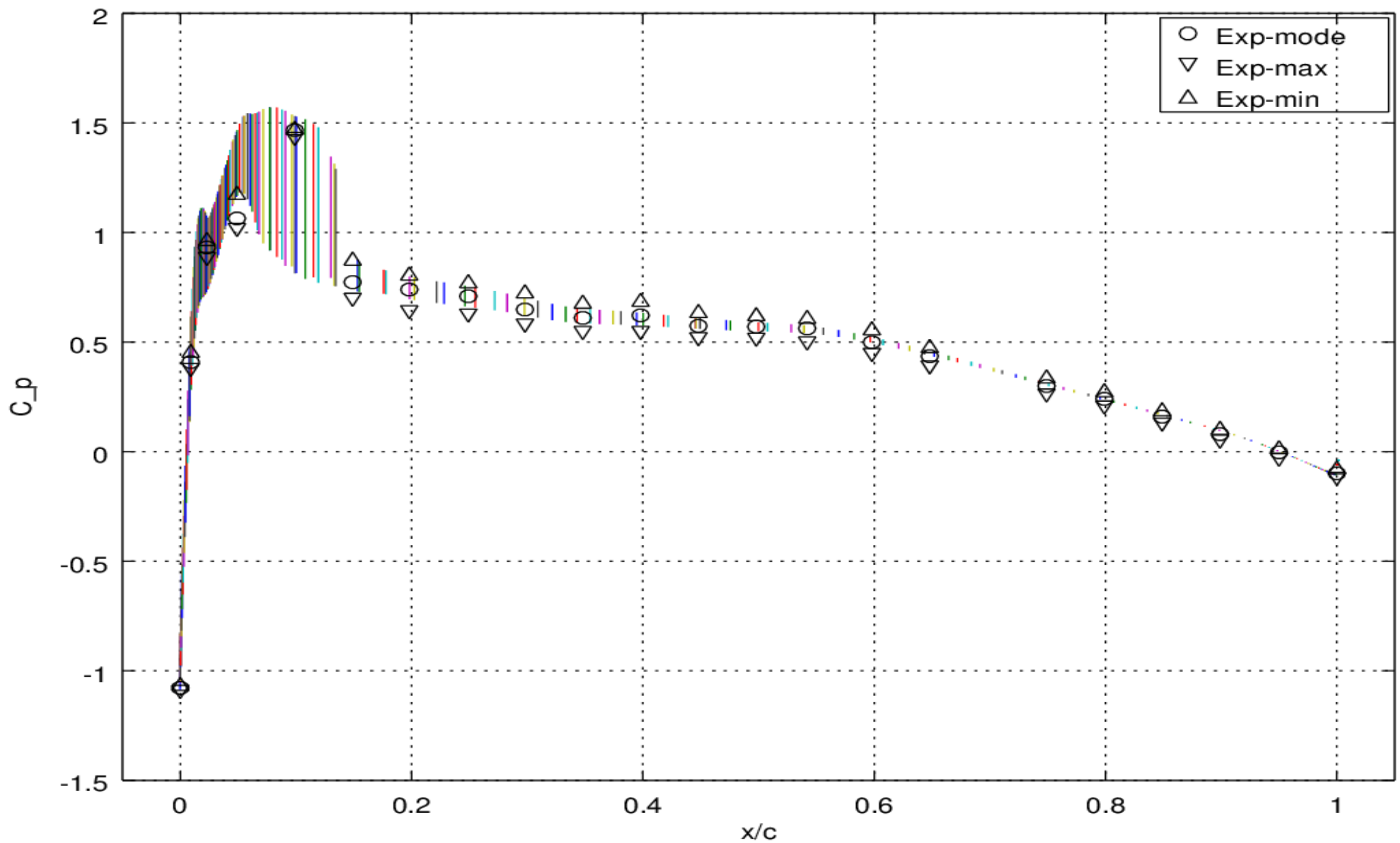






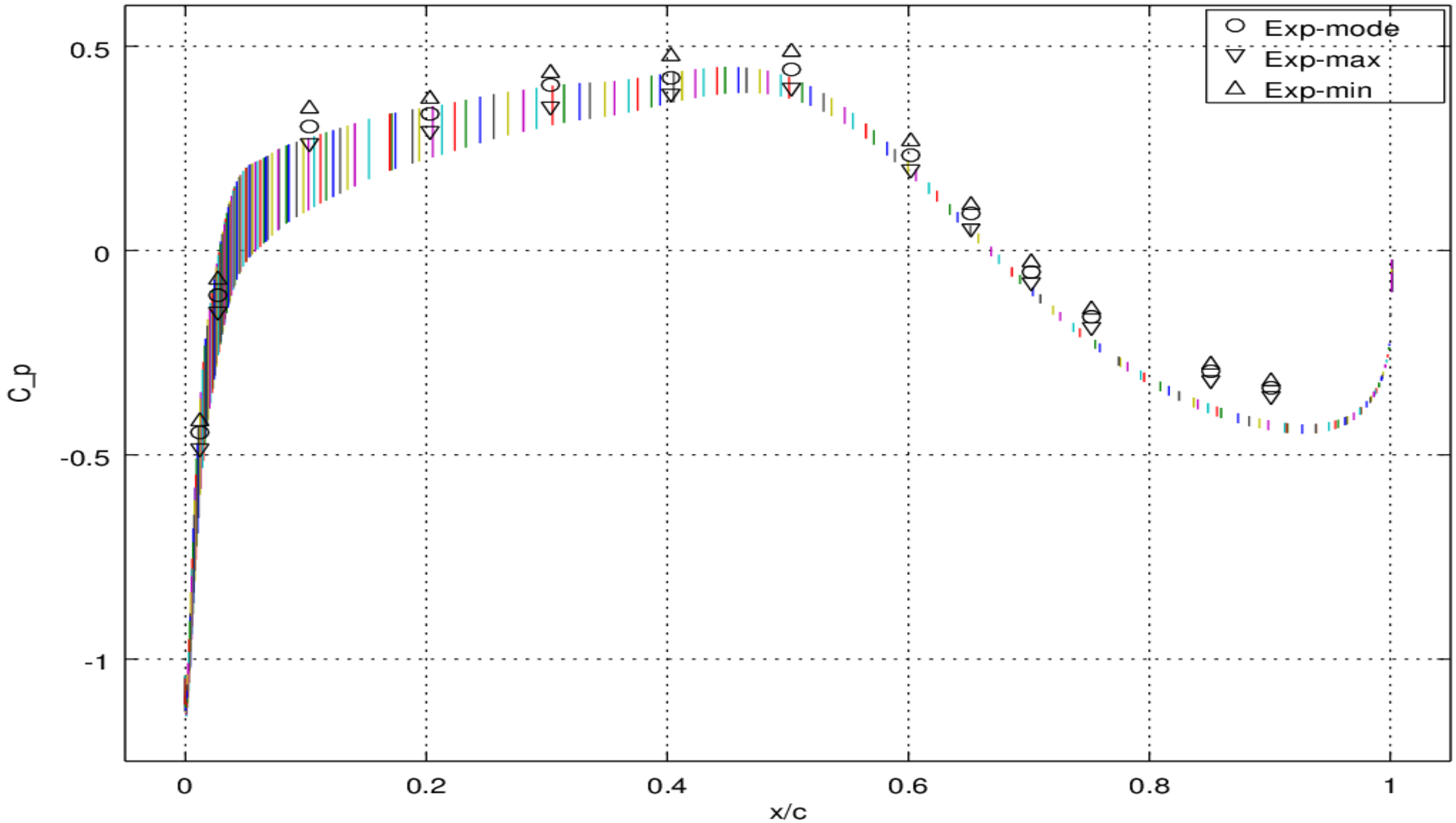
Unsteady Upper Surface Pressure

Medium Grid, 60% Span Location, dt_2=5e-4 sec



Unsteady Lower Surface Pressure

Medium Grid, 60% Span Location, dt_2=5e-4 sec

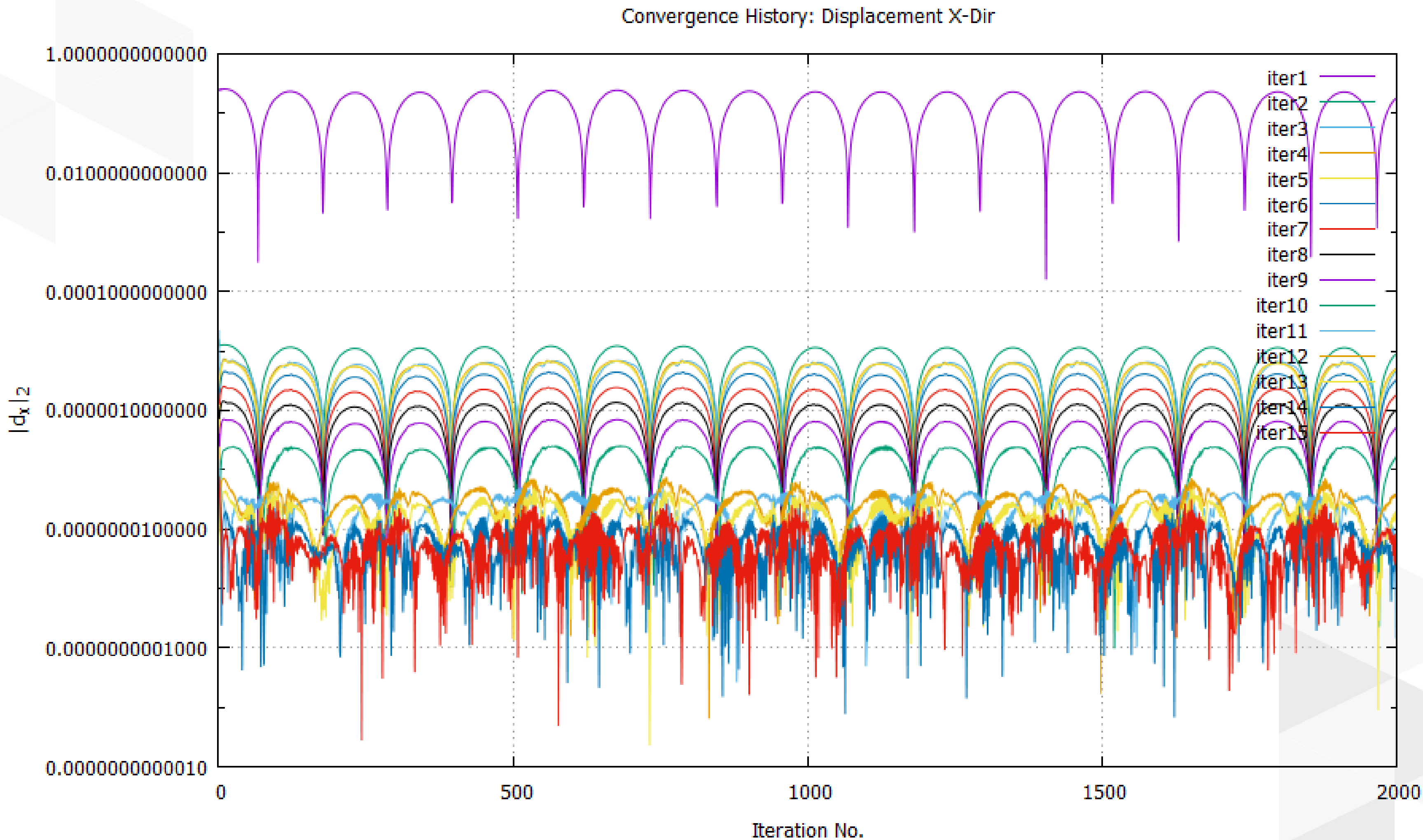


Case 2 Input Parameters

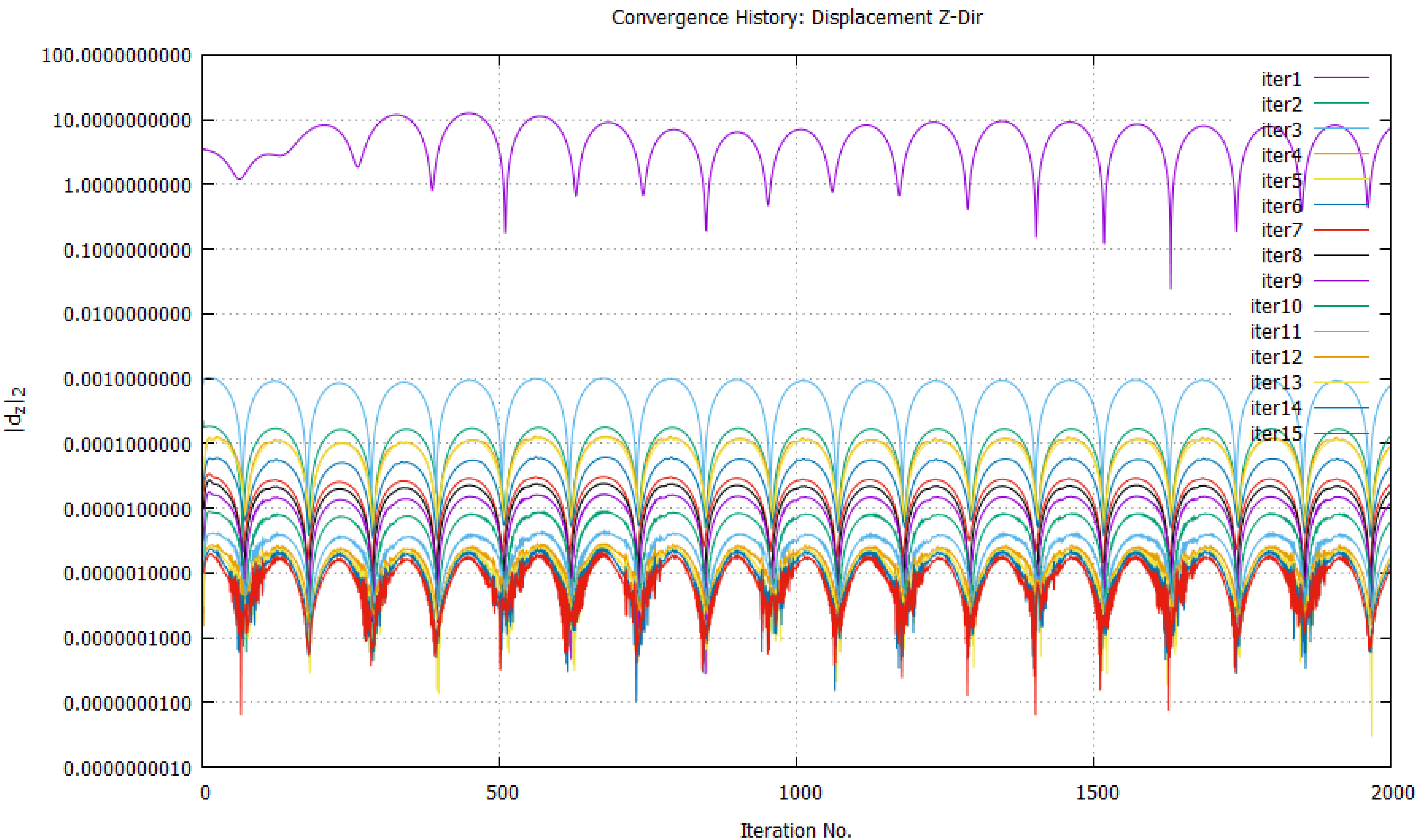
- Time step: Δt = 1e-3 sec
  - Approximately 220 pts per cycle (Strouhal number consideration)
  - Performed time step study on Case 1
    - Δt = 5e-4 sec adequate (200 pts per cycle) and similar flow condition
    - Should perform temporal study for Case 2
  - Balance between accuracy and resources
- Used strong coupling with 15 sub-iterations
  - Displacement converge 4 orders of magnitude
  - Force converge 3-4 orders of magnitude
- Duration: 4 sec → ~18 periods
- Initial conditions: converged rigid solution and perturbation to rotational velocity
- Mesh deformed, not moved rigidly

Sub-iteration Convergence History: dx

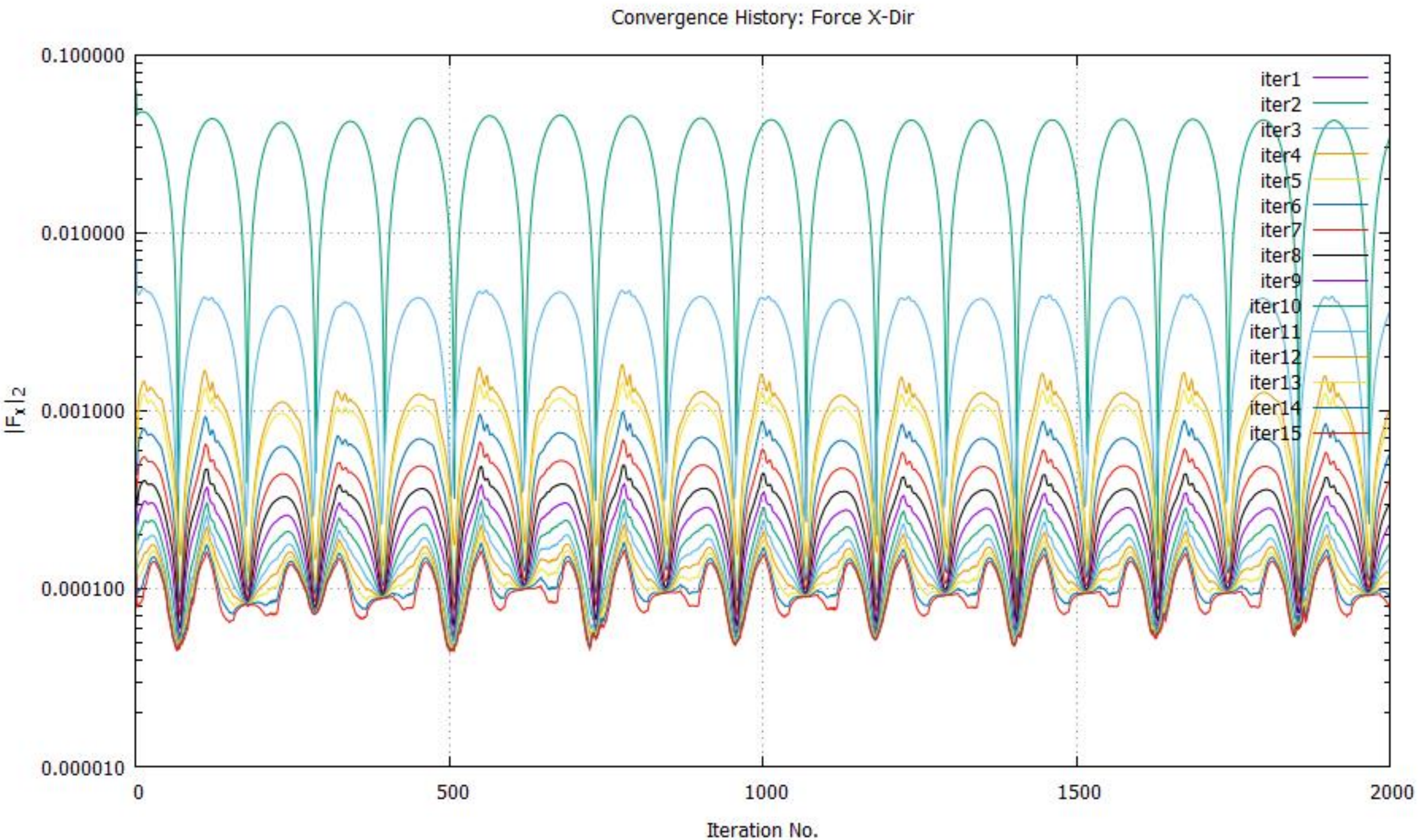
Case 2, Medium Grid, Flutter Condition



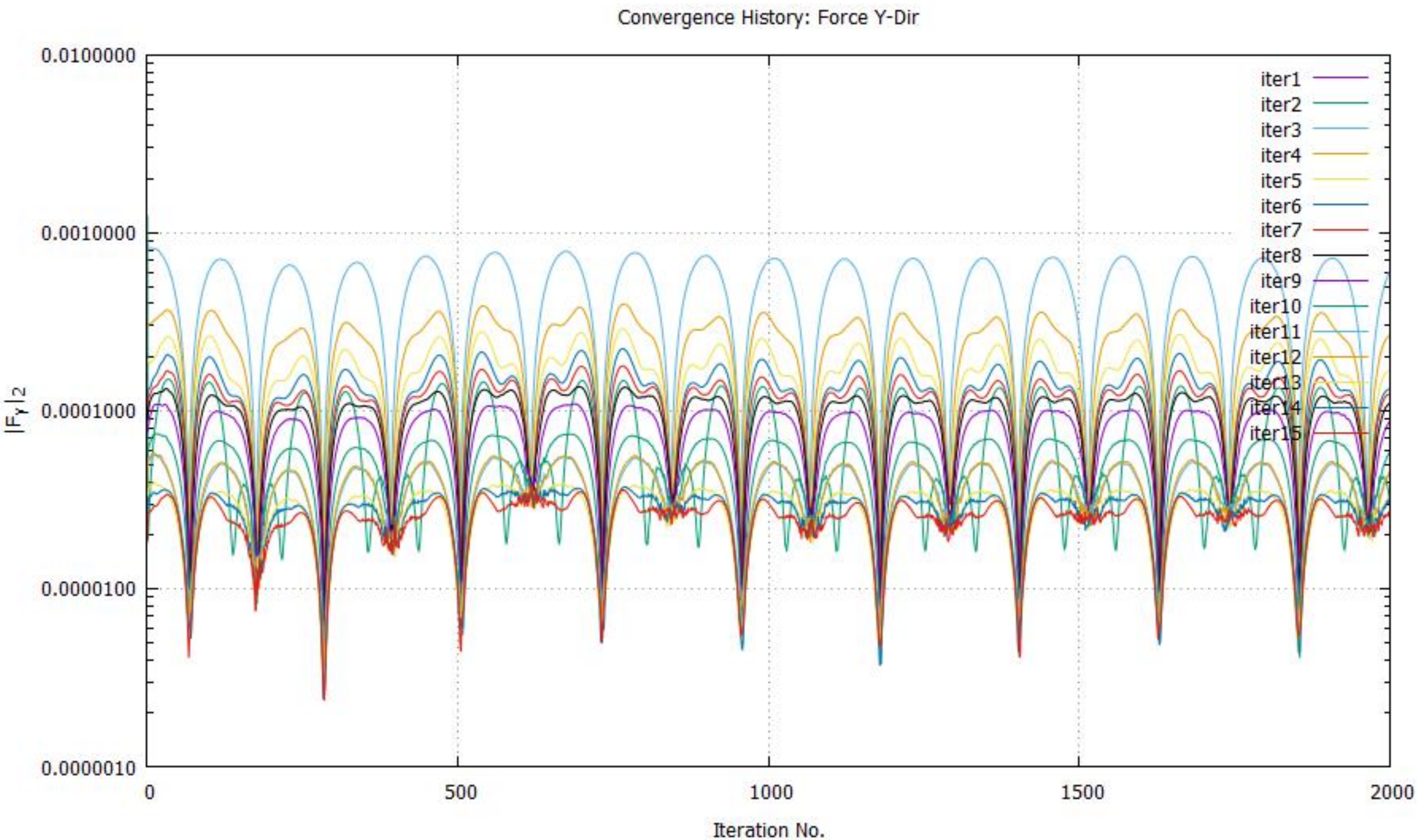
Sub-iteration Convergence History: dz



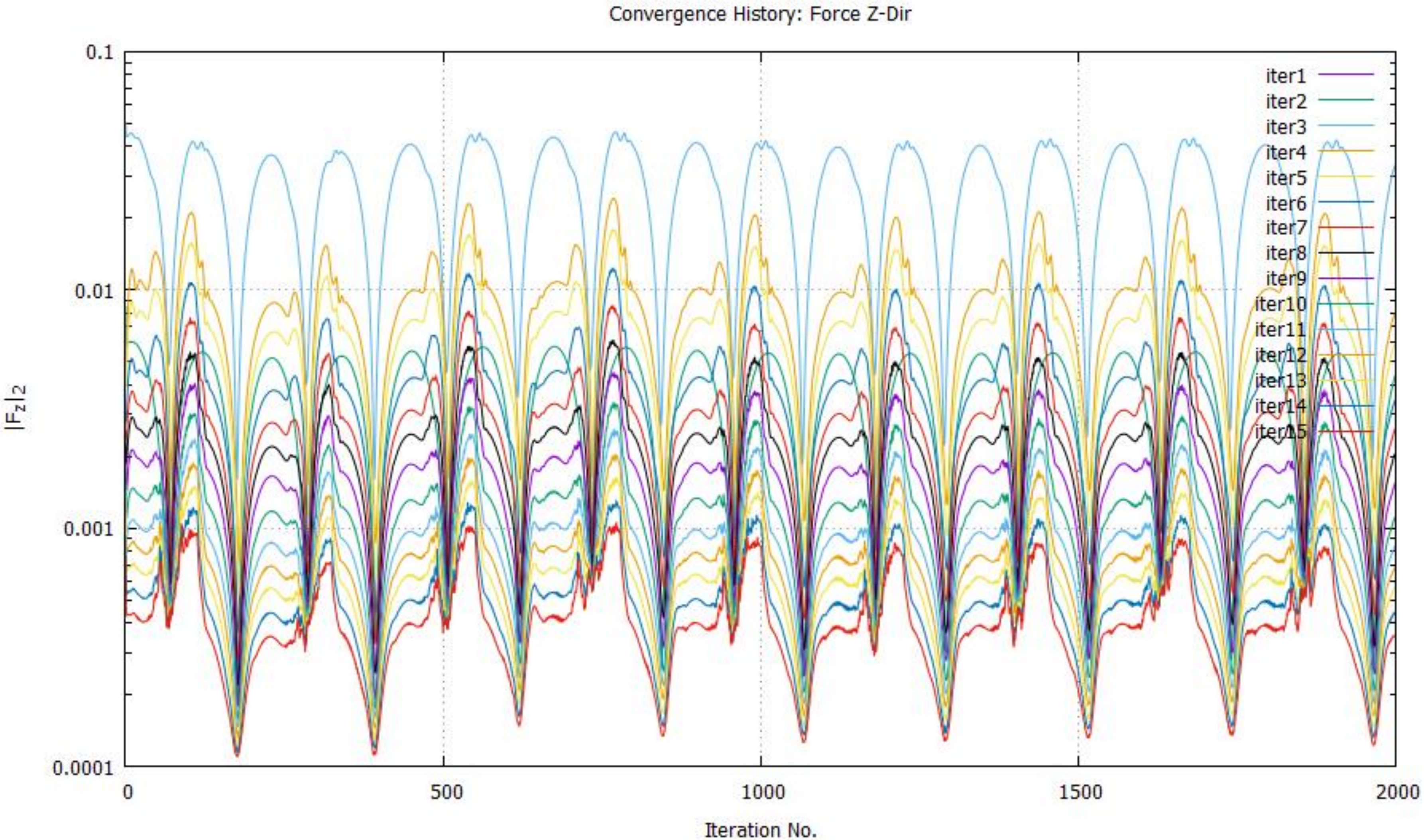
Sub-iteration Convergence History: Fx



Sub-iteration Convergence History: Fy



Sub-iteration Convergence History: Fz

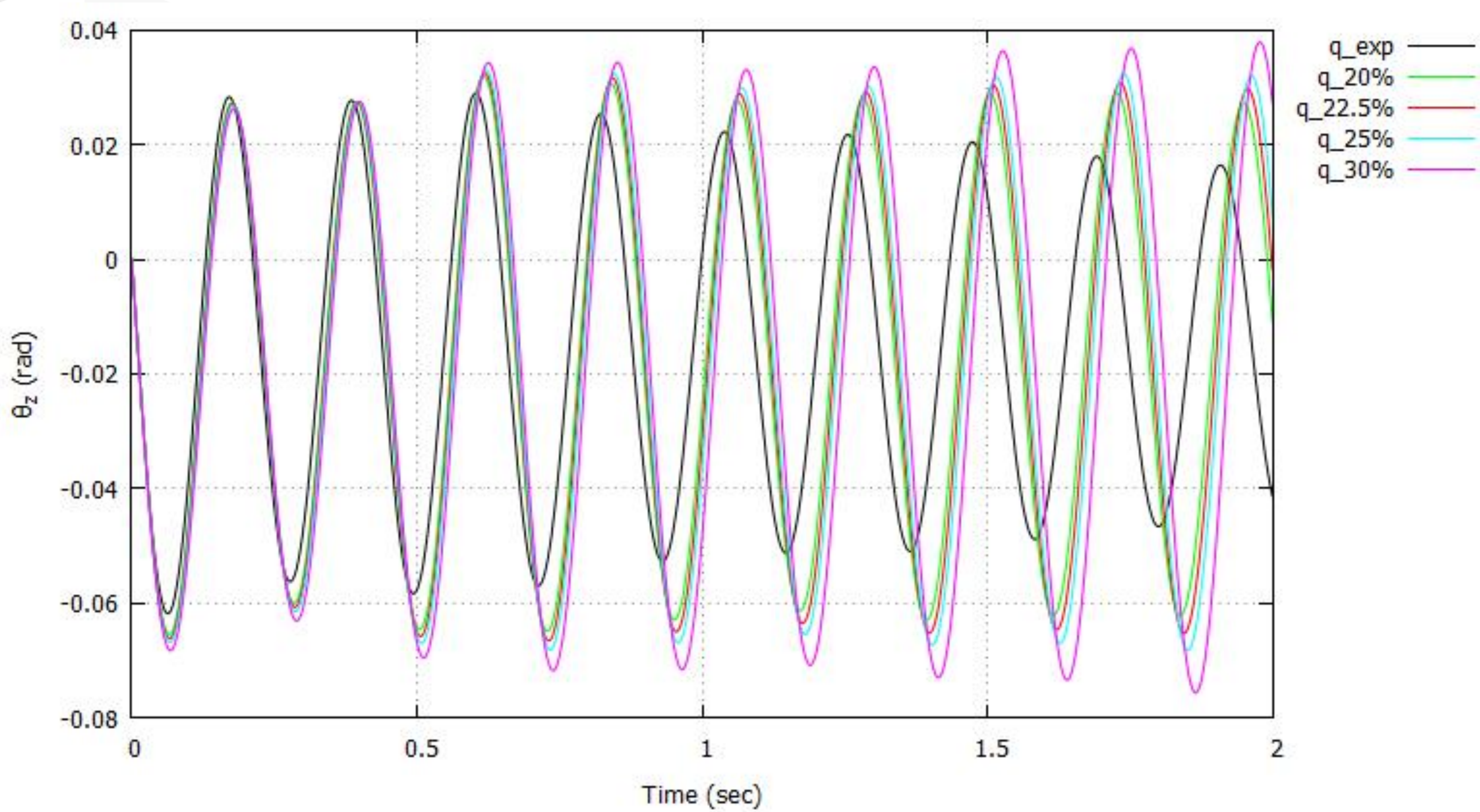


Case 2 Summary

	Experimental Condition F=4.3 Hz, Q=168.8 psf			Predicted Flutter Condition			
	Freq. (Hz)	% Error	Damping	Freq. (Hz)	% Error	Q (psf)	% Error
Coarse	4.63	7.67	0.83%	4.46	3.72	206.78	22.5
Medium	4.59	6.74	0.94%	4.46	3.72	202.56	20
Fine	4.57	6.28	0.91%	4.46	3.72	202.56	20

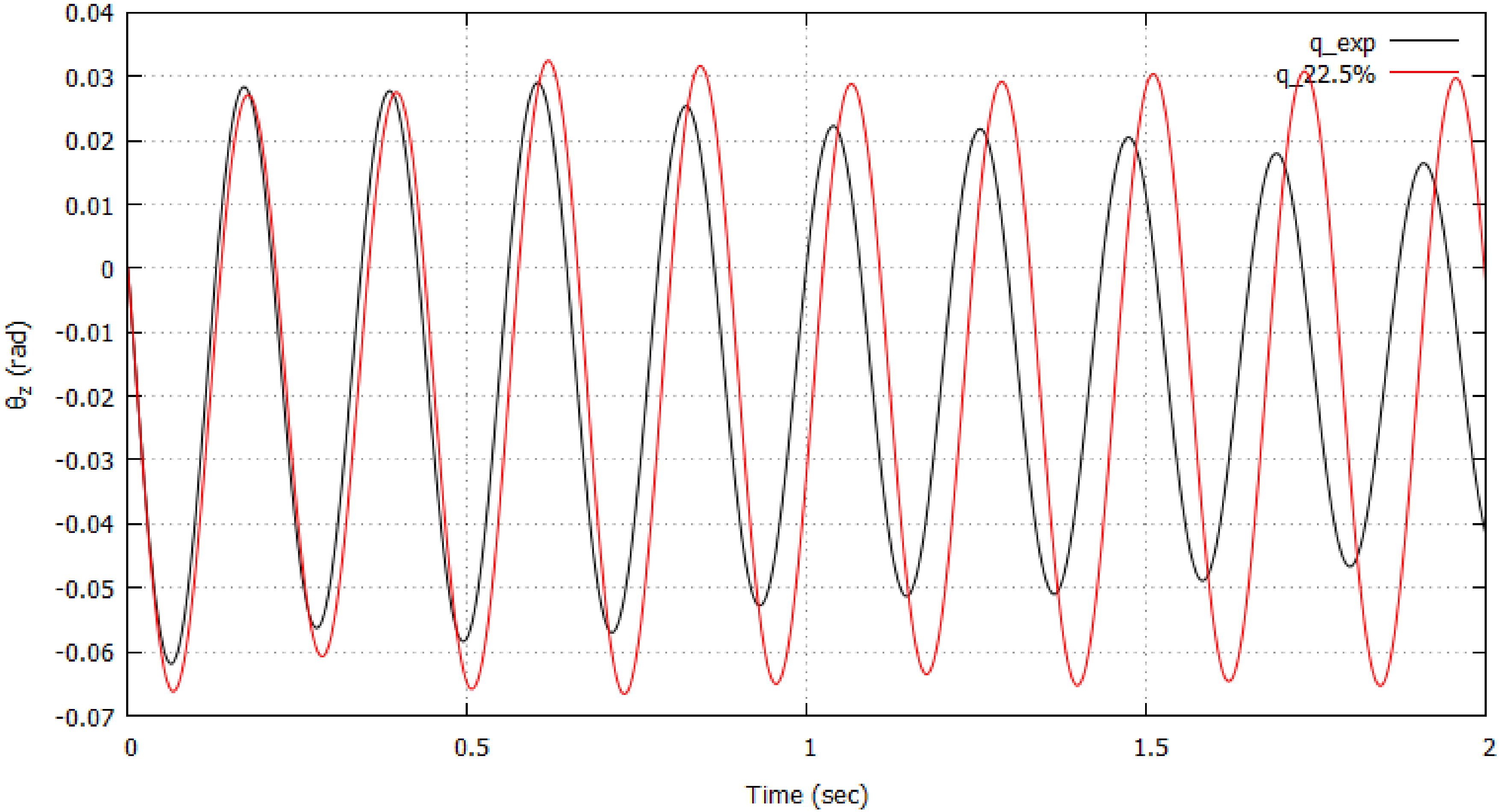
Case 2: Coarse Grid Pitch Response

Flutter Frequency = 4.45 Hz, Q_f=+122.5%



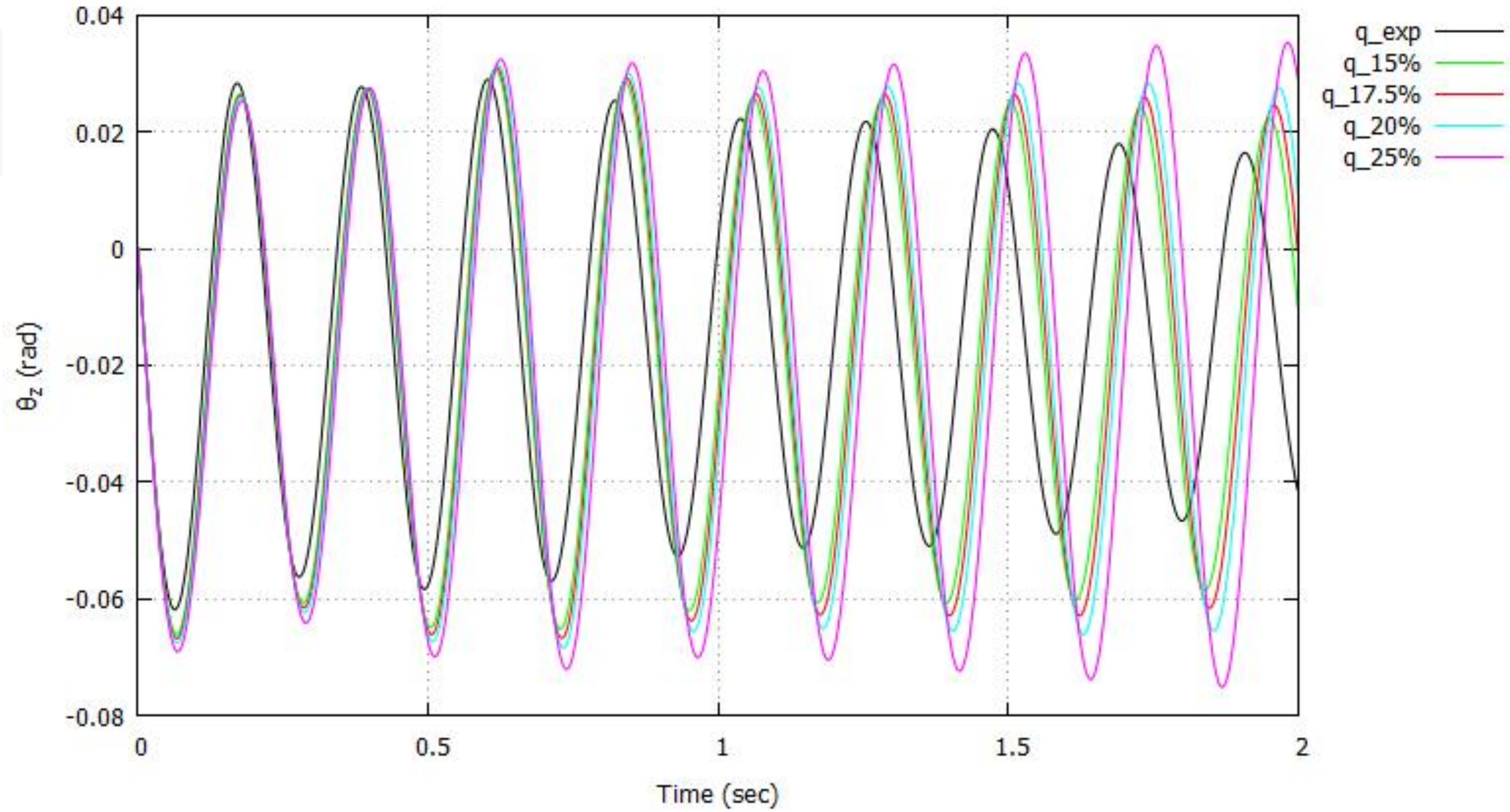
Case 2: Coarse Grid Pitch Response

Flutter Frequency = 4.45 Hz, Q_f=+122.5%



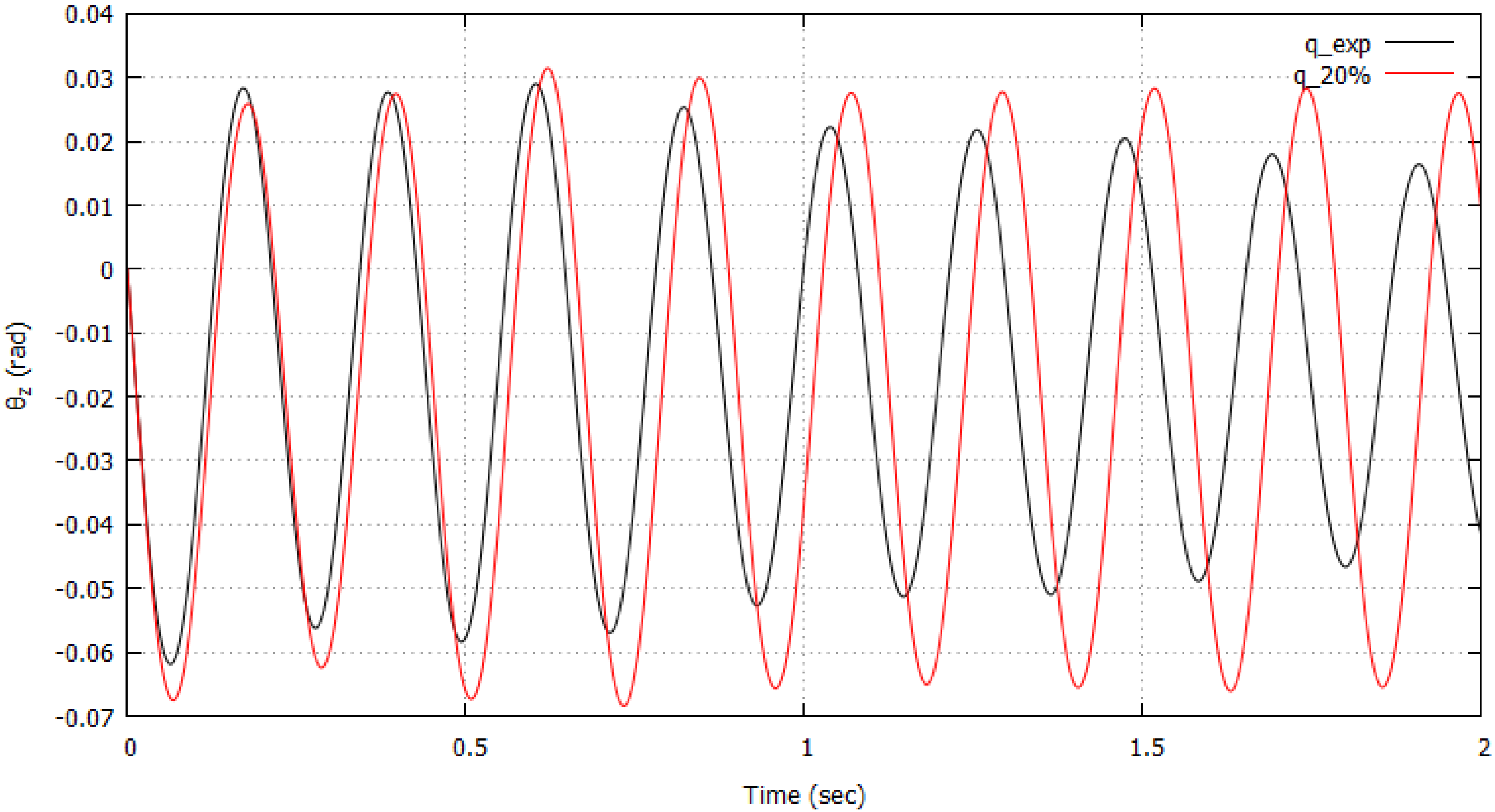
Case 2: Medium Grid Pitch Response

Flutter Frequency = 4.46 Hz, Q_f=+120%



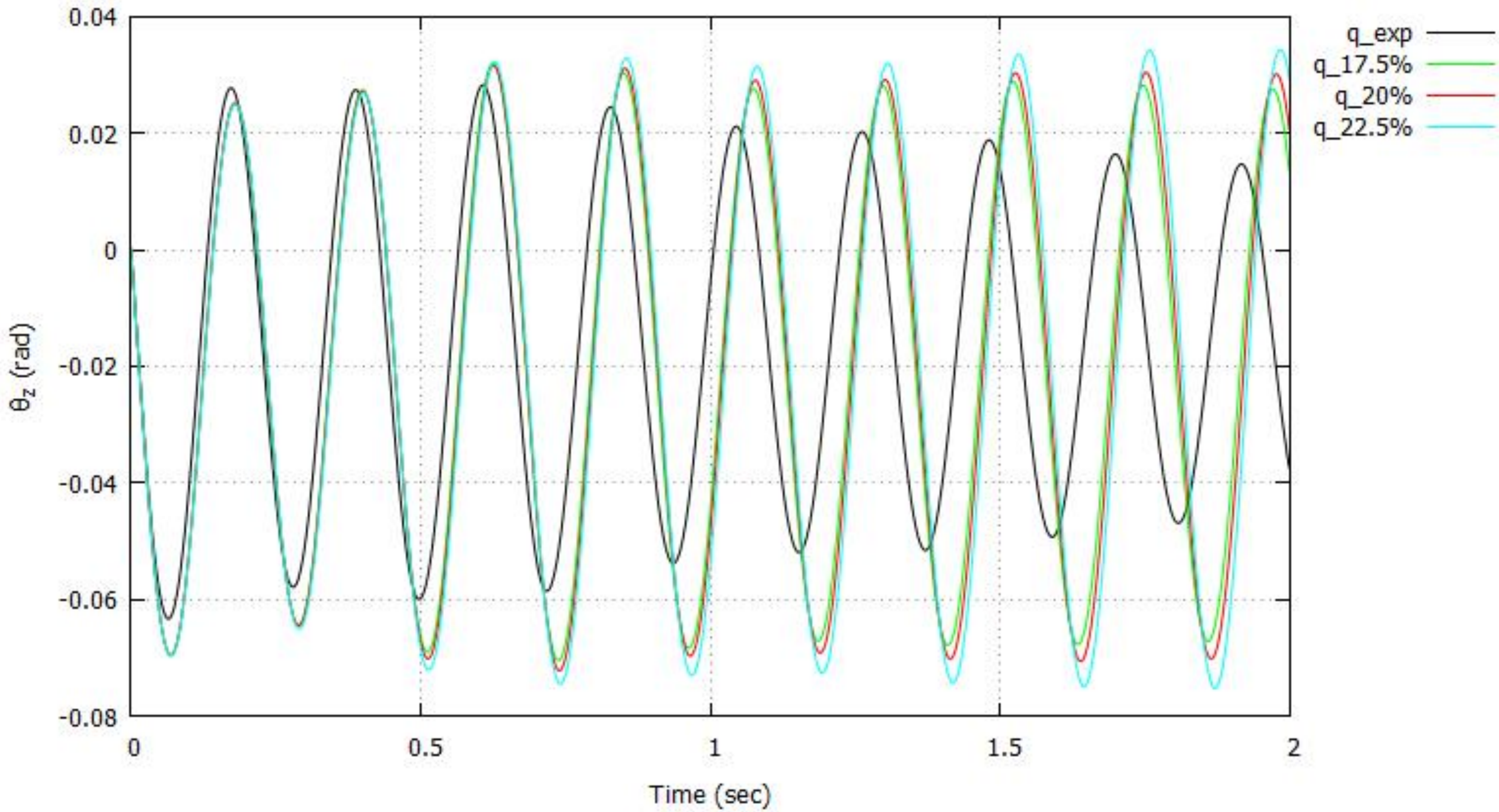
Case 2: Medium Grid Pitch Response

Flutter Frequency = 4.46 Hz, Q_f=+120%



Case 2: Fine Grid Pitch Response

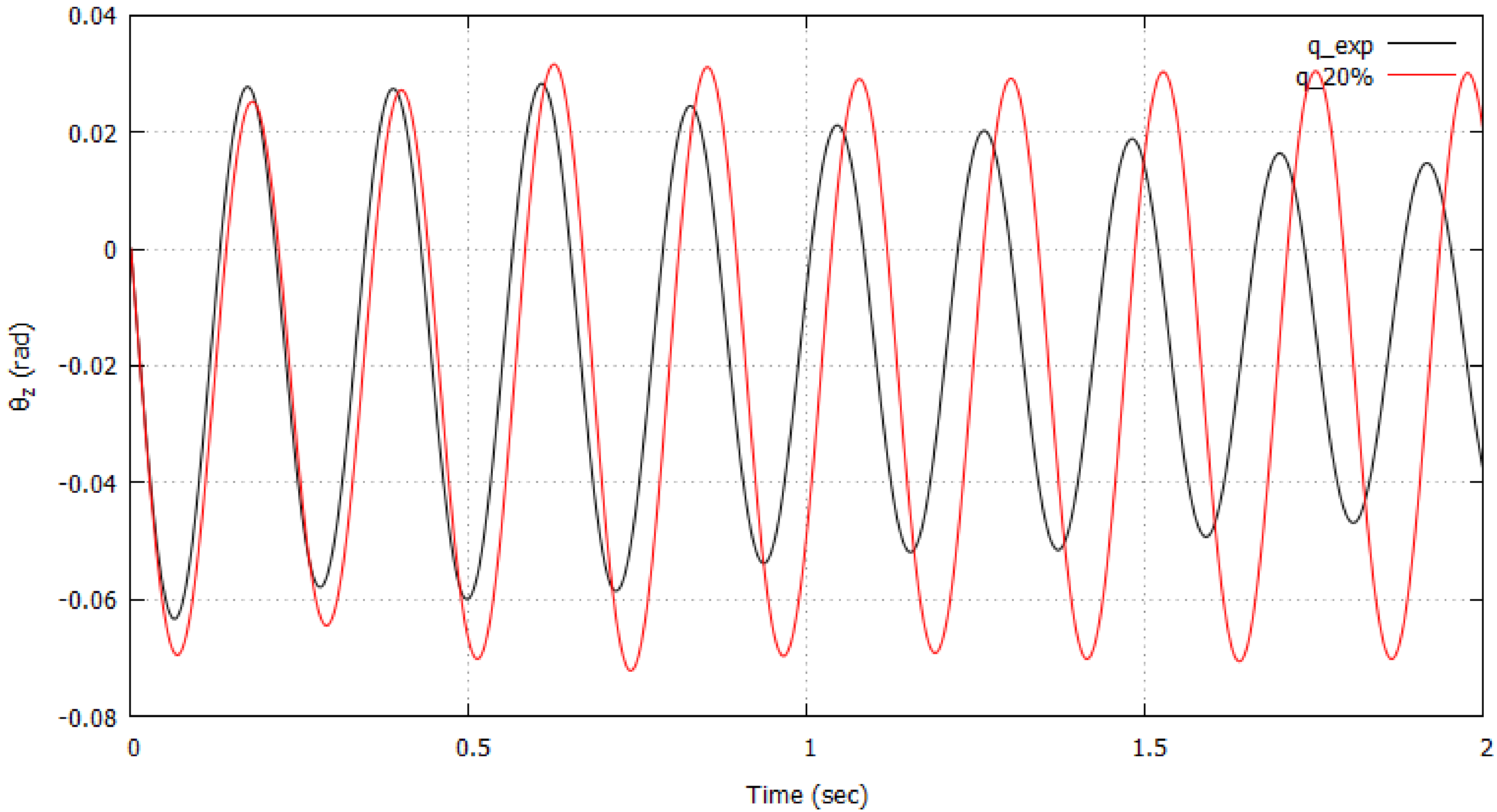
Flutter Frequency = 4.46 Hz, Q_f=+120%



ATA ENGINEERING, INC.

Case 2: Fine Grid Pitch Response

Flutter Frequency = 4.46 Hz, Q_f=+120%



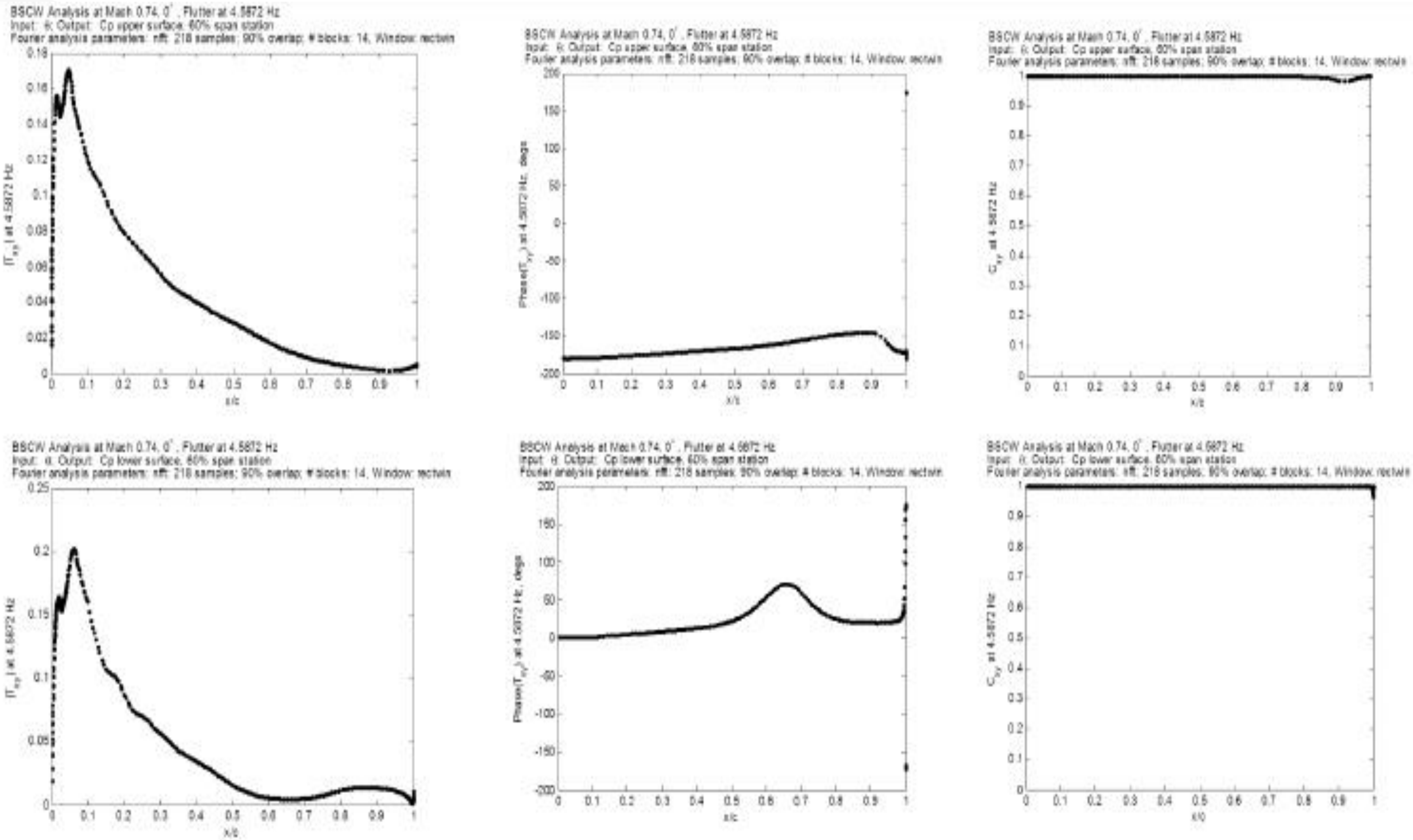
ATA ENGINEERING, INC.

Case 2 FRF Data

Fine Grid, 60% Span

Upper Surface

Lower Surface



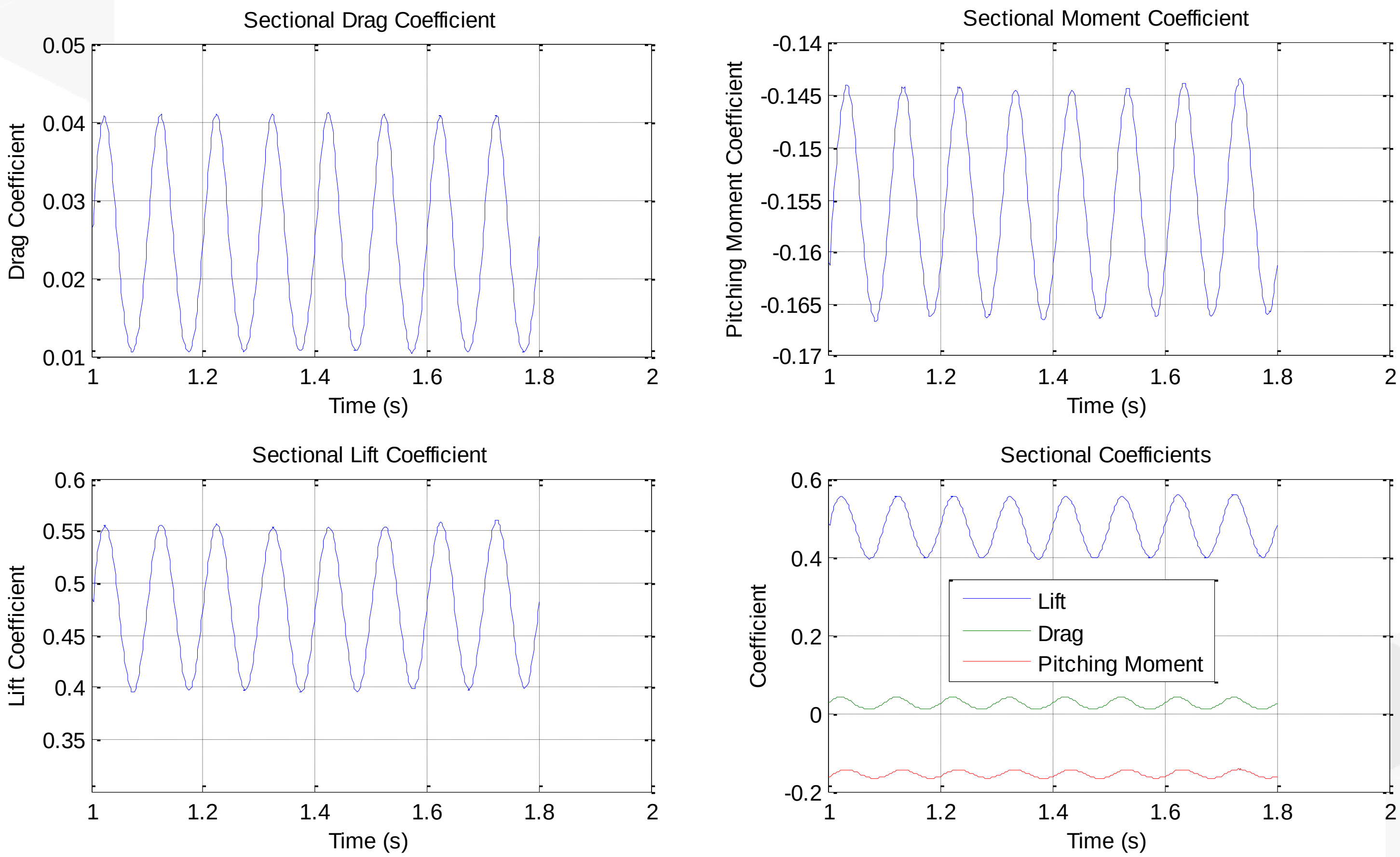
Summary

- Case 1: Unsteady pressure within experimental bounds
- Case 2: Flutter predictions estimate flutter at higher dynamic pressure (~20% higher)
  - Still investigating why
  - Generate own grid to resolve near wing region
  - Repeat without limiter
  - Need to conduct temporal study for Case 2
  - Need to complete steady/unsteady comparisons to experiment
- Coupling example: Strong coupling allows for larger time step than weak coupling

Questions?

Case 1: Coefficient Time Histories

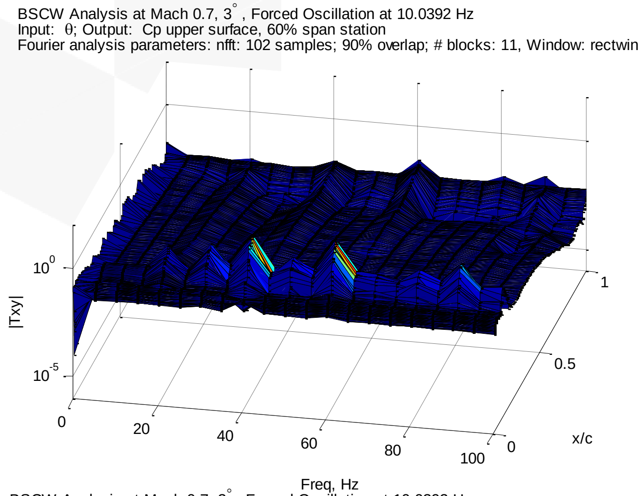
Fine Grid, 60 % Span

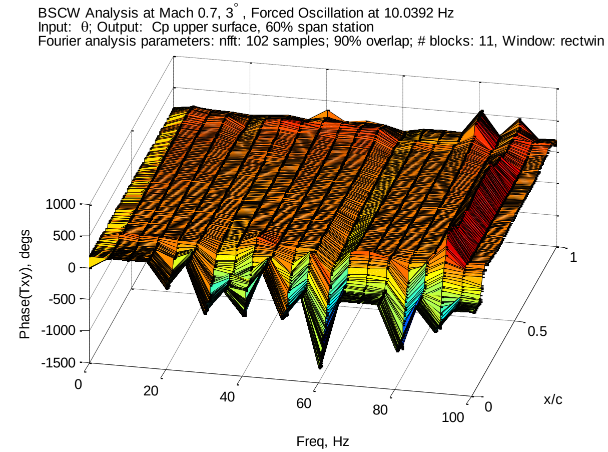


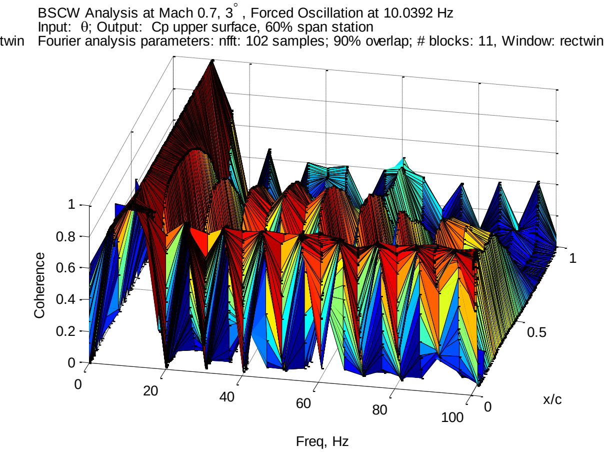
ATA ENGINEERING, INC.

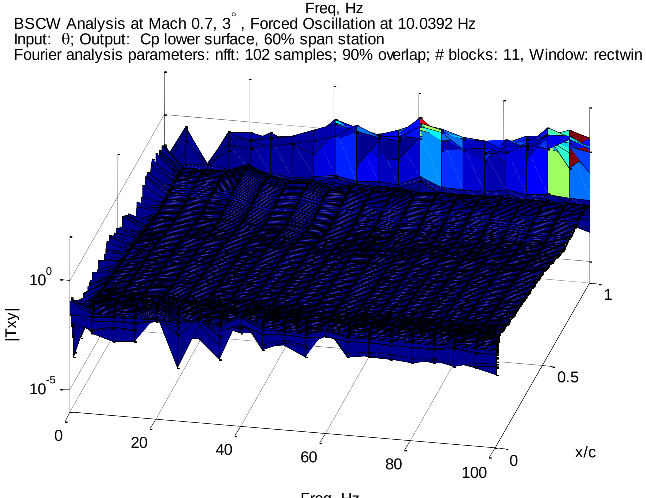
Case 1: FRFs

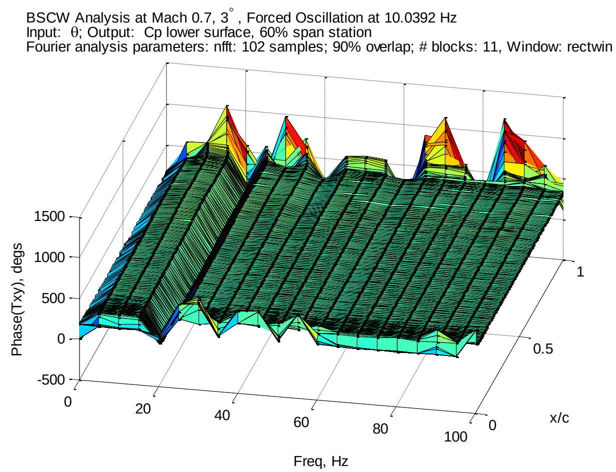
Fine Grid, 60 % Span

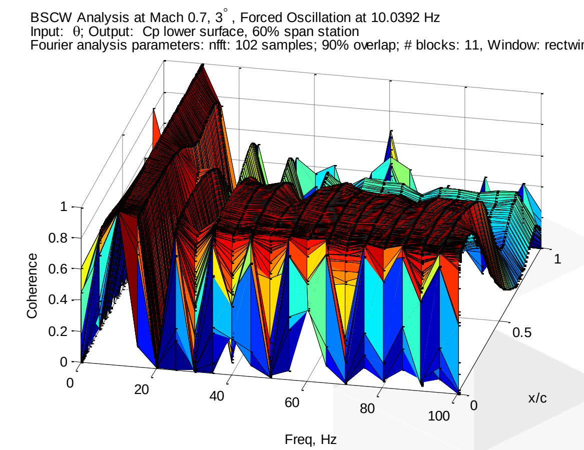












ATA ENGINEERING, INC.

NACA 64A010 Airfoil Flutter Case

➢Equations of motion

$$m\ddot{h} + S_\alpha \ddot{\alpha} + K_h = -L$$

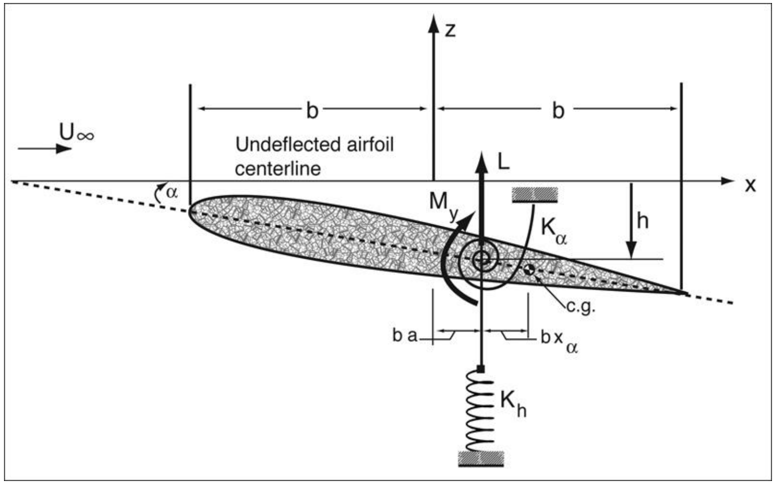
$$S_\alpha \ddot{h} + I_\alpha \ddot{\alpha} + K_\alpha = M_{ea}$$

$$[M] = \begin{bmatrix} 1 & x_\alpha \\ x_\alpha & r_\alpha^2 \end{bmatrix} \qquad [K] = \begin{bmatrix} \left(\frac{\omega_h}{\omega_\alpha}\right)^2 & 0 \\ 0 & r_\alpha^2 \end{bmatrix}$$

$$\mu = \frac{m}{\pi \rho b^2} = 60 \qquad a=-2.0$$

$$r_\alpha = \sqrt{I_\alpha / mb^2} = \sqrt{3.48} \qquad x_\alpha = S_\alpha / mb = 1.8$$

$\omega_h = \omega_\alpha = 100$ rad/sec



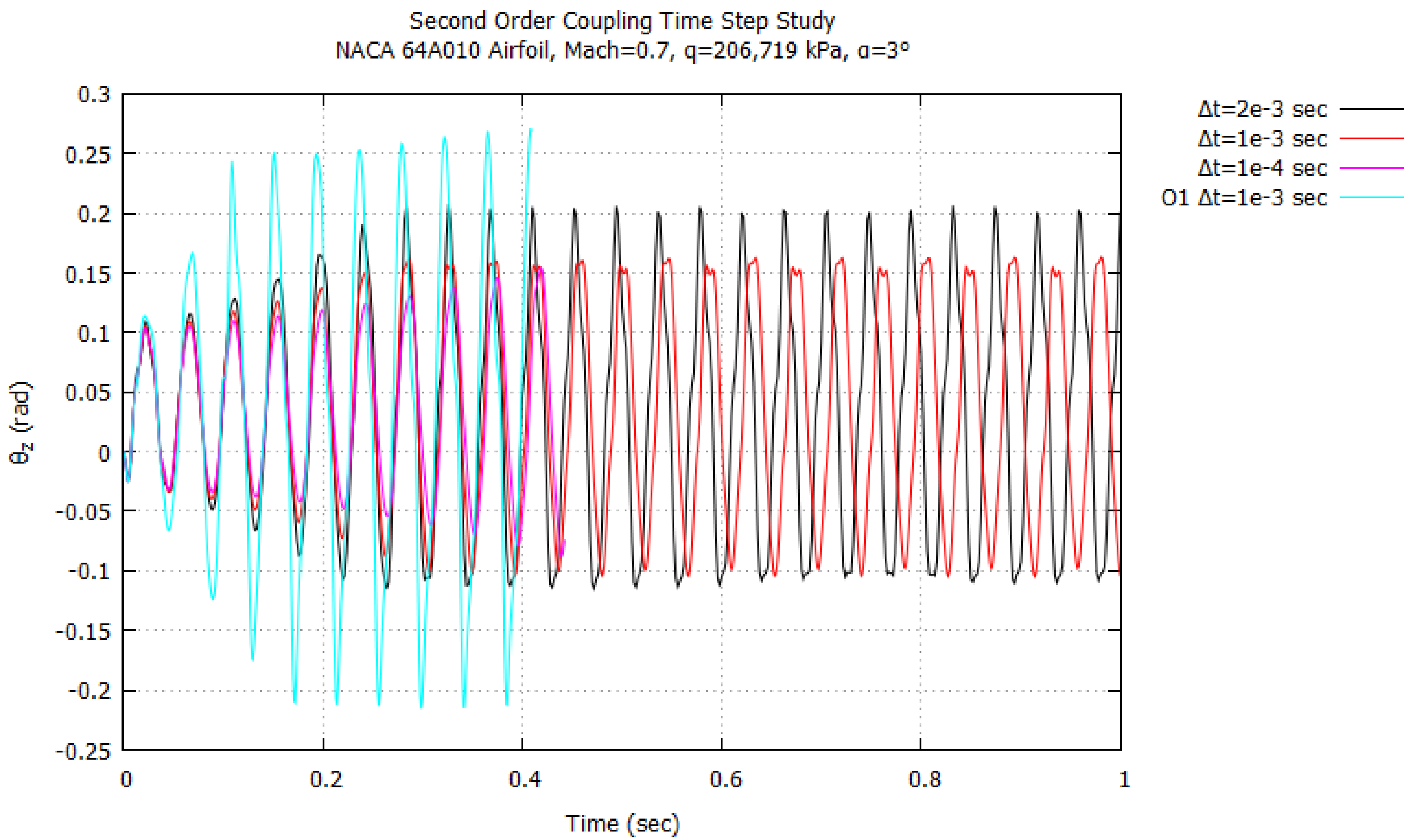
Exhibits a dramatic reduction in the flutter speed in the transonic region

Pertinent Questions for FSI Simulations

- What is appropriate time step?
- When/how frequently to couple/exchange information?
  - How many sub-iterations are required?
- How to perturb the system?
  - Gust
  - Initial rotational velocity
  - Apply impulse force/moment to structure
    - Magnitude
    - Duration

Coupling Scheme Comparison

Neutrally Stable Condition



Case 1: Unsteady Pressure Slices at 60% span

Predicted shock location is forward of experiment