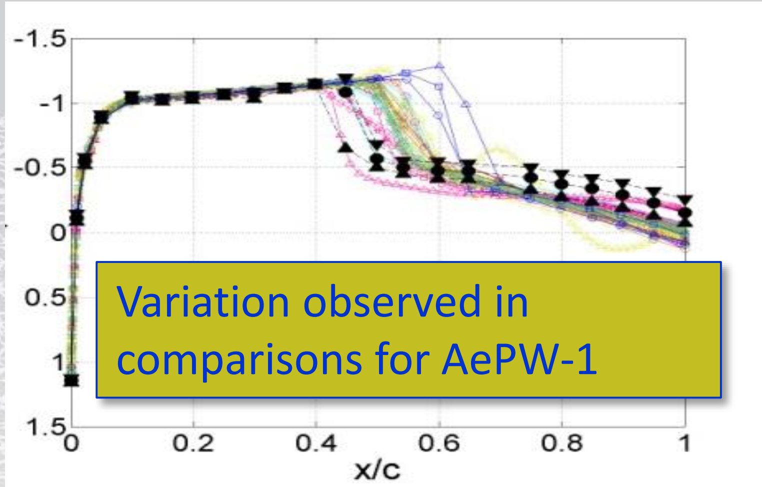
2nd AIAA Aeroelastic Prediction Workshop

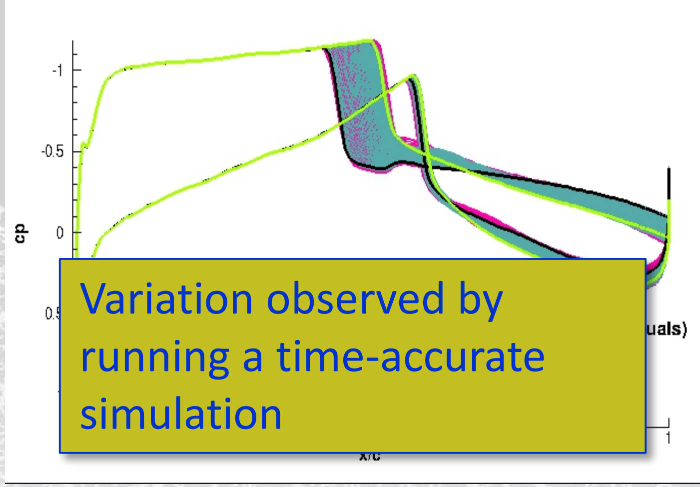
BSCW Analyses: Temporal Effects

Jennifer Heeg & Pawel Chwalowski
NASA Langley Research Center
January 2, 2016

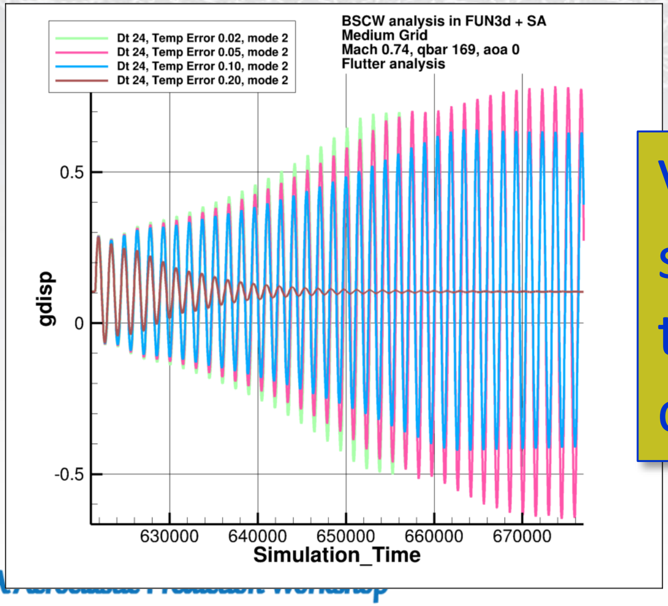
Issues included in this discussion

- Aerodynamic calculations at transonic conditions
  - Time-accurate solution of unforced system cases
  - Variation in unforced distribution due to analyst choice and/or post-processing





- Aeroelastic simulations
  - Time domain simulation guidelines from literature
  - Convergence experiences with BSCW



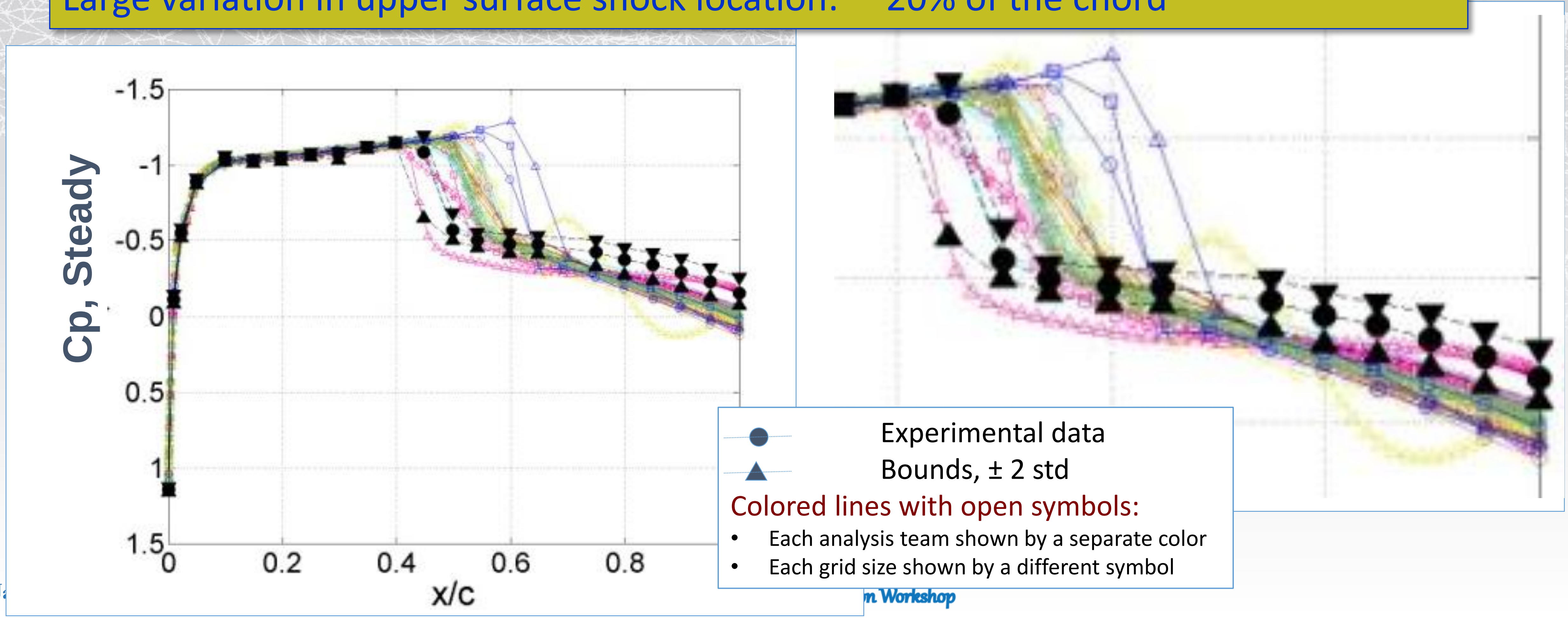
Variations in flutter simulations based on temporal parameter choices

Issues

- Aerodynamic calculations at transonic conditions
  - Time-accurate solution of unforced system cases
  - Variation in unforced distribution due to analyst choice and/or post-processing
- Aeroelastic simulations
  - Time domain simulation guidelines from literature
  - Convergence experiences with BSCW

Revisiting some points from the first workshop: Steady pressure distribution results

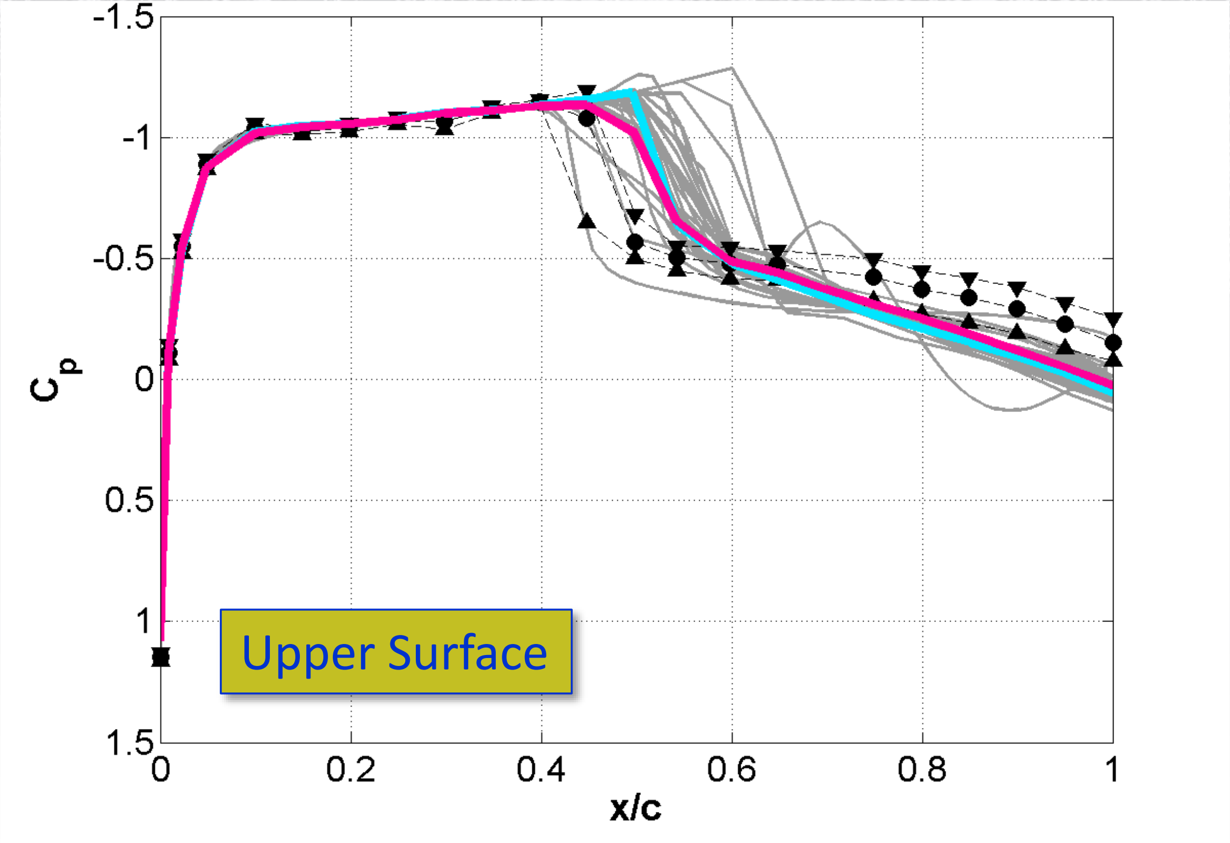
Values submitted were either the last point from steady non-time-accurate analysis
OR the mean value from the forced oscillation case
Large variation in upper surface shock location: ~ 20% of the chord

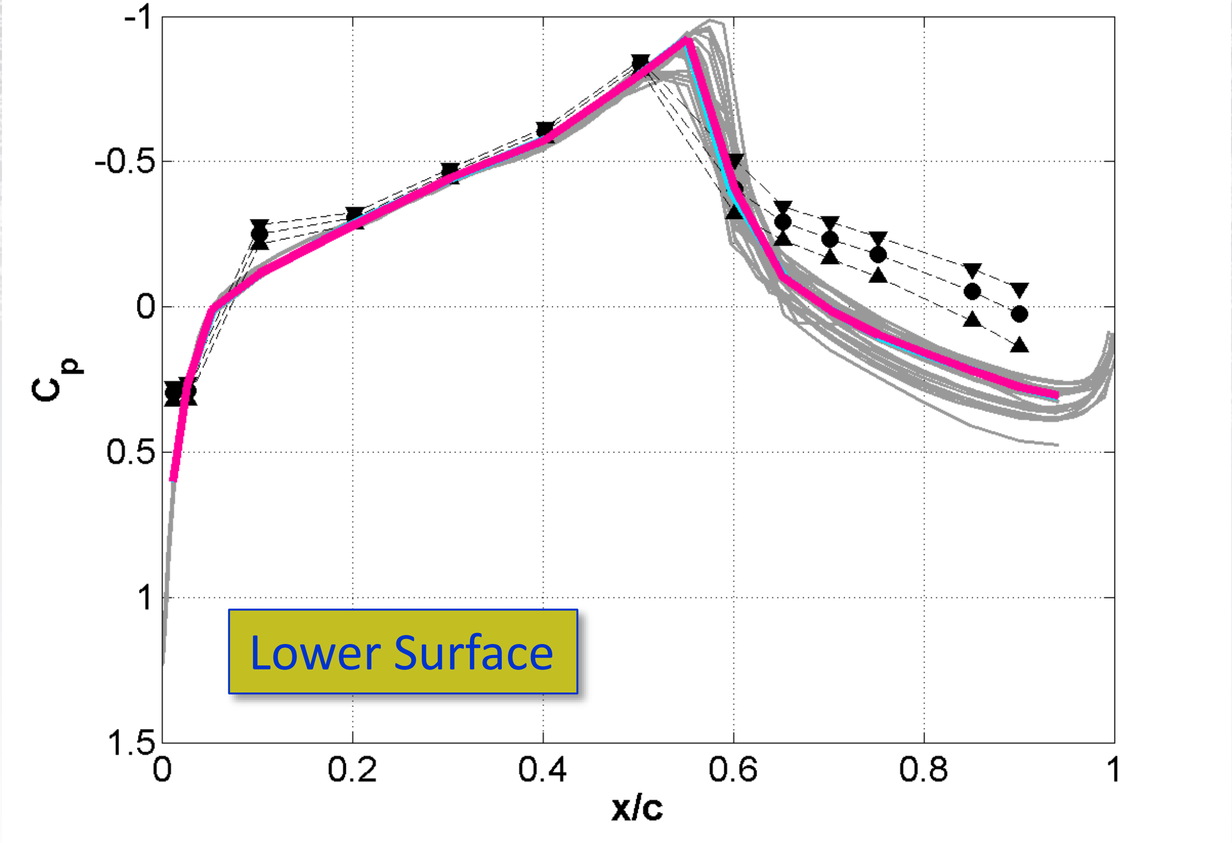


Revisiting some points from the first workshop: Steady pressure distribution results

The medium grid analysis results generated using FUN3D are highlighted below

Last point of the steady analysis
Mean value of the 1 Hz forced oscillation case

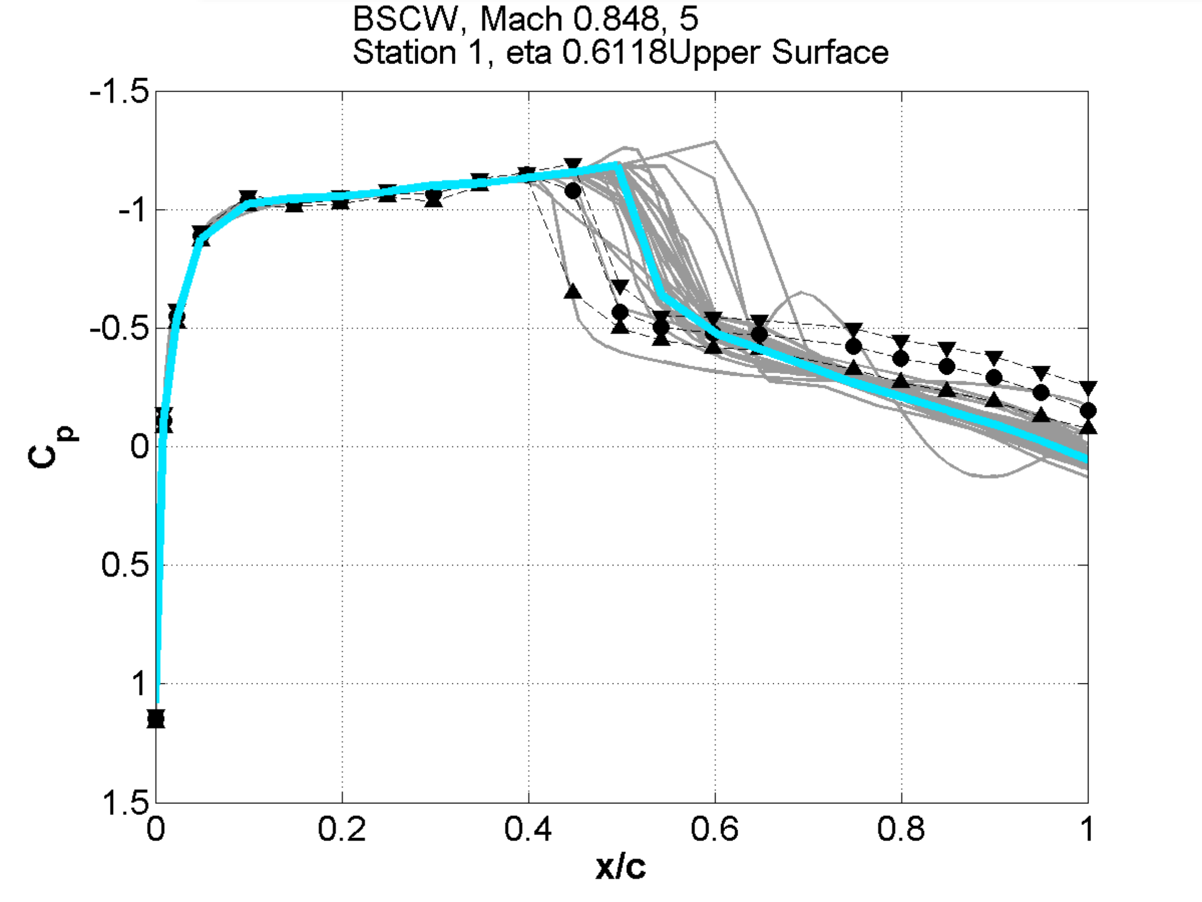


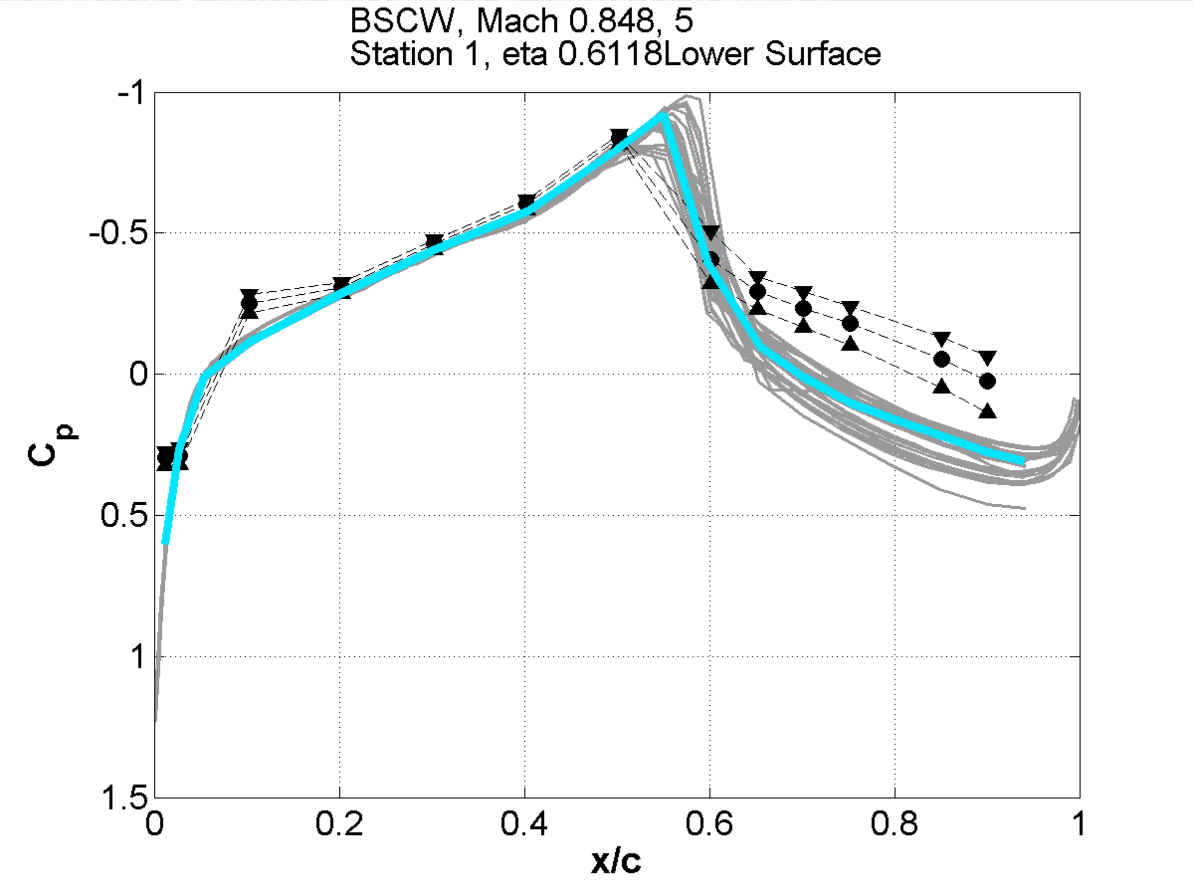


Revisiting some points from the first workshop: Steady pressure distribution results

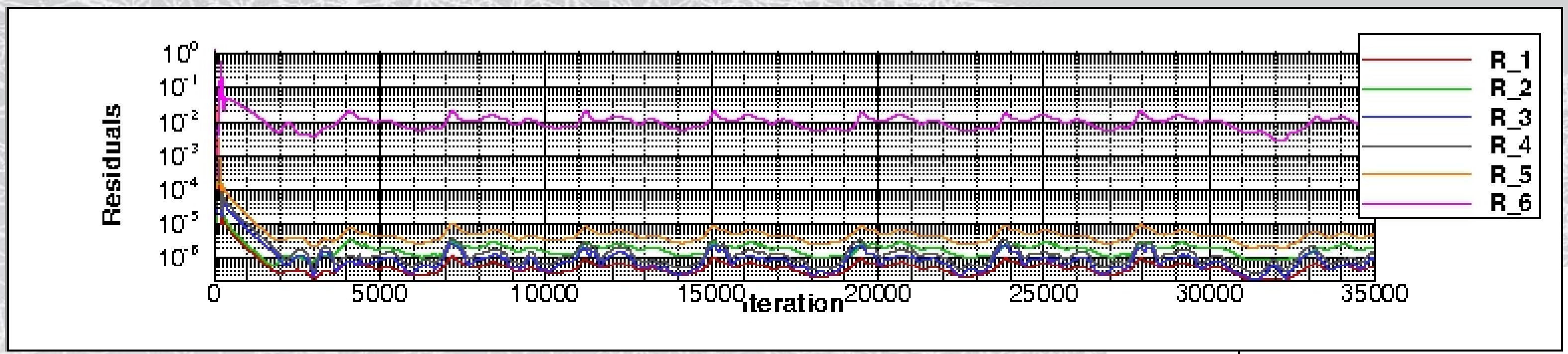
The medium grid analysis results generated using FUN3D are highlighted below

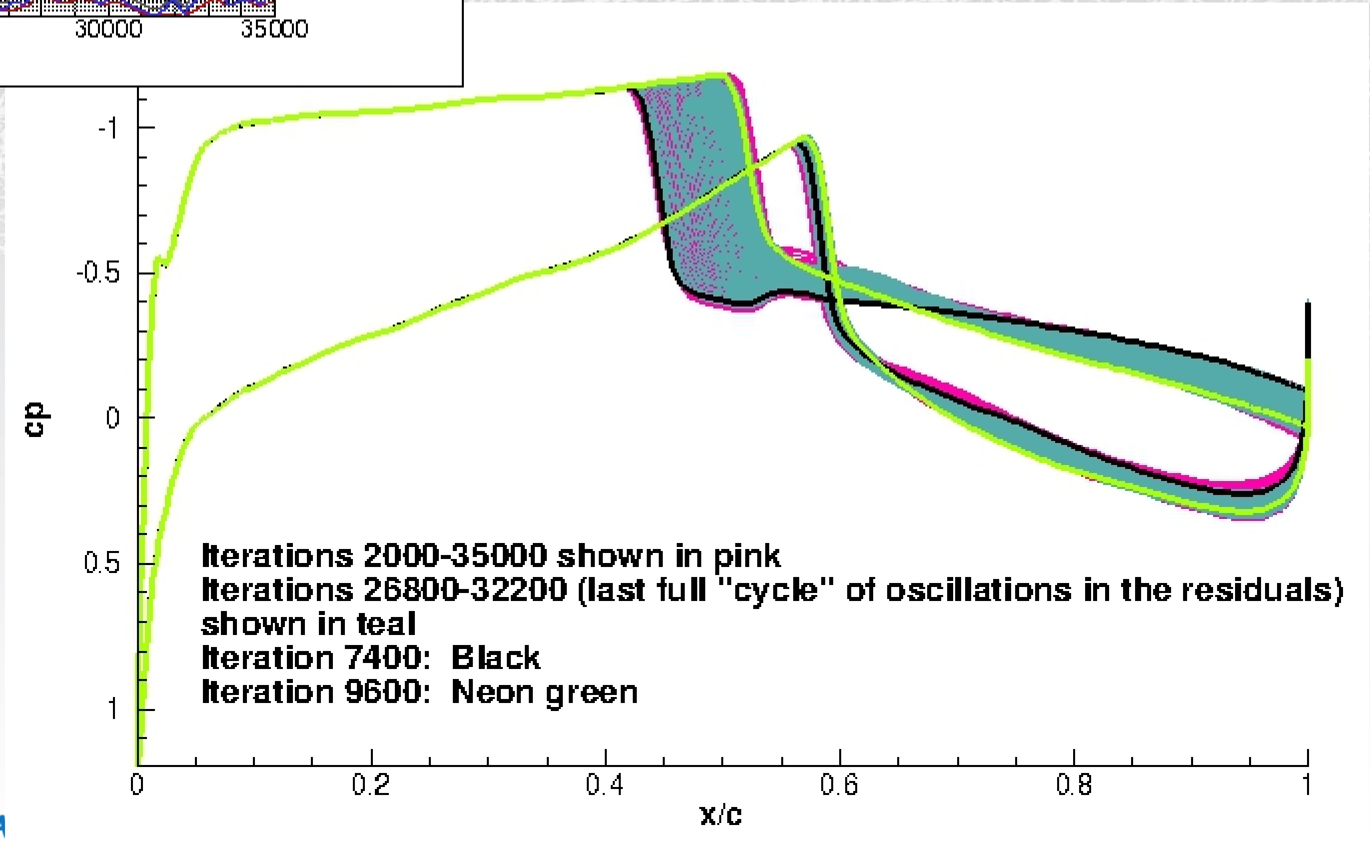
This is the last point of the steady analysis



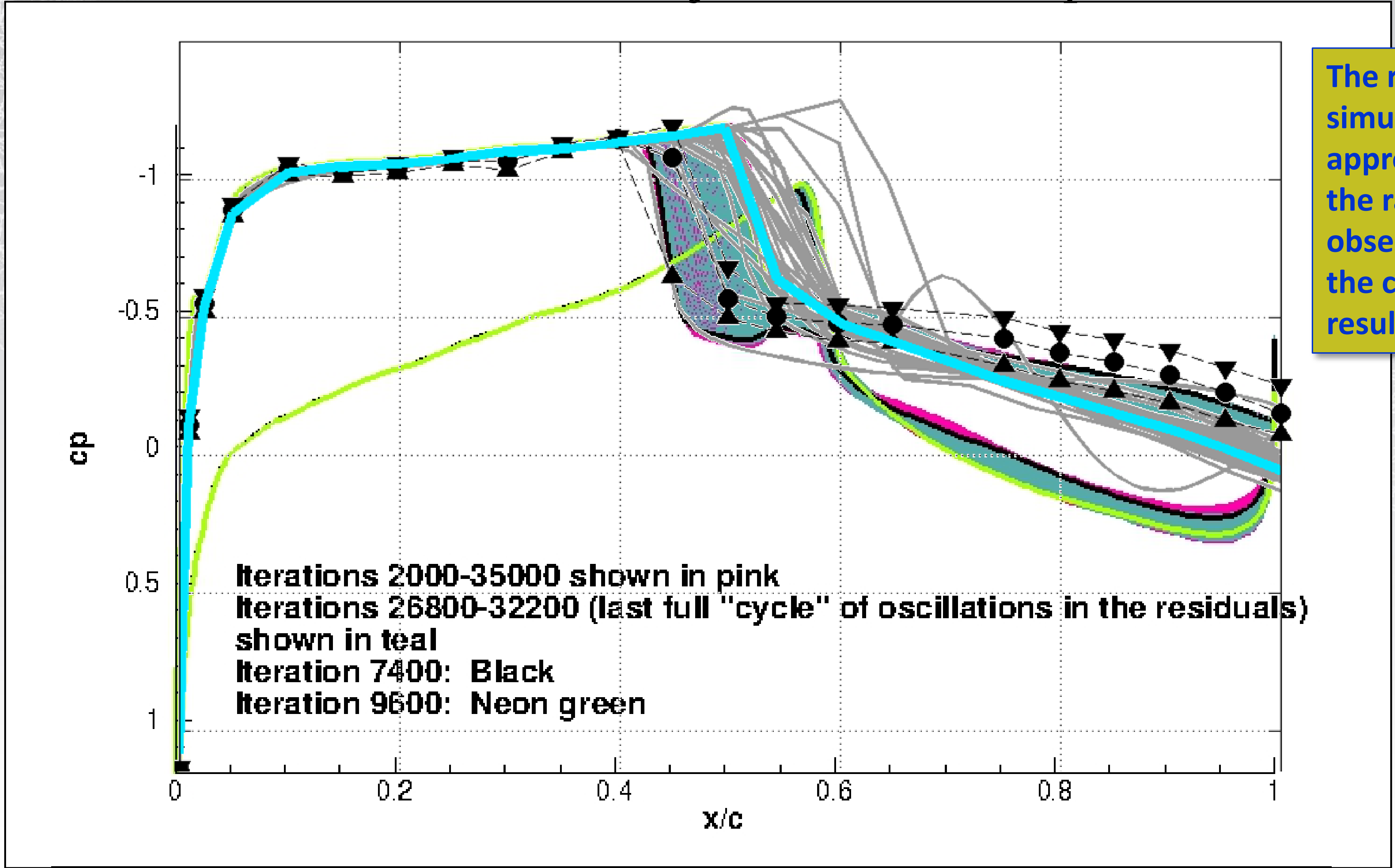


Running the solution for 35000 iterations illustrated that the solution never stabilizes to a fixed value at this condition





Was the variation due to user selection of data submitted? i.e moment at which each analyst chose to stop their solution?



The range of the simulation results is approximately half of the range of variation observed among all of the computational results submitted

Issues

- Aerodynamic calculations at transonic conditions
  - Time-accurate solution of unforced system cases
  - Variation in unforced distribution due to analyst choice and/or post-processing
- Aeroelastic simulations
  - **Time domain simulation guidelines from literature**
  - Convergence experiences with BSCW

Time step Guidelines from FUN3D developers

Taken from:
FUN3D v12.7 Training
Session 16: Aeroelastic Simulations
Bob Biedron, June 2015

Determining the Time Step

- Identify a ***characteristic time*** t^*_{chr} that you need to resolve with some level of accuracy in your simulation; perhaps:
  - Some important shedding frequency f^*_{shed} (Hz) is known or estimated $t^*_{chr} \sim 1 / f^*_{shed}$
  - Periodic motion of the body $t^*_{chr} \sim 1 / f^*_{motion}$
  - A range of frequencies in a DES-type simulation $t^*_{chr} \sim 1 / f^*_{highest}$
  - If none of the above, you can estimate the time it takes for a fluid particle to cross the characteristic length of the body, $t^*_{chr} \sim L^*_{ref} / U^*_{ref}$
  - $t_{chr} = t^*_{chr}\, a^*_{ref}\, (L_{ref}/L^*_{ref})$ (comp) $t_{chr} = t^*_{chr}\, U^*_{ref}\, (L_{ref}/L^*_{ref})$ (incomp)
- Say you want N time steps within the characteristic time:
  - $\Delta t = t_{chr} / N$ = `time_step_nondim`
- Figure an absolute *minimum* of N = 100 for reasonable resolution of t_{chr} with a 2^{nd} order scheme - really problem dependent (*frequencies > f* may be important*); but don't over resolve time if space is not well resolved too

t^* is physical time; t is non-dimensional time; $t = t^* a^*$

Applying these guidelines to AePW-2 analyses

Flutter is around 4 Hz:
~ 1 / 4 cycles/sec =0.25
Nyquist requires 2 samples/cycle, so ½ of this: = 0.125 sec

Highest mode is around 5 Hz:
~ 1 / 5 cycles/sec =0.2
Nyquist requires 0.1 sec

t_{chr}=chord/Velocity 16

Case	V (in/sec)	t_{chr} (sec)
1	4648	0.0034
2	4508	0.0035
3	5628	0.0028

Suggested rule of thumb from Spalart:

"A CFL number of approximately 1 is necessary for accurate prediction of large eddies, which is a requirement in both grid spacing and time step"

$$\Delta x_o / \Delta t = U_{\max}$$

where Δx_o is the grid spacing in the LES focus region and $U_{\max}$ is the maximum velocity in that region

CFL number

- Courant-Friedrichs-Lewy (CFL) condition
- A stability criterion for hyperbolic equations
- From CFD Online Wiki:
  - … for example, if a wave is moving across a discrete spatial grid and we want to compute its amplitude at discrete time steps of equal length then this length must be less than the time for the wave to travel to adjacent grid points
  - When the grid point separation is reduced, the upper limit for the time step also decreases
- Courant number, C: (for 1-dimensional case)

$$C_{\max} \geq u \frac{\Delta t}{\Delta x}$$

CFL number comments from Cummings, Morton & McDaniel

- If the CFL number or the Courant number is less than 1, the grid and time step are sufficiently sized to capture flow phenomena that occur at the given velocity. A very small Courant number may indicate that the time steps are being wasted and could be made longer.
- If the CFL number is greater than 1, the time step is too large relative to the grid size.

$$C_{\max} \geq C = u\frac{\Delta t}{\Delta x}$$

$$u = C\frac{\Delta x}{\Delta t} \qquad C = \frac{u}{u_{\mathrm{lim}it\_of\_grid}}$$

Translating and interpreting

- The items on the past few slides say:

You can’t capture flow structures that are moving across the airfoil (or your grid) faster than you have set your time step (and grid) to observe.

- In aeroelasticity, we consider the reduced frequency which relates the movement (velocity) of the wing relative to the movement (velocity) of the freestream air. We generally think of the airfoil as a single unsteady entity, so we use the wing semi-chord as our reference geometric length instead of a single aerodynamic element.
- In the simulations, we are trying to capture the details of the unsteady flow as the flow structures cross the individual aerodynamic elements, sometimes.

Conclusions from Cummings, Morton and McDaniel

ELSEVIER
Available online at www.sciencedirect.com
ScienceDirect
Progress in Aerospace Sciences 44 (2008) 241–257
PROGRESS IN AEROSPACE SCIENCES
www.elsevier.com/locate/paerosci

Experiences in accurately predicting time-dependent flows

- understand what is important for your calculation and know the physics involved: the grid and time step should be determined by the flow region of interest;
- perform grid and time-step study in conjunction with one another;
- vary time step and number of iterations so that all computations are for the same physical time;
- compute at least 10 cycles of the frequencies of interest;
- use appropriate averaging over a reasonable simulation interval;
- perform PSD for frequency analysis;
- evaluate time-integration method (including sub-iterations) and damping for their impact on accuracy;
- use the least amount of damping possible in simulations;
- use taps instead of integrated forces if necessary for PSD;
- use hybrid turbulence models if possible, including DES or DDES.

Jen's comments (circa 2012)

•I concur with most of the points that I fully understand (points 1,2,3,5,6,9)

•Point 4: why 10 cycles? What tells them that 10 cycles is a good number? I like that they gave CFD boys a rough order of magnitude and it wasn't 1 cycle, which is what CFD boys generally seem to think that they need. This number will likely depend on the quality of the information (e.g. noise content of the data signal and the ratio of the "information" to the noise floor), the complexity of the flow field, the sinusoidality (sinusoidness?) of the phenomena… but as estimates go, this doesn't seem like a bad one. I like 30 ensembles, and if we consider each ensemble to contain 1 full cycle and we use 2/3 of a cycle overlap, we get 28 segments. So, I like the number (10'ish cycles) based on reduced uncertainty in the FRF calculations, but I don't have a number wrt what the uncertainty is reduced to. I really should do that calculation…

•Comment regarding point 9: integrating over a fluctuating flow field isn't necessarily going to capture the fluctuation itself. We use integration a lot of times to "integrate out" the variations, fluctuations, unsteadiness. So, examining individual elemental quantities should be used to produce much richer frequency content and capture the details of the flow structure.

•Items outside my area of expertise: points 7,8,10

In the interim (2012-2015) , we have endeavored to improve our understanding of points 7, 8 and 10

Issues

- Aerodynamic calculations at transonic conditions
  - Time-accurate solution of unforced system cases
  - Variation in unforced distribution due to analyst choice and/or post-processing
- Aeroelastic simulations
  - Time domain simulation guidelines from literature
  - **Convergence experiences with BSCW**

AePW-2 Analysis Teams that submitted data sets explicitly examining temporal effects

Adam Jirasek and Mats Dalenbring (FOI) Jan Navratil (Brno University of Technology, VUT, Czech Republic)
Yannick Hoarau and C.-K. Huang (ICUBE, Strasbourg University, France) A. Gehri and J. Vos (CFS Engineering, Lausanne, Switzerland)
Eric Blades (ATA Engineering)
Tomer Rokita (Aerodynamics department, RD&E Division, RAFAEL)
Balasubramanyam Sasanapuri and Krishna Zore (ANSYS India) Robin Steed (ANSYS Canada) Eric Bish (ANSYS Inc.)
Marcello Righi (Zurich University of Applied Sciences)
Daniella E. Raveh (Technion IIT) Yuval Levy and Yair Mor Yossef (Israeli CFD Center)
Guilherme Begnini Cleber Spode Aluísio V. Pantaleão Bruno Guaraldo Neto, Guilherme O. Marcório, Marcos H.J. Pedras Carlos Alberto Bones (Embraer)
Pawel Chwalowski and Jennifer Heeg (NASA Langley Research Center)
Eirikur Jonsson Charles A. Mader Joaquim R.R.A. Martins (University of Michigan)
Sergio Ricci and Andrea Mannarino (Department of Aerospace Science and Technology of Politecnico di Milano)
Patrick McGah Girish Bhandari Alan Mueller Durrell Rittenberg (CD-adapco)
Amin Fereidooni and Anant Grewal (NRC) Marcel Grzeszczyk (NRC, University of Toronto)
Bimo Pranata and Bart Eussen (NLR)
Eduardo Molena (ITA)

FUN3D

- Backwards Difference Scheme, 2nd order optimized is used in all calculations
- **Dual time-stepping approach**
- Performed brute force convergence studies
- No sensitivity analyses available in FUN3D for aeroelastic simulations
- How did we determine when we were at a converged solution?
- What output parameters should be examined to determine that you have refined the temporal parameters "sufficiently"?

FUN3D implementation of temporal error assessment

- Estimate the temporal error incurred at each time step
  - →Do this by calculating the residual contribution with 2 different levels of approximations of the time derivatives.
- Specify a percentage of the temporal error norm to use as an exit criteria for the subiteration process.

Temporal convergence studies were done for Case 2

	Case 1	Case 2	Optional Case 3		
			A	B	C
Mach	0.7	0.742	0.85	.85	.85
Angle of attack	3°	-0°	5°	5°	5°
Dynamic Data Type	Forced oscillation	Flutter	Unforced Unsteady	Forced Oscillation	Flutter
Notes:	• Attached flow solution. • Oscillating Turn Table (OTT) experimental data. • R-134a	• Pitch and Plunge Apparatus (PAPA) experimental data. • R-12	• Separated flow effects. • Oscillating Turn Table (OTT) experimental data. • R-134a	• Separated flow effects. • Repeat of AePW-1 • Oscillating Turn Table (OTT) experimental data. • R-134a	• No experimental data for comparison. • Separated flow effects on aeroelastic solution. • R-134a

Time step Guidelines from FUN3D developers

Taken from:
FUN3D v12.7 Training
Session 16: Aeroelastic Simulations
Bob Biedron, June 2015

Determining the Time Step

- Identify a ***characteristic time*** t^*_{chr} that you need to resolve with some level of accuracy in your simulation; perhaps:
  - Some important shedding frequency f^*_{shed} (Hz) is known or estimated $t^*_{chr} \sim 1 / f^*_{shed}$
  - Periodic motion of the body $t^*_{chr} \sim 1 / f^*_{motion}$
  - A range of frequencies in a DES-type simulation $t^*_{chr} \sim 1 / f^*_{highest}$
  - If none of the above, you can estimate the time it takes for a fluid particle to cross the characteristic length of the body, $t^*_{chr} \sim L^*_{ref} / U^*_{ref}$
  - $t_{chr} = t^*_{chr}\, a^*_{ref}\, (L_{ref}/L^*_{ref})$ (comp) $t_{chr} = t^*_{chr}\, U^*_{ref}\, (L_{ref}/L^*_{ref})$ (incomp)
- Say you want N time steps within the characteristic time:
  - $\Delta t = t_{chr} / N$ = `time_step_nondim`
- Figure an absolute *minimum* of N = 100 for reasonable resolution of t_{chr} with a 2nd order scheme - really problem dependent (*frequencies > f* may be important*); but don't over resolve time if space is not well resolved too

t* is physical time; t is non-dimensional time; $t = t^* a^*$

Applying these guidelines to AePW-2 analyses

Flutter is around 4 Hz:
~ 1 / 4 cycles/sec =0.25
Nyquist requires 2 samples/cycle, so ½ of this: = 0.125 sec

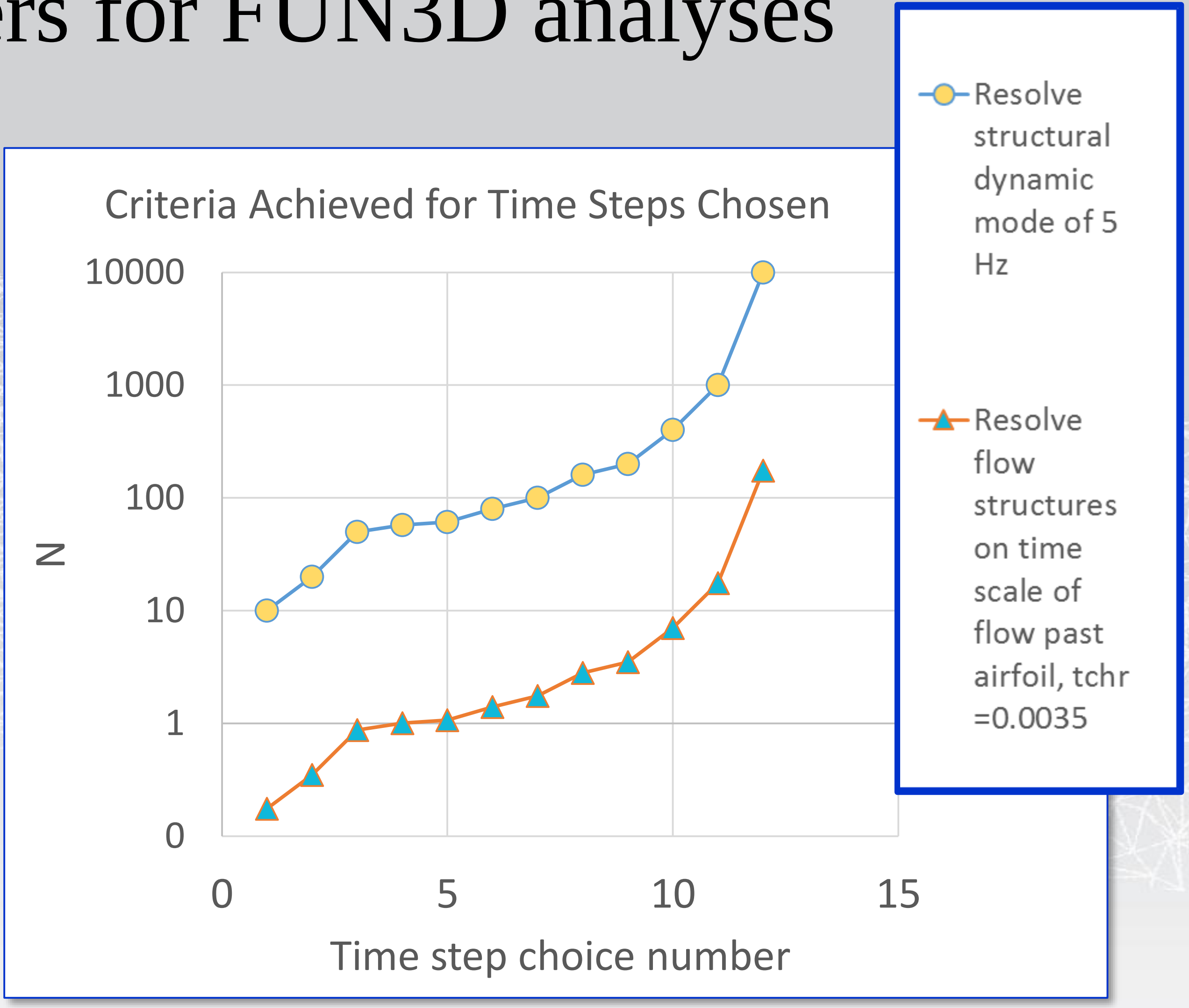
Highest mode is around 5 Hz:
~ 1 / 5 cycles/sec =0.2
Nyquist requires 0.1 sec

t_{chr}=chord/Velocity 16

Case	V (in/sec)	t_{chr} (sec)
1	4648	0.0034
2	4508	0.0035
3	5628	0.0028

Summary of temporal parameters for FUN3D analyses

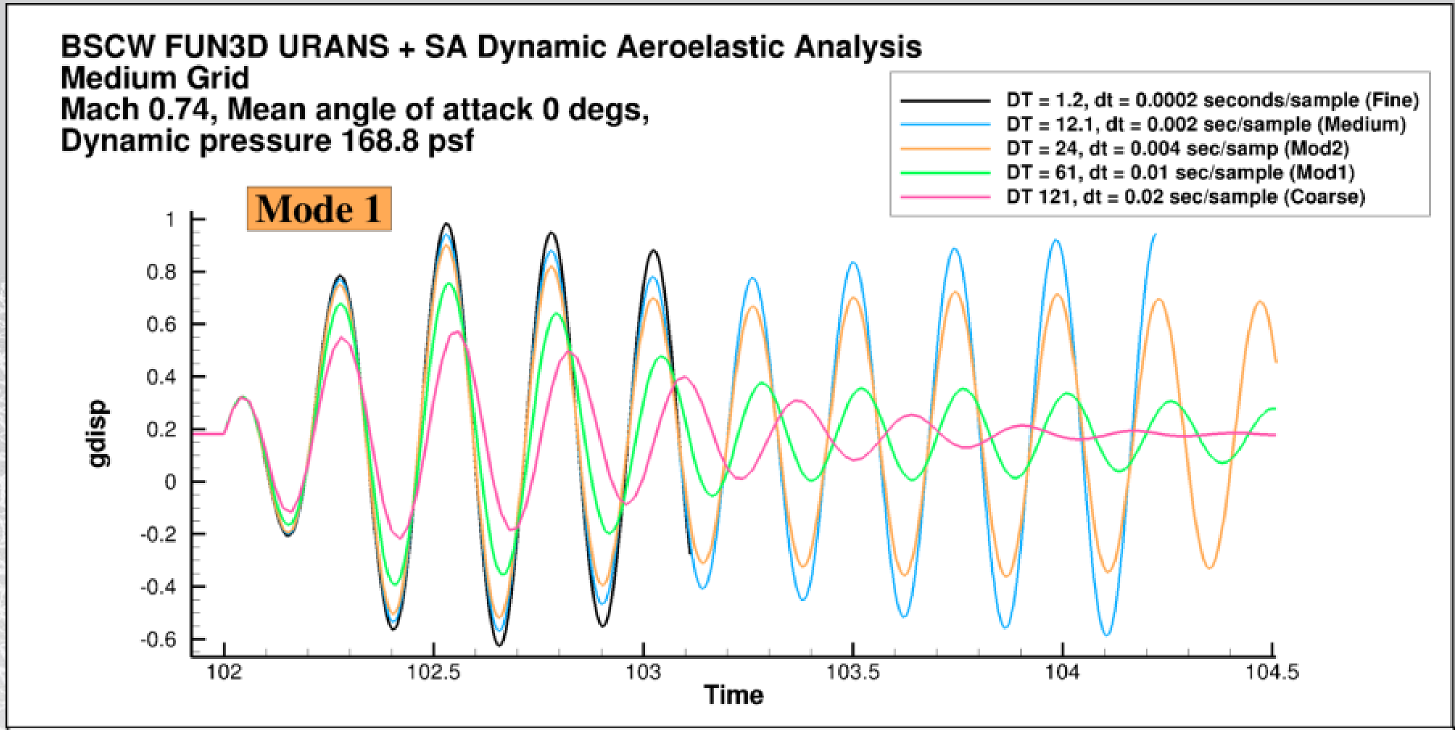
	Time step size		CRITERIA	
			Resolve structural dynamic mode of 5 Hz	Resolve flow structures on time scale of flow past airfoil, t_{chr} =0.0035
	DT	dt		0.0035
	Non Dimen sional	physical	samples / 5 Hz cycle	t_{chr} / dt
	(-)	(sec/sample)	(N)	(N)
1	121.88	0.02	10	0.175
2	60.94	0.01	20	0.350
3	24.38	0.004	50	0.875
4	21.2	0.00348	57	1.006
5	20	0.00328	61	1.066
6	15.23	0.0025	80	1.400
7	12.19	0.002	100	1.750
8	7.62	0.00125	160	2.800
9	6.09	0.001	200	3.500
10	3.05	0.0005	400	7.000
11	1.22	0.0002	1000	17.500
12	0.12	0.00002	10000	175.000

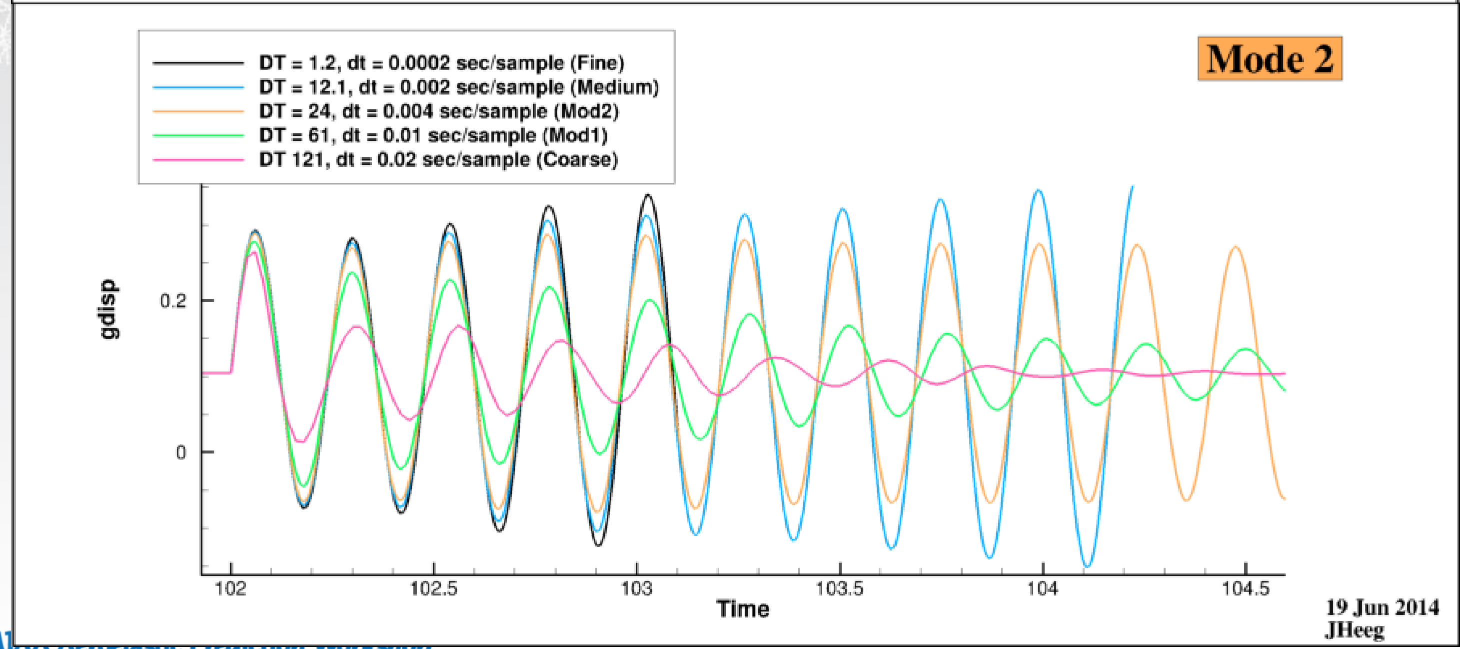


The time step size that was used for the “fine” time step for FUN3D analysis is overly sufficient for resolving the structural dynamic modes, but may fall short of being able to resolve small flow structures

Varying time step size at the experimental flutter condition: q = 168.8 psf; 25 subiterations per global time step

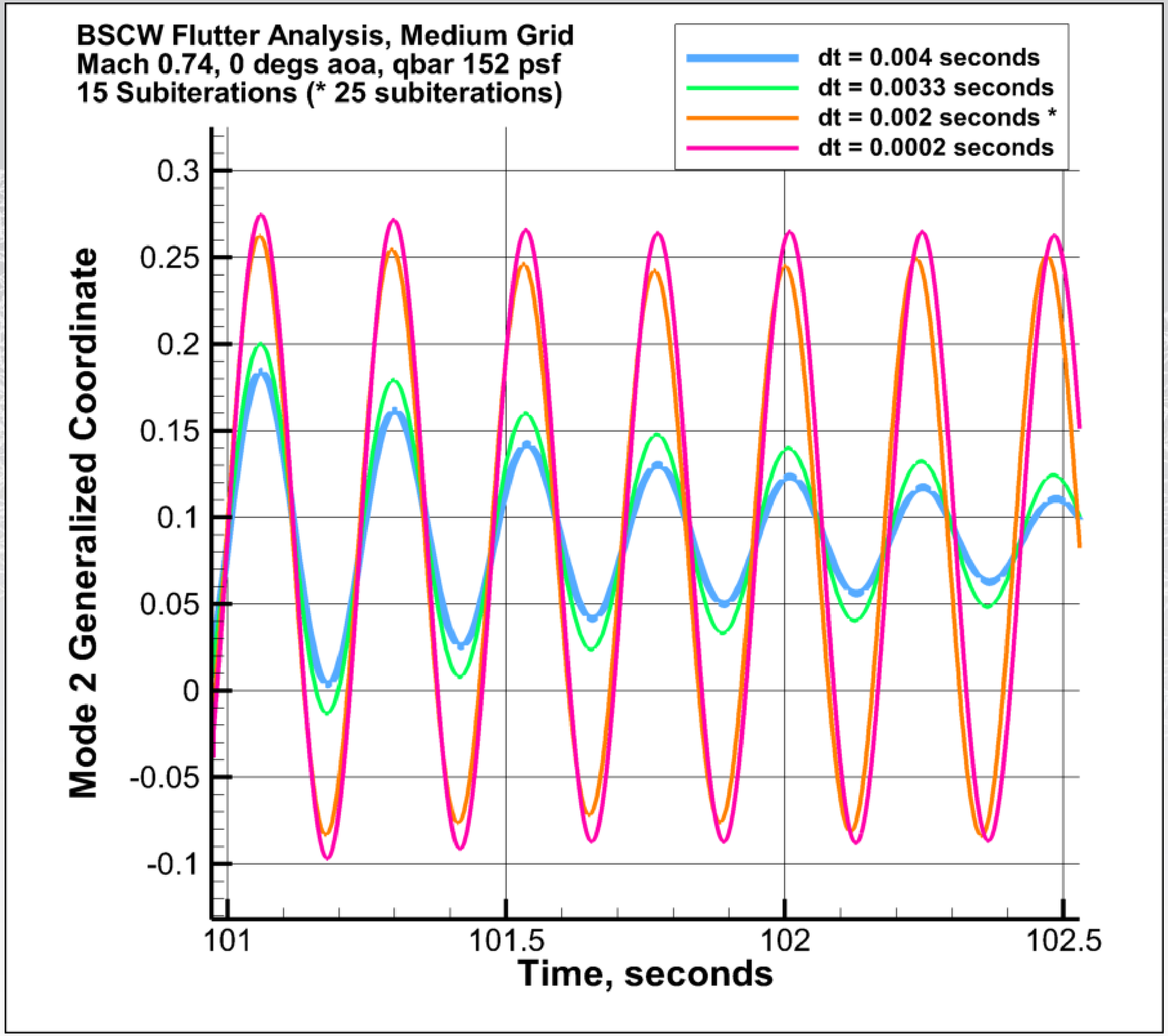
	Time step size		CRITERIA	
			Resolve structural dynamic mode of 5 Hz	Resolve flow structures on time scale of flow past airfoil, t_{chr} =0.0035
	DT	dt		0.0035
	Non Dimen sional	physical	samples / 5 Hz cycle	t_{chr} / dt
	(-)	(sec/sample)	(N)	(N)
1	121.88	0.02	10	0.175
2	60.94	0.01	20	0.350
3	24.38	0.004	50	0.875
4	21.2	0.00348	57	1.006
5	20	0.00328	61	1.066
6	15.23	0.0025	80	1.400
7	12.19	0.002	100	1.750
8	7.62	0.00125	160	2.800
9	6.09	0.001	200	3.500
10	3.05	0.0005	400	7.000
11	1.22	0.0002	1000	17.500
12	0.12	0.00002	10000	175.000



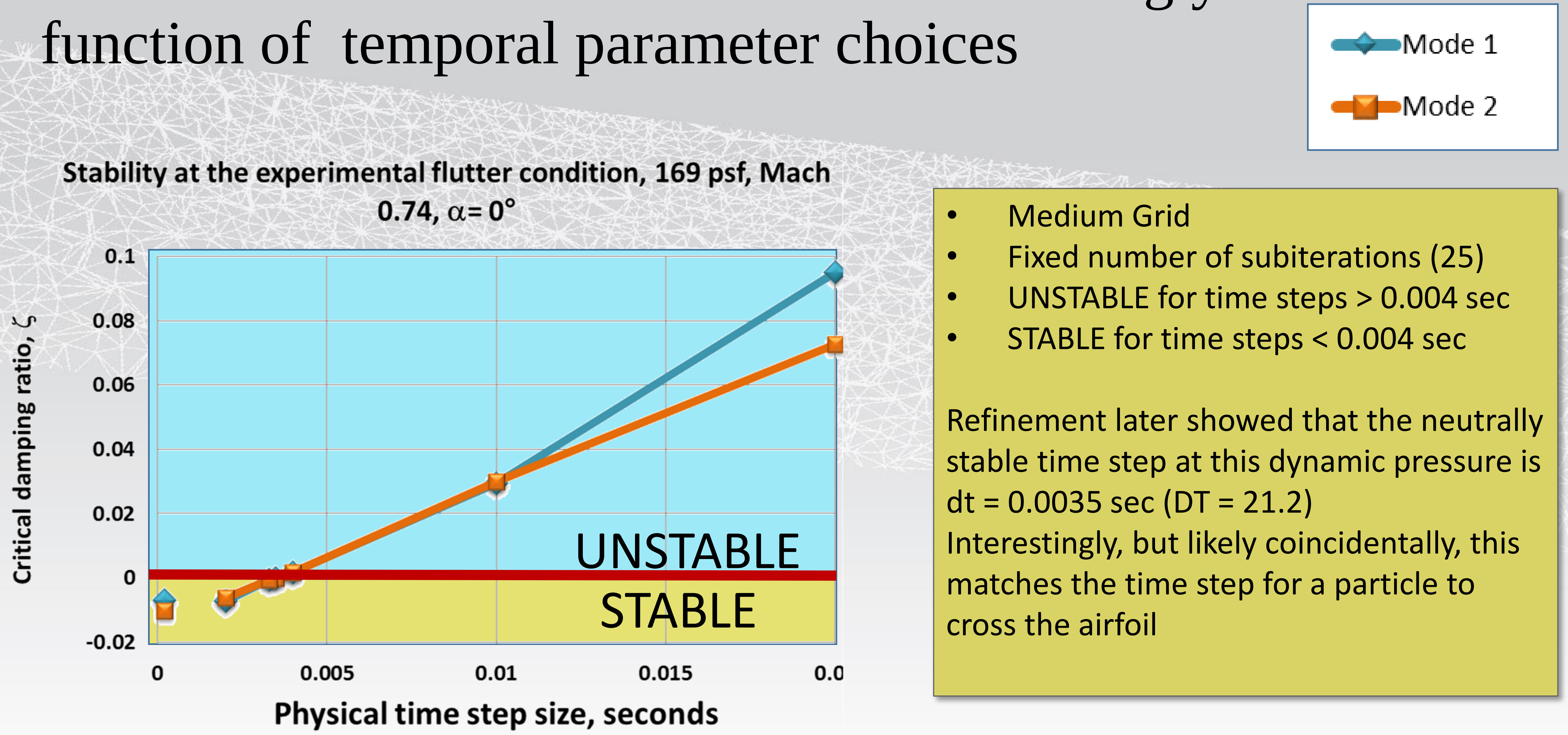


Varying time step size at 152 psf; 15 subiterations per global time step

	Time step size		CRITERIA	
			Resolve structural dynamic mode of 5 Hz	Resolve flow structures on time scale of flow past airfoil, t_{chr} =0.0035
	DT	dt		0.0035
	Non Dimen sional	physical	samples / 5 Hz cycle	t_{chr} / dt
	(-)	(sec/sample)	(N)	(N)
1	121.88	0.02	10	0.175
2	60.94	0.01	20	0.350
3	24.38	0.004	50	0.875
4	21.2	0.00348	57	1.006
5	20	0.00328	61	1.066
6	15.23	0.0025	80	1.400
7	12.19	0.002	100	1.750
8	7.62	0.00125	160	2.800
9	6.09	0.001	200	3.500
10	3.05	0.0005	400	7.000
11	1.22	0.0002	1000	17.500
12	0.12	0.00002	10000	175.000



Prediction of flutter demonstrated to be strongly a function of temporal parameter choices



- Medium Grid
- Fixed number of subiterations (25)
- UNSTABLE for time steps > 0.004 sec
- STABLE for time steps < 0.004 sec

Refinement later showed that the neutrally stable time step at this dynamic pressure is dt = 0.0035 sec (DT = 21.2)
Interestingly, but likely coincidentally, this matches the time step for a particle to cross the airfoil

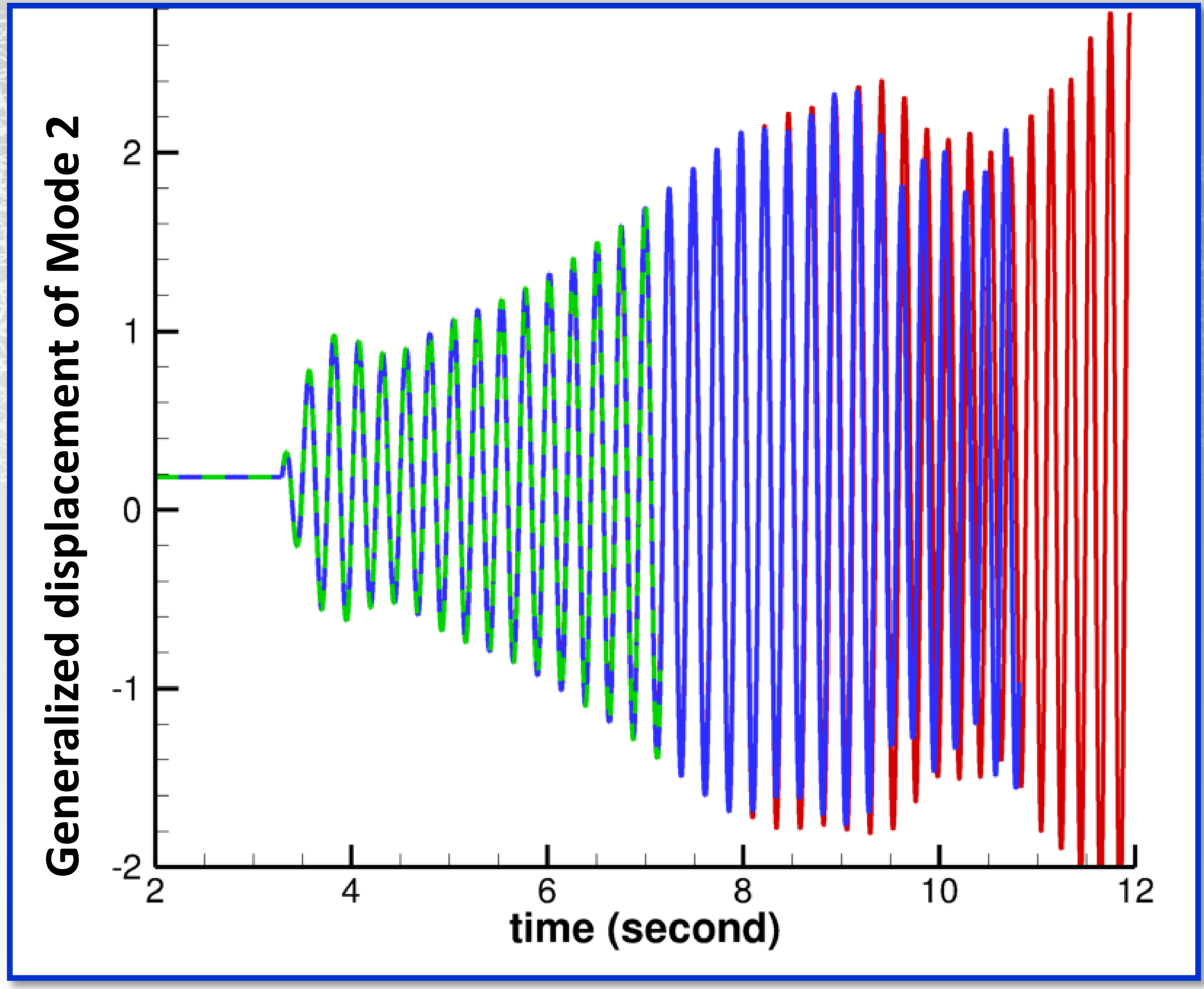
We eliminated using time steps < 0.004 sec

Time step of 0.0002 determined to be our required time step size for temporal convergence

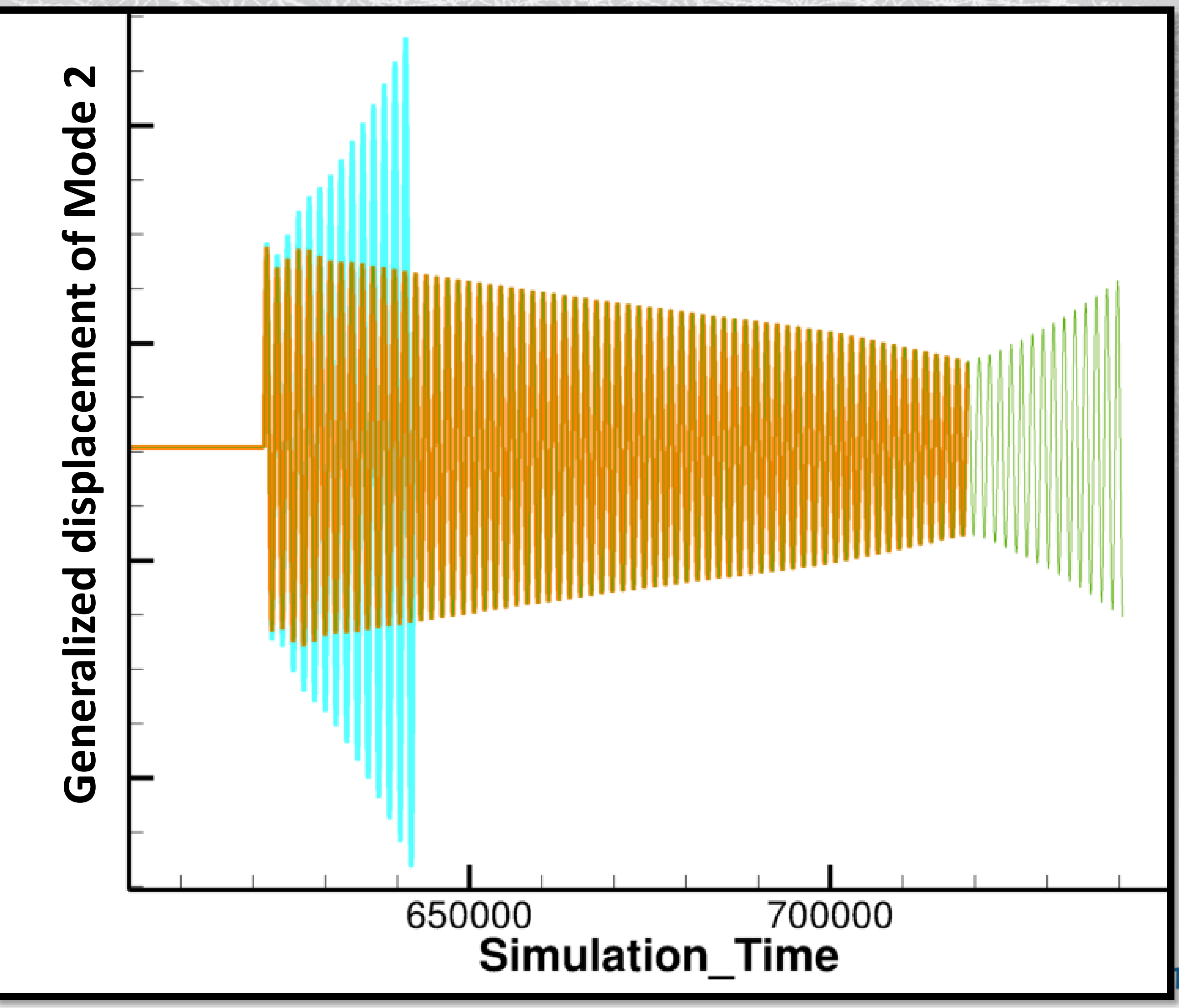
The results did not change in substantial ways when the time step was decreased by an additional order or magnitude

	Time step size		CRITERIA	
			Resolve structural dynamic mode of 5 Hz	Resolve flow structures on time scale of flow past airfoil, t_{chr} =0.0035
	DT	dt		0.0035
	Non Dimen sional	physical	samples / 5 Hz cycle	t_{chr} / dt
	(-)	(sec/sample)	(N)	(N)
1	121.88	0.02	10	0.175
2	60.94	0.01	20	0.350
3	24.38	0.004	50	0.875
4	21.2	0.00348	57	1.006
5	20	0.00328	61	1.066
6	15.23	0.0025	80	1.400
7	12.19	0.002	100	1.750
8	7.62	0.00125	160	2.800
9	6.09	0.001	200	3.500
10	3.05	0.0005	400	7.000
11	1.22	0.0002	1000	17.500
12	0.12	0.00002	10000	175.000

(Ignore the blue line for a moment, please)



Stability changes with subiteration criteria



Results from 3 simulations are shown

All have the same time step size: dt = 0.004 seconds (non-dimensional time = 24)

Simulation 1: 2% Temporal Error Criteria applied to subiteration level (1000 subiterations maximum) (Blue trace)

Simulation 2: 25 Subiterations per global time step. (Brown trace)

Simulation 3: Start with the stable, converging solution of Case 2. Alter the subiteration criteria to match Case 1. (Green trace)

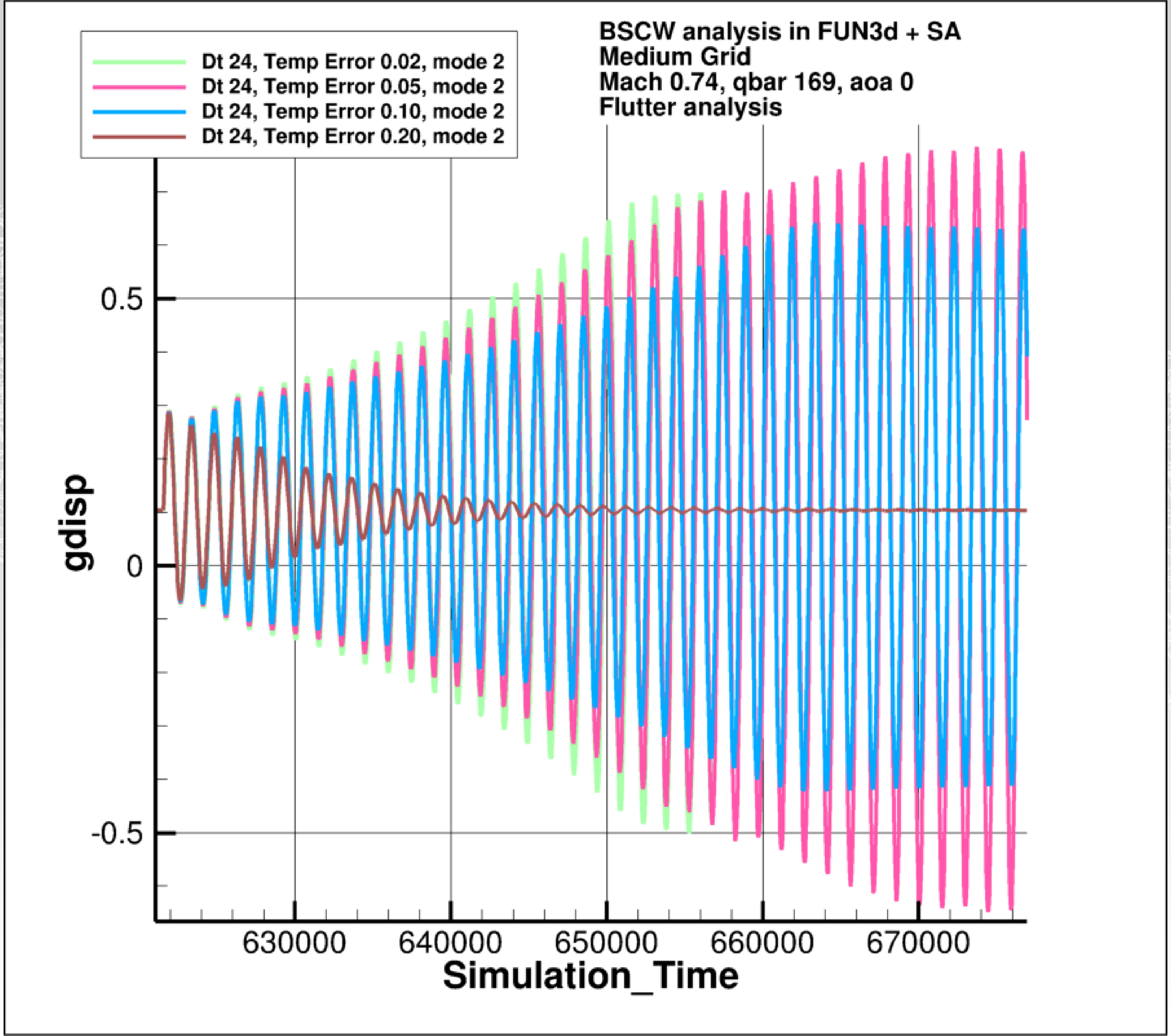
Note that the damping of the system with the changed criteria is different from the damping that occurs if the system is initialized with the 2% temporal error convergence criteria.

- Flutter runs perfomed at Mach 0.74, α=0°
- Medium Grid, no limiter
- For time step sizes ≤ 0.004 seconds (DT ≤ 24)

	Dynamic pressure	zeta of flutter mode	Frequency of flutter mode	Static aeroelastic twist angle	Maximum Subiterations	Temporal Error Criterion	Initial generalized velocity impulse size	Time step size
	psf		Hz	degs addnl twist		%		(seconds)
	169	**-0.01015**	**4.10349**	**-1.03397**	**1000**	**10**	**5**	**0.0002**
	152	**-0.00033**	**4.2214**	**-0.8771**	**1000**	**10**	**5**	**0.0002**
	135	**0.005936**	**4.33625**	**-0.73726**	**1000**	**10**	**5**	**0.0002**
1	135	0.006407	4.33595	-1.14676	25	666	5	0.0002
2	135	0.005973	4.33682	-0.73727	1000	10	5	0.0002
3	135	0.005934	4.33909	-0.73727	1000	2	5	0.0002
4	135	-8.83E-05	4.37716	0.677878	25	666	5	0.002
5	135	0.001901	4.33805	-0.73728	1000	20	5	0.002
6	135	0.002959	4.35919	-0.73726	1000	10	5	0.002
7	135	0.006209	4.33688	-0.73726	1000	5	5	0.002
8	152	0.001121	4.21725	-0.77786	15	666	5	0.0002
9	152	-0.00033	4.2214	-0.8771	1000	10	5	0.0002
10	152				1000	2	5	0.0002
11	152	0.001795	4.2213	0.789304	15	666	5	0.002
12	152	-0.00127	4.25322	0.66719	25	666	5	0.002
13	152	0.001795	4.2213	-2.2281	1000	10	5	0.002
14	152	0.001805	4.22026	-0.92021	1000	10	0.5	0.0002
15	152	-0.00033	4.2214	-0.8771	1000	10	5	0.0002
16	152	-0.01104	4.1959	-1.03573	1000	10	10	0.0002
17	152	0.001255	4.21969	-1.64628	1000	10	5	0.004
18	169	-0.01015	4.10349	-1.03397	25	666	5	0.0002
19	169	-0.00844	4.09026	-1.197	1000	10	5	0.004
20	169	-0.00931	4.08679	0.496025	1000	10	5	0.00125
21	169	-0.01015	4.10349	-1.03397	1000	10	5	0.0002
22	169	-0.00593	4.11216	-0.9892	1000	10	0.5	0.0002
23	169	-0.00881	4.10264	-1.49106	1000	10	2.75	0.0002
24	169	-0.01015	4.10349	-1.03397	1000	10	5	0.0002
25	169	-0.01268	4.20852	0.054148	1000	10	10	0.0002
26	169	-0.0071	4.12665	-1.03397	25	666	5	0.002
27	169	-0.00904	4.08552	-1.23988	1000	10	5	0.002
28	169	0.030952	4.15415	-2.25098	1000	20	5	0.004
29	169	-0.00853	4.08754	-1.66157	1000	10	5	0.004
30	169	-0.00836	4.09151	0.437936	1000	5	5	0.004
31	169	-0.00935	4.08666	-2.80293	1000	2	5	0.004

666 = no criteria used; fixed # subiterations

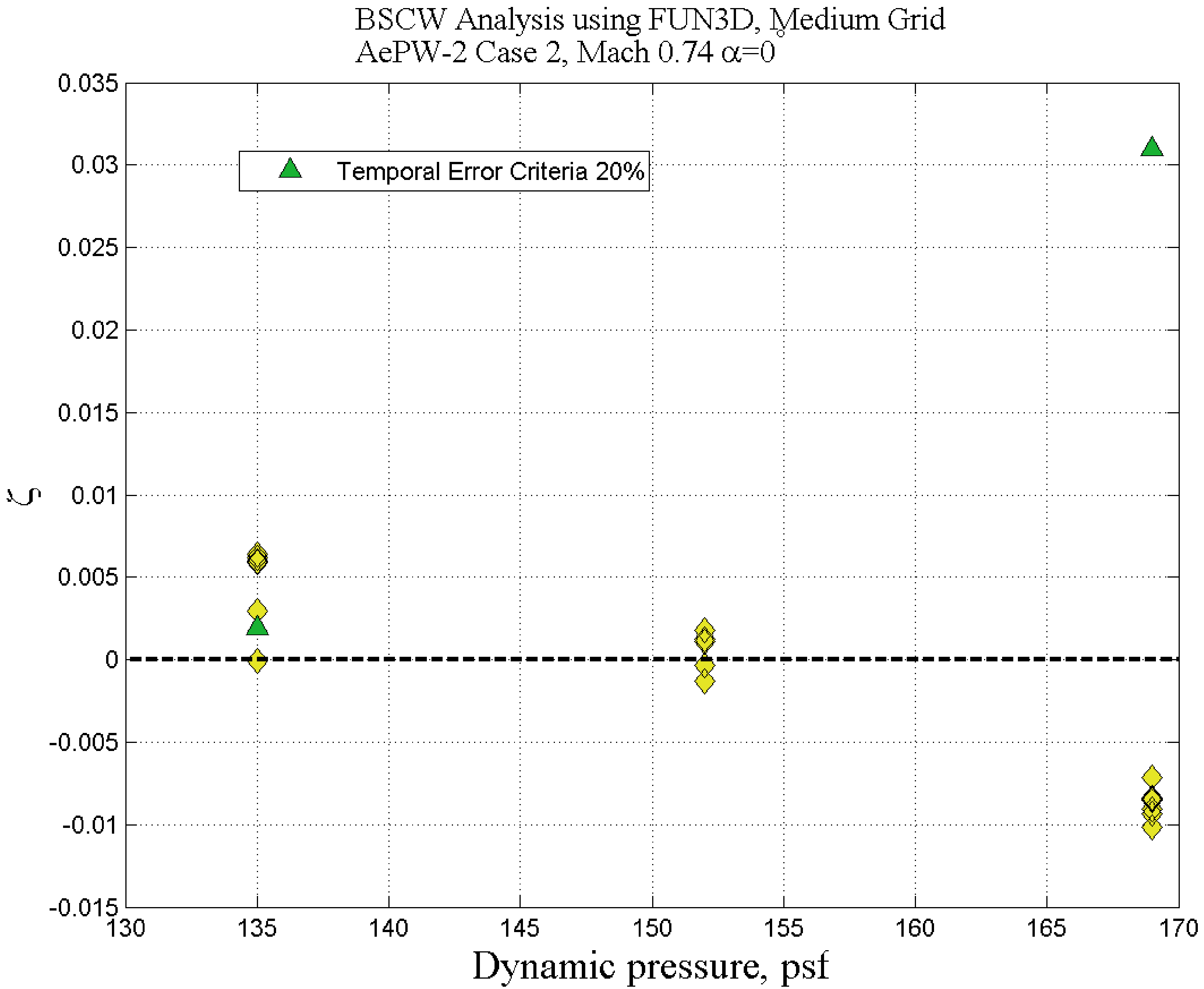
Varying the temporal error criteria



Results shown are
169 psf
dt = 0.004 seconds
Initial velocity kick amplitude 5

Damping results

- Mach 0.74, 0 degs angle of attack
- Medium Grid
- No flux limiter
- Time step sizes ≤ 0.004 seconds (DT ≤ 24)
- Initial generalized velocity kick size = 5

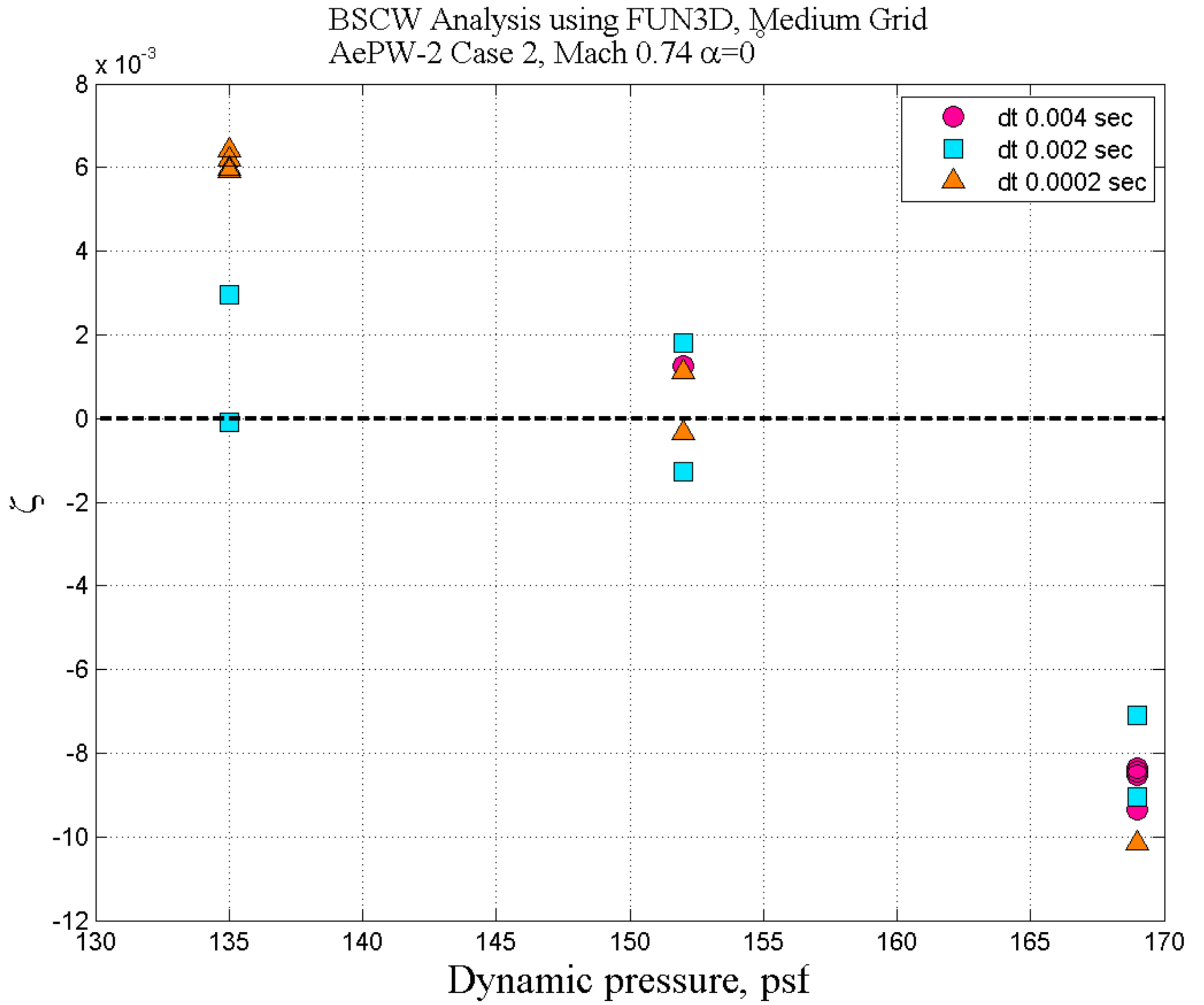


We eliminated using fixed subiterations and temporal error criteria < 10%

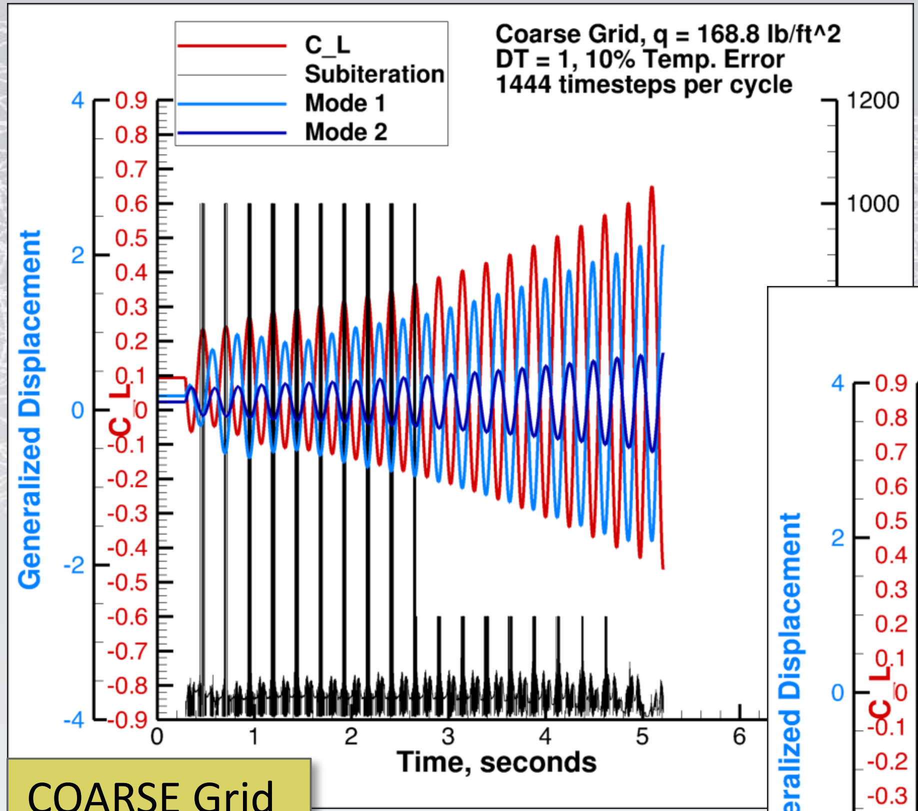
Requirements on our computations for declared temporally converged simulations

- Time step 0.0002 seconds
- 10% temporal error criteria

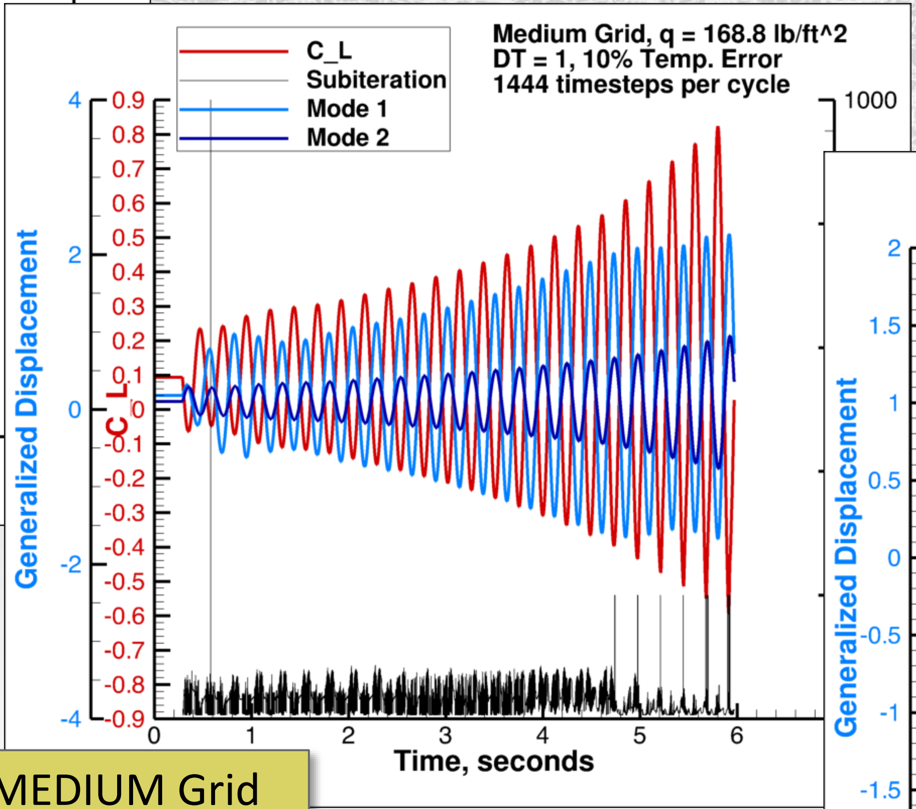
Declared flutter as 152 psf



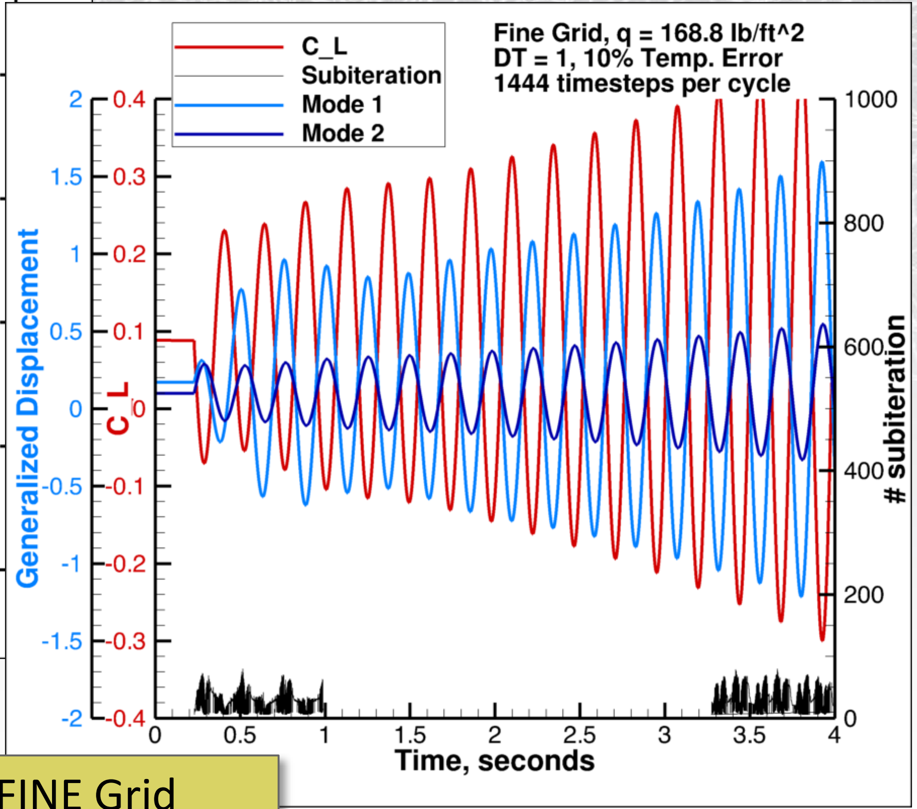
Prediction of flutter demonstrated to be a strongly a function of combined temporal and spatial effects



COARSE Grid



MEDIUM Grid



FINE Grid

Convergence assessment was very expensive:

- The flutter solutions were used for convergence assessment.
- No reliable intermediate parameter was found to represent the goodness of the flutter result.

Coarser grid required more work (subiterations) to converge each time accurate solution (global time step)

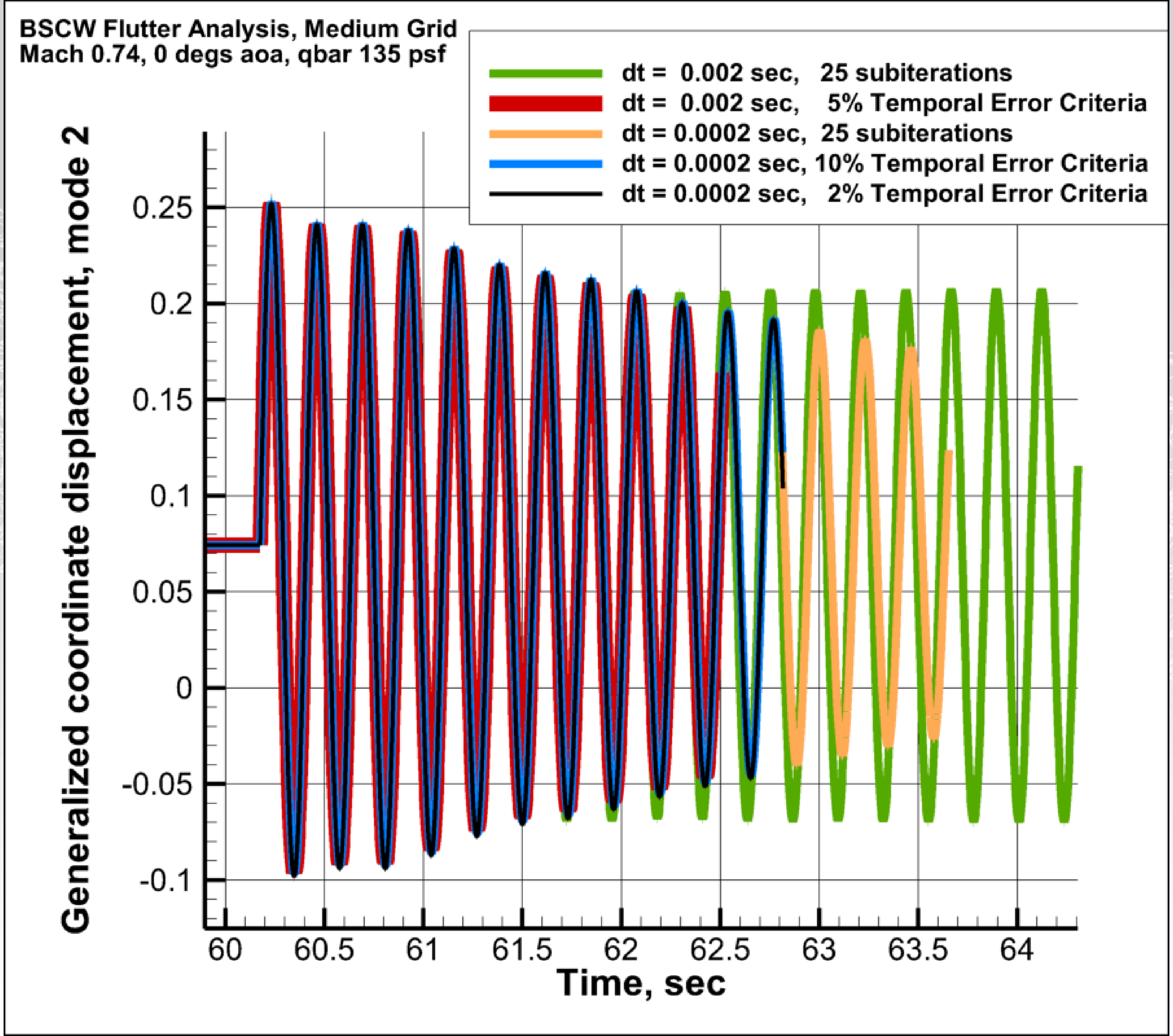
Summary

- Convergence study simultaneously included:
  - Spatial grid refinement
  - Global time step refinement
  - Convergence level per global time step
- Convergence study should be performed for different flow fields (i.e. different Mach number and angle of attack domains)

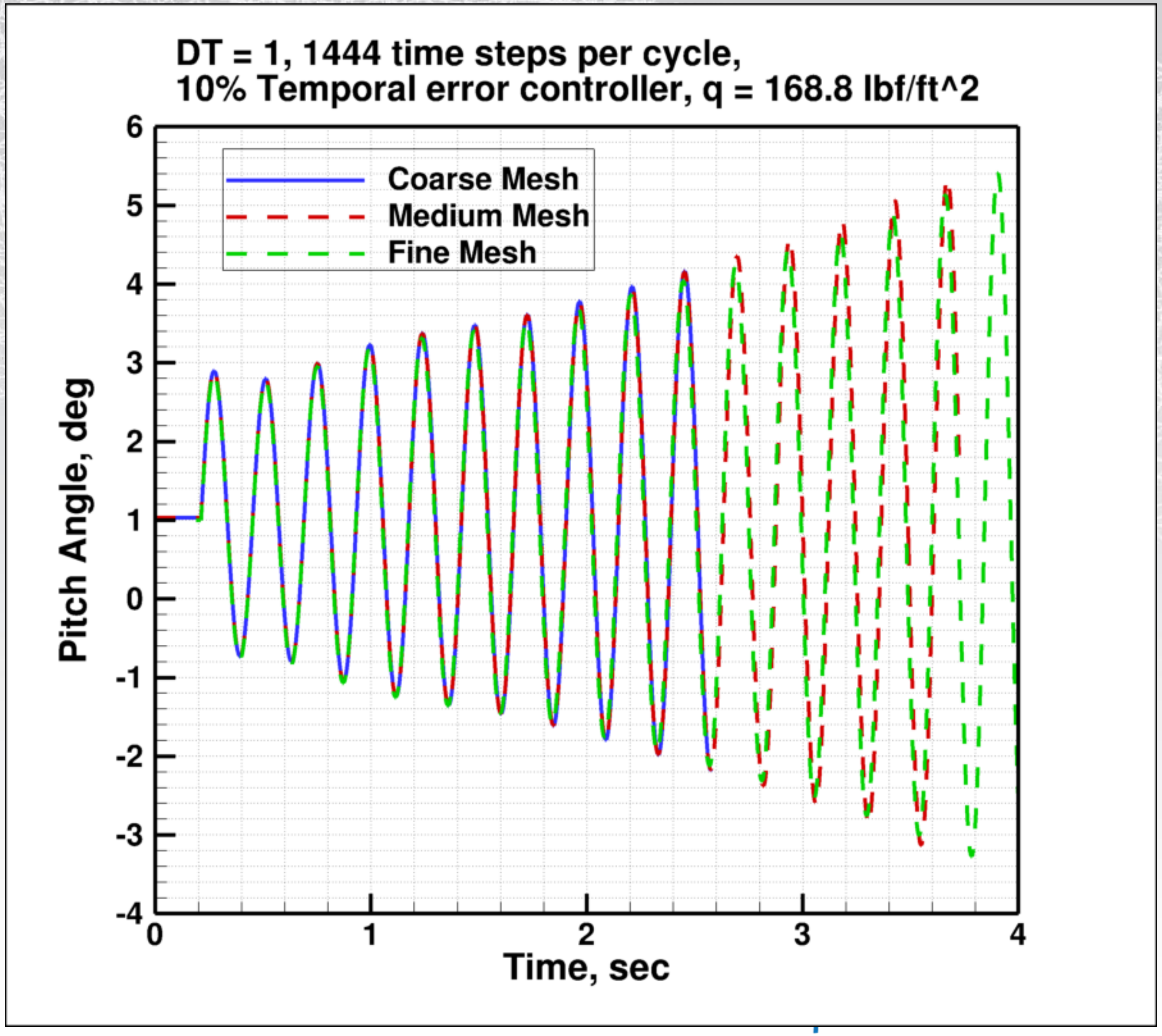
FIN

More results

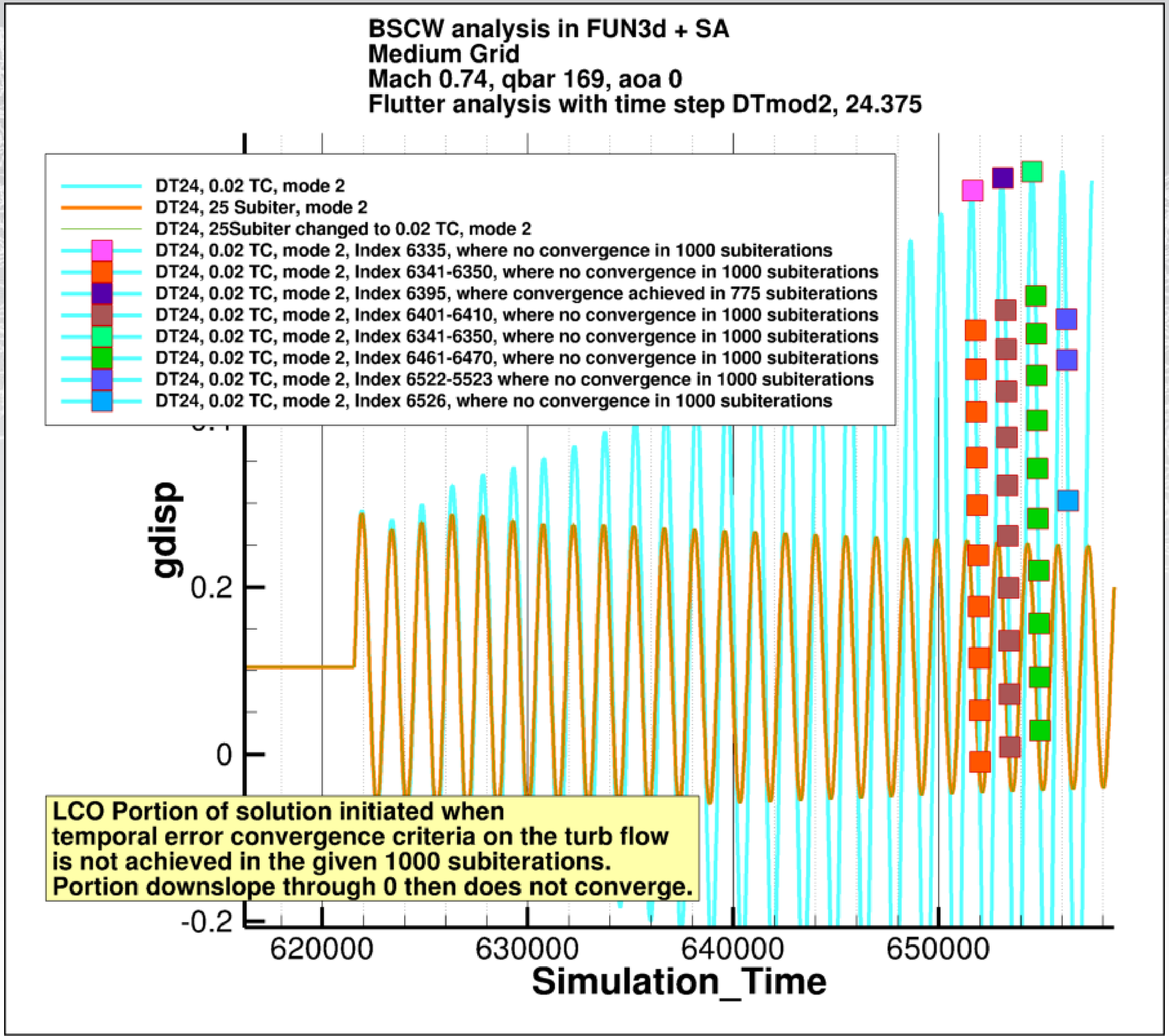
135 psf: Varying time step size and temporal error criteria



Current FUN3D results: spatial and temporal convergence AePW-2 Case#2 Flutter results



Q 169, DT 24: Limit cycle oscillation onset details 2% temporal error convergence



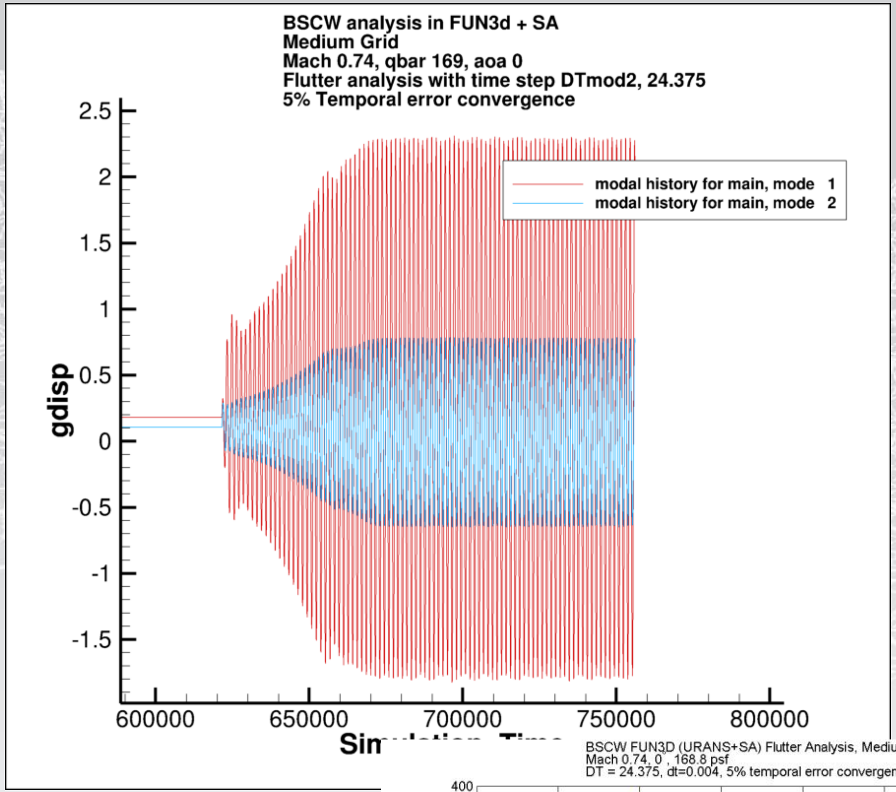
In almost all cases examined, LCO was generated only when the simulation failed to converge.

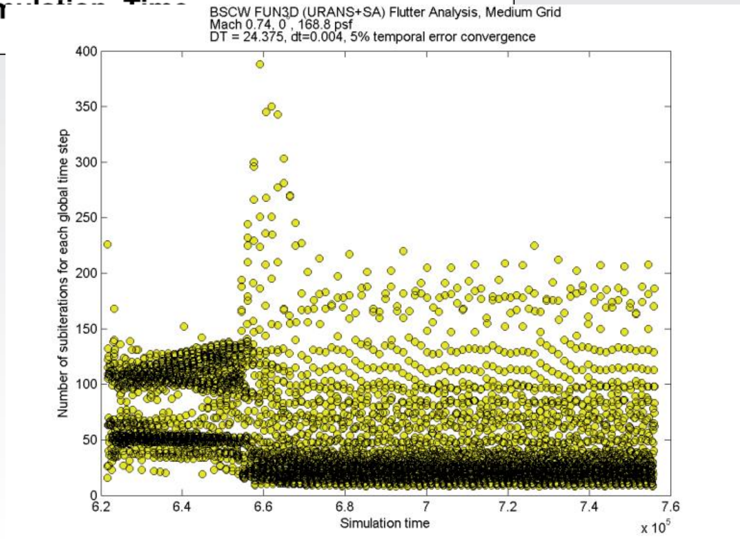
For some solutions, this behavior preceded critical failure of the code (negative volumes), but in most cases it did not.

The damping changes and limit cycle behavior onset occurs just after the temporal error convergence
criterion of 2% is no longer met.

This doesn’t say that the LCO prediction is NEVER a prediction of the physics.
Neutral stability = flutter onset = LCO.

Q 169, DT 24: Limit cycle oscillation onset details
5% temporal error convergence



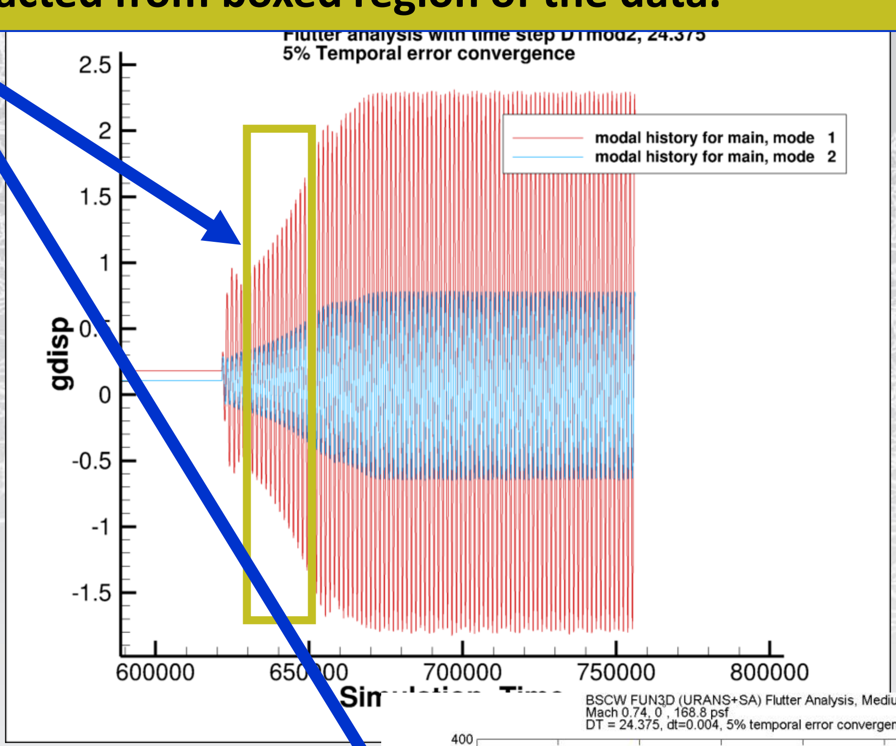


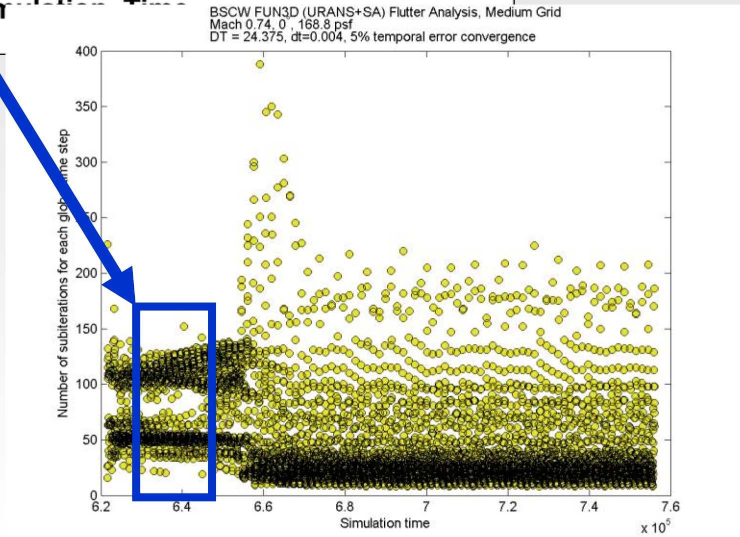
The temporal error convergence criterion is met for as long as I have run this solution. The largest number of subiterations required is nearly 400, just as LCO onset occurs.

<u>Interpretation? #1: Numerical</u>
There is numerical damping that is being added to the system. It is minimized by making the temporal error convergence specification small.
As the amplitude gets larger, the numerical damping that is added is larger.
At a point in the solution, the residuals bounce about rather than converging any further. This bouncing, for this case, is within the 5% temporal error limit, but outside the 2% temporal error limit. HMMM. If it is strictly tied to the amplitude, a solution with a different initial input step will go to the same amplitude (physical amplitude? Modal amplitude?) before the convergence criteria on 2% fails and still the 5% criteria will not fail there?
<u>Interpretation? #2: Physical</u>
Since the structural model implemented here is linear, any physics-based (i.e. real) LCO must be tied to nonlinearities in the aerodynamics.
If this is a real, physical, LCO, then does this say that the aerodynamic behavior has been driven to a resonant behavior?

nit cycle oscillation onset details
5% temporal error convergence

Decision that was made:
Simulation is prediction a physically unstable aeroelastic system.
Damping extracted from boxed region of the data.

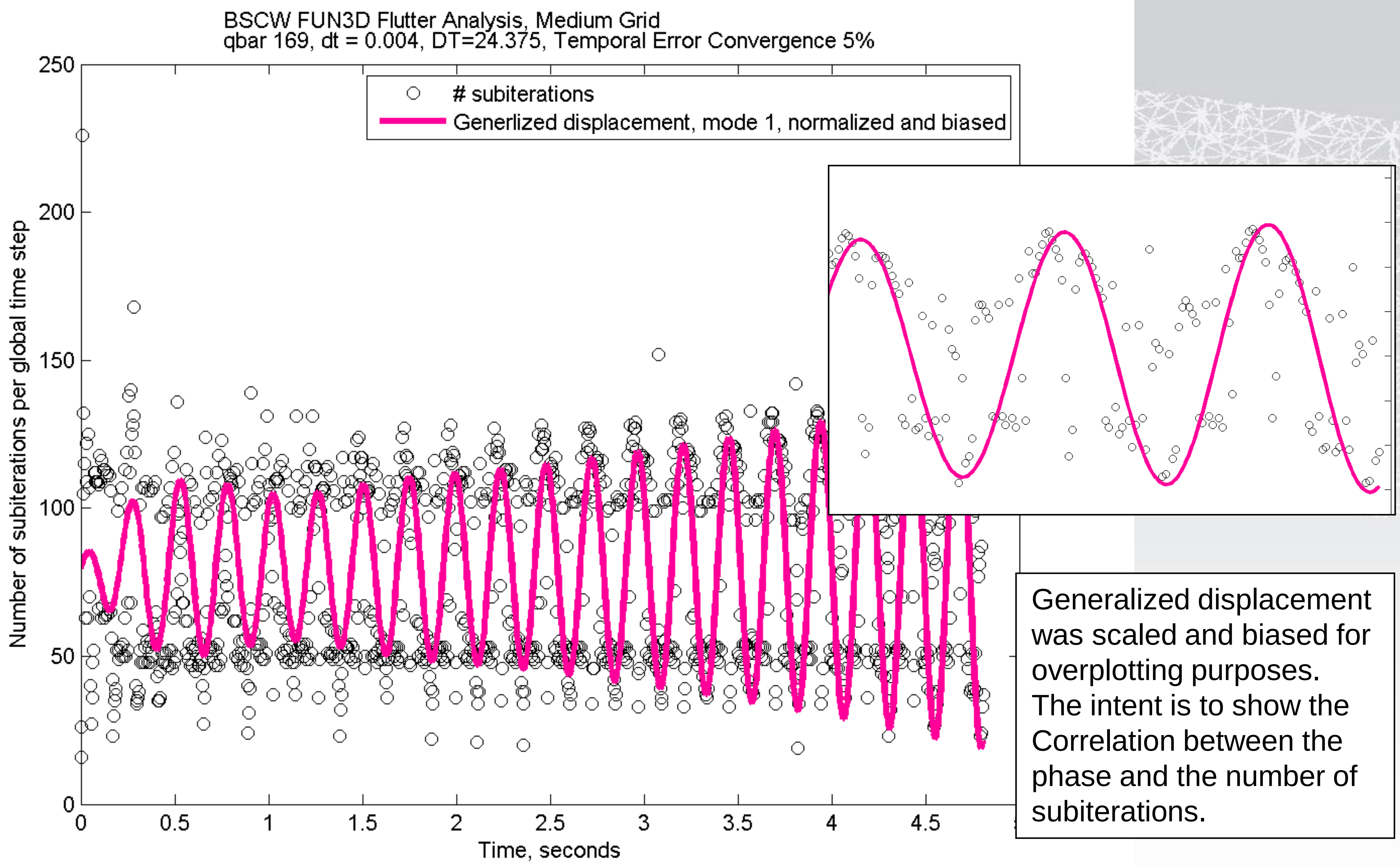




The temporal error convergence criterion is met for as long as I have run this solution. The largest number of subiterations required is nearly 400, just as LCO onset occurs.

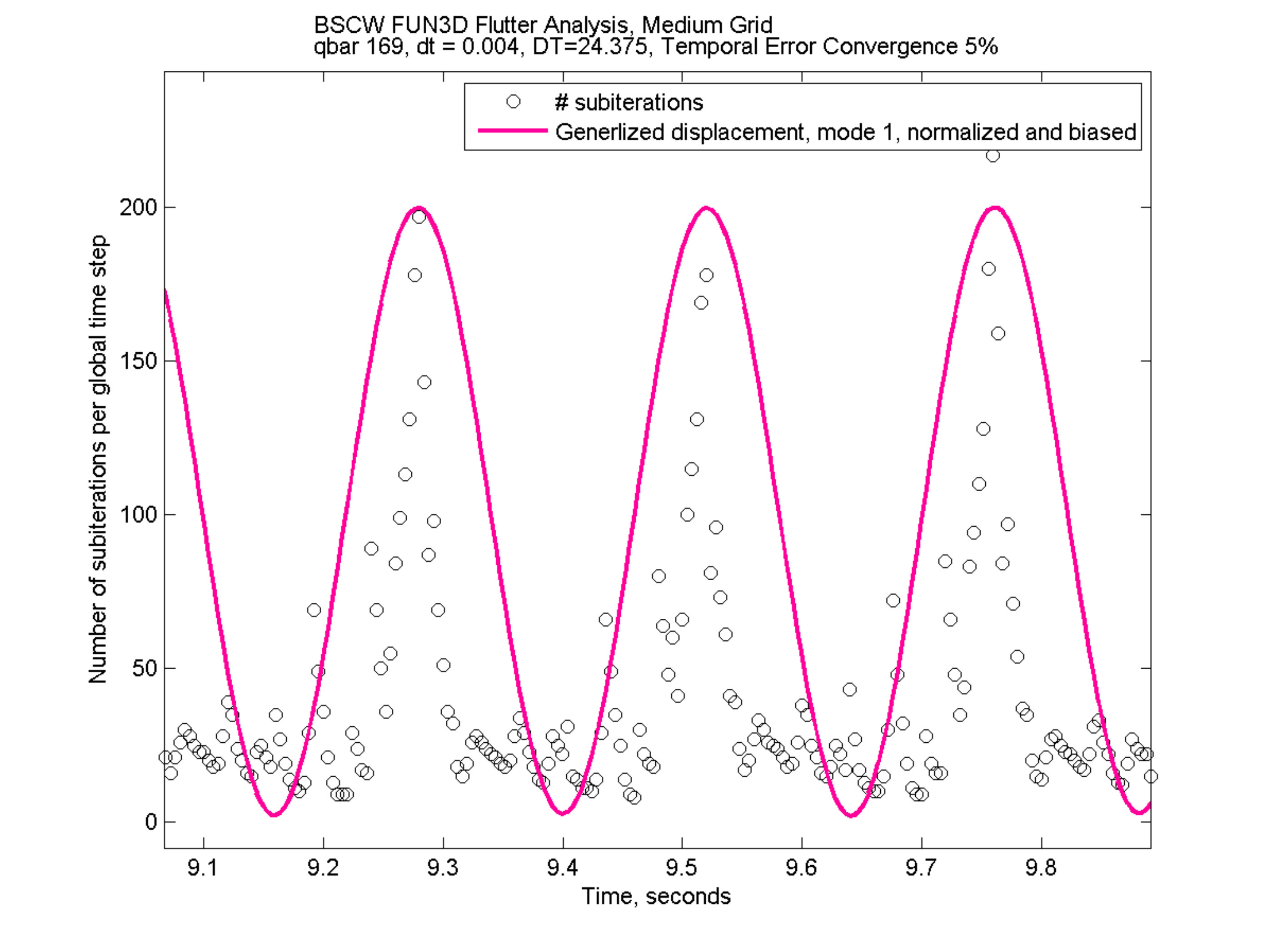
Interpretation? #1: Numerical
There is numerical damping that is being added to the system. It is minimized by making the temporal error convergence specification small. As the amplitude gets larger, the numerical damping that is added is larger.
At a point in the solution, the residuals bounce about rather than converging any further. This bouncing, for this case, is within the 5% temporal error limit, but outside the 2% temporal error limit. HMMM. If it is strictly tied to the amplitude, a solution with a different initial input step will go to the same amplitude (physical amplitude? Modal amplitude?) before the convergence criteria on 2% fails and still the 5% criteria will not fail there?
Interpretation? #2: Physical
Since the structural model implemented here is linear, any physics-based (i.e. real) LCO must be tied to nonlinearities in the aerodynamics.
If this is a real, physical, LCO, then does this say that the aerodynamic behavior has been driven to a resonant behavior?

Q 169, DT 24: 5% temporal error convergence
Number of subiterations required for convergence to 5%; initial divergent section of time history

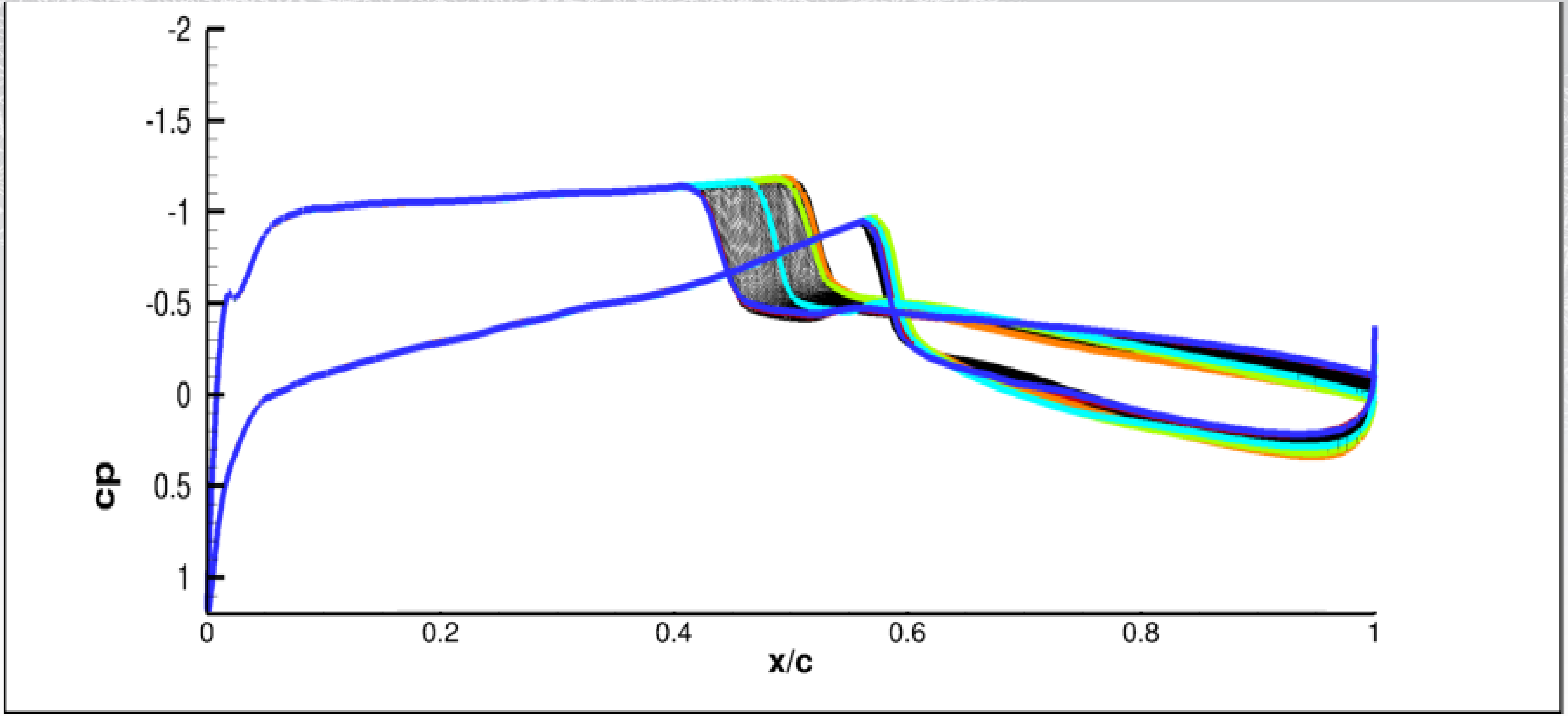


Generalized displacement was scaled and biased for overplotting purposes. The intent is to show the Correlation between the phase and the number of subiterations.

Q 169, DT 24: 5% temporal error convergence
Number of subiterations required for convergence to 5%; portion of time history after LCO onset



Time accurate solutions produced the same range of pressure distribution. Initial location of the shock from the steady distribution was irrelevant to the range of shock motion observed in the time-accurate simulation



FUN3D

FUN3D implementation: Time-discretization

The time change in volumetric quantities is defined as R. The derivative wrt time is approximated by a series expansion. This series is evaluated at a time step, n+1; The quantities at time n+1 are functions of the previous time step. The ϕ in the equation below are the coefficients of that expansion at the denoted time steps. In the initial approximation, given by equation 4 of the reference, the ϕ are generic. However, the sequence of ϕ that is chosen or implemented defines the backward difference formula (BDF) that is applied.

The choice of ϕ governs the accuracy of the temporal discretization.

As the bullet below says, a Nth order BDF is implemented, and it was chosen to linearize R about time level n+1. The equation below is the means by which the solution is advanced in time in FUN3D.

At each m subiteration, the linear system of the equation is iteratively solved using a user-specified number of "Gauss-Seidel sweeps with multi-color ordering."

The pseudo-time term tries to make up for the errors introduced by this linearization. The right hand side of the below equation is the subiteration residual.

- Discretize using Nth order backward differences in time, linearize $\vec{R}$ about time level n+1, and introduce a pseudo-time term:

$$\left[\left(\frac{V^{n+1}}{\Delta\tau}+\frac{V^{n+1}\phi_{n+1}}{\Delta t}\right)\bar{\bar{I}}-\frac{\partial\vec{R}^{n+1,m}}{\partial\vec{Q}}\right]\Delta\vec{Q}^{n+1,m}=\vec{R}^{n+1,m}-\frac{V^{n+1}\phi_{n+1}}{\Delta t}\left(\vec{Q}^{n+1,m}-\vec{Q}^{n}\right)-\ldots+\vec{R}^{n+1}_{GCL}$$

$$=\overline{\vec{R}^{n+1,m}}+O(\Delta t^{N})$$

- Physical time-level t^n ; Pseudo-time level τ^m

Ref: Biedron & Thomas, 2009, Recent Enhancements to the FUN3D Flow Solver for Moving-Mesh Applicatios

FUN3D implementation: Temporal Error Control

- Converging the subiteration residual to machine zero is usually not done.
- Balance must be struck between cost and accuracy (i.e. perform just enough subiterations to obtain a result that is "essentially unchanged" by additional subiterations.
- For the BDF schemes:
  - Estimate the temporal error incurred at each time step (Do this by calculating the residual contribution with 2 different levels of approximations of the time derivatives.).
  - Specify a percentage of the temporal error norm to use as an exit criteria for the subiteration process.
  - Equations to do these steps are shown on the following slide, taken directly from the reference

Ref: Vatsa & Carpenter, 2005, Higher order temporal schemes with error controllers for unsteady Navier-Stokes equations

For the BDF schemes, we can estimate the temporal error incurred at each time step by examining the difference in residual contribution with two different levels of approximations of time derivatives. For example, the time derivatives (up to third order) for dependent variables can be written as

$$\frac{d\mathbf{U}^A}{dt} = \frac{1}{\Delta t}[a_0^A \mathbf{U}^{n+1} + a_1^A \mathbf{U}^n + a_2^A \mathbf{U}^{n-1} + a_3^A \mathbf{U}^{n-2}] \tag{9}$$

$$\frac{d\mathbf{U}^B}{dt} = \frac{1}{\Delta t}[a_0^B \mathbf{U}^{n+1} + a_1^B \mathbf{U}^n + a_2^B \mathbf{U}^{n-1} + a_3^B \mathbf{U}^{n-2}] \tag{10}$$

The superscripts A and B represent two different approximations for the time derivative. For example, superscript A represents the third order accurate BDF3 scheme, whereas superscript B represents the second order BDF2OPT scheme. By subtracting Eq. 10 from Eq. 9, we can estimate the temporal error of the solution when marching from time step n to $n+1$ as follows

$$\frac{d\mathbf{U}^A}{dt} - \frac{d\mathbf{U}^B}{dt} = \frac{1}{\Delta t}[(a_0^A - a_0^B)\mathbf{U}^{n+1} + (a_1^A - a_1^B)\mathbf{U}^n + (a_2^A - a_2^B)\mathbf{U}^{n-1} + (a_3^A - a_3^B)\mathbf{U}^{n-2}] \tag{11}$$

And the temporal error norm is given by

$$E_t = \Delta t \left| \frac{d\mathbf{U}^A}{dt} - \frac{d\mathbf{U}^B}{dt} \right| \tag{12}$$

We can use the temporal error norm, E_t, as an exit criteria for terminating the subiteration loop of the dual time stepping process. Basically, we terminate this subiteration process when the residuals (algebraic errors) drop below a specified fraction of E_t. Such a strategy results in uniform temporal accuracy for all time steps, and eliminates the guess work for selecting iteration count or some other preselected exit criteria. The resulting scheme is also more efficient, because we do not waste computational resources by performing unneeded iterations.

Ref: Vatsa & Carpenter, 2005, Higher order temporal schemes with error controllers for unsteady Navier-Stokes equations

The following slides were taken from

- Bob Biedron’s slides:
  - Fun3D v12.4 Training
  - Session 10: Time dependent simulations
  - March 2014

Governing Equations

- Arbitrary Lagrangian-Eulerian (ALE) Formulation

$$\frac{\partial(\vec{Q}V)}{\partial t} = -\oint_{\partial V}\left(\overline{\overline{F}} - \vec{q}\vec{W}^T\right)\cdot\vec{n}dS - \oint_{\partial V}\overline{\overline{F_v}}\cdot\vec{n}dS = \vec{R} \qquad \vec{Q} = \frac{\oint_V \vec{q}\,dV}{V}$$

$\vec{W} =$ Arbitrary control surface velocity; Lagrangian if $\vec{W} = (u,v,w)^T$ (moves with fluid); Eulerian if $\vec{W} = 0$ (fixed in space)

- Discretize using N^th order backward differences in time, linearize $\vec{R}$ about time level n+1, and introduce a pseudo-time term:

$$\left[\left(\frac{V^{n+1}}{\Delta\tau} + \frac{V^{n+1}\phi_{n+1}}{\Delta t}\right)\overline{\overline{I}} - \frac{\partial\vec{R}^{n+1,m}}{\partial\vec{Q}}\right]\Delta\vec{Q}^{n+1,m} = \vec{R}^{n+1,m} - \frac{V^{n+1}\phi_{n+1}}{\Delta t}\left(\vec{Q}^{n+1,m} - \vec{Q}^{n}\right) - \ldots + \vec{R}^{n+1}_{GCL}$$

$$= \overline{\vec{R}^{n+1,m}} + O(\Delta t^N)$$

- Physical time-level t^n ; Pseudo-time level τ^m
- Want to drive ***subiteration residual*** $\overline{\vec{R}^{n+1,m}} \to 0$ using pseudo-time subiterations at each time step – much more later – otherwise you have more error than the expected $O(\Delta t^N)$ truncation error

KEY POINT

Time Advancement - Namelist Input

- The `&nonlinear_solver_parameters` namelist in the `fun3d.nml` file governs how the solution is advanced in time
- Relevant entries - *default values shown* - some definitely need changing:

```
&nonlinear_solver_parameters
  time_accuracy        = 'steady'   (i.e. not time accurate)
  time_step_nondim     = 0.0
  subiterations        = 0
  schedule_iteration   = 1      50
  schedule_cfl         = 200.0 200.0
  schedule_cflturb     = 50.0   50.0
  pseudo_time_stepping = "on"
  temporal_err_control = .false.
  temporal_err_floor   =  0.1
/
```

- Let's look at these in some detail (defer `time_step_nondim` to last)

Time Advancement - Order of Accuracy

- Currently have several types of backward difference formulae (BDF) that are controlled by the `time_accuracy` component:
  - In order of formal accuracy: BDF1 (`1storder`), BDF2 (`2ndorder`), $BDF2_{OPT}$ (`2ndorderOPT`), BDF3 (`3rdorder`), MEBDF4 (`4thorderMEBDF4`)
  - Can pretty much ignore all but $BDF2_{OPT}$ and BDF2
    - BDF1 is least accurate; little gain in CPU time / step over 2nd order; for moving grids can be helpful to start out with BDF1 (rare)
    - BDF3 not guaranteed to be stable; feeling lucky?
    - MEBDF4 only efficient if working to very high levels of accuracy - *including spatial accuracy* - generally ***not*** for practical problems
    - $BDF2_{OPT}$ *(recommended)* is a stable blend of BDF2 and BDF3 schemes; formally 2nd order accurate but error is ~1/2 that of BDF2; also allows for a more accurate estimate of the temporal error for the error controller (p.8)

Time Advancement - Subiterations (1/4)

- Can think of each time step as a mini steady-state problem
- Subiterations (`subiterations` > 0) are essential
  - Subiteration control in *each time step* operates exactly like iteration control in a steady state case:
    - CFL ramping is available for mean flow and turbulence model – however, be aware that ramping schedule should be < `subiterations` or the specified final CFL won't be obtained
      - *We almost never ramp CFL for time-accurate cases*
      - If used, CFL ramping starts over each time step
    - Caution: the *spatial* accuracy flag, `first_order_iterations`, starts over each time step, so make sure you don't have this on
- Pseudo-time term helpful for large time steps
  - We *always* use it in our applications
  - `pseudo_time_stepping = "on" (default)`

Time Advancement - Subiterations (2/4)

- How many subiterations?
  - In theory, should drive subiteration residual "to zero" each time step – but you cannot afford to do that
  - Otherwise have additional errors other than $O(\Delta t^2)$ (if 2nd order time)
- In a perfect world, the answer is to use the ***temporal error controller***
  - `temporal_err_control = .true.`
  - `temporal_err_floor = 0.1` => iterate until the subiteration residual is 1 order lower than the (estimated) temporal error (0.01 => 2)
  - Subiterations kick out when this level of convergence is reached OR subiteration counter > `subiterations`
  - (empirically) 1 order is about the minimum; 2 orders is better, BUT…
  - Often, either the turbulence residual converges slowly or the mean flow does, and the max subiterations you specify will be reached
  - When it kicks in, the temporal error controller is the best approach, and the most efficient; even if it doesn't kick in, it can be informative

Time Advancement - Subiterations (3/4)

- Be wary reaching conclusions about the effect of time-step refinement unless the subiterations are “sufficiently” converged for each size step
- How to monitor and assess the subiteration convergence:
  - Printed to the screen, so you can “eyeball” it
  - With temporal error controller, if the requested tolerance is not met, message(s) will be output to the screen:
    - **`WARNING: mean flow subiterations failed to converge to specified temporal_err_floor level`**
    - **`WARNING: turb flow subiterations failed to converge to specified temporal_err_floor level`**
    - Note: when starting unsteady mode, first timestep ***never*** achieves target error (no error estimate first step, so target is 0)
    - Note: x-momentum residual (`R_2`) is the mean-flow residual targeted by the error controller
  - Plot it (usually best)

Determining the Time Step

- Identify a ***characteristic time*** t^*_{chr} that you need to resolve with some level of accuracy in your simulation; perhaps:
  - Some important shedding frequency f^*_{shed} (Hz) is known or estimated $t^*_{chr} \sim 1 / f^*_{shed}$
  - Periodic motion of the body $t^*_{chr} \sim 1 / f^*_{motion}$
  - A range of frequencies in a DES-type simulation $t^*_{chr} \sim 1 / f^*_{highest}$
  - If none of the above, you can estimate the time it takes for a fluid particle to cross the characteristic length of the body, $t^*_{chr} \sim L^*_{ref} / U^*_{ref}$
  - $t_{chr} = t^*_{chr}\, a^*_{ref}\, (L_{ref}/L^*_{ref})$ (comp) $t_{chr} = t^*_{chr}\, U^*_{ref}\, (L_{ref}/L^*_{ref})$ (incomp)
- Say you want N time steps within the characteristic time:
  - $\Delta t = t_{chr} / N$ = `time_step_nondim`
- Figure an absolute *minimum* of N = 100 for reasonable resolution of t_{chr} with a 2nd order scheme - really problem dependent (*frequencies > f* may be important*); but don't over resolve time if space is not well resolved too

Example - Unsteady Flow at High Alpha (1/7)

- Consider flow past a (2D) NACA 0012 airfoil at 45^o angle of attack - the flow separates and is unsteady
  - Re_{c^*} = 4.8 million, M_{ref} = 0.6, *assume* a^*_{ref} = 340 m/s
  - chord = 0.1m, chord-in-grid = 1.0 so L_{ref}/L^*_{ref} = 1.0/0.1 = 10 (m^{-1})
  - Say we know from experiment that lift oscillations occur at ~450 Hz
  - $t^*_{chr} = 1 / f^*_{chr}$ = 1 / 450 Hz = 0.002222 s
  - $t_{chr} = t^*_{chr}\, a^*_{ref}\, (L_{ref}/L^*_{ref})$ = (0.002222)(340)(10) = 7.555
  - $\Delta t = t_{chr} / N$ so Δt = 0.07555 for 100 steps / lift cycle
  - By way of comparison, for M = 0.6, a^*_{ref} = 340 m/s, and L^*_{ref} = 0.1 m it takes a fluid particle ~ (0.1)/(204) = 0.00049 s to pass by the airfoil; this leads to smaller, more conservative estimate for the time step, by about a factor of 4

The following were taken from:

- Bob Biedron’s slides:
- FUN3D v12.7 Training
- Session 16: Aeroelastic Simulations
- June 2015

Dynamic Aeroelastic Coupling

- For time-accurate aeroelastic modeling, FUN3D currently relies on a modal decomposition approach
  - *Linear* structural dynamics equation (see AIAA 2009-1360) - appropriate for small deflections (e.g. during flutter onset)
  - Deflection assumed a linear combination of eigenmodes (mode shapes)
    - FEM model used a priori to extract eigenmodes / frequencies
    - Deflection represented as linear combination of eigenmodes (mode shapes)
    - Typically only a limited set of the "important" eigenmodes retained
  - A *nonlinear* aerodynamics model is used (FUN3D), so effects of shocks and viscosity can be captured in the flow field
  - Middleware (e.g. DDFdrive) maps eignenmodes onto CFD surface in a one-time preprocessing step; at startup FUN3D reads these
  - Aerodynamics at current time step determine the weight applied to each eigenmode; current shape is weighted sum of eigenmodes

Dynamic Aeroelastic Coupling

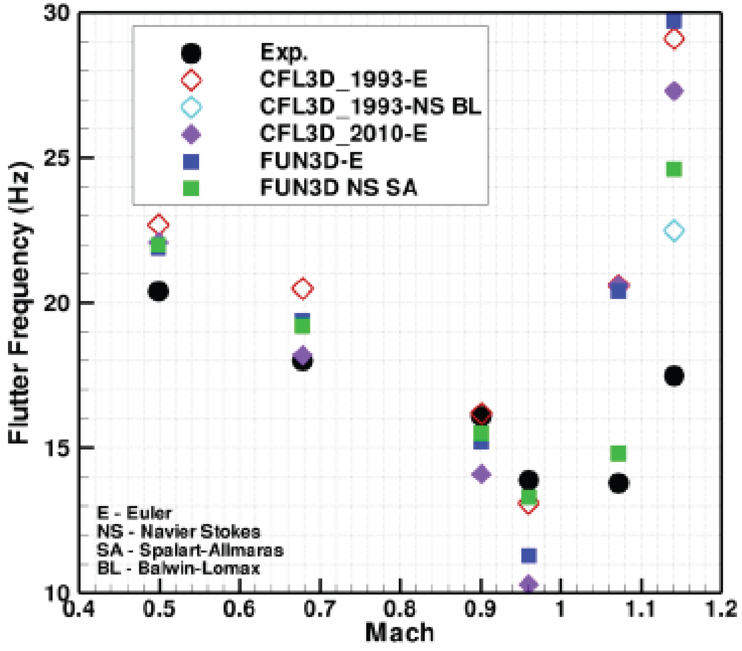
- Typical flutter assessment process
  1. Run FEM to extract and output the desired modes
  2. Run FUN3D in steady-state mode with `--write-massoud` CLO to generate a steady-state solution and provide a file(s) that will serve as a template for subsequent mode-shape files
  3. Map the FEM modes onto the template (DDFdrive can be used) to generate one surface file per mode
  4. Run FUN3D in moving-grid, time-dependent mode, using modal aeroelastic inputs (upcoming slides) with critical damping ratio ~1
     - This yields a static aeroelastic deflection, the starting point for flutter assessment
     - Symmetric configuration at zero AoA can skip this step (as in the case in tutorial example covered later)
  5. Run FUN3D in moving-grid, time-dependent mode, using modal aeroelastic inputs with a initial perturbation to “kick” elastic response; does response grow or decay?

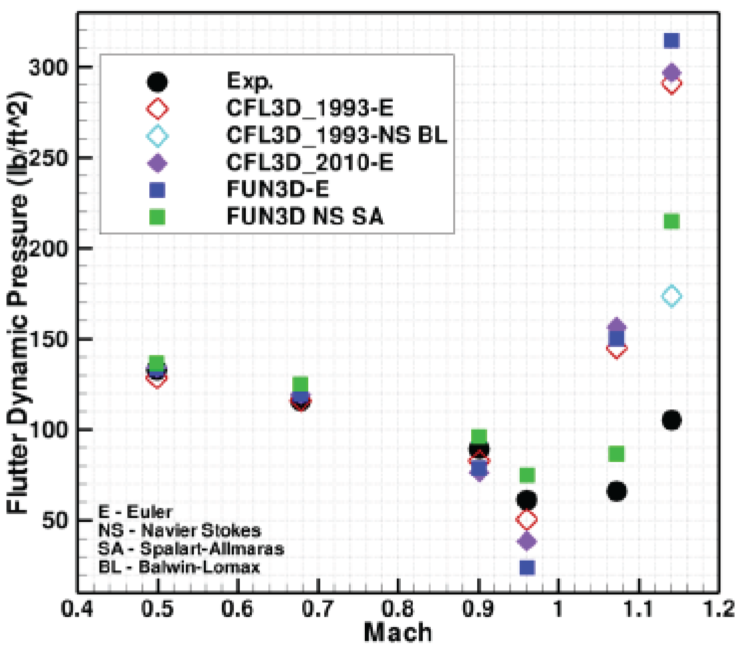
Tutorial Case: AGARD 445 Wing (5/8)

- Step 5 (cont) Setting the FUN3D timestep
  - From experiment, the flutter frequency at Mach 0.9 is $\omega^* \sim 120$ rad/sec, so we'll assume we need to resolve at least up to this frequency
  - From nondimensionalization slides, have
    - $t_{chr} = (1/f^*)\, a^*_{inf}\, (L_{ref}/L^*_{ref}) = (2\pi/\omega^*)\, a^*_{inf}\, (L_{ref}/L^*_{ref})$
    - $\Delta t = t_{chr}/N$
  - Take 200 steps to resolve this frequency; from previous slide have
    - U^*_{inf} = 973 ft/sec so at M=0.9, a^*_{inf} = 1081 ft/sec
    - The grid is in ft so $L_{ref}/L^*_{ref} = 1$
    - Δt = (6.28/120) 1081 (1) / 200 = 0.283 (tutorial uses 0.3)
    - In practice, would need to do a time step refinement to verify this time step is adequate (at this time step, mode 4 resolved with only ~42 steps/period)

Tutorial Case: AGARD 445 Wing (7/8)

- The dynamic pressure (q = 75 psf) in the tutorial does not lead to flutter at M = 0.9 – so we would need to increment q and repeat until we found a response that grows with time (M = 78.6 psf) – then repeat over the Mach range
- Pawel Chwalowski, Aeroelasticity Branch, NASA Langley has carried out this exercise and provided these plots (not part of tutorial):





General

Consider the subtleties of the higher dimensional cases:

- Consider the two-dimensional case

$$C_{\max} \geq u_x \frac{\Delta t}{\Delta x} + u_y \frac{\Delta t}{\Delta y} = \Delta t \left(\frac{u_x}{\Delta x} + \frac{u_y}{\Delta y} \right)$$

- and the n-dimensional case

$$C_{\max} \geq \Delta t \sum_{i=1}^{n} \frac{u_i}{\Delta x_i}$$

Notes on interpreting and applying these concepts:

The grid spacing isn't required to be the same for the different grid directions, but should be related to the velocity in that component direction, with consideration of the chosen time step.

Can the Courant number criteria be applied as an independent assessment for each of the component directions?
The implication in using the combined criterion is that the non-flow-wise directions contribute little to the Courant number.

An approach is to use the grid density in one spatial direction to set the time step and then use that time step to set the grid density in the other directions (or verify that those directions comply with the observability rule).

The additive relationship above provides a conservative methodology for considering the directions simultaneous.

And further notes from CFD online

- The value of Cmax changes with the method used to solve the discretized equation.
- Explicit (time-marching) solvers typically use (or require) Cmax =1 <to produce a stable solution>
- Implicit (matrix) solvers are usually less sensitive to numerical instability so larger values of Cmax may be tolerated

The above requirements on CFL number are solely to address the issue of stability of hyperbolic equations.
They do not address the requirements for resolving flow features.

Suggested rule of thumb from Spalart:

"A CFL number of approximately 1 is necessary for accurate prediction of large eddies, which is a requirement in both grid spacing and time step"

$$\Delta x_o / \Delta t = U_{\max}$$

where Δx_o is the grid spacing in the LES focus region and $U_{\max}$ is the maximum velocity in that region

Jen's notes to self: Parenthetically, CMM says that Umax can safely be assumed to be 1.5 to 2 times higher than the freestream velocity. In general, this seems like a safe assumption- the locally-accelerated flow will result in a shock and prevent it exceeding this 1.5 to 2 times limit? Under what circumstances might you get supersonic bubble for very low subsonic flow? To me this seems like one case where the parenthetical might be wrong. The other case that occurs to me is for vortical flow. Something swirling around has a higher point velocity because it is rotationally moving in addition to translationally moving. But U is the stream-wise component? So this isn't a contributor? Or is U being used in a more generic sense? What about x? Is it being used in a generic sense or only in the steam-wise direction?

CFL number

- Courant-Friedrichs-Lewy (CFL) condition
- A stability criterion for hyperbolic equations
- From CFD Online Wiki:
  - … for example, if a wave is moving across a discrete spatial grid and we want to compute its amplitude at discrete time steps of equal length then this length must be less than the time for the wave to travel to adjacent grid points
  - When the grid point separation is reduced, the upper limit for the time step also decreases
- Courant number, C: (for 1-dimensional case)

$$C_{\max} \geq u \frac{\Delta t}{\Delta x}$$

CFL number comments from Cummings, Morton & McDaniel

- If the CFL number or the Courant number is less than 1, the grid and time step are sufficiently sized to capture flow phenomena that occur at the given velocity. A very small Courant number may indicate that the time steps are being wasted and could be made longer.
- If the CFL number is greater than 1, the time step is too large relative to the grid size.

$$C_{\max} \geq C = u\frac{\Delta t}{\Delta x}$$

$$u = C\frac{\Delta x}{\Delta t} \qquad C = \frac{u}{u_{\mathrm{lim}it\_of\_grid}}$$

On grid sizing within the unsteady flow structure

- Cummings, Morton & McDaniel suggest that at least 5 grid points (or cells) *are required to* model a large-scale flow feature correctly, meaning that the grid needs to be at least 5 times less than the smallest flow structure resolved by LES (referencing Spalart[7] and Schiff[8]).
- Interpretation and experiences from experimental data processing:
    - This is akin to the data processing rule of thumb of using a factor relative to the Nyquist frequency to discern the highest frequency of interest. Here, they use a factor of 2.5 (My personal guideline is a factor of between 2 and 5 depending on the data quality and the resources available.) on the spatial frequency that they are trying to resolve, relative to the spatial Nyquist. They are only using a factor of 2.5 on the spatial Nyquist. They are saying that you need 5 grid points per cycle. Nyquist would say that you need 2 grid points per cycle. 5/2 = 2.5. So they are actually at the lower end of my personal guideline.
    - But… they should be able to resolve a flow structure with only 2 samples per period. In a nice perfect CFD world without any noise, this should be more achievable than with experimental data. This also, however, implies that the flow structure is sinusoidal. Need to examine the quoted references to see if they just weren't confident or satisifed with their results generated with 2 samples per period…

On grid sizing within the unsteady flow structure

- Cummings, Morton & McDaniel suggest that at least 5 grid points (or cells) *are required to* model a large-scale flow feature correctly, meaning that the grid needs to be at least 5 times less than the smallest flow structure resolved by LES (referencing Spalart[7] and Schiff[8]).
- Interpretation and experiences from experimental data processing:
  - Akin to data processing practices of using a factor relative to the Nyquist frequency to discern the highest frequency of interest. Practitioners generally use a factor between 2 and 5 based on personal preference, quality of data set, resources available. (here, this is temporal)
  - The CMM comment is regarding the spatial frequency, relative to the spatial Nyquist. They are saying that you need 5 grid points per cycle. Nyquist would say that you minimally need 2 grid points per cycle. 5/2 = 2.5. Compare this to the 2-5 factor used by data processing practitioners

AePW-2 Analysis Codes Utilized

Linear	RANS, Uncoupled	Euler, Coupled	RANS, Coupled	Hybrid RANS/LES
• MSC NASTRAN	• SU2	• OpenFoam	• CFD++ • Aero • EZNSS • Edge • FUN3D • EZAir • Star_CCM+ • Loci/Chem • Fluent • CFX • SUMAD • ENFLOW • NSMB	• Edge • FUN3D • EZAir