

#### **Introduction to ANSYS CFX**

**Realize Your Product Promise®** 

#### **ANSYS** Introduction

- Lecture Theme:
  - Performing a transient calculation is similar to performing a steady-state calculation, but there are additional considerations. More data is generated and extra inputs are required.
- Learning Aims you will learn:
  - Transient flow calculations are becoming increasingly common due to advances in high performance computing (HPC) and reductions in hardware costs. You will understand what transient calculations involve and be able to perform them with confidence





- Unsteadiness can be
  - Natural growth of instabilities, non-equilibrium initial state
    - natural convection flows, turbulent eddies, fluid waves
  - Forced time-dependent boundary conditions, source terms, accumulation/depletion...
    - pulsing flow, rotor-stator interaction, tank filling
- Transient simulations allow extraction of
  - Natural frequencies (e.g. Strouhal Number)
  - Spectral data fast Fourier transform (FFT)
  - Time-averaged and/or RMS values
  - Time-related parameters (e.g. time to cool a hot solid)
- More expensive to run & complex to analyse than steady state

**Motivation** 





Output

Introduction

© 2015 ANSYS, Inc. March 13, 2015 3

Setup

**Time Steps** 

**ANSYS** Confidential

# **ANSYS** How to Solve a Transient Case

4

- Transient simulations are solved by computing a solution for many discrete points in time
- At each time point we must iterate to the solution



## **ANSYS** How to Solve a Transient Case

- Similar setup to steady state
- The general workflow is:
  - Set the Analysis Type to Transient
  - Specify the transient time duration to solve and the time step size
  - Set up physical models and boundary conditions as usual
    - Boundary conditions may change with time
  - Prescribe initial conditions
    - Use physically realistic initial conditions, such as a steady solution
  - Assign solver settings
  - Configure transient results files, transient statistics, monitor points
  - Run the solver

Inti	oduction	Motivation	Setup	Time Steps	Output
© 2015 A	NSYS, Inc. Ma	arch 13, 2015	ANSYS Confidential		

Analysis Type		
Option	Steady State	~
	Steady State	
	Transient	

# **ANSYS** Time Duration and Time Step

- Set the Time Duration
  - Specify time period, either per run or overall
- Set the Time Step size

6

- Can be fixed, a list or expression
- Time Duration/Timesteps
  - Should resolve in a reasonable number of outer iterations

Outline Analysis Type	
Details of Analysis Type i	n Flow Analysis 1
Basic Settings	
External Solver Couplin	g 🖂 🖂
Option	None 🖌
Analysis Type	
Option	Transient 🛛 💙
Option	Total Time 🔽
Total Time	10 [s]
Time Steps	Θ
Option	Timesteps 🖌
Timesteps	0.1 [s]
Initial lime	
Option	Automatic with Value
Time	0 [s]

Σ	Introduction	Motivation	Setup	Time Steps	Output
	© 2015 ANSYS, Inc.	March 13, 2015	ANSYS Confidential		

## **ANSYS** Time Duration and Time Step

- The Time Step size is an important parameter in transient runs
  - It must be small enough to resolve time-dependent features



7

# **ANSYS** Time Duration and Time Step

- The quantity of interest may be changing very slowly (temperature in a solid), but you may not be able to use a large timestep if other quantities (velocity) have smaller timescales
- The Courant Number can be used to estimate a time step:

$$CourantNumber = \frac{Velocity \times \Delta t}{ElementSize}$$

- Number of mesh elements the fluid passes through in one timestep
- Average and maximum Courant number is reported in the Solver out file
- Recommended average values are 2 10, in some cases higher values are acceptable
- A smaller timestep will typically improve convergence

8

 Introduction
 Motivation
 Setup
 Time Steps
 Output

 © 2015 ANSYS, Inc.
 March 13, 2015
 ANSYS Confidential
 Output

# **ANSYS** Boundary Conditions

- If required, boundary conditions can be functions of time instead of constant values
  - Velocities, Mass flows, pressure conditions, temperatures, etc. can all be expressed as functions
  - In CEL expressions use "t" or "Time"
  - Can read in time varying experimental data through User FORTRAN







- Physically realistic initial conditions should be used
  - A converged steady state solution is often used as the starting point
- If a transient simulation is started from an approximate initial guess, the early timesteps will not be accurate
  - The first few timesteps may not converge
  - A smaller time step may be needed initially to maintain solver stability
  - For cyclic behavior the first few cycles can be ignored until a repeatable pattern is obtained



IntroductionMotivationSetupTime StepsOutput10© 2015 ANSYS, Inc.March 13, 2015ANSYS Confidential



- Transient scheme numerical algorithm for transient term
  - Default is recommended
- Timestep Initialisation controls starting point for next timestep
  - Default: Automatic is recommended
- Aim at 3-5 loops to converge each timestep
  - Complex physics may need more loops
  - Reduce timestep size rather than increase number of loops
- Solution proceeds to next timestep regardless of whether the convergence criteria was met → monitor the solution

Outline Solver	r Control ontrol in Flow Analysis 1
Basic Settings	Equation Class Settings Advanced Optio
-Advection Sche	me
Option	High Resolution 🔽
Transient Schen	ne
Option	Second Order Backward Euler 🗸 🗸
-Timestep Initia	lisation
Option	Automatic 😪
C Lower Co	urant Number (
C Upper Co	urant Number [
-Turbulence Num	erics
Option	First Order 😽 😽
Convergence C	ontrol
Min. Coeff. Loop:	5 1
Max. Coeff. Loop	ns 10
	Service 1
Timescale Conti	rol Coefficient Loops 😪
-Convergence C	riteria
Recidual Type	RMS V
Conservet	I.C-4
- Elansed Wa	Il Clock Time Control
	ontrol

IntroductionMotivationSetupTime StepsOutput11© 2015 ANSYS, Inc.March 13, 2015ANSYS Confidential



- Transient Results
  - Need to define under Output Control
- Transient Results Option
  - Standard (full result file)
  - Smallest (restart possible)
  - Selected Variables
    - Pick only variables of interest → smaller files
    - Specify whether or not to Include Mesh
- Output Frequency
  - Controls how often results are written

Outline Output Co etails of Output Cont	ntrol rol in Flow Anal	ysis 1	E
Results Backup	Trn Results	Trn Stats	Monitor
- Transient Results			
Transient Results	1		<ul><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li></ul>
Transient Results 1 Option	Standard		
File Compression	Default tion Residuals —		<b>∼</b>
Output Frequenc	Veriebles List		
Option	Timestep I	Interval	*
Timestep Interval	Timestep Every Tim Time Inter Timestep Time List None	Interval estep rval List	

Σ	Introduction	Motivation	Setup	Time Steps	Output
.2	© 2015 ANSYS, Inc.	March 13, 2015	ANSYS Confidential		



- Transient Statistics
  - Used to generate running statistics for solution variables
- Arithmetic Average, RMS, Minimum, Maximum, Standard Deviation and Full (everything) are available options
- Pick the variables of interest
- Start and Stop Iteration List defines when to begin and end collecting the statistics



Σ	Introduction	Motivation	Setup	>	Time Steps	$\boldsymbol{\boldsymbol{\lambda}}$	Output
13	© 2015 ANSYS, Inc.	March 13, 2015	ANSYS Confidential				



- Monitor Coefficient Loop Convergence
  - To check that solution is converging within a timestep
- Interval Definition for Monitor Statistics is in terms of Time or Timesteps

Tip: Monitors create a transient history chart in the CFX Solver Manager. This is easier and quicker than creating the chart in CFD-Post and does not require writing a series of transient results files

Outline Output Co	ntrol	×						
Details of Output Cont	rol in Flow Analysis 1							
Results Backup Trn Results Trn Stats Monitor								
Monitor Options								
Monitor Coefficie	Monitor Coefficient Loop Convergence							
Monitor Balances -		• 🗄 🚽 🚽						
-Monitor Forces		• <b>E</b>						
_Monitor Residuals -		• <b>=</b>						
-Monitor Totals		• <b>E</b> _						
-Monitor Particles -		.⊞						
Monitor Point 1								
Option	Expression •							
Expression Value	volumeAve(Temperature)@Domain 1							
Coordinate Frame	Coord 0 🗸							
Monitor Statistics								
Interval Option	Moving Interval							
Statistics List Arithmetic Average 👻 📖								
Interval Definition								
Option	Timesteps 🗸							
Number of Timesteps	Time							
	Timesteps							

Σ	Introduction	Motivation	Setup	Time Steps	Output	
14	© 2015 ANSYS, Inc.	March 13, 2015	ANSYS Confidential			



- Each time step now contains coefficient loop output
- Courant number information at the start of each timestep
- Make sure convergence has been achieved by the end of the timestep by monitoring the RMS and MAX residual plots

	Timestepping Information	
Timestep	RMS Courant Number	Max Courant Number
2.5000E-04	0.12	2.42
TIME STEP = 945 SIMUL (THIS RUN: 236	ATION TIME = 5.9000E-02 5.9000E-02	CPU SECONDS = 8.384E+04 8.384E+04

COEFFICIENT LOOP ITERATI	[ON] =	1	CPU	SECONDS = 8.384E+0
Equation	Rate	RMS Res	Max Res	Linear Solution
U-Mom   V-Mom   W-Mom   P-Mass	0.89 0.94 0.83 0.94	4.0E-07 3.6E-07 3.1E-07 4.0E-06	1.2E-05 1.0E-05 1.1E-05 4.6E-04	1.7E-01 ok 2.6E-02 OK 1.1E-01 ok 6.2 3.6E-02 OK
K-TurbKE   E-Diss.K	1.00 0.97	4.7E-05 2.9E-05	9.2E-04 7.7E-04	5.8 2.8E-06 OK 7.5 8.7E-07 OK
COEFFICIENT LOOP ITERATI	[ON =	2	CPU	SECONDS = 8.395E+04
Equation	Rate	RMS Res	Max Res	Linear Solution
U-Mom   V-Mom   W-Mom   P-Mass	1.05 0.93 1.23 0.72	4.2E-07 3.4E-07 3.9E-07 2.9E-06	1.6E-05 1.1E-05 1.5E-05 3.7E-04	1.2E-01 ok 7.4E-02 OK 1.1E-01 ok 6.2 4.6E-02 OK

IntroductionMotivationSetupTime StepsOutput15© 2015 ANSYS, Inc.March 13, 2015ANSYS Confidential

#### **ANSYS**

#### Workshop 06 Vortex Shedding

- Simulation of vortex shedding behind a cylinder
  - Transient analysis
  - Fast Fourier Transform
  - Calculation of Strouhal number for comparison with experiment



Error Types

Introduction

**Best Practices for Meshing**