NASA/TM-2006-214301

CFL3D Version 6.4—General Usage and Aeroelastic Analysis

Robert E. Bartels, Christopher L. Rumsey, and Robert T. Biedron
Langley Research Center, Hampton, Virginia

April 2006

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the lead center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers, but having less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or co-sponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized databases, organizing and publishing research results ... even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at ***http://www.sti.nasa.gov***

- E-mail your question via the Internet to help@sti.nasa.gov

- Fax your question to the NASA STI Help Desk at (301) 621-0134

- Phone the NASA STI Help Desk at (301) 621-0390

- Write to:
  NASA STI Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076-1320

NASA/TM-2006-214301

CFL3D Version 6.4—General Usage and Aeroelastic Analysis

Robert E. Bartels, Christopher L. Rumsey, and Robert T. Biedron
Langley Research Center, Hampton, Virginia

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23681-2199

April 2006

Available from:

NASA Center for AeroSpace Information (CASI)
7121 Standard Drive
Hanover, MD 21076-1320
(301) 621-0390

National Technical Information Service (NTIS)
5285 Port Royal Road
Springfield, VA 22161-2171
(703) 605-6000

Abstract

This document contains the course notes on the computational fluid dynamics code CFL3D version 6.4. It is intended to provide from basic to advanced users the information necessary to successfully use the code for a broad range of cases. Much of the course covers capability that has been a part of previous versions of the code, with material compiled from a CFL3D v5.0 manual and from the CFL3D v6 web site prior to the current release. This part of the material is presented to users of the code not familiar with computational fluid dynamics. There is new capability in CFL3D version 6.4 presented here that has not previously been published. There are also outdated features no longer used or recommended in recent releases of the code. The information offered here supersedes earlier manuals and updates outdated usage. Where current usage supersedes older versions, notation of that is made. These course notes also provides hints for usage, code installation and examples not found elsewhere.

Table of Contents

Table of Contents

Table of Contents

Introduction

CFL3D is a Reynolds-averaged Navier-Stokes flow solver for structured grids. The original version, developed in the 1980's was given its name to denote its origin in the Computational Fluids Laboratory. CFL3D solves the time-dependent conservation law form of the equations using a semi-discrete finite-volume approach with upwind-biasing of the convective and pressure terms and central differencing of the shear stress and heat transfer terms. Numerous turbulence models are provided. Grids must be supplied external to the code.

The present document is an outgrowth of a course that was presented on the computational fluid dynamics code CFL3D version 6.4. Publication of this material in the present form makes it available to many more users of the code. This document should provide the information necessary to successfully use the code for a broad range of cases. The target audience ranges from basic to advanced users.   New users should find useful the discussion of general features of the code and the many options that are available, code set up, creation of grids and input for steady and unsteady computations. New features that are available in CFL3D version 6.4 will also be discussed. There is a lengthy discussion of issues related to unsteady computations, moving and deforming meshes, aeroelastic simulations and parallel computing using the message passing interface (MPI). Within these discussions there are detailed instructions on input parameters, their use within the code, as well as illustrative examples.

Much of the course covers capability that has been a part of previous versions of the code, with material compiled from a CFL3D v5.0 manual and from the CFL3D v6 web site prior to the current release.   This part of the material is presented to users of the code not familiar with computational fluid dynamics. There is also new capability in CFL3D v6.4 that has not previously been published. This course intends to acquaint users with this new capability. There are also outdated features no longer used or recommended in recent releases of the code. The information offered here supersedes earlier manuals and updates outdated usage. Where current usage supersedes older versions, notation of that is made.   This document also provides hints for usage and code installation not found elsewhere.

Introduction

There is much information in the CFL3D v5.0 manual that is not presented in these notes. The use of patched, overset or embedded grids is not discussed here. Since the intention is to provide users a practical guide on code usage, there is very little discussion of the fluid dynamics equations and computational method used. This information is available in the CFL3D v5.0 manual.

The attempt is to organize this material in an intuitive way. Topics are presented in the order they would be encountered in the process of building up a real test case. The ordering of the information reflects the course instructor's own learning experience with CFL3D. Others may order the material differently. This course is not comprehensive. Because of the vast number of ways in which CFL3D can be used there are many input options that are not discussed and none are discussed in complete detail. Those that are discussed are the more commonly used features. By the end of the course the reader should be able to perform a number of different analyses with the code. If the reader is interested in more detail also consult the CFL3D v6 web page and the CFL3D v5.0 user's manual. These references are listed at the back of this document.

What's New in CFL3D v6.4

There are new capabilities in CFL3D v6.4 presented in this document. They are:

- New mesh deformation scheme with more options available.
- New aeroelastic analyses not available in previous versions of CFL3D
- 2nd order time accuracy in turbulence modeling (default)
- New keywords are available
  - 1st time accurate turbulence modeling (default is 2nd order)
  - New options in turbulence modeling
  - Full Navier-Stokes terms available
  - Option to exercise mesh deformation without full flow solver
  - Calculation of CFL number can be modified for axisymmetric cases to increase convergence rate
- Changes in the input for prescribed modal motion

CFL3D Overview

- Major features
  - Euler
  - Laminar thin-layer Navier-Stokes
  - Reynolds-Averaged thin-layer Navier-Stokes (RANS)
  - Structured grid
  - Single or multi-block
  - Dynamic memory
  - Parallel (MPI) capability
  - Moving grid and mesh deformation capability
  - CGNS (CFD General Notation System) capability for CFD output
- Discretization and numerical method
  - Conservation law form of the Euler or RANS equations
  - Spatial discretization is semi-discrete finite-volume approach
  - Upwind-Biasing is used for the convective and pressure terms
  - Solves either the steady or unsteady form of the equations
  - Time advancement is implicit with dual time stepping and sub-iterations

CFL3D Overview

- Discretization and numerical method (…continued)
  - Approximate-Factorized (AF) numerical scheme
  - Explicit block boundary conditions
  - Multigrid
  - Grid sequencing
- Block structures
  - 1-1 blocking (preferred)
  - Patching
  - Overlapping
  - Embedding
  - Sliding patched zone interfaces
  - Grids must have been created prior to execution of CFL3D

CFL3D Overview

- Turbulence models for RANS computation
  - 0-equation models: Baldwin-Lomax, Baldwin-Lomax with Degani-Schiff modification
  - 1-equation models: Baldwin-Barth, Spalart-Almaras, including Detached Eddy Simulation (DES)
  - 2-equation models: Wilcox k-ω model, Menter's k-ω Shear Stress Transport (SST) model, Abid k-ω model, k-ω and k-ε Explicit Algebraic Stress Models (EASM), k-enstrophy model
- Computing modes
  - Sequential or single processor (single or multiple blocks)
  - Parallel processing
    - Message Passing Interface (MPI)
      - Requires multi-block structure
      - May be run on distributed memory machines. (PC clusters or parallel supercomputer)

CFL3D Overview

- Computing modes (…continued)
  - Complex computation
    - Allows computation of sensitivity derivatives due to static and dynamic variables (e.g. $dC_L/d\alpha$)
    - Requires compiling of the complex executable for static and dynamic sensitivity calculations
    - Dynamic sensitivity calculations require additional keyword input
- Code developers and points of contact:
  - Many developers have contributed to CFL3D
  - Most recent primary NASA LaRC developers (POC's) are:

    Dr. Robert T. Biedron (757-864-2156, r.t.biedron@larc.nasa.gov) general flow solver, multiblock, MPI

    Dr. Christopher Rumsey (757-864-2165,c.l.rumsey@larc.nasa.gov) – turbulence models

    Dr. Bob Bartels (757-864-2813, r.e.bartels@larc.nasa.gov) –

    aeroelastic modules and deforming mesh

CFL3D Overview

- Online and printable documentation: http://cfl3d.larc.nasa.gov/Cfl3dv6/cfl3dv6.html
  - Recommend printing the Version 5.0 manual for reference (found as a link at the web site above)
- Acquiring the code:
  - Version 6 is currently available for general distribution to U.S. citizens within the United States. **The code cannot be released outside of the United States.** If you would like a copy of the code, please follow the request procedure below:
  - Send e-mail or FAX (757-864-8816) to one of the POC's requesting CFL3D Version 6, along with a brief description of your planned usage of the code, your phone number, and FAX number.
  - Your request will be forwarded internally to a NASA Software Releasing Authority (SRA). The SRA will determine whether or not the code may be released to you; if so, the SRA will e-mail or FAX a Usage Agreement to you to fill out, sign and return to the SRA.

CFL3D Overview

- After the SRA has granted permission, the code will be provided to you electronically. In addition, you will be added to the Version 6 user list, and will receive any updates and/or corrections that occur.
- Note: even if you are a registered Version 5 user you must still follow the formal request procedure for Version 6.
- Conditions of use:
  - Do not distribute any part of the code outside of your working group
  - Report any bugs you may find
  - CFL3D is restricted to use within the United States
  - Abide by any additional conditions in the usage agreement

Getting Started

- To install CFL3Dv6 on a particular machine, you must have the following file:

  cfl3dv6.tar.DATE.gz (tarred and gzipped version 6 package)

  Note: DATE indicates the release date in the form MMM_DD_YYYY. For example, cfl3dv6.tar.Sep_12_2003 indicates the code as of September 12, 2003.

- Make sure that: ./ is in your path; if not, you will have to explicitly prepend ./ to all the commands below

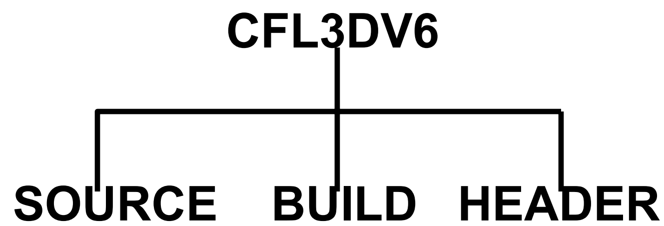
  Type:

  gunzip cfl3dv6.tar.DATE.gz

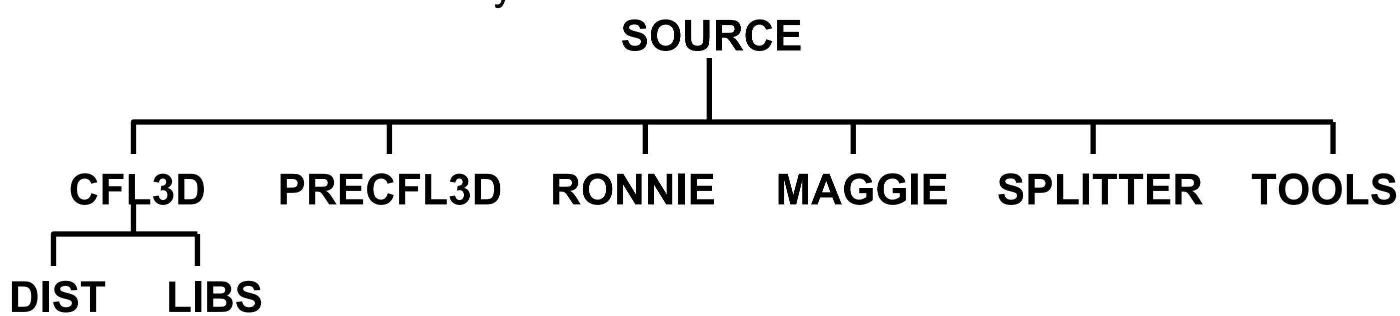
  tar -xvf cfl3dv6.tar.DATE

Getting Started

You should end up with the following directory structure:

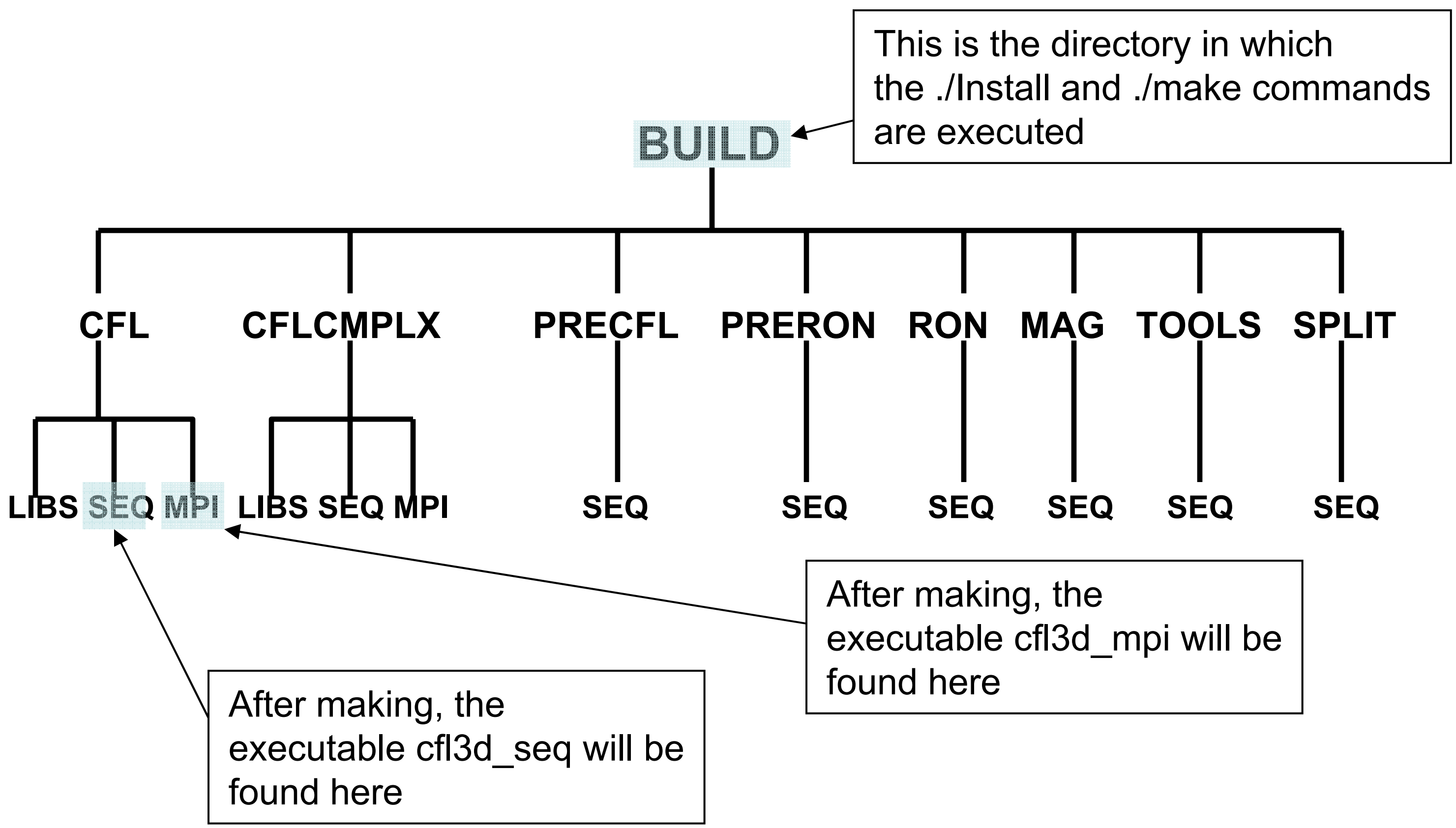


Within the source directory:



Getting Started

Within the build directory:



Getting Started

– In the subdirectory build, type:

Install [options]   or ./Install [options]

Where [options] may be blank or one or more of the following:

-no_opt
- create executables with little optimization but fast compilation

-single
- create single precision executables

-noredirect
- disallow redirected input file; needed only for SP2 and sometimes on Linux with MPI

-mpichdir=*dir1*
- use MPICH on a workstation cluster; *dir1* is the directory where mpich is located - not used on MPP machines

-linux_compiler_flags=*flag*
- sets up to compile using special compiler flags for use on Linux operating systems only; *flag* is currently Intel, PG, Lahey, or Alpha (Intel is currently the default) Example: To use the Portland Group compiler MUST install with: ./Install -linux_compiler_flags=PG

-help
- print out the Install options

Getting Started

- Note: the directory paths for either the mpichdir or cgnsdir options should be either absolute paths or paths relative to the installation directory; the use of ~ to denote a home directory is not allowed.
- If -no_opt is not specified, various compiler optimization levels are used to speed execution but results in slower compilation.
- If -mpichdir=dir1 is not used, then it is assumed "native" MPI is available, and will use a default location for the necessary MPI libraries.
- If -single is not used, then double precision executables will be created at the make [ ] command.
- Once installation is complete, a makefile will automatically be created for the machine platform on which the code is installed.
- Go to the build directory.
- By typing “make” you will see all the make options available.

Getting Started

– Several of the most common make options are:

  make cfl3d_seq - make the sequential (single processor) version of the code

  make cfl3d_mpi - make the MPI (multiprocessor) version of the code

  make splitter - make the block splitter executable

  make cfl3d_tools - make some of the cfl3d utilities

– Within the build directory, type the make option for the executable you want.

– To execute the sequential code type:

```
./cfl3d_seq < cfl3d.inp
```

– To execute the MPI code type:

```
mpirun –np <noprocessors> ./cfl3d_mpi < cfl3d.inp
```

where <noprocessors> is typically one greater than the number of blocks*

* The MPI process requires an extra administrative processor beyond those that perform the computation. (e.g. For a 12 block grid, all with equal numbers of grid points, to be run on 3 processors, noprocessors = 4)

Equations and Dimensions
Reference parameters

- The governing equations are the Euler or Navier-Stokes equations combined with a turbulence model for RANS computation
- The governing equations are non-dimensionalized based on the following parameters:

$\tilde{L}_R$	–	Reference length used by the code (dimensional)
$\tilde{\rho}_\infty$	–	Free-stream density, mass/unit length cubed
$\tilde{a}_\infty$	–	Free-stream speed of sound, length/time
$\tilde{\mu}_\infty$	–	Free-stream molecular viscosity, mass/length-time

Equations and Dimensions

- Since there is no standard system of units for CFD models the non-dimensionalization in CFL3D removes the necessity of converting grids into units compatible with the code. The way in which the non-dimensionalization is accomplished will be presented later in this document.

- Note that the term *free-stream* is used in the non-dimensionalization. CFL3D was developed primarily as an external flow solver. It has the capability to perform computations for internal flows as well. Therefore a more general term *reference state* should probably be used, but the term f*ree-stream* is used throughout the documentation.

Equations and Dimensions

Non-dimensional variables

In CFL3D the non-dimensionalizations are performed as follows:

$$x = \frac{\tilde{x}}{\tilde{L}_R} \qquad y = \frac{\tilde{y}}{\tilde{L}_R} \qquad z = \frac{\tilde{z}}{\tilde{L}_R} \qquad t = \frac{\tilde{t}\,\tilde{a}_\infty}{\tilde{L}_R}$$

Time nondim-
ensionalized by
speed of sound
and ref length

$$\rho = \frac{\tilde{\rho}}{\tilde{\rho}_\infty} \qquad u = \frac{\tilde{u}}{\tilde{a}_\infty} \qquad v = \frac{\tilde{v}}{\tilde{a}_\infty} \qquad w = \frac{\tilde{w}}{\tilde{a}_\infty}$$

Velocities nondim-
ensionalized by
speed of sound

Non-dimensionalizing by speed of sound makes transonic the natural flow regime for CFL3D, although low speed and hypersonic flows can be computed, with modified input, as well.

Problem Formulation and Setup

Overview

- There are five steps in problem formulation and setup for steady and unsteady computation:
    - Condition definition
    - Grid generation
    - Block splitting (if necessary)
    - Blocking and boundary conditions
    - Input development

- Parameters that define a condition are:
    - Mach number
    - Reynolds number
    - Ambient temperature
    - Grid orientation (angle of attack, side slip, etc…)

Input for these parameters will be discussed later. For the moment several of these parameters are required for the proper construction of the grid…

Problem Formulation and Setup

Grid generation

Considerations that are important for generation of a grid:

- Reynolds number sets permissible Δy^+ at the surface.
  - For most turbulent computations typically want a $y^+ \sim 1$ for first grid off the surface
  - For turbulent computations with wall function, typically want a $y^+ \sim$ 50-100 for first grid off the surface
  - Setting Δy^+ requires an estimate of the wall shear stress prior to computing

Note that:

$$y^+ = \frac{y}{\nu}\sqrt{\frac{\tau_w}{\rho}}$$

where y is normal distance to surface, τ_w is wall shear stress, ρ is density and ν is kinematic viscosity.

Problem Formulation and Setup

Grid generation

- After the first converged successful run with a coarse grid, y^+ of the first grid can be checked. This value is found at the end of the cfl3d.out file.
  (See Y+ MAX, Y+ MIN and Y+AVG below)

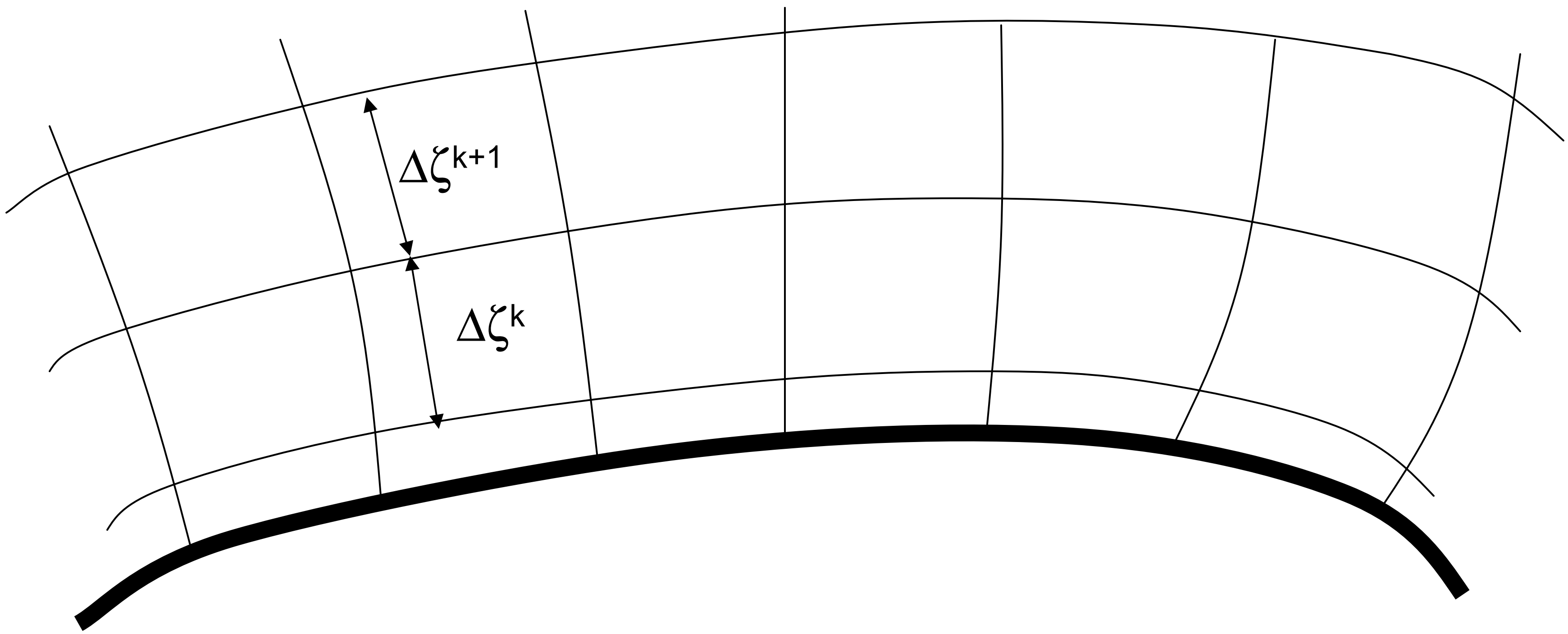
```
YPLUS STATISTICS (endpts not included) - BLOCK  1 (GRID  1)

   K=1    SURFACE:
     Y+ MAX    JLOC    ILOC       Y+ MIN    JLOC    ILOC
   0.535E+00    151       1      0.261E-01    217       1
      DN MAX    JLOC    ILOC       DN MIN    JLOC    ILOC
   0.152E-05    228       1      0.149E-05    219       1
      Y+ AVG    Y+ STD DEV      NY+ > 5    NPTS
   0.264E+00    0.373E+00            0      199

  YPLUS STATISTICS (endpts not included) - ALL GLOBAL BLOCKS
     Y+ MAX    ILOC    JLOC    KLOC  BLOCK   GRID
   0.535E+00      1     151       1      1      1
     Y+ MIN    ILOC    JLOC    KLOC  BLOCK   GRID
   0.261E-01      1     217       1      1      1
```

etc…

Problem Formulation and Setup

Grid generation

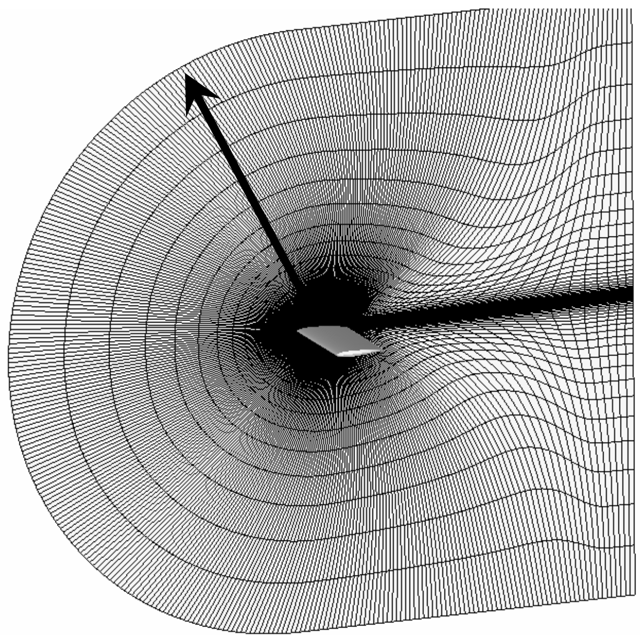
- Grid stretching away from a surface.
    - *Rule of thumb:* $\Delta\zeta^{k+1}$ should be no more than 1.2 to 1.5 times $\Delta\zeta^{k}$



Problem Formulation and Setup

Grid generation

- Outer extent of grid
  - *Rule of thumb:* The outer boundary of the grid should be at least 15 body lengths away (3D) and at least 20 body lengths away (2D). This is not a hard and fast rule and there are some notable exceptions. Note that the grid below would not be considered a fully acceptable grid.



Problem Formulation and Setup
Grid generation

- Grid quality
  - Grid metric smoothness. CFL3D assesses the size of local variations in grid metrics. Warnings are printed to the cfl3d.out file. Any messages of the following form indicate a problem with the grid:

```
FATAL si grid normal direction change near j,k,i,i+1=   23    5  164  165
        ... suspect bad grid
FATAL sj grid normal direction change near j,k,i,i+1=   23    5  164  165
        ... suspect bad grid
```

Etc… Or

```
WARNING: Dramatic si grid norm direction change (>120deg)
WARNING: Dramatic sj grid norm direction change (>120deg)
```

Etc…

Problem Formulation and Setup

Grid generation

- Grid quality (...continued)
  - Negative grid volumes. CFL3D checks whether there are negative volumes in the grid. Under normal operating procedures the code will exit with an error message in the cfl3d.error file.*
- Grid clustering to resolve flow gradients
  - Resolving a wake. Although angle of attack is specified in the input, it does result in the possibility of flow separation and wing stall and resulting wake. The wake may need grid clustering.
  - Resolving a shock or curvature effect. Mach number effects such as a shock or surface curvature may result in gradients that require resolving.
  - These steps must be performed prior to running CFL3D.

* There is a keyword option that allows computing to continue with negative volumes. This option will be discussed later in the course under “Keyword Input”.

Problem Formulation and Setup

Grid generation

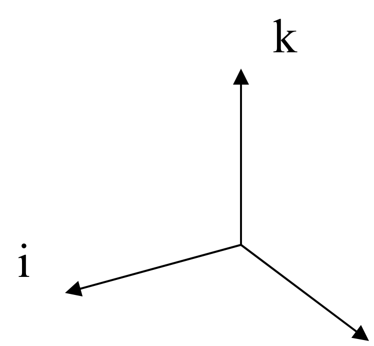
- Grid file format
  - The grid file format must be unformatted
  - Two grid data formats are possible, plot3d and cfl3d. These formats are presented in the CFL3D version 5.0 manual.
  - If CFL3D is compiled in double precision, the grid file must be written as double precision real
  - Example of multi-platform issue: If a Linux compiler is used to compile CFL3D to read an SGI unformatted grid file, the grid file must be generated with the same compile options

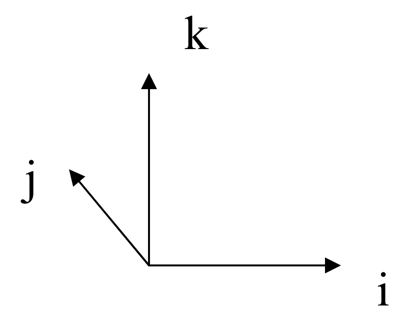
Example: Suppose the code 'hygrid' is used to generate the unformatted grid file. On a Linux based PC platform using the Portland Group compiler, the compile option –byteswapio swaps bytes from big-endian to little-endian for input compatibility with a Sun or SGI system. This compiler option will allow CFL3D to read the grid file created either on the PC cluster or on an SGI machine.

Problem Formulation and Setup

Grid generation

CFL3D requires that the right-hand rule be observed in both the *x,y,z* orientation and the *i,j,k* index directions. Also, *i,j* and *k* do not have to be in the *x,y* and *z* directions. Any permutation is valid as long as the right-hand rule is upheld. Caveat: When using turbulence models there are direction preferences as will be discussed.





Problem Formulation and Setup

Multigridable dimensions

To use multigrid, grid dimensions including all b.c. segments must be multigridable

Table 7–2. Grid sizes multigridable to three additional level.

Grid:	Coarser Levels:			Grid:	Coarser Levels:			Grid:	Coarser Levels:		
9	5	3	2	345	173	87	44	673	337	169	85
17	9	5	3	353	177	89	45	681	341	171	86
25	13	7	4	361	181	91	46	689	345	173	87
33	17	9	5	369	185	93	47	697	349	175	88
41	21	11	6	377	189	95	48	705	353	177	89
49	25	13	7	385	193	97	49	713	357	179	90
57	29	15	8	393	197	99	50	721	361	181	91
65	33	17	9	401	201	101	51	729	365	183	92
73	37	19	10	409	205	103	52	737	369	185	93
81	41	21	11	417	209	105	53	745	373	187	94
89	45	23	12	425	213	107	54	753	377	189	95
97	49	25	13	433	217	109	55	761	381	191	96
105	53	27	14	441	221	111	56	769	385	193	97
113	57	29	15	449	225	113	57	777	389	195	98
121	61	31	16	457	229	115	58	785	393	197	99
129	65	33	17	465	233	117	59	793	397	199	100
137	69	35	18	473	237	119	60	801	401	201	101
145	73	37	19	481	241	121	61	809	405	203	102
153	77	39	20	489	245	123	62	817	409	205	103
161	81	41	21	497	249	125	63	825	413	207	104

From CFL3D User's Manual, 7.1.2, pg 129

Problem Formulation and Setup

Multigrid dimensions

169	85	43	22	505	253	127	64	833	417	209	105
177	89	45	23	513	257	129	65	841	421	211	106
185	93	47	24	521	261	131	66	849	425	213	107
193	97	49	25	529	265	133	67	857	429	215	108
201	101	51	26	537	269	135	68	865	433	217	109
209	105	53	27	545	273	137	69	873	437	219	110
217	109	55	28	553	277	139	70	881	441	221	111
225	113	57	29	561	281	141	71	889	445	223	112
233	117	59	30	569	285	143	72	897	449	225	113
241	121	61	31	577	289	145	73	905	453	227	114
249	125	63	32	585	293	147	74	913	457	229	115
257	129	65	33	593	297	149	75	921	461	231	116
265	133	67	34	601	301	151	76	929	465	233	117
273	137	69	35	609	305	153	77	937	469	235	118
281	141	71	36	617	309	155	78	945	473	237	119
289	145	73	37	625	313	157	79	953	477	239	120
297	149	75	38	633	317	159	80	961	481	241	121
305	153	77	39	641	321	161	81	969	485	243	122
313	157	79	40	649	325	163	82	977	489	245	123
321	161	81	41	657	329	165	83	985	493	247	124
329	165	83	42	665	333	167	84	993	497	249	125
337	169	85	43								

From CFL3D User's Manual, 7.1.2, pg 129

Problem Formulation and Setup

Blocking and boundary conditions

Blocking and boundary conditions are specified at the following boundaries:

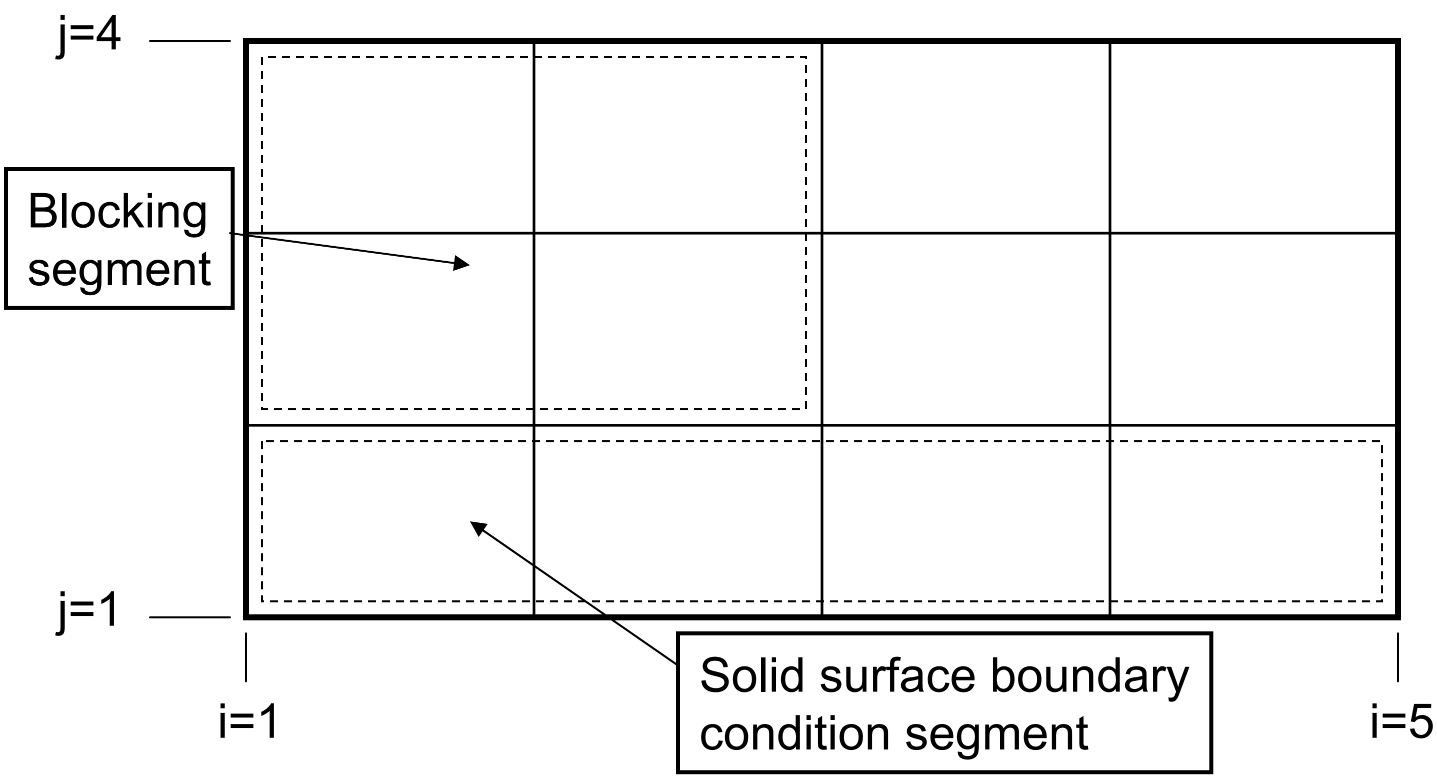
i0	(i=1) and	idim
j0	(j=1) and	jdim
k0	(k=1) and	kdim

where idim, jdim and kdim are the block dimensions in the ijk-directions. Blocking and boundary condition data can be composed of multiple segments but the combined segments must span each of the six block faces. Note that to perform multigrid computations, the boundary and blocking segments must be multigridable integers.

Problem Formulation and Setup

Blocking and boundary conditions

Example of possible blocking or boundary condition segments on the k0 face. Suppose that part of the k0 face below represents the surface of a wing.

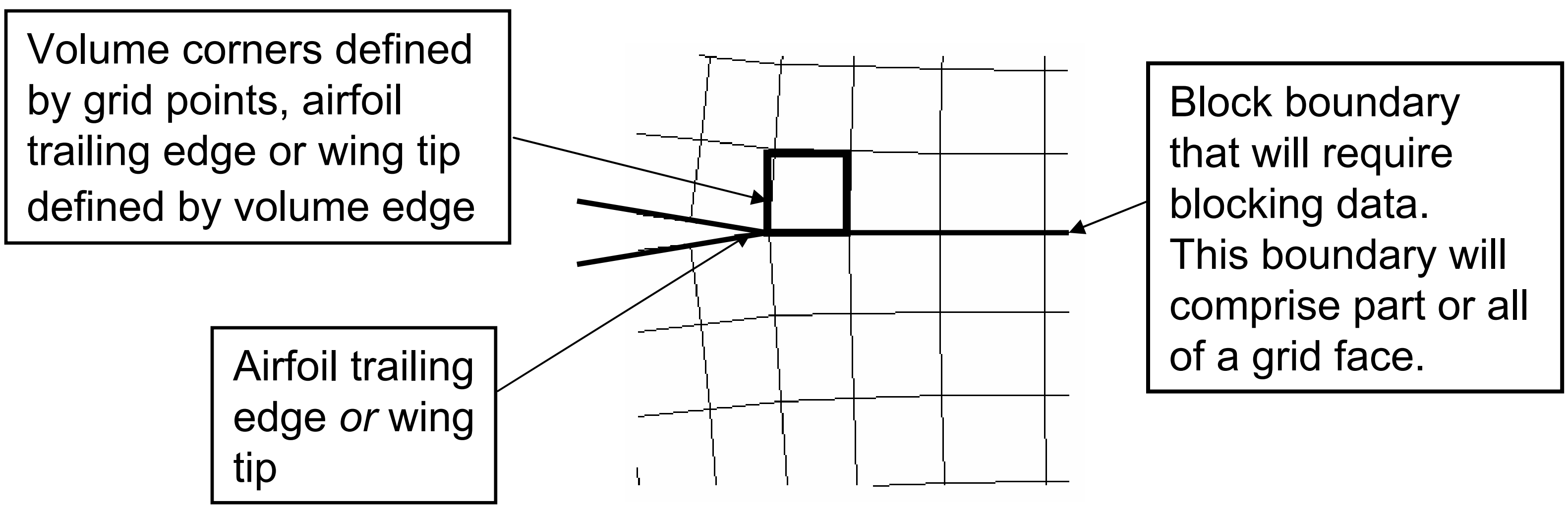


Problem Formulation and Setup

Blocking and boundary conditions

Volume *edges* define geometric extremities. The volume edges will also be the start and end points of blocking pairs. All blocking and boundary conditions will be on external surfaces of grid blocks.

Example: Trailing edge of an airfoil or tip of a wing.

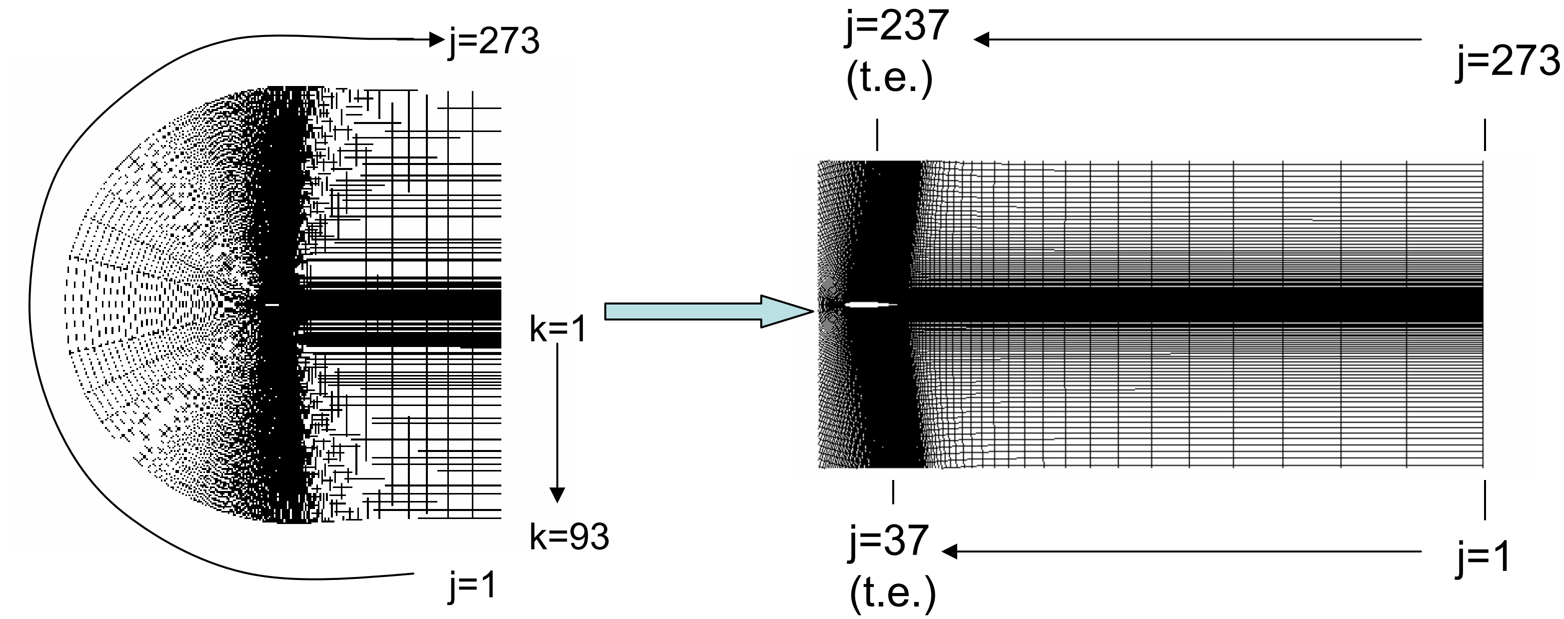


Problem Formulation and Setup

Blocking and boundary conditions

Blocking defines the start and ending indices of 1-1 interfaces between one or more corresponding grid blocks.

Consider the example of a 2D airfoil using a single block C-grid with dimension 2x273x93. CFL3D is a finite volume code and therefore requires 2 grid points in the span-wise direction (always i-dir for a 2D grid). *Note that the arrows in the right hand figure below denotes the end of a blocking segment.* The meaning of this statement will be made clear in the following pages.



Problem Formulation and Setup

Blocking and boundary conditions

The following is the steady input file for the single block C-grid 2D airfoil. Highlighted sections are the blocking and boundary condition input:

Boundary conditions

```
input/output files:
grid.bin
plot3dg.bin
plot3dq.bin
cfl3d.out
cfl3d.res
cfl3d.turres
cfl3d.blomax
cfl3d.out15
cfl3d.prout
cfl3d.out20
ovrlp.bin
patch.bin
restart.bin
    NLR7301 airfoil, cfl3d type grid
    Xmach     alpha      beta      ReUe   Tinf,dR     ialph    ihstry
    0.753      1.10       0.0    5.7567     460.0         0         0
     sref      cref      bref       xmc       ymc       zmc
      1.0       1.0       1.0     0.075       0.0       0.0
       dt     irest   iflagts      fmax     iunst   cfl_tau
     -2.0         0         0       1.0         0       5.0
    ngrid   nplot3d    nprint    nwrest      ichk       i2d    ntstep       ita
        1         1         1      1000         0         1         1        -2
      ncg       iem  iadvance    iforce  ivisc(i)  ivisc(j)  ivisc(k)
        2         0         0         1         0         0         5
     idim      jdim      kdim
        2       273        93
   ilamlo    ilamhi    jlamlo    jlamhi    klamlo    klamhi
        0         0         0         0         0         0
    inewg    igridc        is        js        ks        ie        je        ke
        0         0         0         0         0         0         0         0
 idiag(i)  idiag(j)  idiag(k)  iflim(i)  iflim(j)  iflim(k)
        1         1         1         4         4         4
  ifds(i)   ifds(j)   ifds(k)  rkap0(i)  rkap0(j)  rkap0(k)
        1         1         1    0.3333    0.3333    0.3333
     grid     nbci0   nbcidim     nbcj0   nbcjdim     nbck0   nbckdim    iovrlp
        1         1         1         1         1         3         1         0
i0:   grid   segment    bctype      jsta      jend      ksta      kend     ndata
         1         1      1002         0         0         0         0         0
idim:grid    segment    bctype      jsta      jend      ksta      kend     ndata
         1         1      1002         0         0         0         0         0
j0:   grid   segment    bctype      ista      iend      ksta      kend     ndata
         1         1      1003         0         0         0         0         0
jdim:grid    segment    bctype      ista      iend      ksta      kend     ndata
         1         1      1003         0         0         0         0         0
k0:   grid   segment    bctype      ista      iend      jsta      jend     ndata
1        1         0         0         0         0         1        37         0
1        2      2004         0         0         0        37       237         2
   tw/tinf        cq
        0.        0.
         1         3         0         0         0       237       273         0
```

Problem Formulation and Setup

Blocking and boundary conditions

```
 kdim:grid  segment    bctype     ista    iend      jsta     jend    ndata
         1        1      1003        0       0         0        0        0
      mseq   mgflag    iconsf      mtt    ngam
         1        1         0        0       2
      issc epsssc(1) epsssc(2) epsssc(3)  issr  epsssr(1) epsssr(2) epsssr(3)
         0      0.3       0.3       0.3      0       0.3       0.3       0.3
      ncyc   mglevg     nemgl    nitfo
      2000        3         0        0
      mit1     mit2      mit3     mit4    mit5 ...
         1        1         1
   1-1 blocking data:
      nbli
         1
    number     grid      ista     jsta    ksta     iend    jend    kend  isva1  isva2
         1        1         1        1       1        2      37       1      1      2
    number     grid      ista     jsta    ksta     iend    jend    kend  isva1  isva2
         1        1         1      273       1        2     237       1      1      2
   patch interface data:
    ninter
         0
plot3d output:
      grid    iptyp      ista     iend    iinc     jsta    jend    jinc   ksta   kend   kinc
         1        0         1        1       1        1     999       1      1    999      1
     movie
         0
  print out:
      grid    iptyp      ista     iend    iinc     jsta    jend    jinc   ksta   kend   kinc
         1        0         1        1       1        1     999       1      1    999      1
  control surfaces
       ncs
         0
      grid     ista      iend     jsta    jend     ksta    kend   iwall  inorm
```

Blocking data

Problem Formulation and Setup

Blocking and boundary conditions

For this example, format of the blocking data in the input file:

```
1-1 blocking data:
      nbli
         1
 number   grid    ista    jsta    ksta    iend    jend    kend    isva1   isva2
      1      1       1       1       1       2      37       1       1       2
 number   grid    ista    jsta    ksta    iend    jend    kend    isva1   isva2
      1      1       1     273       1       2     237       1       1       2
```

Number of lines of blocking data

No. of lines in each data must equal *nbli*

Number of the block (in the present example there is only 1 block)

Number of the blocking data line

Note: The text cards must be present, but the text within those lines is arbitrary, and is for user information only. All lines with data are in free field format throughout the input file.

Problem Formulation and Setup

Blocking and boundary conditions

Blocking data

1-1 blocking data:

nbli

1

number	grid	ista	jsta	ksta	iend	jend	kend	isva1	isva2
1	1	1	1	1	2	37	1	1	2
number	grid	ista	jsta	ksta	iend	jend	kend	isva1	isva2
1	1	1	273	1	2	237	1	1	2

i – start indices

i – end indices

First index variation on both sides is in the i-direction

j – start indices

j – end indices

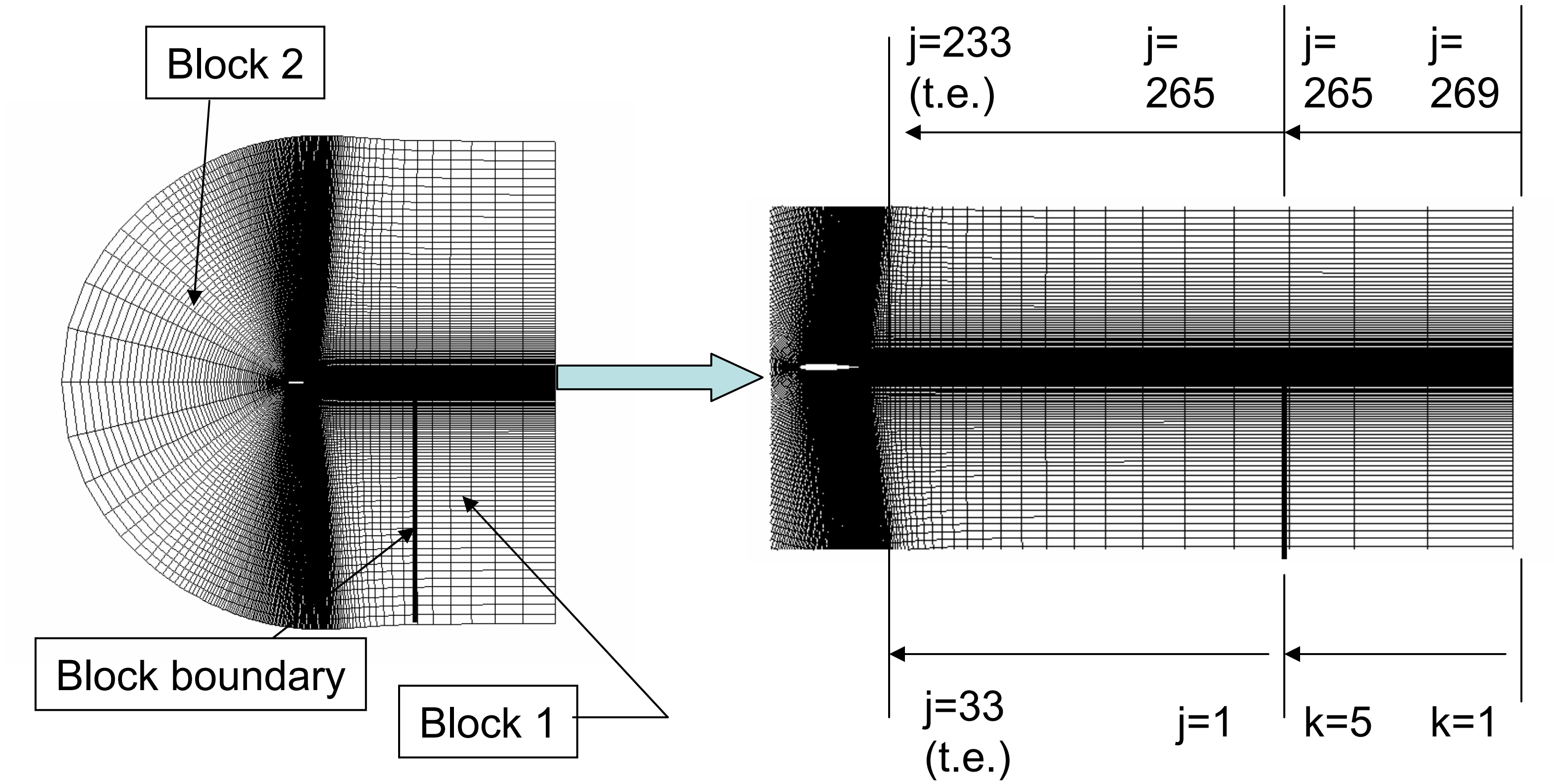
Second index variation on both sides is in the j-direction

Because this is a volume grid, the blocking will always define a two-dimensional interface in index space

Problem Formulation and Setup

Blocking and boundary conditions

Consider a second example of a 2D airfoil using two blocks to compose a C-grid. Block 1 has dimensions 2x93x5. Block 2 has dimensions 2x269x93. *Note again that the arrows in the right hand figure below denotes the end of a blocking segment.* This fact is made clear by the following page.



Problem Formulation and Setup

Blocking and boundary conditions

Blocking data

1-1 blocking data:

3 blocking data sets now

nbli
3

number	grid	ista	jsta	ksta	iend	jend	kend	isva1	isva2
1	1	1	1	1	2	1	5	1	3
2	2	1	1	1	2	33	1	1	2
3	1	1	1	1	2	97	1	1	2

number	grid	ista	jsta	ksta	iend	jend	kend	isva1	isva2
1	2	1	269	1	2	265	1	1	2
2	2	1	265	1	2	233	1	1	2
3	2	1	1	1	2	1	97	1	3

k-index of block 1 now varies with the j-index of block 2

A new blocking boundary appears that previously did not exist

Problem Formulation and Setup

Blocking and boundary conditions

Blocking faces require corresponding boundary condition data

In the first example above, the blocking interface is at the k=1 boundary. Therefore, the boundary condition data for that blocking interface is in the 'k0' boundary data.

k0:	grid	segment	bctype	ista	iend	jsta	jend	ndata
	1	1	0	1	2	1	37	0
		.						
		.						
		.						
	1	3	0	1	2	237	273	0

Boundary condition type for a blocking interface is 0

Problem Formulation and Setup

Blocking and boundary conditions

CFL3D will stop if the number of grid points across a blocking interfaces does not match.

Suppose the following blocking data had been specified for example 1 above:

Erroneous jend value

number	grid	ista	jsta	ksta	iend	jend	kend	isva1	isva2
1	1	1	1	1	2	35	1	1	2

number	grid	ista	jsta	ksta	iend	jend	kend	isva1	isva2
1	1	1	273	1	2	237	1	1	2

Execution will terminate with the following error message at the end of the file 'precfl3d.out':

.

.

the limits of ind2 are not the same for both sides for 1:1 plane 1

Problem Formulation and Setup

Blocking and boundary conditions

CFL3D also checks the input connection data by computing the geometric mismatch between both sides of the interface. A true 1-1 interface will have zero (machine zero) mismatch. Any mismatches larger than ε (where ε is the larger of 10^{-9} or 10x(machine zero)) will cause a warning message.

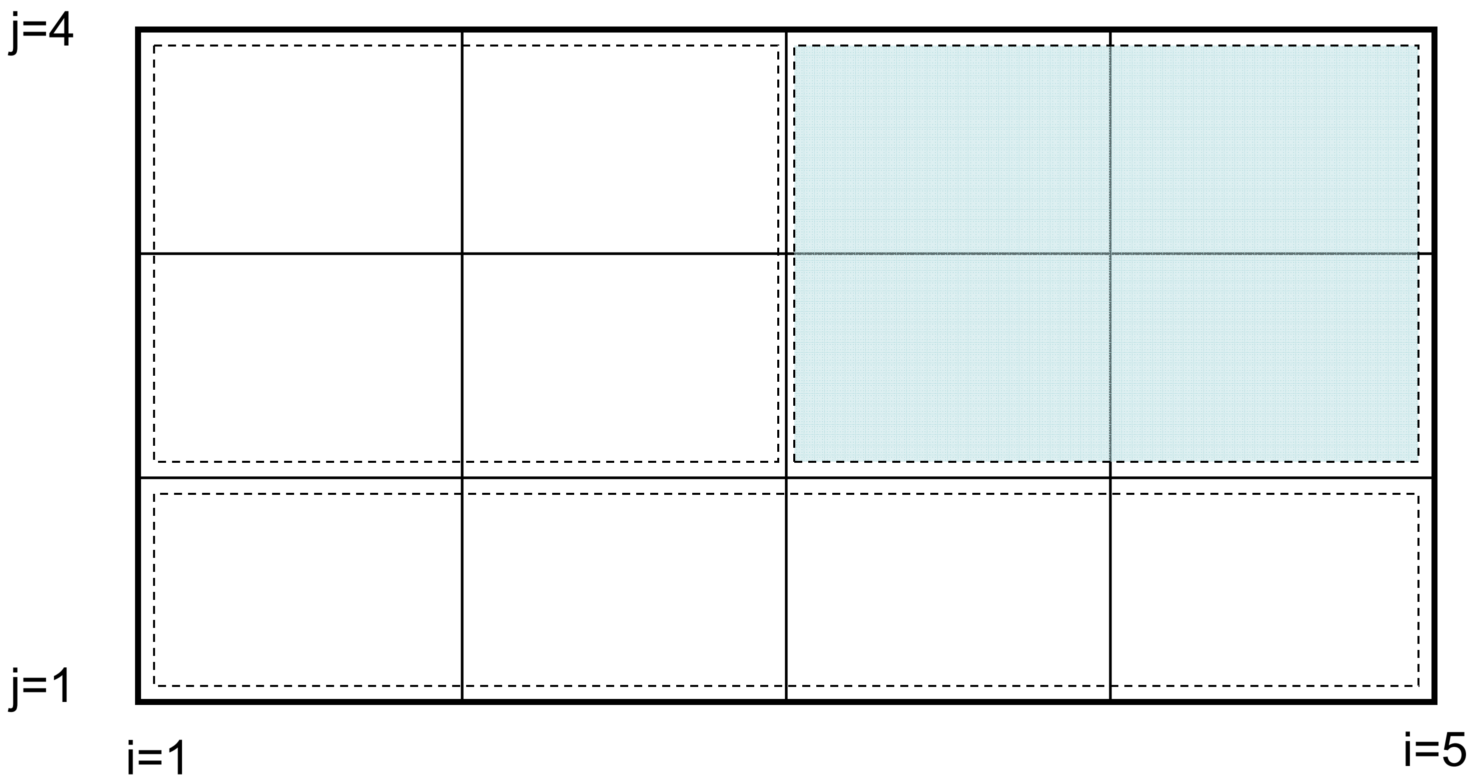
Example of the output in ‘cfl3d.out’:

```
j=   1    1-1 blocking                    type      0     i=  1,  31    k=137,   69
          connects to j =  1 of block  2
          blocking check....geometric mismatch =  0.2166272E-03
```

Problem Formulation and Setup

Blocking and boundary conditions

Example of possible boundary condition segments on the k0 face. Suppose that the k0 face below represents the surface of a wing.

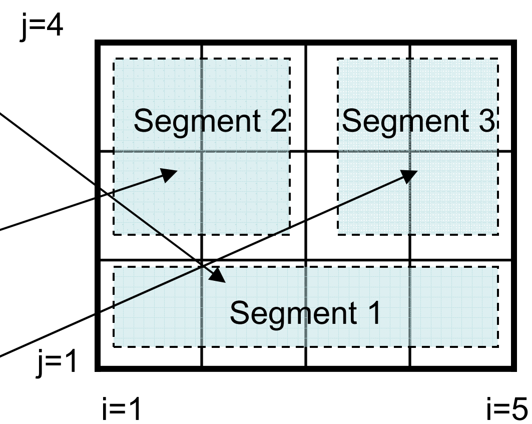


Problem Formulation and Setup

Blocking and boundary conditions

At the unshaded cells, it is desired to apply a heated wall boundary condition, while at the shaded cells it is desired to apply an adiabatic wall boundary condition. One way to accomplish this objective is to divide the boundary into the segments shown. The CFL3D input file would have input that looks like this:

```
k0: grid  segment  bctype    ista    iend    jsta    jend   ndata
       1        1    2004       1       5       1       2       2
  tw/tinf      cq
 1.60000  0.00000
       1        2    2004       1       3       2       4       2
  tw/tinf      cq
 1.60000  0.00000
       1        3    2004       3       5       2       4       2
  tw/tinf      cq
 0.00000  0.00000
```



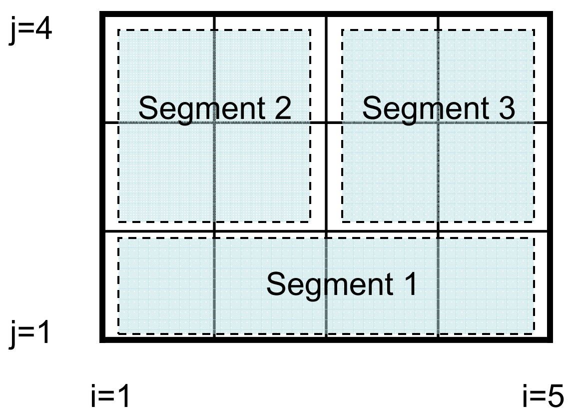
Note that for segment 1, for instance, the grid points i = 1 to 5, j = 1 to 2 define the boundary of the cells at which the condition type is to be applied.

Problem Formulation and Setup

Blocking and boundary conditions

Setting $ista = iend = 0$ and/or $jsta = jend = 0$ is a shorthand way of specifying the entire range. In other words, an alternate boundary condition input with identical outcome is:

k0: grid	segment	bctype	ista	iend	jsta	jend	ndata
1	1	2004	0	0	1	2	2
tw/tinf	cq						
1.60000	0.00000						
1	2	2004	1	3	2	4	2
tw/tinf	cq						
1.60000	0.00000						
1	3	2004	3	5	2	4	2
tw/tinf	cq						
0.00000	0.00000						



Problem Formulation and Setup

Blocking and boundary conditions

The following 1000 series boundary conditions are available:

bctype	boundary condition
1000	free stream
1001	general symmetry plane
1002	extrapolation
1003	inflow/outflow
1005	inviscid surface
1006	inviscid surface (using normal momentum)
1008	tunnel inflow
1011	singular axis – half-plane symmetry
1012	singular axis – full plane
1013	singular axis – partial plane

Refer to the Version 5.0 Manual and Version 6.0 web page for more information on these boundary conditions

Problem Formulation and Setup

Blocking and boundary conditions

The following 2000 series boundary conditions are available:

bctype	boundary condition
2002	specified pressure ratio
2003	inflow with specified total conditions
2004	no-slip wall
2005	periodic in space
2006	set pressure to satisfy the radial equilibrium equation
2007	set all primitive variables

Refer to the Version 5.0 Manual and Version 6.0 web page for more information on these boundary conditions

Problem Formulation and Setup

Blocking and boundary conditions

The following 2000 series boundary conditions are available:

bctype	boundary condition
2008	user specifies density and velocity components, pressure extrapolated from interior
2009	sets total pressure and total temperature. Inflow pressure extrapolated from interior
2014	user specifies transpiration through the boundary
2018	user specifies temperature and momentum components, pressure extrapolated from interior
2028	user specifies frequency and maximum momentum components, density and pressure extrapolated
2102	pressure ratio specified as a sinusoidal function of time

Refer to the Version 5.0 Manual and Version 6.0 web page for more information on these boundary conditions

Problem Formulation and Setup

Blocking and boundary conditions

Boundary condition 1000 - Free stream. Extrapolation points just outside the boundary are set to initial free stream values, which are:

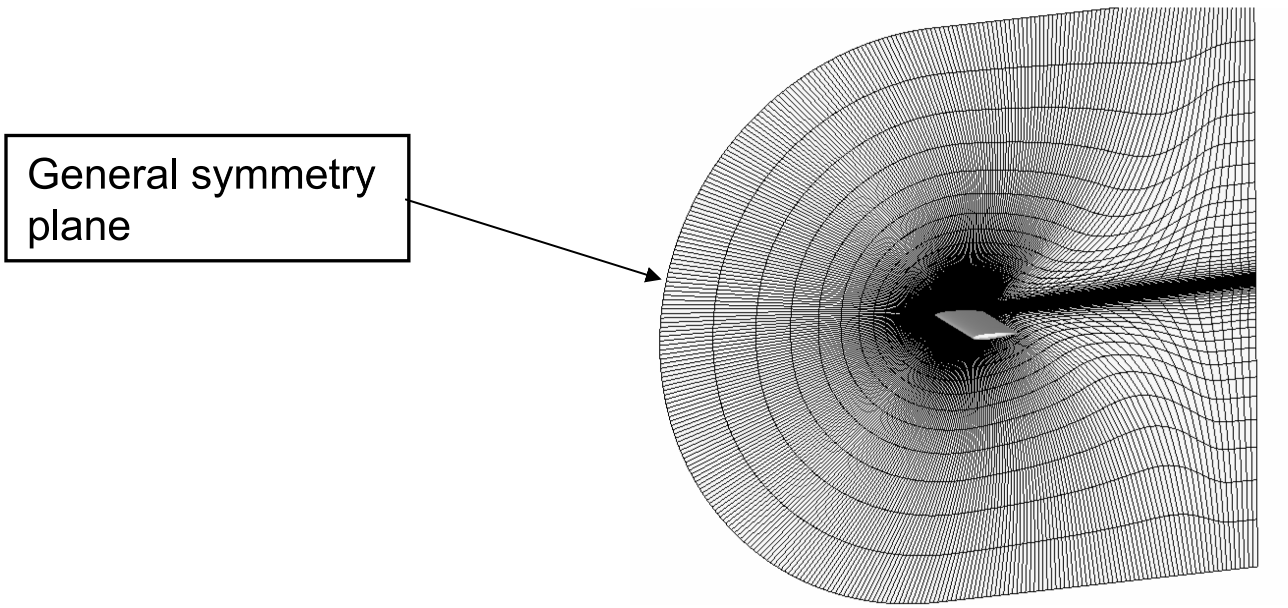
$$\rho_{initial} = 1.0$$
$$u_{initial} = M_{\infty} \cos\alpha \cos\beta$$
$$v_{initial} = -M_{\infty} \sin\beta$$
$$w_{initial} = M_{\infty} \sin\alpha \cos\beta$$
$$p_{initial} = \rho_{initial} a_{initial}^{2} / \gamma$$

where ρ is density, u,v,w are the x,y,z components of velocity, p is pressure, a is speed of sound and γ is ratio of specific heats. M is Mach number, α is the angle of attack and β is the side slip angle.

Problem Formulation and Setup

Blocking and boundary conditions

Boundary condition 1001 - General symmetry plane. Suppose we wish to simulate a 3D wing using the half wing shown. If only one type of maneuver is performed (i.e. with aircraft maneuver symmetry in the x-y plane, x-z plane or y-z plane only) the symmetry plane boundary condition can be used.



Problem Formulation and Setup

Blocking and boundary conditions

Boundary condition 1002 - Extrapolation. Ghost points outside the flow field domain are extrapolated from the interior.

Boundary condition 1003 - Inflow/Outflow. This condition uses Riemann invariants to calculate inflow and outflow at the boundary cell face. It effectively sets total pressure.

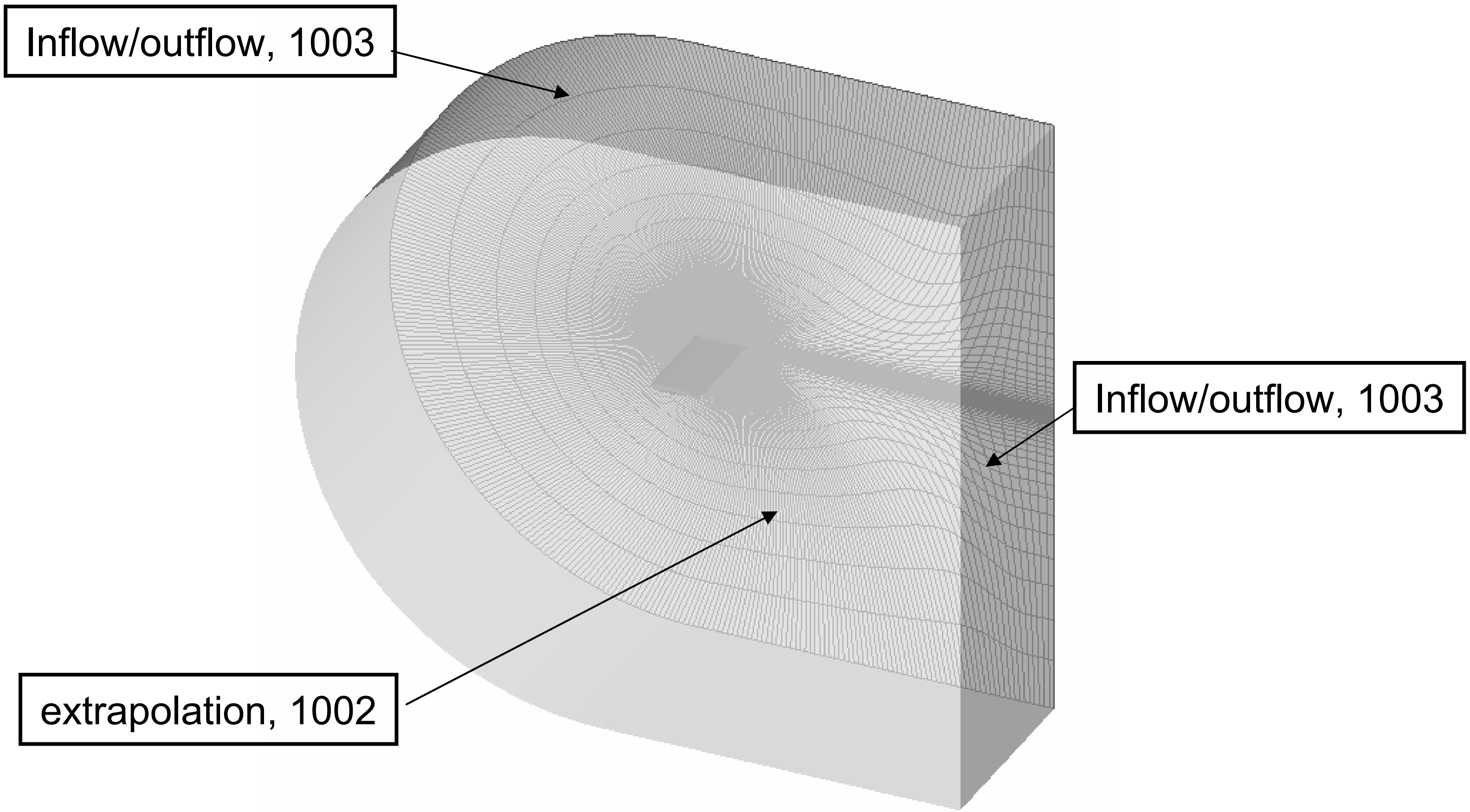
Boundary condition 1005 - Inviscid surface. Velocity components normal to the surface are set to zero. Density and pressure gradients are set to zero.

Boundary condition 1006 - Inviscid surface. Similar to b.c. 1005 except that the normal momentum equation is used to obtain wall pressure. Generally results in a smoother solution near an inviscid surface.

Boundary condition 2004 - No slip wall. Viscous boundary conditions are set at surface cell face, i.e. flow velocity equals the surface velocity.

Problem Formulation and Setup

Example of typical “outer” boundary conditions



Problem Formulation and Setup

Blocking and boundary conditions

Boundary condition 1005: Inviscid surface

.
.
.

i0:	grid	segment	bctype	ista	iend	jsta	jend	ndata
	1	1	1005	1	5	1	2	0
	1	2	0	1	3	2	4	0
idim:	grid	segment	bctype	ista	iend	jsta	jend	ndata

.
.
.

Problem Formulation and Setup

Blocking and boundary conditions

Note that the b.c. 1005 has no auxiliary data, while the b.c. 2004 has two additional lines

.
.

k0:	grid	segment	bctype	ista	iend	jsta	jend	ndata
	1	1	1005	1	5	1	2	0

.
.

Specifies no additional data entries

…versus…

.
.

k0:	grid	segment	bctype	ista	iend	jsta	jend	ndata
	1	1	2004	1	5	1	2	2

tw/tinf	cq
1.60000	0.00000

Specifies two additional auxiliary data entries

Problem Formulation and Setup

Blocking and boundary conditions

- Series 1000 boundary conditions require no auxiliary data
- Number of auxiliary data entries for series 2000 boundary conditions are shown below

b.c. type	No. of auxiliary data
2002	1
2003	5
2004	2
2005	5
2006	4
2007	5*
2008	4*
2009	4*
2014	3
2016	7
2018	4*
2028	4*
2102	4

* Means turbulence data can also be specified, adding either 1 or 2 additional aux. data inputs

See the CFL3D version 5.0 manual and CFL3D Version 6 web page for discussion of these boundary conditions

Problem Formulation and Setup

Blocking and boundary conditions

Example of a boundary condition with 5 auxiliary data entries: 2003 - "Engine inflow", inflow with specified total conditions:

```
                                          .
                                          .
k0:   grid   segment    bctype       ista        iend        jsta       jend      ndata
         1           1      2003            1            5          1          2          5
   Mach        Pt/Pinf    Tt/Tinf   Alphae     Betae
     0.30        4.000   1.1755          0.0         0.0
                                          .
                                          .
```

Problem Formulation and Setup

Blocking and boundary conditions

Input data so far for the 2D airfoil using a single block C-grid

Number of k0 segments

Boundary condition data

i-boundary data

j-boundary data

k-boundary data

```
                .
                .
                .
     grid   nbci0  nbcidim  nbcj0  nbcjdim  nbck0  nbckdim  iovrlp
        1       1        1      1        1      3        1       0
i0:  grid  segment  bctype   jsta    jend    ksta    kend   ndata
        1        1    1002      0       0       0       0       0
idim: grid segment  bctype   jsta    jend    ksta    kend   ndata
        1        1    1002      0       0       0       0       0
j0:  grid  segment  bctype   ista    iend    ksta    kend   ndata
        1        1    1003      0       0       0       0       0
jdim: grid segment  bctype   ista    iend    ksta    kend   ndata
        1        1    1003      0       0       0       0       0
k0:  grid  segment  bctype   ista    iend    jsta    jend   ndata
        1        1       0      0       0       1      37       0
        1        2    2004      0       0      37     237       2
  tw/tinf       cq
       0.       0.
        1        3       0      0       0     237     273       0
kdim: grid segment  bctype   ista    iend    jsta    jend   ndata
        1        1    1003      0       0       0       0       0
                .
                .
                .
1-1 blocking data:
   nbli
    1
number  grid  ista  jsta  ksta  iend  jend  kend  isva1  isva2
     1     1     1     1     1     2    37     1      1      2
number  grid  ista  jsta  ksta  iend  jend  kend  isva1  isva2
     1     1     1   273     1     2   237     1      1      2
                .
                .
```

Blocking data

Setting Up a Steady Run

Input/output file specifications

Some of the key input, output files:

input/output files:	
grid.bin	Grid file (Input) (Unit 1)
plot3dg.bin	Plot3D output for the grid and q-array (Units 3 and 4)
plot3dq.bin	
cfl3d.out	Main CFL3D output (Unit 11)
cfl3d.res	Flow field residual history
cfl3d.turres	Turbulence model residual history
cfl3d.blomax	
cfl3d.out15	
cfl3d.prout	Flow field, flow field and surface data print out file
cfl3d.out20	
ovrlp.bin	
patch.bin	
restart.bin	Restart file (Input and Output) (Unit 2)

Setting Up a Steady Run

Input/output file specifications

- These names can be changed by the user.
- Input/output redirects are permitted. (e.g. ../../grid.bin or ./cflout/cfl3d.out)
- Additional files are printed out not contained in this list. (e.g. precfl3d.out, precfl3d.error, cfl3d.error, cfl3d.subit_res and cfl3d.subit_turres) These files cannot be renamed or redirected
- The restart file name that is read at the start of the computation is the same name used for output at the end. Scripting that saves restart files to another name will be required if the user wishes to save the input restart.

Setting Up a Steady Run

Navigating diagnostic output

Diagnostic output:

- Initial input syntax and completeness are checked in the preprocessor ‘precfl3d’. This is an initial step automatically performed by CFL3D. Output from this check will be in the files ‘precfl3d.error’ and ‘precfl3d.out’. Input errors will cause the output in ‘precfl3d.out’ to stop at the line at which the error occurred. Often informative diagnostics will be output there.
- When the checker ‘precfl3d’ has determined that the input is properly configured, the top of ‘cfl3d.out’ will show the input values it has read.
- Other checks (e.g. grid dimension, blocking, incompatibility of a restart file) are performed in ‘cfl3d’. Error output including the suspected cause of the termination will be found in ‘cfl3d.error’. Sometimes additional insight into the cause of the error can be found by checking the main output in ‘cfl3d.out’ although frequently there is little additional diagnostic output in ‘cfl3d.out’ if the code terminates.

Setting Up a Steady Run

Title line and condition data

```
input/output files:
grid.bin
plot3dg.bin
plot3dq.bin
cfl3d.out
cfl3d.res
cfl3d.turres
cfl3d.blomax
cfl3d.out15
cfl3d.prout
cfl3d.out20
ovrlp.bin
patch.bin
restart.bin
    NLR7301 airfoil, cfl3d type grid
    Xmach    alpha    beta     ReUe  Tinf,dR   ialph   ihstry
    0.753     1.10     0.0   5.7567    460.0       0        0
     sref     cref    bref      xmc      ymc     zmc
      1.0      1.0     1.0    0.075      0.0     0.0
       dt    irest  iflagts    fmax    iunst  cfl_tau
     -2.0        0        0     1.0        0     5.0
```

We will now focus on these and subsequent lines

```
      ngrid   nplot3d    nprint   nwrest     ichk      i2d   ntstep      ita
          1         1         1     1000        0        1        1       -2
        ncg       iem  iadvance   iforce  ivisc(i) ivisc(j) ivisc(k)
          2         0         0        1        0        0        5
       idim      jdim      kdim
          2       273        93
     ilamlo    ilamhi    jlamlo   jlamhi   klamlo   klamhi
          0         0         0        0        0        0
      inewg    igridc        is       js       ks       ie       je       ke
          0         0         0        0        0        0        0        0
   idiag(i)  idiag(j)  idiag(k) iflim(i) iflim(j) iflim(k)
          1         1         1        4        4        4
    ifds(i)   ifds(j)   ifds(k) rkap0(i) rkap0(j) rkap0(k)
          1         1         1   0.3333   0.3333   0.3333
       grid    nbci0    nbcidim   nbcj0  nbcjdim   nbck0  nbckdim   iovrlp
          1         1         1        1        1        3        1        0
i0:    grid   segment    bctype     jsta     jend     ksta     kend    ndata
          1         1      1002        0        0        0        0        0
idim:grid     segment    bctype     jsta     jend     ksta     kend    ndata
          1         1      1002        0        0        0        0        0
j0:    grid   segment    bctype     ista     iend     ksta     kend    ndata
          1         1      1003        0        0        0        0        0
jdim:grid     segment    bctype     ista     iend     ksta     kend    ndata
          1         1      1003        0        0        0        0        0
k0:    grid   segment    bctype     ista     iend     jsta     jend    ndata
1         1         0         0        0        0        1       37        0
1         2      2004         0        0        0       37      237        2
    tw/tinf        cq
         0.        0.
          1         3         0        0        0      237      273        0
```

Setting Up a Steady Run

Title line and condition data

Condition title line

Angle of attack, Deg.

Sideslip, Deg.

```
NLR7301 airfoil, cfl3d type C-grid
       Xmach    alpha    beta    ReUe   Tinf,dR  ialph  ihstry
       0.753     1.10     0.0   5.7567   460.0       0       0
```

Condition data line

Free-stream temperature, degrees Rankine

ialph – indicator to determine whether angle of attack is measured in the x-z plane or the x-y plane

ihstry – determines which variables are to be tracked for convergence history. Default is C_l, C_d, C_y (or C_z), $C_{m.}$

Input of ReUe (Reynolds number) requires some additional explanation….

Setting Up a Steady Run

Calculation of *ReUe*

Reference length

Recall the nondimensionalizations:

$$x = \frac{\tilde{x}}{\tilde{L}_R} \qquad y = \frac{\tilde{y}}{\tilde{L}_R} \qquad z = \frac{\tilde{z}}{\tilde{L}_R} \qquad t = \frac{\tilde{t}\,\tilde{a}_\infty}{\tilde{L}_R}$$

$$\rho = \frac{\tilde{\rho}}{\tilde{\rho}_\infty} \qquad u = \frac{\tilde{u}}{\tilde{a}_\infty} \qquad v = \frac{\tilde{v}}{\tilde{a}_\infty} \qquad w = \frac{\tilde{w}}{\tilde{a}_\infty}$$

Reynolds number based on reference length:

$$\mathrm{Re}_{\tilde{L}_R} = \frac{\tilde{\rho}_\infty \left|\tilde{V}_\infty\right| \tilde{L}_R}{\tilde{\mu}_\infty}$$

Setting Up a Steady Run

Calculation of *ReUe*

Calculation of $ReUe$

$$ReUe = \mathrm{Re}_{\widetilde{L}_R} \times 10^{-6} = \frac{\widetilde{\rho}_\infty \left|\widetilde{V}_\infty\right| \widetilde{L}_R}{\widetilde{\mu}_\infty} \times 10^{-6} = \frac{\widetilde{\rho}_\infty M_\infty \sqrt{\gamma R T_\infty}\, \widetilde{L}_R}{\widetilde{\mu}_\infty} \times 10^{-6}$$

Example: Suppose we have a grid that is in inches, and we wish to retain that length scale so that the grid remains compatible with a finite element model of the wing structure that is also in inches. Suppose the Reynolds number is 1 million based on chord length of 20 inches.

Set $\widetilde{L}_R = 1\ \ inch$, then $\mathrm{Re}_{\widetilde{L}_R} = \mathrm{Re}_c(\widetilde{L}_R / c) = 50{,}000, \quad ReUe = 0.05$

$ReUe$ is the Reynolds number per unit grid length in millions

Setting Up a Steady Run

Reference data input

Center for moment calculations, in grid units

sref	cref	bref	xmc	ymc	zmc
1.0	1.0	1.0	0.075	0.0	0.0

Reference length used in calculation of roll moment coefficient, in grid units

Reference length used in calculation of pitch moment coefficient, in grid units

Reference area used in calculation of force coefficients, in grid units

Setting Up a Steady Run

Steady solution cycling input

```
input/output files:
grid.bin
plot3dg.bin
plot3dq.bin
cfl3d.out
cfl3d.res
cfl3d.turres
cfl3d.blomax
cfl3d.out15
cfl3d.prout
cfl3d.out20
ovrlp.bin
patch.bin
restart.bin
   NLR7301 airfoil, cfl3d type grid
   Xmach     alpha      beta      ReUe   Tinf,dR     ialph     ihstry
   0.753      1.10       0.0    5.7567     460.0         0          0
    sref      cref      bref       xmc       ymc       zmc
     1.0       1.0       1.0     0.075       0.0       0.0
      dt     irest   iflagts      fmax     iunst   cfl_tau
    -2.0         0         0       1.0         0       5.0
```

We will now want to focus on these three lines

```
 ngrid   nplot3d    nprint    nwrest      ichk       i2d    ntstep       ita
     1         1         1      1000         0         1         1        -2
   ncg       iem  iadvance    iforce  ivisc(i)  ivisc(j)  ivisc(k)
     2         0         0         1         0         0         5
  idim      jdim      kdim
     2       273        93
ilamlo    ilamhi    jlamlo    jlamhi    klamlo    klamhi
     0         0         0         0         0         0
 inewg    igridc        is        js        ks        ie        je        ke
     0         0         0         0         0         0         0         0
idiag(i)  idiag(j)  idiag(k)  iflim(i)  iflim(j)  iflim(k)
     1         1         1         4         4         4
ifds(i)   ifds(j)   ifds(k)  rkap0(i)  rkap0(j)  rkap0(k)
     1         1         1    0.3333    0.3333    0.3333

                         .
                         .
                         .

  mseq    mgflag    iconsf       mtt      ngam
     1         1         0         0         2
  issc epsssc(1) epsssc(2) epsssc(3)      issr epsssr(1) epsssr(2) epsssr(3)
     0       0.3       0.3       0.3         0       0.3       0.3       0.3
  ncyc    mglevg     nemgl     nitfo
  2000         3         0         0
  mit1      mit2      mit3      mit4      mit5  ...
     1         1         1
```

Setting Up a Steady Run

Steady solution cycling input

Time step parameters:

CFL number
(for steady run)

dt	irest	iflagts	fmax	iunst	cfl_tau
-2.0	0	0	1.0	0	5.0

Number of time step advances, and time accuracy:

ngrid	nplot3d	nprint	nwrest	ichk	i2d	ntstep	ita
1	1	1	1000	0	1	1	-2

Number of
time steps

Cycle control:

Number of cycles

ncyc	mglevg	nemgl	nitfo
2000	3	0	0

Setting Up a Steady Run

Steady solution cycling input

dt	irest	iflagts	fmax	iunst	cfl_tau		
-2.0	0	0	1.0	0	5.0		
		.					
		.					
ngrid	nplot3d	nprint	nwrest	ichk	i2d	ntstep	ita
1	1	1	1000	0	1	1	-2
		.					
		.					
ncyc	mglevg	nemgl	nitfo				
2000	3	0	0				

Note:

– when $dt < 0$, local time stepping is used, i.e. $CFL = |dt|$. This is used for converging a steady state solution. For steady state computations

$$\Delta\tau = CFL \cdot \Delta r$$

where Δr is a measure of local grid spacing and $\Delta\tau$ is the local pseudo time step size.

– *cfl_tau* is not used when $dt < 0$. The value input for that parameter is a placeholder.

– *iunst* is set to *0* in the code when $dt < 0$.

– *ntstep* is set to *1* in the code when $dt < 0$.

– *ncyc* controls the number of steady solution cycles computed.

– Values of *dt* of -2.0 to -10.0 are typical. Lower values will be required for a stiffer problem.

Setting Up a Steady Run

Grid sequencing

Grid sequencing can and should be used to accelerate convergence to a steady state solution. The following input sequences through three grid levels.

Number of coarser levels to be created

```
          .
          .
ncg      iem  iadvance    iforce    ivisc(i)   ivisc(j)  ivisc(k)
  2        0         0         1          0          0         5

          .
          .
          .
mseq     mgflag    iconsf        mtt     ngam
  3           1         0          0        2
issc epsssc(1) epsssc(2) epsssc(3)    issr   epsssr(1) epsssr(2) epsssr(3)
  0       0.3       0.3       0.3        0         0.3       0.3       0.3
ncyc   mglevg     nemgl         nitfo
2000        1         0             0
1000        2         0             0
 500        3         0             0
mit1      mit2      mit3       mit4     mit5  ...
   1
   1         1
   1         1         1
          .
          .
          .
```

Number of sequence levels

Sequencing from coarsest to finest grid level, *mseq* lines required

mseq lines required

Setting Up a Steady Run

Grid sequencing output

The following grid level information will be found in the cfl3d.out on the completion of the 3D single block C-grid airfoil computation:

Because ncg = 2, two coarser levels created

```
.
.
reading grid   1 of dimensions (I/J/K) :   2 273  93
   creating coarser block   2 of dimensions (I/J/K) :   2 137  47
   creating coarser block   3 of dimensions (I/J/K) :   2  69  24
.
.
.
***** BEGINNING TIME ADVANCEMENT, iseq = 1 *****

steady-state computations

***** BEGINNING MULTIGRID CYCLE *****

iseq=    1
level top =  1
level bottom =  1
number of global grid levels =  1
lglobal=  1
.
.
.
```

Coarsest to mid level

```
.
.
.
***** BEGINNING SEQUENCING TO FINER LEVEL *****

interpolating solution on coarser block   3  to   finer block   2 (grid   1)
  jdim,kdim,idim (finer grid)=  137   47    2
  jj2,kk2,ii2    (coarser grid)=    69   24    2
  interpolating turb quantities from coarser to finer block

***** ENDING SEQUENCING TO FINER LEVEL *****

***** BEGINNING TIME ADVANCEMENT, iseq = 2 *****

steady-state computations

***** BEGINNING MULTIGRID CYCLE *****

     iseq=    2
     level top =  2
     level bottom =  1
     number of global grid levels =  2
     lglobal=  2
.
.
.
```

Setting Up a Steady Run

Grid sequencing output

```
.
.
.

    ***** BEGINNING SEQUENCING TO FINER LEVEL *****

 interpolating solution on coarser block   2  to   finer block   1 (grid   1)
  jdim,kdim,idim (finer grid)=  273   93    2
  jj2,kk2,ii2     (coarser grid)= 137   47    2
  interpolating turb quantities from coarser to finer block

***** ENDING SEQUENCING TO FINER LEVEL *****

***** BEGINNING TIME ADVANCEMENT, iseq = 3 *****

 steady-state computations

***** BEGINNING MULTIGRID CYCLE *****

 iseq=    3
 level top =  3
 level bottom =  1
 number of global grid levels =  3
 lglobal=  3
```

Mid to finest level

Setting Up a Steady Run

Grid sequencing

```
 ncg      iem  iadvance    iforce    ivisc(i)   ivisc(j)  ivisc(k)
  2         0         0        1          0          0         5
                      .
                      .
 idim     jdim      kdim
  2        273        93
                      .
                      .
.
mseq    mgflag    iconsf       mtt     ngam
   3         1         0         0        2
 issc epsssc(1) epsssc(2) epsssc(3)   issr   epsssr(1) epsssr(2) epsssr(3)
   0       0.3       0.3       0.3      0        0.3       0.3       0.3
 ncyc   mglevg    nemgl      nitfo
2000         1        0          0
1000         2        0          0
 500         3        0          0
 mit1      mit2      mit3      mit4     mit5  ...
   1
   1         1
   1         1         1
```

These dimensions support up to four multigrid levels. See version 5.0 manual for a table of multigridable dimensions. Note that *idim* is not multigridded for a 2D grid.

Note:

- The number of grid levels that will have been created are the coarser levels (*ncg*) plus the finest level. Therefore, *mseq* must be equal to or less than $ncg + 1$. Setting *mseq* higher than this will result in a termination and an error message in *precfl3d.out*.
- The permissible value of *ncg* will depend on the dimensions of the grid. It is usually good to have three to four possible levels of multi-grid. For example, since four levels of multi-grid are possible with this grid, we could have set $ncg = 3$.

Setting Up a Steady Run

Grid sequencing

Note:

- Many more cycles will be done at the coarser levels. The computing required for a 3D grid will be a factor of 8 cheaper at each coarser level. For the present problem, the coarsest level would be 64 times cheaper than the finest level if a 3D grid had been used. Since it is a 2D grid it will be 16 times cheaper.
- It is usually good to completely converge the coarser levels before proceeding to the finer level. However, some problems will not compute well at a coarse level, but will compute at a finer level.
- *Mglevg* is always starting from the finest level … as the following example will show…

Setting Up a Steady Run

Grid sequencing

Example: We wish to compute on only the two coarser levels with the grid used in the previous example. The following input has been set up:

.
.

ncg	iem	iadvance	iforce	ivisc(i)	ivisc(j)	ivisc(k)
2	0	0	1	0	0	5

.
.
.

mseq	mgflag	iconsf	mtt	ngam
2	1	0	0	2

issc	epsssc(1)	epsssc(2)	epsssc(3)	issr	epsssr(1)	epsssr(2)	epsssr(3)
0	0.3	0.3	0.3	0	0.3	0.3	0.3

ncyc	mglevg	nemgl	nitfo
2000	1	0	0
1000	2	0	0

mit1	mit2	mit3	mit4	mit5 ...
1				
1	1			

.
.
.

Value of ncg is unchanged, but now set mseq = 2

Based on this input, you would expect CFL3D to compute on the two coarsest levels, but it actually computes on the second and finest levels…

Setting Up a Steady Run

Grid sequencing

…Here is what is actually output in cfl3d.out:

```
***** BEGINNING TIME ADVANCEMENT, iseq = 1 *****

 steady-state computations

***** BEGINNING MULTIGRID CYCLE *****

 iseq=    1
 level top =  2
 level bottom =  2
 number of global grid levels =  1
 lglobal=  2
                    .
                    .
                    .

***** BEGINNING SEQUENCING TO FINER LEVEL *****

 interpolating solution on coarser block   2  to   finer block   1 (grid   1)
   jdim,kdim,idim (finer grid)=  273   93    2
   jj2,kk2,ii2     (coarser grid)= 137   47    2
   interpolating turb quantities from coarser to finer block

***** ENDING SEQUENCING TO FINER LEVEL *****

***** BEGINNING TIME ADVANCEMENT, iseq = 2 *****

 steady-state computations

***** BEGINNING MULTIGRID CYCLE *****

 iseq=    2
 level top =  3
 level bottom =  2
 number of global grid levels =  2
 lglobal=  3
```

Computations performed on the middle and finest grids

Setting Up a Steady Run

Grid sequencing at coarsest levels only

Here is how to compute only on the two coarsest levels:

```
                .
                .
  ncg       iem  iadvance     iforce     ivisc(i)    ivisc(j)  ivisc(k)
    2         0         0          1          0           0         5

                .
                .
                .
mseq     mgflag     iconsf        mtt     ngam
    3         1          0          0        2
  issc epsssc(1) epsssc(2) epsssc(3)    issr   epsssr(1) epsssr(2) epsssr(3)
    0       0.3       0.3       0.3        0        0.3       0.3       0.3
 ncyc    mglevg     nemgl       nitfo
 2000         1         0          0
 1000         2         0          0
    0         3         0          0
  mit1      mit2      mit3       mit4      mit5 ...
     1
     1         1
     1         1         1
               .
               .
               .
```

The finest level is included but with zero cycles

Setting Up a Steady Run

Grid sequencing at coarsest levels only

….and here is the output:

```
***** BEGINNING TIME ADVANCEMENT, iseq = 1 *****

 steady-state computations

***** BEGINNING MULTIGRID CYCLE *****

 iseq=    1
 level top =  1
 level bottom =  1
 number of global grid levels =  1
 lglobal=  1
                        .
                        .
                        .

***** BEGINNING SEQUENCING TO FINER LEVEL *****

 interpolating solution on coarser block   3  to   finer block   2 (grid   1)
   jdim,kdim,idim (finer grid)=  137   47    2
   jj2,kk2,ii2     (coarser grid)=   69   24    2
   interpolating turb quantities from coarser to finer block

***** ENDING SEQUENCING TO FINER LEVEL *****

***** BEGINNING TIME ADVANCEMENT, iseq = 2 *****

 steady-state computations

***** BEGINNING MULTIGRID CYCLE *****

 iseq=    2
 level top =  2
 level bottom =  1
 number of global grid levels =  2
 lglobal=  2
```

Computations performed on the coarsest and middle levels

Setting Up a Steady Run

Grid sequencing at coarsest levels only

Why is it sometimes valuable to compute on the coarser levels only?

- Cost effectiveness of coarser levels
- Sometimes it is not possible to converge the finest level
- Many times you will want to compute unsteady solutions on coarser levels only, especially when debugging. Computing unsteady solutions on coarser levels only requires the steady starting point be on a coarser level.

Setting Up a Steady Run

Ramping up *dt*

Sometimes it is useful for stiff problems to ramp up the time step size. Ramping up the time step size is accomplished with the following input:

No. of cycles over which time step ramping occurs

dt	irest	iflagts	fmax	iunst	cfl_tau
-2.0	0	1000	5.0	0	5.0

$dt_{ending} = fmax * dt_{initial}$

$dt_{initial}$

In this example, the final CFL value of 10 is obtained after 1000 cycles. Note that this counter is reset with each restart. Therefore, $dt_{initial}$ will have to be reset to the dt_{ending} of the previous run.

Setting Up a Steady Run

Additional input

irest = 0 - do not read restart
irest = 1 - read restart file

dt	irest	iflagts	fmax	iunst	cfl_tau		
-2.0	0	1000	5.0	0	5.0		
ngrid	nplot3d	nprint	nwrest	ichk	i2d	ntstep	ita
1	1	1	1000	0	1	1	-2

No. of cycles (or time steps) between restart file writes

Parameter controlling accuracy of unsteady solution

No. of grid blocks to be read from the grid file

Controls checks for negative values. Usually set to 0.

i2d = 0 - 3D case
i2d = 1 - 2D case
i2d =-1 - 2D case with far-field vortex correction

Setting Up a Steady Run

Additional input

Flag for residual/update usually set to 0

Embedded mesh flag, usually 0

Parameters controlling level of turbulence modeling in the i, j, k directions

ncg	iem	iadvance	iforce	ivisc(i)	ivisc(j)	ivisc(k)
2	0	0	1	0	0	5
idim	jdim	kdim				
2	273	93				

This card repeated *ngrid* times

This card repeated *ngrid* times

Flag controlling force computations on block Faces. Format is IJK, e.g. 100 calculates force On solid i=1 surfaces, 10 calculates force on solid j=1 surfaces, etc.… See version 5 manual for more

Setting Up a Steady Run

Turbulence model input

There are more than 13 turbulence models available, but the following are the most common turbulence models and the corresponding parameter input values:

0	-	inviscid
1	-	laminar
3	-	turbulent, Baldwin-Lomax with Degani-Schiff option (not recommended)
5	-	turbulent, Spalart-Allmaras model
6	-	turbulent, Wilcox k-ω
7	-	turbulent, k-ω SST (Menter's version)
13	-	nonlinear EASM k-ε model
14	-	nonlinear EASM k-ω model

See the CFL3D Version 5.0 manual (Appendix H) and the CFL3D Version 6 web page (under `New Features') for descriptions of these and other models. See also under the 'Keywords' discussion in these notes for parameters that turn turbulence model features on.

Setting Up a Steady Run

Turbulence model

Several key notes on turbulence models:

1. If $ivisc(m) < 0$, a wall function is employed
2. Thin-layer viscous terms (laminar or turbulent) can be included in the i,j or k directions separately or combined. Cross-derivatives are not included. For the Baldwin-Lomax model, terms are allowed simultaneously in two directions only, either j-k or i-k.
3. Using the Baldwin-Lomax model with multi-zonal grids, wall distances are calculated only within a given zone.
4. It is preferable to let k be the primary viscous direction and i be secondary viscous direction.
5. The minimum distance function *smin* is computed from viscous walls only, not inviscid walls.

Setting Up a Steady Run

Turbulence model

6. Note that the field equation turbulence models may or may not transition to turbulent flow. Whether they transition will largely be determined by the free stream value of turbulence. Free stream turbulence level can be set in the key word input.
7. There are several places in which the turbulence level can be checked
   - There is an option allows the output of turbulence quantities in the plot3d file.
   - The file ‘cfl3d.prout’ contains the value of the turbulent viscosity. This is shown in the next slide.

See the CFL3D User’s Manual, Version 5.0, Section 3.7 for more complete discussion

Setting Up a Steady Run

Turbulence model output

The top of the ‘cfl3d.prout’ file is shown here:

Turbulent viscosity

```
 NASA Langley BACT Model: NACA 0012 af, AR=1.5 wing,.75TE Flap
   Mach     alpha      beta      ReUe   Tinf,dR      time
 0.82000   0.00000   0.00000 0.236E+07 486.00000   0.03839


BLOCK   1  (GRID   1)    IDIM,JDIM,KDIM=   73  345   73
NOTE: endpts may not be reliable

 I   J  K          X              Y               Z              U/Uinf        V/Vinf        W/Winf         P/Pinf        T/Tinf          MACH           cp           tur. vis.
 1   1   1  0.70000E+01  0.00000E+00  0.18698E-09  0.10000E+01 -0.38013E-18  0.72322E-13  0.10000E+01  0.10000E+01  0.82000E+00  0.50654E-07  0.90000E-02
 1   2   1  0.68895E+01  0.00000E+00  0.18866E-09  0.10000E+01 -0.16458E-16 -0.14259E-15  0.10000E+01  0.10000E+01  0.82000E+00  0.50654E-07  0.90000E-02
                                      .
                                      .
```

Data lines will be printed out for all flow field points specified by the user in the ‘print out’ portion of the input file.

Setting Up a Steady Run

Miscellaneous input

Lower and upper i,j,k indices of laminar region

ilamlo	ilamhi	jlamlo	jlamhi	klamlo	klamhi
0	0	0	0	0	0

This card repeated *ngrid* times

inewg	igridc	is	js	ks	ie	je	ke
0	0	0	0	0	0	0	0

This card repeated *ngrid* times

Embedded mesh specifications. Zero if no embedded mesh. See version 5.0 manual for more information

Setting Up a Steady Run

Miscellaneous input

Flux limiter flag in the i,j,k directions.
iflim = 3 was recommended in Version 5.0
iflim = 4 is recommended in Version 6.0

idiag(i)	idiag(j)	idiag(k)	iflim(i)	iflim(j)	iflim(k)
1	1	1	4	4	4

This card repeated *ngrid* times

ifds(i)	ifds(j)	ifds(k)	rkap0(i)	rkap0(j)	rkap0(k)
1	1	1	0.3333	0.3333	0.3333

This card repeated *ngrid* times

Spatial differencing in the i,j,k directions.
ifds = 1 – flux-difference splitting (Roe's) (recommended)

Spatial differencing parameter for Euler fluxes in the i,j,k directions.
rkap0 = 1/3 - upwind-biased third order (recommended)

Setting Up an Unsteady Run

Input for time advancement

We will again focus on these three lines

```
input/output files:
grid.bin
plot3dg.bin
plot3dq.bin
cfl3d.out
cfl3d.res
cfl3d.turres
cfl3d.blomax
cfl3d.out15
cfl3d.prout
cfl3d.out20
ovrlp.bin
patch.bin
restart.bin
   NLR7301 airfoil, cfl3d type grid
   Xmach    alpha     beta     ReUe   Tinf,dR   ialph   ihstry
   0.753     1.10      0.0   5.7567     460.0       0        0
    sref     cref     bref      xmc       ymc     zmc
     1.0      1.0      1.0    0.075       0.0     0.0
      dt    irest  iflagts     fmax     iunst cfl_tau
     .05        1        0      1.0         0     5.0
```

```
  ngrid  nplot3d   nprint  nwrest    ichk      i2d   ntstep    ita
      1        1        1    1000       0        1        1     -2
    ncg      iem iadvance  iforce  ivisc(i) ivisc(j) ivisc(k)
      2        0        0       1       0        0        5
   idim     jdim     kdim
      2      273       93
 ilamlo   ilamhi   jlamlo  jlamhi  klamlo   klamhi
      0        0        0       0       0        0
  inewg   igridc       is      js      ks       ie       je     ke
      0        0        0       0       0        0        0      0
idiag(i) idiag(j) idiag(k) iflim(i) iflim(j) iflim(k)
      1        1        1       4       4        4
 ifds(i)  ifds(j)  ifds(k) rkap0(i) rkap0(j) rkap0(k)
      1        1        1  0.3333  0.3333   0.3333
                         .
                         .
                         .
   mseq   mgflag   iconsf     mtt    ngam
      1        1        0       0       2
   issc epsssc(1) epsssc(2) epsssc(3)  issr epsssr(1) epsssr(2) epsssr(3)
      0      0.3       0.3       0.3      0       0.3       0.3       0.3
   ncyc   mglevg    nemgl   nitfo
      4        3        0       0
   mit1     mit2     mit3    mit4    mit5 ...
      1        1        1
```

Setting Up an Unsteady Run

Input for time advancement

Time step parameters:

Non-dimensional time step size

dt	irest	iflagts	fmax	iunst	cfl_tau
.05	1	0	1.0	0	5.0

Number of time step advances, and time accuracy:

ngrid	nplot3d	nprint	nwrest	ichk	i2d	ntstep	ita
1	1	1	1000	0	1	100	-2

Number of time steps

Parameter controlling time accuracy and dual time stepping

Iterative control:

ncyc	mglevg	nemgl	nitfo
4	3	0	0

Number of sub-iterations

Setting Up an Unsteady Run

Input for time advancement

Order of time-accuracy, dual time scheme flag (*ita*)

ita = +1	First order accurate in time; physical time term only (t-TS) method
ita = +2	Second order accurate in time; physical time term only (t-TS) method
ita = -1	First order accurate in time; physical time and pseudo time term (τ-TS) method
ita = -2	Second order accurate in time; physical time and pseudo time term (τ-TS) method

Setting Up an Unsteady Run

Input for time advancement

Note:

- The approximate factorization scheme used to advance the solution in time introduces first order errors in time. Furthermore, if the diagonal version is utilized ($idiag = 1$), additional errors of order $\Delta\tau$ are introduced. Sub-iterations can be used to drive these factorization errors to zero. Therefore, if a formally second-order (in time) solution is desired, sub-iterations must be used.
- The inclusion of a pseudo time term increases (often dramatically) the maximum allowable time step one can take for a particular problem. However, sub-iterations ($ncyc > 1$) are therefore mandatory and multi-grid is highly recommended.
- Larger time steps imply greater error, therefore second order is recommended.
- You will almost never want to use the t-TS method of time stepping.

Setting Up an Unsteady Run

Equations for τ-TS time advancement

Non-dimensional time step increment

Sub-iteration index

$$\left[\left(\frac{1+\phi'}{J\Delta\tau}+\frac{1+\phi}{J\Delta t}\right)I+\delta_\xi A+\delta_\eta B+\delta_\zeta C\right]\Delta Q^m=$$

$$\frac{\phi'\Delta Q^{m-1}}{J\Delta\tau}+\frac{\phi\Delta Q^{n-1}}{J\Delta t}-\frac{(1+\phi)(Q^m-Q^n)}{J\Delta t}+R(Q^m)$$

Pseudo time step increment

Current time step index

Setting Up an Unsteady Run

Equations for t-TS time advancement

The pseudo time terms are omitted for t-TS time advancement:

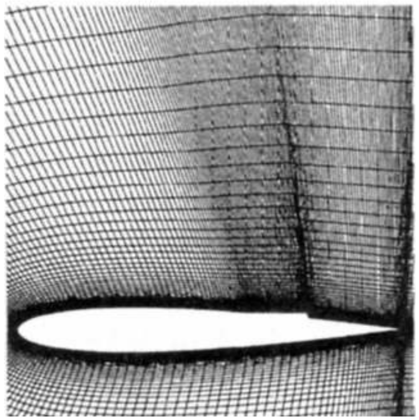
$$\left[\left(\frac{1+\phi}{J\Delta t}\right)I + \delta_\xi A + \delta_\eta B + \delta_\zeta C\right]\Delta Q^m =$$

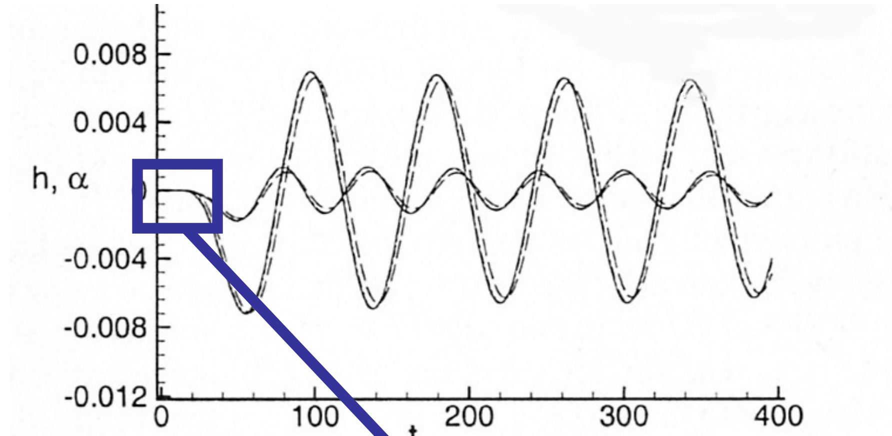
$$\frac{\phi \Delta Q^{n-1}}{J\Delta t} - \frac{(1+\phi)(Q^m - Q^n)}{J\Delta t} + R(Q^m)$$

Non-dimensional time step increment

Setting Up an Unsteady Run

Case study: The t-TS and τ-TS schemes, oscillating spoiler





The solution using the t-TS scheme blows up even at a very small time step size

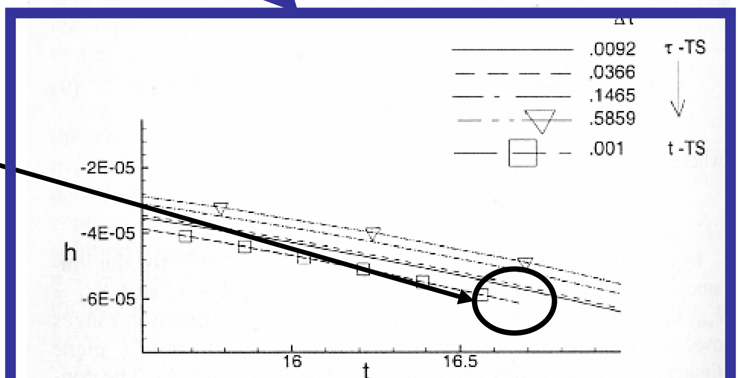


Fig. 6 Comparison of t-TS and τ-TS time stepping for aeroelastic response (same model and condition as Fig. 4).

From: Bartels, R. E., "Mesh Strategies for Accurate Computation of Unsteady Spoiler and Aeroelastic Problems," Journal of Aircraft, Vol. 37, No. 3, pp. 521-525.

Setting Up an Unsteady Run

Speeding up execution time

Parameters controlling the form of the Jacobian matrices used on the left hand side of the equations

idiag(i)	idiag(j)	idiag(k)	iflim(i)	iflim(j)	iflim(k)
1	1	1	4	4	4
ifds(i)	ifds(j)	ifds(k)	rkap0(i)	rkap0(j)	rkap0(k)
1	1	1	0.3333	0.3333	0.3333

Setting idiag(i), idiag(j), idiag(k) to 1 results in a very efficient trigiagonal inversion of the left hand side of the equations in the i, j and k directions.
However, be aware of the implications of setting this …..

Setting Up an Unsteady Run

Diagonalized versus full Jacobian matrices

idiag controls the form of the matrices *A, B, C* on the left hand side only. If *idiag = 0*, the full 5x5 matrix is used. If *idiag = 1*, the matrix is diagonalized (i.e. Very efficient scalar tridiagonal inversion of the left hand side of this equation).

$$\left[\left(\frac{1+\phi'}{J\Delta\tau}+\frac{1+\phi}{J\Delta t}\right)I+\delta_\xi A+\delta_\eta B+\delta_\zeta C\right]\Delta Q^m =$$

$$\frac{\phi'\Delta Q^{m-1}}{J\Delta\tau}+\frac{\phi\Delta Q^{n-1}}{J\Delta t}-\frac{(1+\phi)(Q^m-Q^n)}{J\Delta t}+R(Q^m)$$

Since $\Delta Q^m \rightarrow 0$ when the solution converges, setting *idiag = 1* does not affect accuracy, … *assuming* the solution has been adequately converged.

Setting Up an Unsteady Run

Sizing Δt, number of subiterations

Recall the non-dimensionalization of time:

$$\Delta t = \frac{\Delta \tilde{t}\ \tilde{a}_\infty}{\tilde{L}_R}$$

The reference length $\tilde{L}_R$ will be determined by the grid. For instance, if a wing with a 5 inch physical chord length is modeled with a grid that has a non-dimensional chord length of 5, then

$$\tilde{L}_R = \frac{5\ \ inches}{5} = 1\ \ inch$$

Note that in this case speed of sound, $\tilde{a}_\infty$ must be in inches/second.

Setting Up an Unsteady Run

Sizing Δt, number of subiterations

- One criteria for time step sizing is the time scale required to resolve a phenomenon at some frequency. Another is the number of time steps for a flow field particle to pass over a chord length. Consider 100 time steps per cycle or 100 time steps to pass over a chord length as the absolute minimum, which ever is smaller.

- The time step size and the number of sub-iterations may have to be set lower/higher respectively by either accuracy or robustness requirements. Short test runs should be performed to ensure adequate convergence.

Setting Up an Unsteady Run

Sizing Δt, number of subiterations

- Indicators that the time step size is too large:
  - The solution converges very slowly or does not converge at all.
  - The solution simply blows up.
  - There are large numbers of negative turbulence parameter values in the file 'cfl3d.subit_turres' the number of which is not converging toward zero at the end of each time step.
- Indicator that the number of sub-iterations is too small:
  - The force coefficients have not leveled out to an acceptable convergence level.
  - The residuals have dropped only by an insufficient magnitude. This can also be a sign that the time step is too large.
  - The solution has been converging, but eventually blows up or starts to gradually diverge.
- Note that these symptoms can also be due to problems with the grid, boundary conditions or turbulence model, so first ensure these issues are settled.

Setting Up an Unsteady Run

Sub-iterative output – checking convergence

The file 'cfl3d.subit_res' contains the following sub-iterative output

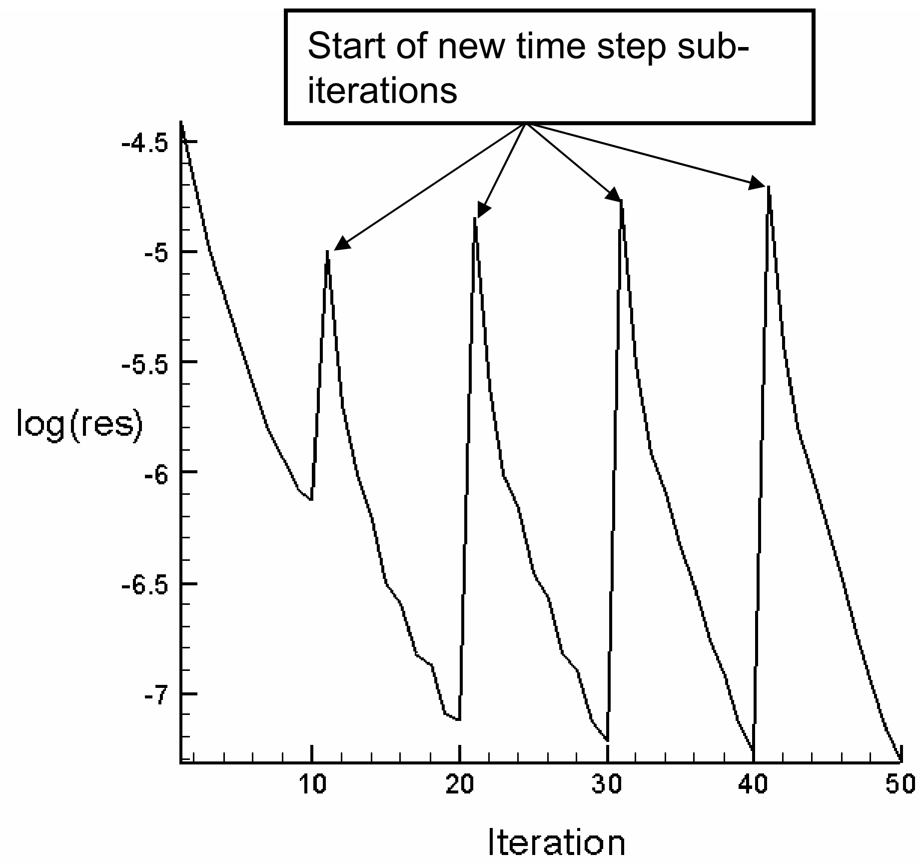
ncyc = 10 so there are 10 lines output per time step

subit	log(subres)	cl	cd	cy	cmy
1	-0.44098E+01	-0.56246E-02	0.29632E+00	0.00000E+00	0.14528E-02
2	-0.45238E+01	0.28737E-01	-0.12683E-01	0.00000E+00	-0.50177E-02
3	-0.49884E+01	0.26860E-01	0.19477E+00	0.00000E+00	-0.47901E-02
4	-0.48541E+01	0.25869E-01	0.80380E-01	0.00000E+00	-0.42342E-02
5	-0.54203E+01	0.26254E-01	0.10470E+00	0.00000E+00	-0.42906E-02
6	-0.53829E+01	0.27267E-01	0.98269E-01	0.00000E+00	-0.44789E-02
7	-0.58126E+01	0.27020E-01	0.10995E+00	0.00000E+00	-0.44088E-02
8	-0.57635E+01	0.26710E-01	0.10469E+00	0.00000E+00	-0.43687E-02
9	-0.60754E+01	0.26657E-01	0.10302E+00	0.00000E+00	-0.43724E-02
10	-0.61285E+01	0.26713E-01	0.10312E+00	0.00000E+00	-0.43877E-02
11	-0.49984E+01	0.26728E-01	0.10431E+00	0.00000E+00	-0.43800E-02
12	-0.56927E+01	0.26415E-01	0.92217E-01	0.00000E+00	-0.42151E-02
13	-0.60126E+01	0.26287E-01	0.83844E-01	0.00000E+00	-0.40628E-02
14	-0.62182E+01	0.26167E-01	0.82317E-01	0.00000E+00	-0.40236E-02
15	-0.65022E+01	0.26110E-01	0.82955E-01	0.00000E+00	-0.40152E-02
16	-0.65972E+01	0.26076E-01	0.83164E-01	0.00000E+00	-0.40164E-02
17	-0.68247E+01	0.26050E-01	0.82959E-01	0.00000E+00	-0.40162E-02
18	-0.68719E+01	0.26052E-01	0.82589E-01	0.00000E+00	-0.40151E-02
19	-0.70916E+01	0.26059E-01	0.82439E-01	0.00000E+00	-0.40141E-02
20	-0.71274E+01	0.26055E-01	0.82404E-01	0.00000E+00	-0.40133E-02

Note that all iterations are output sequentially

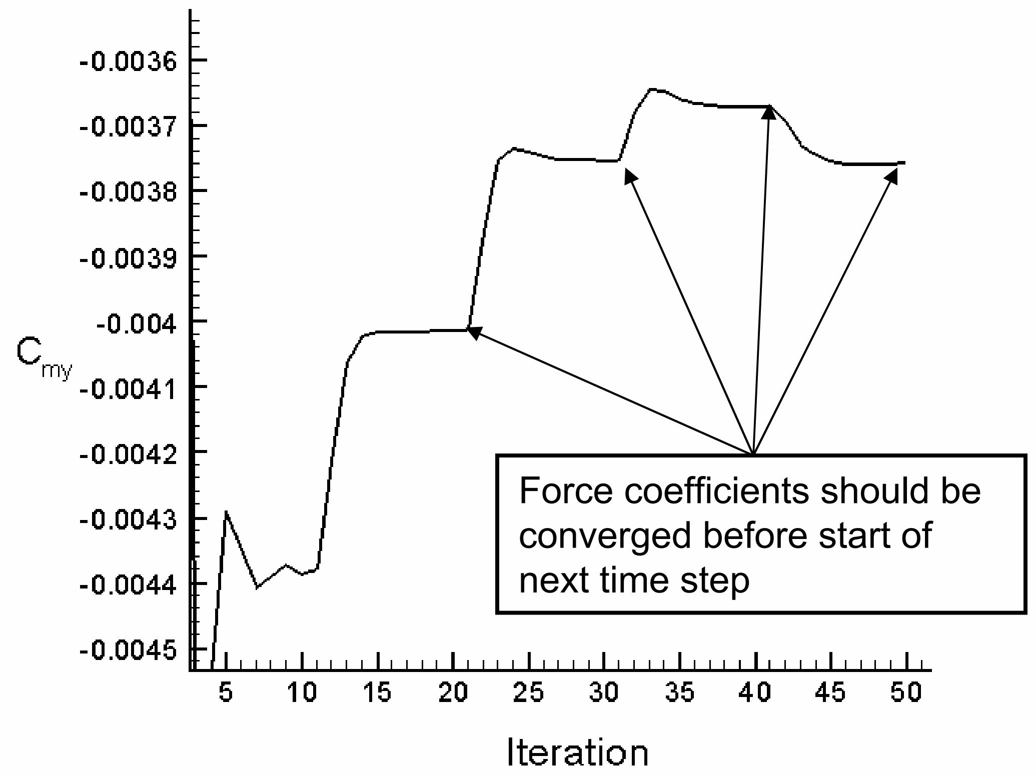
Setting Up an Unsteady Run

Sub-iterative output– checking convergence



Setting Up an Unsteady Run

Sub-iterative output– checking convergence



Setting Up an Unsteady Run

Sub-iterative turbulence output

The file 'cfl3d.subit_turres' contains the following sub-iterative output for Menter's shear stress transport (SST) k-w turbulence model:

In this case ncyc = 10 so there are 10 turbulence model iterations per time step.

subit	log(turres1)	log(turres2)	nneg1	nneg2
1	-0.73658E+01	-0.92553E+01	0	710
2	-0.74563E+01	-0.91092E+01	0	82
3	-0.76424E+01	-0.90767E+01	0	2
4	-0.80379E+01	-0.90899E+01	0	0
5	-0.82466E+01	-0.93470E+01	0	8
6	-0.84600E+01	-0.93751E+01	0	30
7	-0.86186E+01	-0.95757E+01	0	58
8	-0.88672E+01	-0.97150E+01	0	56
9	-0.89497E+01	-0.98376E+01	0	48
10	-0.91579E+01	-0.99516E+01	0	38
.				
.				
.				
.				
51	-0.95921E+01	-0.88827E+01	2498	2149
52	-0.95925E+01	-0.90172E+01	2340	2693
53	-0.95509E+01	-0.91643E+01	2124	2603
54	-0.99381E+01	-0.90386E+01	1959	1193
55	-0.98511E+01	-0.91025E+01	2244	1252
56	-0.99244E+01	-0.92361E+01	3529	1393
57	-0.10161E+02	-0.91691E+01	2373	1486
58	-0.10217E+02	-0.91525E+01	1395	1360
59	-0.10304E+02	-0.92210E+01	1266	1460
60	-0.10377E+02	-0.93327E+01	1109	1218

Note that there are a few grid points that have negative values of k and ω initially…

…however, large numbers of negative values of turbulence model parameters indicate a potential problem

Even though the turbulence model appears to be converging well, a large number of negative values may mean that the time step size is too large for the turbulence model. Usually reducing time step size will fix this problem.

Setting Up an Unsteady Run

Multigrid strategies

- Multigrid is a must for unsteady computations. The following input section establishes four multigrid sub-iterations each on three levels, the third being the finest:

mseq	mgflag	iconsf	mtt	ngam
1	1	0	0	2

issc	epsssc(1)	epsssc(2)	epsssc(3)	issr	epsssr(1)	epsssr(2)	epsssr(3)
0	0.3	0.3	0.3	0	0.3	0.3	0.3

ncyc	mglevg	nemgl	nitfo
4	3	0	0

mit1	mit2	mit3	mit4	mit5 ...
1	1	1		

Mesh sequencing and multigrid parameters

Correction and residual smoothing, typically not used (*issc=issr=0*)

Multigrid cycling parameters

Number of iterations for each level, $mitL = 1$ recommended

Setting Up an Unsteady Run

Multigrid strategies

mseq	mgflag	iconsf	mtt	ngam			
1	1	0	0	2			
issc	epsssc(1)	epsssc(2)	epsssc(3)	issr	epsssr(1)	epsssr(2)	epsssr(3)
0	0.3	0.3	0.3	0	0.3	0.3	0.3
ncyc	mglevg	nemgl	nitfo				
4	3	0	0				
mit1	mit2	mit3	mit4	mit5 ...			
1	1	1					

Note:

- *iconsf* is a parameter for setting conservative flux treatment for embedded grids. For most computations it is set to zero.
- *mtt* is a flag for additional iterations on the up portion of the multigrid. Recommend setting to zero.
- *ngam* is the multigrid cycle flag. $ngam = 1$ sets V-cycle, $ngam = 2$ sets a W-cycle. The W-cycle is not recommended for overlapped grids.
- *mglevg* is the number of grids to use in multigrid cycling. E.g. $mglevg = 1$ sets the finest grid level only, $mglevg = 2$ sets two grid levels, etc…
- *nemgl* is set to zero when there are no embedded grids.
- *nitfo1* is the number of first order iterations. Zero is recommended.

Setting Up an Unsteady Run

Multigrid strategies

What if you want to compute an unsteady solution using multigrid on coarser levels only? Assume that the steady starting solution has been performed on coarser levels only, as we previously discussed. The following input will allow you to perform the unsteady run:

```
 mseq   mgflag   iconsf      mtt   ngam
    2        1        0        0      2
  issc epsssc(1) epsssc(2) epsssc(3)   issr  epsssr(1) epsssr(2) epsssr(3)
    0       0.3       0.3       0.3      0        0.3       0.3       0.3
  ncyc  mglevg    nemgl      nitfo
     4       2        0        0
     0       3        0        0
  mit1     mit2     mit3      mit4    mit5 ...
     1        1
     1        1        1
```

Note that a line with 0 sub-iterations is included for a 3 level multigrid

Setting Up an Unsteady Run

Multigrid strategies

….and here is the output:

```
.
.
reading grid   1 of dimensions (I/J/K) :   2 273  93
   creating coarser block   2 of dimensions (I/J/K) :   2 137  47
   creating coarser block   3 of dimensions (I/J/K) :   2  69  24
.
.
.
 reading restart file for block   2  (grid   1)
 reading vist3d data from restart file, block    2
 reading field eqn turb quantities from restart file, block    2
.
.
.
***** BEGINNING MULTIGRID CYCLE *****

  iseq=    1
  level top =  2
  level bottom =  1
  number of global grid levels =  2
  lglobal=  2
```

The full grid is read, and two coarser levels created

This level is the finest on which computations are performed

Restart data is read for coarser block 2 only

Setting Up an Unsteady Run

Multigrid strategies

```
                              .
                              .
interpolating correction from coarser block   3 to   finer block   2 (grid   1)
  jdim,kdim,idim (finer grid)=  137   47    2
  jj2,kk2,ii2    (coarser grid)=    69   24    2
                              .
                              .
writing restart file for block    2
  writing vist3d data to restart file, block    2
  writing field eqn turb quantities to restart file, block    2
writing 2nd order time data to restart file, block   2

***** ENDING TIME ADVANCEMENT, iseq = 1 *****


writing plot3d file for JDIM X KDIM =  137 x    47 grid
  plot3dg file is an xyz file at grid points
  plot3dq file is a    q file at grid points
  plot3d files to be read with /mgrid/blank/2d qualifiers

writing printout file for IDIM X JDIM X KDIM =    2 x   137 x    47 grid
```

Multigrid performed on the two coarser levels only

Only the coarser level solution is written to the restart file

Plot3D and print out data written for coarser level

User Specified Grid Motion

Overview

CFL3D has the capability to perform computations for prescribed surface motion in two ways

1. Prescribed, or user specified rigid grid motion. In this mode, the entire grid or set of grids translates or rotates in a manner prescribed by user input.

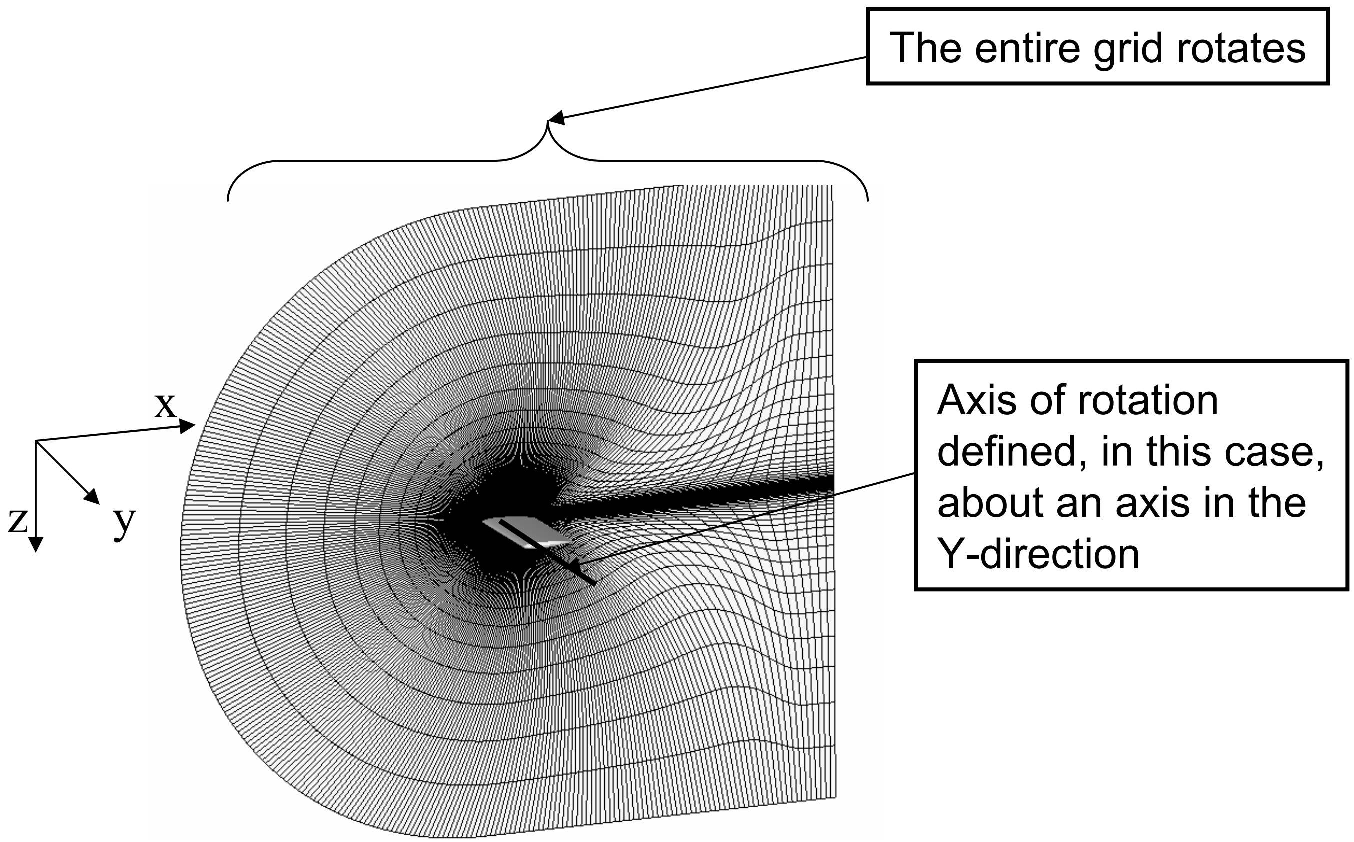
2. Prescribed surface motion with deforming mesh. In this mode, the surface(s) prescribed by the user translate or rotate and the mesh deforms accordingly.

These types of motion are only available when the code is running in unsteady mode.

User Specified Grid Motion

Rigid grid rotation

As an example consider the wing shown:



User Specified Rigid Grid Motion

Rigid grid rotation

The following unsteady input file performs rotation about the axis shown:

```
input/output files:
 wbgrid.cfl
 plot3dg.bin
 plot3dq.bin
 cfl3d.out
 cfl3d.res
 cfl3d.turres
 cfl3d.blomax
 cfl3d.out15
 cfl3d.prout
 cfl3d.out20
 ovrlp.bin
 patch.bin
 restart.bin
 NASA Langley BACT Model: NACA 0012 af, AR=1.5 wing,.75TE Flap
    Mach     alpha      beta      ReUe   Tinf,dR     ialph    ihstry
  0.82000  0.00000  0.00000 0.236E+07   486.00         1         0
     sref      cref      bref       xmc       ymc       zmc
    1.000  1.00000  1.00000   0.25000   0.00000   0.00000
       dt     irest    iflagts      fmax     iunst   cfl_tau
  0.04000         0      3000   1.00000         1   2.00000
    ngrid   nplot3d     nprint    nwrest      ichk       i2d    ntstep    ita
        1         1          1      1000         0         0         1     -2
```

Note that *iunst = 1* for rigid translation or rotation

User Specified Rigid Grid Motion

Rigid grid rotation

	ncg	iem	iadvance	iforce	ivisc(i)	ivisc(j)	ivisc(k)	
	2	0	0	1	5	5	5	
	idim	jdim	kdim					
	73	345	73					
	ilamlo	ilamhi	jlamlo	jlamhi	klamlo	klamhi		
	0	0	0	0	0	0		
	inewg	igridc	is	js	ks	ie	je	ke
	0	0	0	0	0	0	0	0
	idiag(i)	idiag(j)	idiag(k)	iflim(i)	iflim(j)	iflim(k)		
	1	1	1	3	3	3		
	ifds(i)	fds(j)	ifds(k)	rkap0(i)	rkap0(j)	rkap0(k)		
	1	1	1	0.3333	0.3333	0.3333		
	grid	nbci0	nbcidim	nbcj0	nbcjdim	nbck0	nbckdim	iovrlp
	1	1	1	1	1	5	1	0
i0:	grid	segment	bctype	jsta	jend	ksta	kend	ndata
	1	1	1001	1	345	1	73	0
idim:	grid	segment	bctype	jsta	jend	ksta	kend	ndata
	1	1	1002	1	345	1	73	0
j0:	grid	segment	bctype	ista	iend	ksta	kend	ndata
	1	1	1003	1	73	1	73	0
jdim:	grid	segment	bctype	ista	end	ksta	kend	ndata
	1	1	1003	1	73	1	73	0
k0:	grid	segment	bctype	ista	iend	jsta	jend	ndata
	1	1	0	1	49	1	33	0
	1	2	2004	1	49	33	313	2
	tw/tinf	cq						
	0.00000	0.00000						
	1	3	0	1	49	313	345	0
	1	4	0	49	73	1	173	0
	1	5	0	49	73	173	345	0
kdim:	grid	segment	bctype	ista	iend	jsta	jend	ndata
	1	1	1003	1	73	1	345	0

User Specified Rigid Grid Motion

Rigid grid rotation

```
  mseq     mgflag     iconsf        mtt   ngam
     1          2          1          0      2
   issc epsssc(1) epsssc(2) epsssc(3)    issr epsssr(1) epsssr(2) epsssr(3)
     0    0.3000    0.3000    0.3000       0    0.3000    0.3000    0.3000
   ncyc    mglevg     nemgl      nitfo
     8          3         0          0
   mit1      mit2      mit3       mit4    mit5  ...
     1          1         1
1-1 blocking data:
   nbli
     2
number      grid      ista       jsta   ksta    iend      jend      kend   isva1  isva2
     1         1         1          1      1      49        33         1       1      2
     2         1        49          1      1      73       173         1       1      2
number      grid      ista       jsta   ksta    iend      jend      kend   isva1  isva2
     1         1         1        345      1      49       313         1       1      2
     2         1        49        345      1      73       173         1       1      2
patch interface data:
  ninter
     0
plot3d output:
   grid     iptyp      ista       iend   iinc    jsta      jend      jinc    ksta   kend   kinc
     1         0         1         49      1       1       345         1       1      1      1
 movie
   0
print out:
   grid     iptyp      ista       iend   iinc    jsta      jend      jinc    ksta   kend   kinc
     1         0         1         49      1       1       345         1       1      1      1
```

User Specified Rigid Grid Motion

Rigid grid rotation input

```
control surfaces:
      ncs
        0
     grid      ista      iend       jsta      jend      ksta    kend    iwall    inorm
moving grid data - rigid translation (forced motion):
   ntrans
       0
     lref
    grid      itrans       rfreq      utrans      vtrans     wtrans
    grid    dxmax     dymax     dzmax
moving grid data - rigid rotation (forced motion):
   nrotat
       1
    lref
    1.0
   grid      irotat        rfreq    omegax  omegay  omegaz     xorig     yorig      zorig
      1           2         0.05        0.00        5.00        0.00       0.25     0.00       0.00
   grid dthxmx      dthymx     dthzmx
      1        10.          10.          10.
Patched data:
ninter2
      0
```

The following lines must be included when $iunst = 1$

Rigid translation input. Note that $ntrans = 0$, so that only remaining header lines are included.

Rigid rotation input

User Specified Rigid Grid Motion

Rigid grid rotation input

Focusing attention on the rigid rotation input:

```
moving grid data - rigid rotation (forced motion):
  nrotat
       1
   lref
     1.0
   grid    irotat     rfreq   omegax  omegay  omegaz   xorig   yorig   zorig
      1         2      0.05     0.00    5.00    0.00    0.25    0.00    0.00
   grid dthxmx      dthymx   dthzmx
      1       10.       10.      10.
```

Number of grid blocks to be rotated

Reference length for reduced frequency

Line repeated *nrotat* times

Line repeated *nrotat* times

User Specified Rigid Grid Motion

Rigid grid rotation input

Focusing on the last two lines of input on the last slide:

```
                  .
                  .
grid    irotat       rfreq   omegax  omegay  omegaz   xorig   yorig   zorig
   1         2        0.05     0.00    5.00    0.00    0.25    0.00    0.00
grid   dthxmx      dthymx    dthzmx
   1       10.        10.       10.
                  .
                  .
```

grid - Grid block to be rotated

irotat - Type of rotation

= 0 - no rotation

= 1 - rotation with constant angular speed

= 2 - sinusoidal variation of angular displacement

= 3 - smooth increase in displacement, asymptotically reaching a maximum angle

rfreq - reduced frequency when *irotat = 2*; growth rate to maximum angular displacement when *irotat = 3*

User Specified Rigid Grid Motion

Rigid grid rotation input

```
                        .
                        .
grid    irotat       rfreq   omegax  omegay  omegaz   xorig   yorig   zorig
   1         2        0.05     0.00    5.00    0.00    0.25    0.00    0.00
grid  dthxmx      dthymx    dthzmx
   1       10.         10.       10.
                        .
                        .
```

omegax, omegay, omegaz - x,y,z components of rotational velocity when *irotat = 1*; maximum angular displacements about x,y,z-axes when *irotat > 1*

xorig, yorig, zorig - *x,y,z* coordinate of origin of the rotational axis

dthymx, dthymx,dthzmx - maximum (absolute) rotational displacement about the x,y,z-axes to be allowed for this grid (set *dthxmx,dthymx, dthzmx = 0* if no restriction is required)

User Specified Rigid Grid Motion

Rigid grid rotation input

Example of sinusoidal rotational motion *irotat = 2*. The following terms are defined:

$$rfreq = k_r \quad , \quad lref = L_{ref}$$

$$omegax = \tilde{\theta}_{x,\max}, \deg. \quad , \quad omegay = \tilde{\theta}_{y,\max}, \deg. \quad , \quad omegaz = \tilde{\theta}_{z,\max}, \deg.$$

The rotational displacement (radians) within the code is governed by

$$\theta_x = \tilde{\theta}_{x,\max} \frac{\pi}{180} \sin(2\pi k_r \frac{t}{L_{ref}}) \quad , \quad \theta_y = \tilde{\theta}_{y,\max} \frac{\pi}{180} \sin(2\pi k_r \frac{t}{L_{ref}})$$

$$\theta_z = \tilde{\theta}_{z,\max} \frac{\pi}{180} \sin(2\pi k_r \frac{t}{L_{ref}})$$

Based on these equations of sinusoidal motion,

$$\Delta t = \frac{L_{ref}}{k_r N}$$

where N is the desired number of time steps per cycle. Consult Chapter 4 of the Version 5.0 User's Manual pp. 55-62 for details on all types of motion.

User Specified Rigid Grid Motion

Rigid grid rotation

The following diagnostic information on the rotation of the surface(s) will be printed in 'cfl3d.out':

```
                    .
                    .
                    .
rotating block      1 to new position
  creating coarser block   2 of dimensions (I/J/K) :  37 173  37
    restricting grid speeds from finer block    1 to coarser block   2
  creating coarser block   3 of dimensions (I/J/K) :  19  87  19
    restricting grid speeds from finer block    2 to coarser block   3
                    .
                    .
                    .
writing restart file for block    1
   writing vist3d data to restart file, block     1
   writing field eqn turb quantities to restart file, block     1
 writing 2nd order time data to restart file, block    1
 writing dynamic mesh data to restart file,    block    1
                    .
                    .
                    .
```

Grid speed information computed for moving grid

Note that new dynamic mesh data has been written to the restart file

User Specified Rigid Grid Motion

Rigid grid translation input

```
                    .
                    .
control surfaces:
      ncs
        0
     grid      ista      iend      jsta      jend      ksta    kend    iwall    inorm
moving grid data - rigid translation (forced motion):
  ntrans
       1
     lref
     1.0
    grid     itrans     rfreq     utrans     vtrans    wtrans
       1          2      0.05       0.00       0.00      5.00
    grid    dxmax    dymax    dzmax
       1      10.      10.      10.
moving grid data - rigid rotation (forced motion):
  nrotat
       0
     lref
grid     irotat     rfreq   omegax  omegay  omegaz    xorig    yorig    zorig
grid dthxmx     dthymx    dthzmx
Patched data:
ninter2
      0
```

Rigid translation input

Rigid rotation input. Note that *nrotat = 0*, so that only remaining header lines are included.

User Specified Rigid Grid Motion

Rigid grid translation input

Focusing attention on the rigid translation input:

```
moving grid data - rigid translation (forced motion):
  ntrans
       1
    lref
     1.0
    grid    itrans     rfreq    utrans    vtrans    wtrans
       1         2      0.05      0.00      0.00      5.00
    grid     dxmax     dymax     dzmax
       1       10.       10.       10.
```

Number of grid blocks to be translated

Reference length for reduced frequency

Line repeated *ntrans* times

Line repeated *ntrans* times

User Specified Rigid Grid Motion

Rigid grid translation input

Focusing on the last two lines of input from the last slide:

```
                      .
                      .
 grid    itrans    rfreq    utrans    vtrans   wtrans
    1         2     0.05      0.00      0.00     5.00
 grid    dxmax    dymax    dzmax
    1       10.      10.      10.
                      .
```

grid - Grid block to be rotated

itrans - Type of translation

= 0 - no translation

= 1 - translation with constant speed

= 2 - sinusoidal variation of displacement

= 3 - smooth increase in displacement, asymptotically reaching a maximum displacement

rfreq - reduced frequency when $itrans = 2$; growth rate to maximum displacement when $itrans = 3$

utrans, vtrans, wtrans - x,y,z components of translation velocity when $itrans = 1$; maximum displacements in the x,y,z directions when $itrans > 1$

dymax, dymax,dzmax - maximum (absolute) translation displacement in the x,y,z directions to be allowed for this grid.

Surface Motion - Deforming Mesh

Overview

- CFL3D can perform several types of user specified surface motion by deforming the mesh, i.e. surface rotation and/or translation of all or partial segments of the solid surfaces as well as modal motion of surfaces.

- Aeroelastic, user defined deforming mesh surface and user defined rigid grid motion can be performed in any combination.

- There are two methods of deforming the mesh.
  - Exponential decay combined with Trans-Finite Interpolation (TFI) of interior mesh points.
  - Finite macro-element deformation combined with TFI.
- Note that deforming surface motion can only be performed with the code running in unsteady mode.

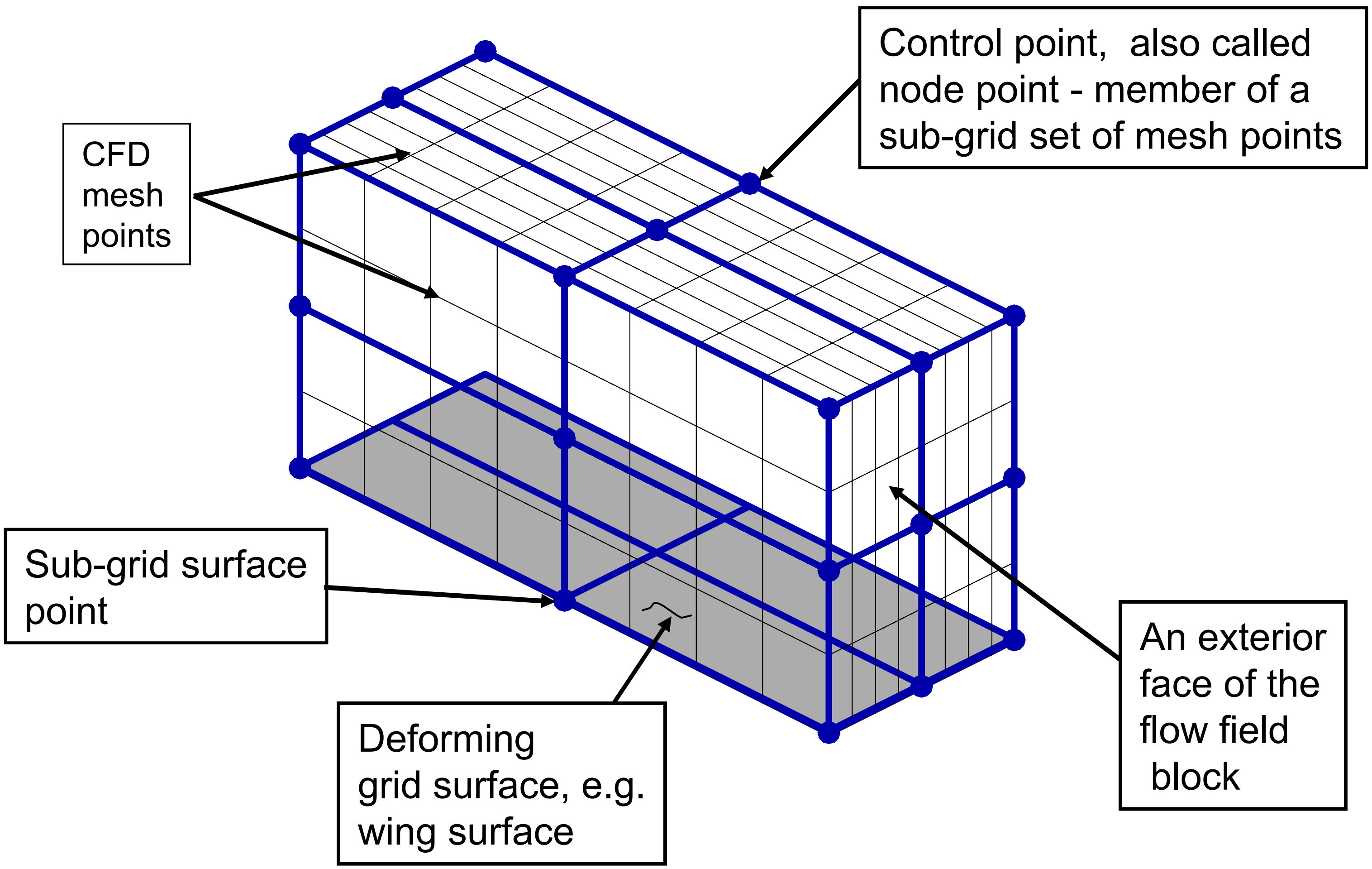
Surface Motion - Deforming Mesh

Overview

- In the first mesh movement option (exponential decay method) deformation is performed in two steps.
  - The first step is exponential decay of control points away from the moving surface. The rate of the exponential decay is controlled by user input.
  - The second step is a TFI of mesh points interior to the control points.
- Advantage of the exponential decay method is that it is computationally efficient
- In the second mesh movement option (finite macro-element method) deformation is also performed in two steps.
  - The first step is a finite element solution of macro-element points. The resulting solution transmits surface motion to the element node points. The element stiffness varies with distance from the surface. User specified input controls the rate at which the element stiffness decays away from surfaces.
  - The second step is a TFI of mesh points interior to the element node (or control) points.
  - See Bartels, R. E., "Finite Macro-Element Mesh Deformation in a Structured Multi-Block Navier-Stokes Code," NASA/TM-2005-213789, July 2005.
- Advantage of the finite macro-element method is that it maintains mesh quality, but is significantly more computationally time consuming.

Surface Motion - Deforming Mesh

Deforming mesh terminology



Surface Motion - Deforming Mesh

Deforming mesh using exponential decay method

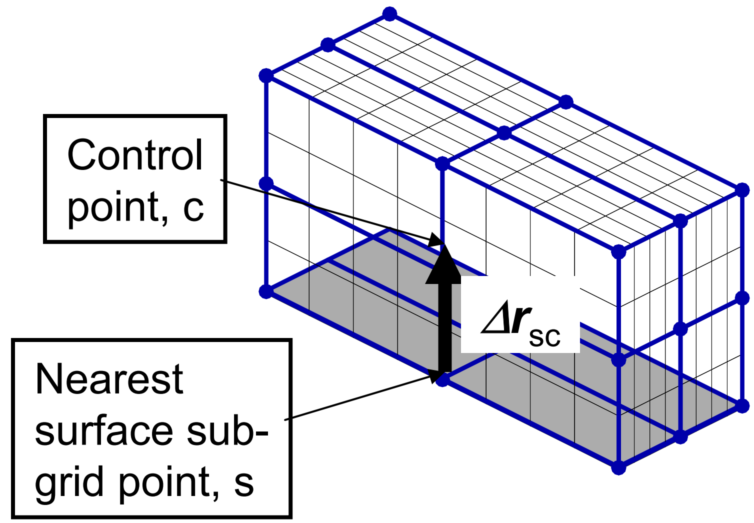
$$\vec{r}_c^{\,n+1} - \vec{r}_c^{\,n} = D_{sc}(\vec{r}_s^{\,n+1} - \vec{r}_s^{\,n})$$

where

$$D_{sc} = \min[1, e^{-A}]$$

and

$$A = \beta_2\,(|\Delta\vec{r}_{sc}| / \Delta r_{\max} - \alpha_2)$$



The movement of surface points is transmitted into the flow field sub-grid through an exponential decay function D_{sc}. The rate of decay is controlled by the parameters β_2 and α_2.

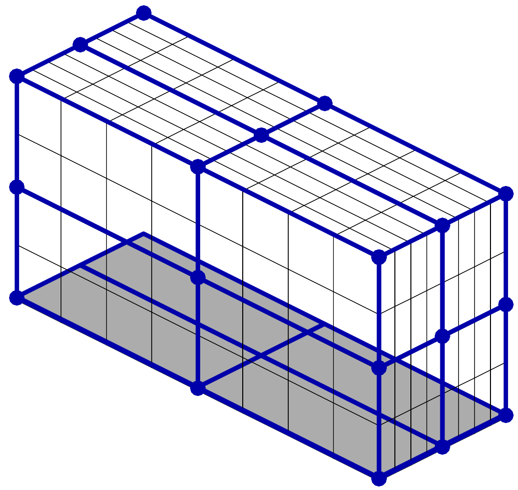
Surface Motion - Deforming Mesh

Deforming mesh with exponential decay method

Note several potential draw backs to this approach:

- Too rapid a rate of decay (β_2 too large, α_2 too small) results in the possibility of the surface points moving through nearby control points.
- Too low a rate of decay (β_2 too small, α_2 too large) results in the possibility of surface deformation being transmitted too far into the flow field with possible penetration of opposing surfaces.
- Typical values for decay parameters are:

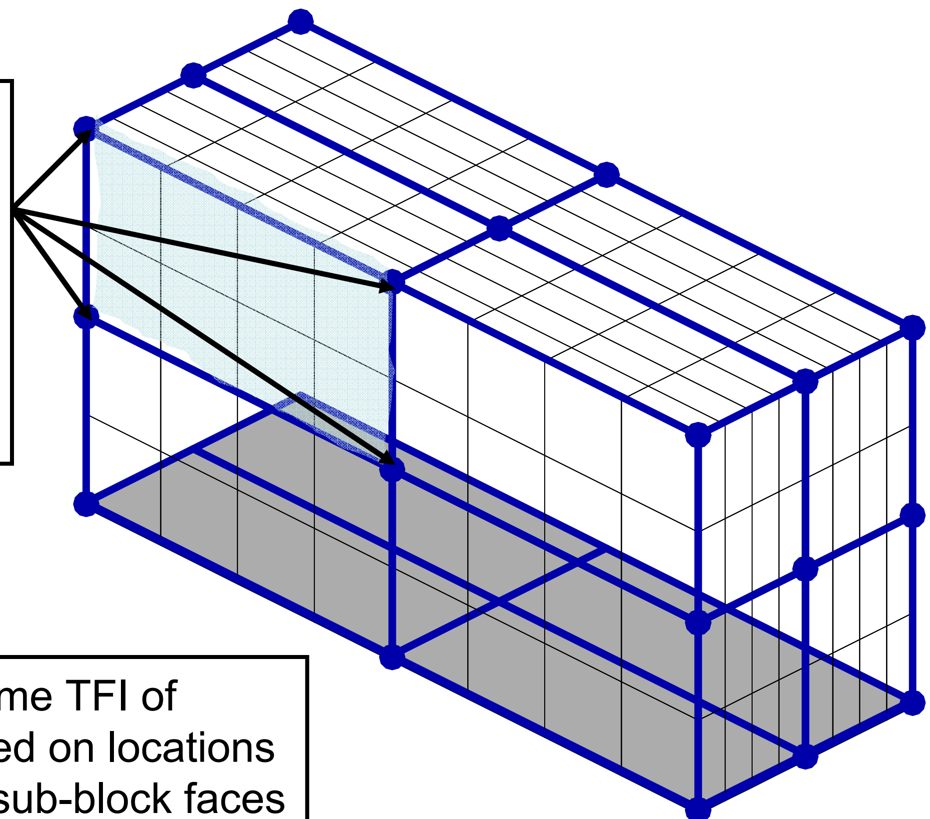
$$\beta_2 = 1 - 10 \quad , \quad \alpha_2 = 0.005 - 0.05$$



Surface Motion - Deforming Mesh

Trans-Finite Interpolation (TFI) of interior points

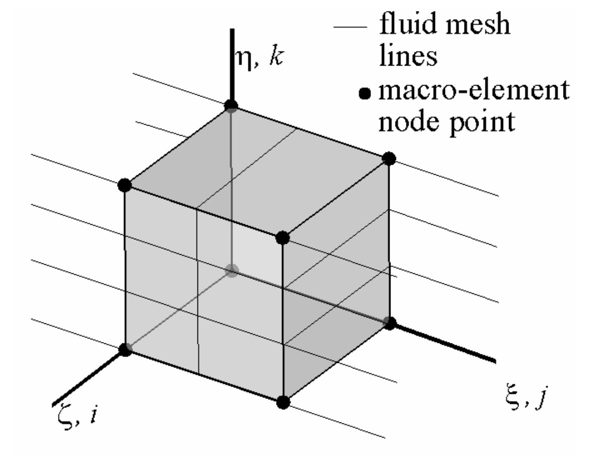
Mesh points interior to the sub-block face are inter-polated using deflection of four corner control points



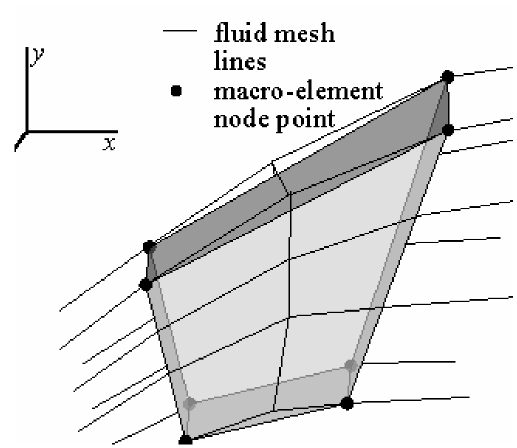
The final step is a volume TFI of interior grid points based on locations of mesh points on the sub-block faces

Surface Motion - Deforming Mesh

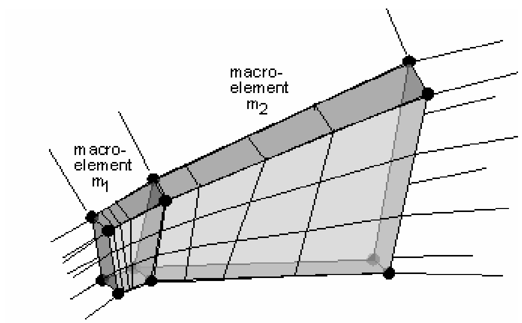
Coordinate systems and terminology for finite macro-element method



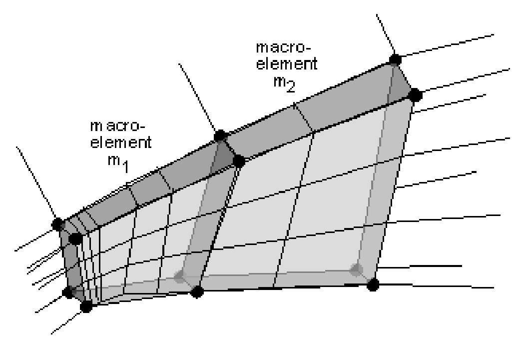
Computational domain



Physical domain



Nodes using constant skip values



Arbitrary node placement

Surface Motion - Deforming Mesh

Finite macro-element method

The equations of elasticity are solved using Hooke's law for element *m*

$$\vec{\sigma}_m = C_m \vec{\varepsilon}_m$$

where

$$\vec{\sigma}_m = \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{yz} \\ \sigma_{xz} \end{bmatrix}_m , \quad \vec{\varepsilon}_m = \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \varepsilon_{xy} \\ \varepsilon_{yz} \\ \varepsilon_{xz} \end{bmatrix}_m , \quad C_m = \begin{bmatrix} E_m & 0 & 0 & 0 & 0 & 0 \\ 0 & E_m & 0 & 0 & 0 & 0 \\ 0 & 0 & E_m & 0 & 0 & 0 \\ 0 & 0 & 0 & G_m & 0 & 0 \\ 0 & 0 & 0 & 0 & G_m & 0 \\ 0 & 0 & 0 & 0 & 0 & G_m \end{bmatrix}$$

$$E_m = E_0 f_m \quad , \quad G_m = G_0 f_m \qquad\qquad f_m = \frac{1}{1 - \exp(-\beta_1 \, \Delta r_m / \Delta r_{\max})}$$

Δr_m is computed as

$$\Delta r_m = \sqrt{(\Delta x_{cs,m})^2 + (\Delta y_{cs,m})^2 + (\Delta z_{cs,m})^2}$$

The user controls the rate of decay of material properties by the parameter β_1. Typical values of β_1 are in the range of *1 – 2*.

Surface Motion - Deforming Mesh

Input for deforming mesh

.

.

Moving grid data – data for field/multiblock mesh movement

nskip	isktyp	beta1	alpha1	beta2	alpha2	nsprgit
4	-1	2.0	1.1	10.0	0.01	0

grid	iskip	jskip	kskip
1	4	4	2

Moving grid data – multi-motion coupling

ncoupl

0

Slave master xorig yorig zorig

nskip - number of blocks for which skip value data is input. If nskip = 0 the code computes default skip values (*isktyp = -1,1*) or control point index values (*isktyp = -2,2*).

isktyp - Parameter defining the mesh deformation approach

$$\left.\begin{array}{l} = -2 \\ = -1 \end{array}\right\} \text{exponential decay method}$$

$$\left.\begin{array}{l} = 1 \\ = 2 \end{array}\right\} \text{finite macro-element method}$$

Surface Motion - Deforming Mesh

Input for deforming mesh

.

.

Moving grid data – data for field/multiblock mesh movement

nskip	isktyp	beta1	alpha1	beta2	alpha2	nsprgit
1	-1	2.0	1.0	10.0	0.01	0

grid	iskip	jskip	kskip
1	4	4	2

Moving grid data – multi-motion coupling

ncoupl

0

Slave master xorig yorig zorig

beta1 - Parameter controlling macro-element stiffness decay (typically 1.0-2.0)

alpha1 - Relaxation parameter for Gauss-Seidel solver (typically 0.8-1.2).

beta2 - Decay parameter for the exponential decay method (typically 1 - 10).

alpha2 - Decay parameter for the exponential decay method (typically 0.005-0.05).

nsprgit - Number of spring analogy smoothing steps performed with the exponential decay method. This step applies *nsprgit* spring analogy steps to the control points after application of the exponential decay step (typically 0-2).

Surface Motion - Deforming Mesh

Input for deforming mesh

preferred method

- There are 4 options for the construction of control points.
  - Option 1: Code generated minimum number of control points.
  - Option 2: Code generated default skip values.
  - Option 3: User input of *i,j,k* skip values for each block.
  - Option 4: User defined input of control point *i,j,k* indices for each block.
- These options depend on the value of *nskip* and the value of *isktyp*
  - Option 1: $isktyp$ = -2, 2 and $nskip = 0$
  - Option 2: $isktyp$ = -1, 1 and $nskip = 0$
  - Option 3: $isktyp$ = -1, 1 and $nskip = ngrid$ (Note: $ngrid$ = number of grid blocks)
  - Option 4: $isktyp$ = -2, 2 and $nskip = ngrid$
- Option 1 creates the minimum number of control points (at non-constant intervals) by placing control point points only at each boundary segment extremity. *This is the preferred method.*
- Options 2 creates skip values that result in control points at constant intervals through out each of the grids, with control points at each boundary segment extremity. Sometimes this is more robust than option 1, but can create many more control points.

Surface Motion - Deforming Mesh

Option 1 – Code generated minimum number of control points

It is possible to have the code calculate the minimum number of control points. This is the preferred method. The following lines of input accomplish that:

.
.

```
Moving grid data – data for field/multiblock mesh movement
nskip     isktyp     beta1     alpha1    beta2    alpha2    nsprgit
   0         -2        2.0        1.1     10.0      0.01        0
 grid      iskip      jskip     kskip
Moving grid data – multi-motion coupling
ncoupl
    0
Slave    master    xorig      yorig      zorig
```

nskip = 0 and *isktyp = -2*

Note that the data input line following the header 'grid ….' is omitted. The code calculates the minimum number of control points possible consistent with placing control points at each boundary segment extremity. The values it calculates will be found in the 'cfl3d.out' section that reflects input. Note that the value of *isktyp* must be either 2 or -2. In general control points will not be at constant intervals.

Surface Motion - Deforming Mesh

Option 2 – Code generated skip values

It is possible to have the code calculate default skip values. The following lines of input accomplish that:

```
                              .
                              .
Moving grid data – data for field/multiblock mesh movement
nskip      isktyp       beta1      alpha1     beta2     alpha2     nsprgit
    0          -1         2.0         1.1      10.0       0.01           0
 grid       iskip        jskip      kskip
Moving grid data – multi-motion coupling
ncoupl
    0
Slave    master     xorig        yorig       zorig
```

nskip = 0 and *isktyp = -1*

Note that the data input line following the header 'grid ….' is omitted. The code calculates the largest values of *iskip, jskip, kskip* possible. The values it calculates will be found in the 'cfl3d.out' section that reflects input. Note that the value of *isktyp* must be either 1 or -1.

Surface Motion - Deforming Mesh

Option 3 – User *i,j,k* skip input

.
.
Moving grid data – data for field/multiblock mesh movement

nskip	isktyp	beta1	alpha1	beta2	alpha2	nsprgit
4	-1	2.0	1.1	10.0	0.01	0

grid	iskip	jskip	kskip
1	4	4	2
2	4	8	2
3	4	8	2
4	4	4	2

Moving grid data – multi-motion coupling
ncoupl
0
Slave master xorig yorig zorig

Note: It is required that
nskip = ngrid

nskip lines are required

grid - The block number for which skip values are input
iskip - Skip value for control points in the i-direction
jskip - Skip value for control points in the j-direction
kskip - Skip value for control points in the k-direction

Surface Motion - Deforming Mesh

Permissible skip values

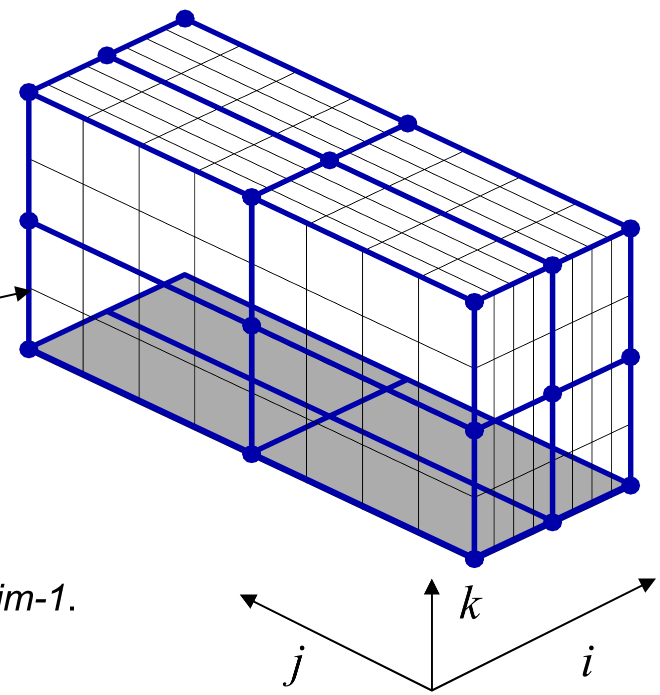
iskip, jskip, kskip values determine the i, j, k skip intervals for creating the sub-grid

For this grid:

$idim = 9, \ jdim = 9, kdim = 5$

and

$iskip = 4, jskip = 4, \ kskip = 2$



Skip values must evenly divide into one minus the dimension of the grid. *jskip* must divide evenly into *jdim-1*. *iskip* must divide evenly into *idim-1* , etc…

With *idim* = 9, permissible values of *iskip* are *2, 4* and *8.*
With *jdim* = 9, permissible values of *jskip* are *2, 4* and *8.*
With *kdim* = 5, permissible values of *kskip* are *2* and *4*.

Surface Motion - Deforming Mesh

Option 4 – User input of i,j,k control point indices

```
                                        .
                                        .
 Moving grid data – data for field/multiblock mesh movement
 nskip        isktyp      beta1    alpha1      beta2       alpha2       nsprgit
    2              -2        2.0          1.1       10.0          0.01           0
Control point input section
   GRID      NIND      NJND      NKND
        1          3           5           3
************************** I NODE INDICES ***************************************************
        1        73         81
************************** J NODE INDICES ***************************************************
        1        33        173        313       345
************************** K NODE INDICES ***************************************************
        1        25         73
   GRID      NIND      NJND      NKND
        2          3           5           3
************************** I NODE INDICES ***************************************************
        1        73         81
************************** J NODE INDICES ***************************************************
        1        33        173        313       345
************************** K NODE INDICES ***************************************************
        1        25         73
 Moving grid data – multi-motion coupling
 ncoupl
        0
 Slave     master      xorig          yorig         zorig
```

$nskip = ngrid$
$isktyp$ must equal -2 or 2

$nskip$ input sets are required

Surface Motion - Deforming Mesh

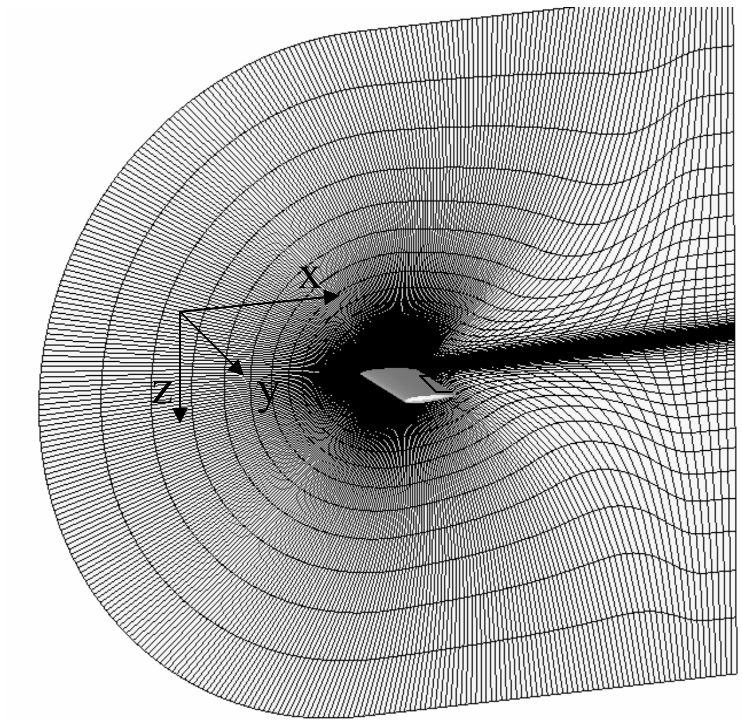
Option 4 – User input of i,j,k control point indices

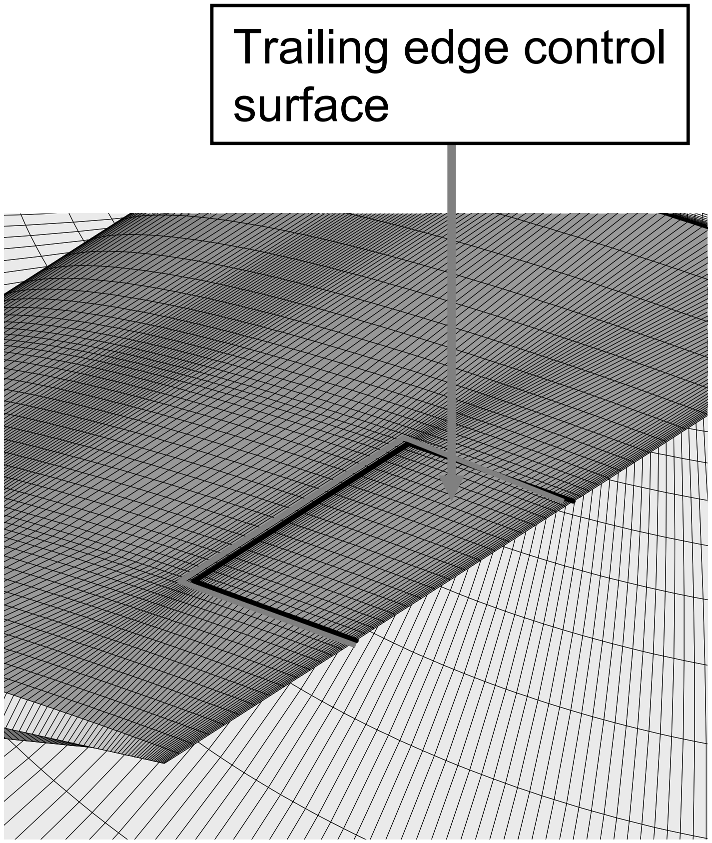
- This option is used when there are problem areas in the surface motion that require customized control point placement. e.g. significant surface motion restricted to a small portion of the entire surface area or if the finite macro-element method is used and added control points are needed to define affine element shapes.
- Note that a control point must be placed at the extremities of all boundary condition segments, 1-1 blocking segments and all block corners.
- The code will do a check at 1-1 blocking segments to see if the control points you have selected result in continuity in control placement between 1-1 blocking boundaries. It will add points as necessary to maintain control point continuity. *This is a very powerful feature that can be very useful when adding control points.*
- The code will not tell you if a b.c. segment extremity or block corner does not have a control point assigned to it. *It will simply cause the grid motion to be messed up and produce negative volumes!*

Surface Motion - Deforming Mesh

Example 1: 3D Control surface rotation with exponential decay method

As an example consider the wing shown undergoing control surface rotation:





Surface Motion - Deforming Mesh

Example 1: 3D Control surface rotation with exponential decay method

The following unsteady input file performs the control surface rotation about the hinge point:

```
input/output files:
 wbgrid.cfl
 plot3dg.bin
 plot3dq.bin
 cfl3d.out
 cfl3d.res
 cfl3d.turres
 cfl3d.blomax
 cfl3d.out15
 cfl3d.prout
 cfl3d.out20
 ovrlp.bin
 patch.bin
 restart.bin
 NASA Langley BACT Model: NACA 0012 af, AR=1.5 wing,.75TE Flap
      Mach     alpha      beta      ReUe   Tinf,dR     ialph    ihstry
   0.82000   0.00000   0.00000 0.236E+07    486.00         1         0
      sref      cref      bref       xmc       ymc       zmc
     1.000   1.00000   1.00000   0.25000   0.00000   0.00000
        dt     irest   iflagts      fmax     iunst   cfl_tau
   0.04000         0      3000   1.00000         2   2.00000
     ngrid   nplot3d    nprint    nwrest      ichk       i2d    ntstep       ita
         1         1         1      1000         0         0         1        -2
```

Note that *iunst = 2* for deforming mesh

Surface Motion - Deforming Mesh

Example 1: 3D Control surface rotation with exponential decay method

	ncg	iem	iadvance	iforce	ivisc(i)	ivisc(j)	ivisc(k)	
	2	0	0	1	5	5	5	
	idim	jdim	kdim					
	81	345	73					
	ilamlo	ilamhi	jlamlo	jlamhi	klamlo	klamhi		
	0	0	0	0	0	0		
	inewg	igridc	is	js	ks	ie	je	ke
	0	0	0	0	0	0	0	0
	idiag(i)	idiag(j)	idiag(k)	iflim(i)	iflim(j)	iflim(k)		
	1	1	1	4	4	4		
	ifds(i)	fds(j)	ifds(k)	rkap0(i)	rkap0(j)	rkap0(k)		
	1	1	1	0.3333	0.3333	0.3333		
	grid	nbci0	nbcidim	nbcj0	nbcjdim	nbck0	nbckdim	iovrlp
	1	1	1	1	1	5	1	0
i0:	grid	segment	bctype	jsta	jend	ksta	kend	ndata
	1	1	1005	1	345	1	73	0
idim:	grid	segment	bctype	jsta	jend	ksta	kend	ndata
	1	1	1002	1	345	1	73	0
j0:	grid	segment	bctype	ista	iend	ksta	kend	ndata
	1	1	1003	1	81	1	73	0
jdim:	grid	segment	bctype	ista	end	ksta	kend	ndata
	1	1	1003	1	81	1	73	0
k0:	grid	segment	bctype	ista	iend	jsta	jend	ndata
	1	1	0	1	73	1	33	0
	1	2	2004	1	73	33	313	2
	tw/tinf	cq						
	0.00000	0.00000						
	1	3	0	1	73	313	345	0
	1	4	0	73	81	1	173	0
	1	5	0	73	81	173	345	0
kdim:	grid	segment	bctype	ista	iend	jsta	jend	ndata
	1	1	1003	1	81	1	345	0

Surface Motion - Deforming Mesh

Example 1: 3D Control surface rotation with exponential decay method

```
   mseq    mgflag    iconsf      mtt   ngam
      1         2         1        0      2
   issc epsssc(1) epsssc(2) epsssc(3)   issr epsssr(1) epsssr(2) epsssr(3)
      0    0.3000    0.3000    0.3000      0    0.3000    0.3000    0.3000
   ncyc    mglevg     nemgl     nitfo
      8         3         0         0
   mit1      mit2      mit3      mit4   mit5 ...
      1         1         1
1-1 blocking data:
   nbli
      2
 number      grid      ista      jsta   ksta     iend     jend     kend   isva1   isva2
      1         1         1         1      1       73       33        1       1       2
      2         1        73         1      1       81      173        1       1       2
 number      grid      ista      jsta   ksta     iend     jend     kend   isva1   isva2
      1         1         1       345      1       73      313        1       1       2
      2         1        73       345      1       81      173        1       1       2
patch interface data:
 ninter
      0
plot3d output:
   grid     iptyp      ista      iend   iinc     jsta     jend     jinc    ksta    kend    kinc
      1         0         1        73      1       33      313        1       1       1       1
 movie
     0
print out:
   grid     iptyp      ista      iend   iinc     jsta     jend     jinc    ksta    kend    kinc
      1         0         1        73      1       33      313        1       1       1       1
```

Surface Motion - Deforming Mesh

Example 1: 3D Control surface rotation with exponential decay method

```
Control Surfaces:
ncs
0
   Grid      ista    iend      jsta     jend     ksta     kend    iwall   inorm
Moving grid data – deforming surface (forced motion):
ndefrm
       2
     lref
     1.0
   Grid    idefrm   rfreq  u/omegax v/omegay w/omegaz    xorig   yorig   zorig
      1        2    0.05      0.00     5.00     0.00     0.75    0.00    0.00
      1        2    0.05      0.00     5.00     0.00     0.75    0.00    0.00
   Grid     icsi     icsf     jcsi     jcsf     kcsi     kcsf
      1       29       53       33       72        1        1
      1       29       53      274      313        1        1
Moving grid data – aeroelastic surface (aeroelastic motion):
 naesrf
       0
 Iaesrf    ngrid    grefl     uinf     qinf   nmodes  iskyhook
   Freq    gmass     damp  x0(2n-1)   xo(2n)  gf0(2n)
 Moddfl      amp     freq       t0
   Grid     iaei     iaef     jaei     jaef     kaei     kaef
Moving grid data – data for field/multiblock mesh movement
  nskip   isktyp    beta1   alpha1    beta2   alpha2  nsprgit
      0       -2      1.0      1.1      1.0    0.005        0
   Control point index input
Moving grid data – multi-motion coupling
ncoupl
      0
Slave   master    xorig    yorig    zorig
```

The following lines must be included when *iunst = 2*

User specified surface motion input

Aeroelasticity input. Note that only header cards are input when *naesrf = 0*

Mesh deformation input

Surface Motion - Deforming Mesh

Example 1: 3D Control surface rotation with exponential decay method

Note that *ndefrm = 2* because the trailing edge control surface is defined by an upper wing surface segment and a lower wing surface segment

```
                              .
                              .
Moving grid data – deforming surface (forced motion):
ndefrm
    2
 lref
  1.0
 Grid   idefrm   rfreq   u/omegax  v/omegay  w/omegaz    xorig   yorig   zorig
    1        2    0.05       0.00      5.00      0.00     0.75    0.00    0.00
    1        2    0.05       0.00      5.00      0.00     0.75    0.00    0.00
 Grid     icsi    icsf       jcsi      jcsf      kcsi     kcsf
    1       29      53         33        72         1        1
    1       29      53        274       313         1        1
                              .
                              .
```

ndefrm lines required

ndefrm lines required

Grid - grid block containing the moving surface

idefrm - type of surface motion

= 1 - translation

= 2 - rotation

rfreq - reduced frequency of the surface motion

u/omegax, v/omegay, w/omegaz - x,y,z-components of surface translational velocity if *idefrm = 1*
- x,y,z-components of surface rotational velocity if *idefrm = 2*

xorig, yorig, zorig - x,y,z coordinates of the origin of the rotation axis (note: value must be input even when *idefrm = 1)*

Surface Motion - Deforming Mesh

Example 1: 3D Control surface rotation with exponential decay method

.
.

Moving grid data – deforming surface (forced motion):
ndefrm
2
lref
1.0

Grid	idefrm	rfreq	u/omegax	v/omegay	w/omegaz	xorig	yorig	zorig
1	2	0.05	0.00	5.00	0.00	0.75	0.00	0.00
1	2	0.05	0.00	5.00	0.00	0.75	0.00	0.00
Grid	icsi	icsf	jcsi	jcsf	kcsi	kcsf		
1	29	53	33	72	1	1		
1	29	53	274	313	1	1		

.
.

1[st] grid point aft of X_{orig} = 0.75

Starting and ending i-indices of moving surfaces

Starting and ending j-indices of moving surfaces

Starting and ending k-indices of moving surfaces

Note that the two surface definitions actually comprise a single control device (upper and lower surfaces of the trailing edge control device).

Surface Motion - Deforming Mesh

Example 1: 3D Control surface rotation with exponential decay method

Short cut: If all the solid surfaces are to be rotated or translated in an identical manner, an input shortcut could have been applied:

.
.
Moving grid data – deforming surface (forced motion):
Ndefrm
-1
lref
1.0

Grid	idefrm	rfreq	u/omegax	v/omegay	w/omegaz	xorig	yorig	zorig
1	2	0.05	0.00	5.00	0.00	0.75	0.00	0.00
Grid	icsi	icsf	jcsi	jcsf	kcsi	kcsf		
1	0	0	0	0	0	0		

1 line only

1 line only

.
.

Setting *ndefrm = -1* applies the input values to all surfaces. Input values of *grid*, and *icsi, icsf, jcsi, jcsf, kcsi, kcsf* are placeholders.

Surface Motion - Deforming Mesh

Example 1: 3D Control surface rotation with exponential decay method

Control point index input using Exponential Decay Method (*isktyp* < *0*)

Note that $\alpha_2 * \Delta r_{max}$ is the distance to which surface motion is transmitted unabated into the flow field

Moving grid data – data for field/multiblock mesh movement

nskip	isktyp	beta1	alpha1	beta2	alpha2	nsprgit
0	-2	1.0	1.1	1.0	0.005	0

Control point index input

β_2 - rate at which surface motion decays away from a moving surface (outside of inner region controlled by α_2)

nskip = *0* forces automatic generation of the minimum number of control points (with *isktyp* = *-2,2*) at all segment and block extremities, while maintaining continuity at all blocking boundaries

Surface Motion - Deforming Mesh

Example 1: 3D Control surface rotation with exponential decay method

```
Moving grid data – data for field/multiblock mesh movement
   nskip      isktyp      beta1     alpha1     beta2     alpha2    nsprgit
      0           -2        2.0        1.1        1.0      0.005          0
    Control point index input
Moving grid data – multi-motion coupling
ncoupl
       0
Slave     master      xorig          yorig        zorig
```

Control point option 1 is used here

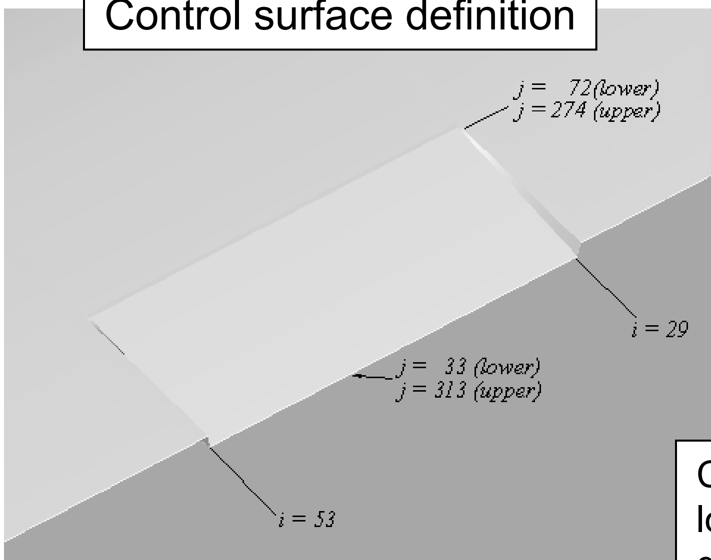
- This input option automatically creates the following control points: (This format is how it would look if you were to input these control points by hand (i.e. using Option 4))

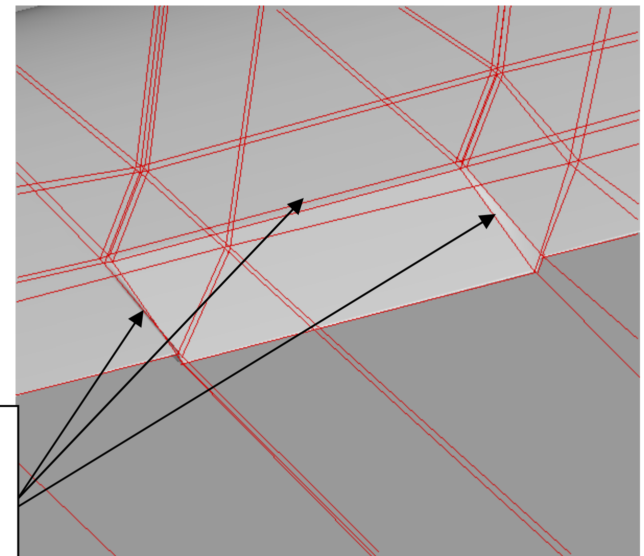
```
     GRID     NIND     NJND       NKND
        1          7          8           2
************************* I NODE INDICES **************************************************
        1        28         29          53         54          73          81
************************* J NODE INDICES **************************************************
        1        33         72          73        273         274         313         345
************************* K NODE INDICES **************************************************
        1        73
```

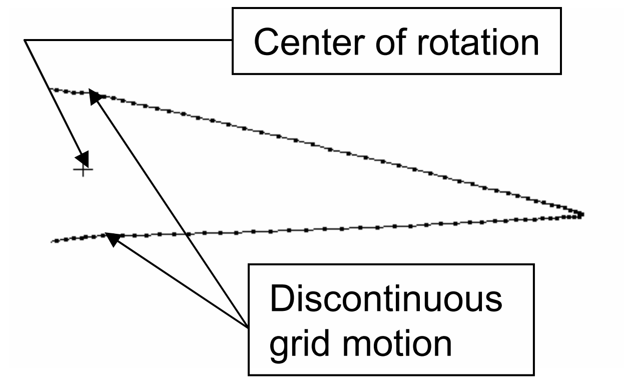
- Note that i node indices, j node indices, k node indices span the entire block. (i.e. $idim = 81, jdim = 345, kdim = 73$)
- Boundary segments have a control point. The trailing edge at $j = 33$ and 313 has control points assigned. The wing tip at $i = 73$ has a control point assigned.
- Other control points have been assigned at discontinuities in the surface movement. (e.g. at $i = 28, 29$ and $53, 54$ and $j = 72, 73$ and $273, 274$) See the next slide.

Surface Motion - Deforming Mesh

Example 1: 3D Control surface rotation with exponential decay method







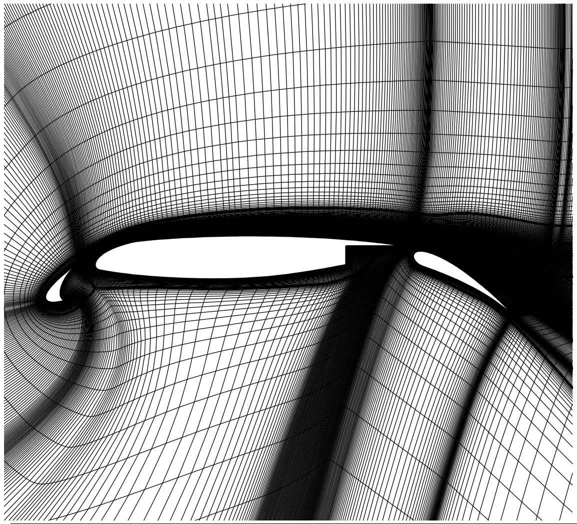
Control points selected

```
  GRID    NIND    NJND    NKND
     1       7       8       3
************************** I NODE INDICES **************************
     1      28      29      53      54      73      81
************************** J NODE INDICES **************************
     1      33      72      73     273     274     313     345
************************** K NODE INDICES **************************
     1      25      73
```

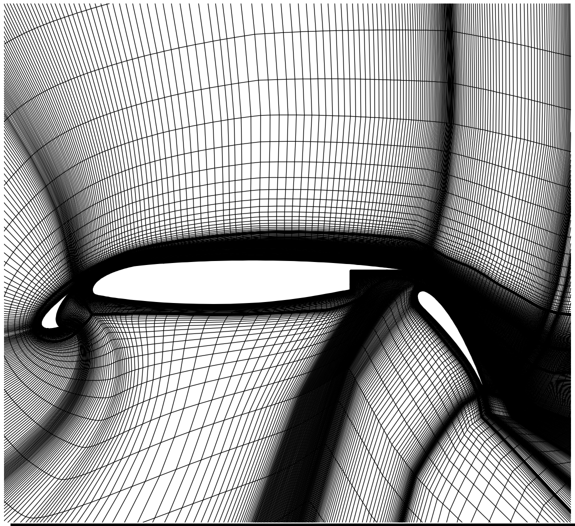
Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation with finite macro-element method

Consider the 2D three element airfoil with rotation and translation of the trailing edge flap.



Initial mesh, flap 30 degrees



Final mesh, flap 60 degrees

From Bartels, R. E., “Finite Macro-Element Mesh Deformation in a Structured Multi-Block Navier-Stokes Code,” NASA/TM-2005-213789, July 2005.

Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation with finite macro-element method

```
                               .
                               .
MOVING GRID DATA - DEFORMING SURFACE (FORCED MOTION):
NDEFRM
      1
  LREF
    1.0
  GRID  IDEFRM   RFREQ  U/OMEGAX  V/OMEGAY  W/OMEGAZ    XORIG    YORIG    ZORIG
     3       2    0.05      0.00     25.00      0.00     0.80     0.00     0.00
  GRID    ICSI    ICSF      JCSI      JCSF      KCSI     KCSF
     3       1       2        49       217         1        1
 MOVING GRID DATA - AEROELASTIC SURFACE (AEROELASTIC MOTION):
 NAESRF
      0
 IAESRF    NGRID    GREFL       UINF       QINF    NMODES  ISKYHOK
   FREQ    GMASS     DAMP   X0(2N-1)     X0(2N)    GF0(2N)
MODDFL       AMP     FREQ         T0
  GRID      IAEI     IAEF       JAEI       JAEF      KAEI     KAEF
MOVING GRID DATA - DATA FOR FIELD/MULTIBLOCK MESH MOVEMENT
  NSKIP   ISKTYP    BETA1     ALPHA1      BETA2     ALPHA2  ISPRNIT
      4        2    1.000      1.000     20.000      0.005        0
 CONTROL  POINT INDEX INPUT
  GRID      NIND     NJND      NKND
     1         2       33         2
********************************************** I NODE INDICES **********************************************
     1         2
```

This section defines the rotation of the trailing edge flap

Number of mesh blocks

Finite Macro-Element Method with user input of control point indices

Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation with finite macro-element method

```
********************************************* J NODE INDICES **********************************************
     1      10      34      49      75     101     113     137     161     201
   237     273     299     317     333     349     380     395     410     433
   445     473     509     545     585     609     633     645     671     697
   712     736     745
********************************************* K NODE INDICES **********************************************
     1      57
  GRID    NIND    NJND    NKND
     2       2      27       2
********************************************* I NODE INDICES **********************************************
     1       2
********************************************* J NODE INDICES **********************************************
     1      10      34      49      75     101     113     137     145     157
   185     225     261     281     299     325     361     397     437     461
   485     497     523     549     564     588     597
********************************************* K NODE INDICES **********************************************
     1      89
  GRID    NIND    NJND    NKND
     3       2      16       2
********************************************* I NODE INDICES **********************************************
     1       2
********************************************* J NODE INDICES **********************************************
     1      10      34      49      75     101     116     121     129     153
   165     191     217     232     256     265
********************************************* K NODE INDICES **********************************************
     1      65
  GRID    NIND    NJND    NKND
     4       2      32       5
```

Up to 10 per line, 500 total allowed

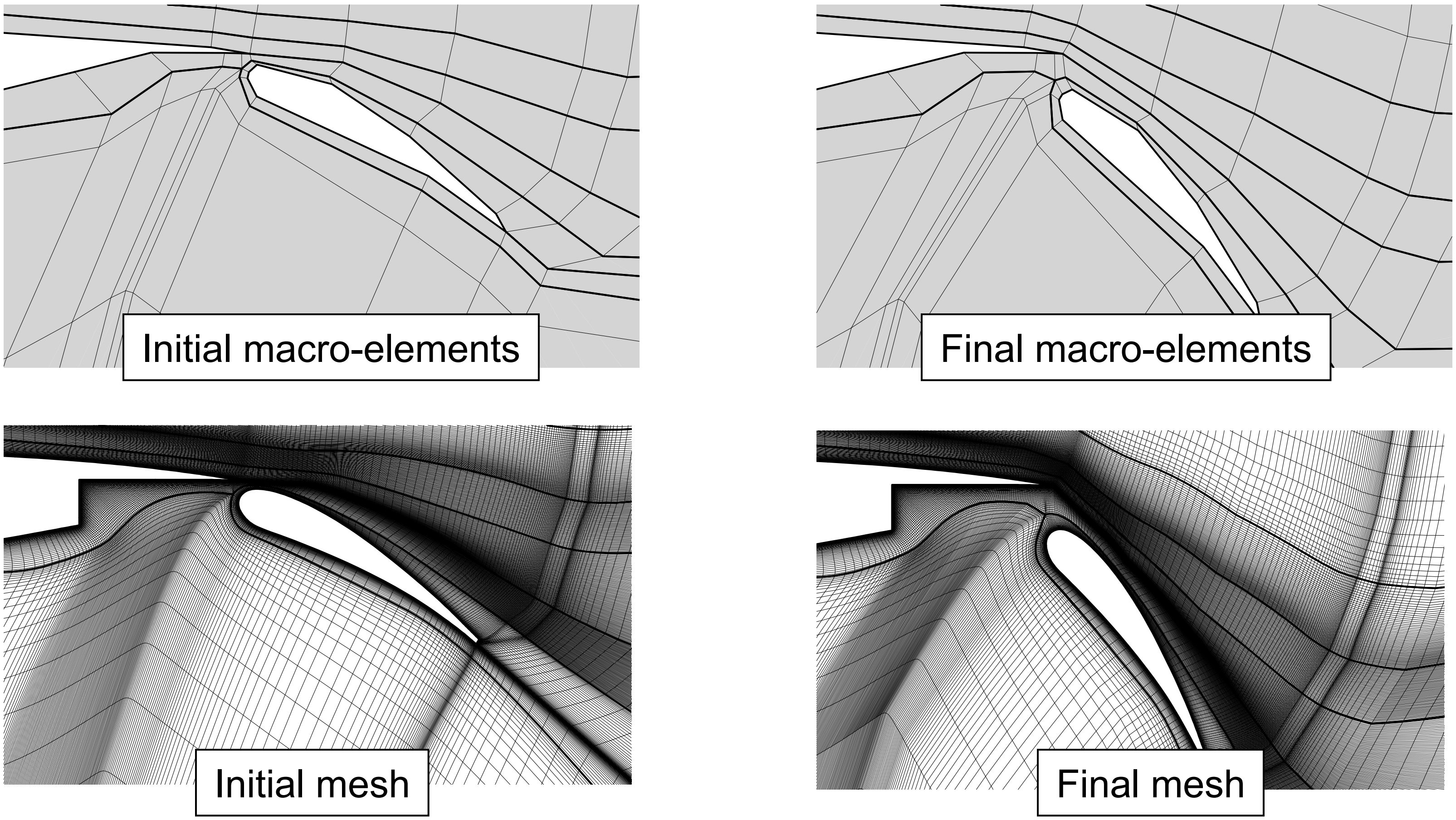
Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation with finite macro-element method

```
******************************************** I NODE INDICES ********************************************
        1          2
******************************************** J NODE INDICES ********************************************
        1         10         34         49         75        101        116        121        133        161
      201        237        257        273        289        320        335        350        373        385
      413        449        485        525        549        573        585        611        637        652
      676        685
******************************************** K NODE INDICES ********************************************
        1         10         17         24         33
MOVING GRID DATA - MULTI-MOTION COUPLING
NCOUPL
        0
SLAVE   MASTER   XORIG  YORIG  ZORIG
```

Surface Motion - Deforming Mesh

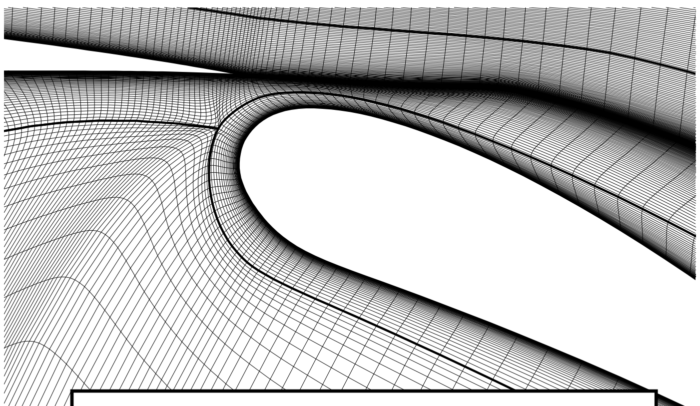
Example 2 : 2D Flap rotation with finite macro-element method



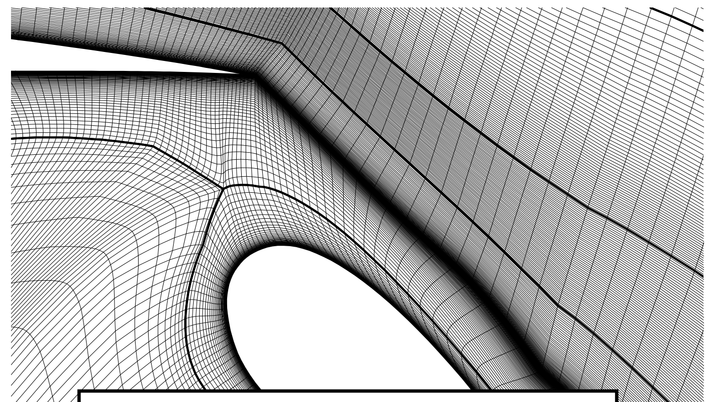
From Bartels, R. E., “Finite Macro-Element Mesh Deformation in a Structured Multi-Block Navier-Stokes Code,” NASA/TM-2005-213789, July 2005.

Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation with finite macro-element method



Flap cove detail, undeflected

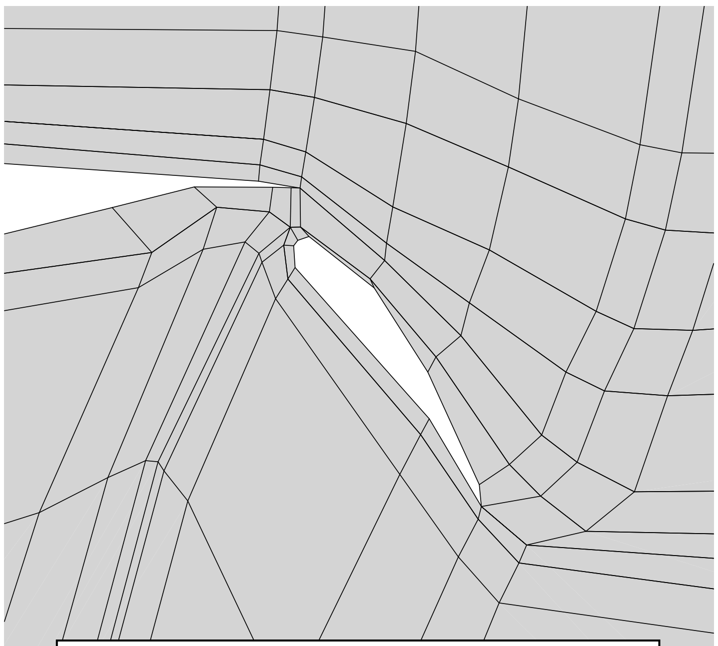


Flap cove detail, deflected

From Bartels, R. E., “Finite Macro-Element Mesh Deformation in a Structured Multi-Block Navier-Stokes Code,” NASA/TM-2005-213789, July 2005.

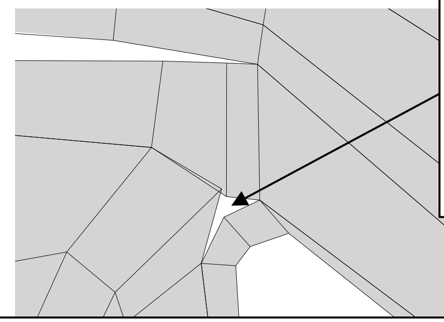
Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation – 1-1 block point checking

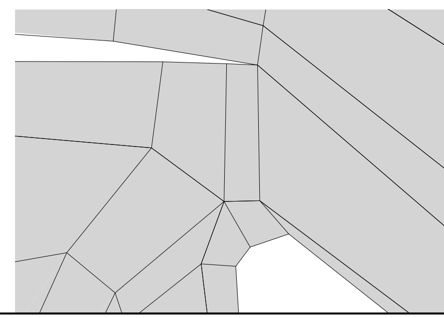


Control point orientation after flap is deflected

Block boundaries separate due to high strain rates in cove region.



Control points without 1-1 point blocking check

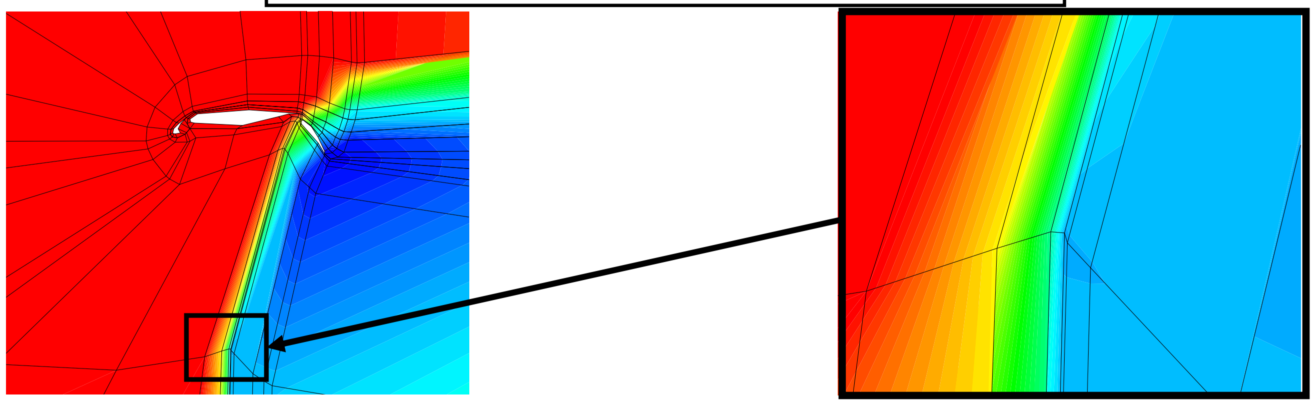


Control points with 1-1 point blocking check

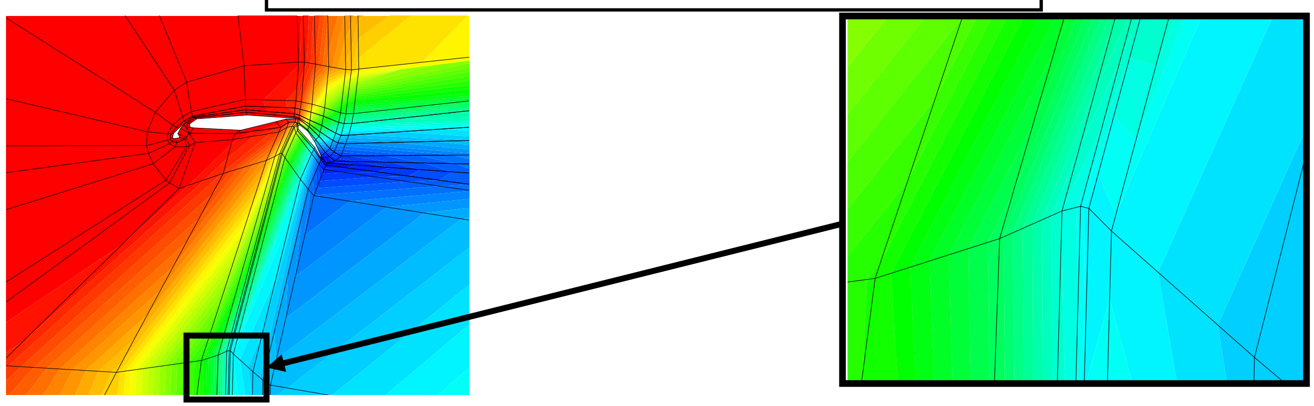
Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation using exponential decay method

Without spring analogy smoothing steps



With 5 spring analogy smoothing steps



Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation using exponential decay method

An alternate approach is to allow automatic creation of the minimum number of control points. (Option 1) The input below accomplishes that by setting $nskip = 0$. Note that the exponential decay method is used ($isktyp < 0$).

```
                         .
                         .
MOVING GRID DATA - DATA FOR FIELD/MULTIBLOCK MESH MOVEMENT
  NSKIP    ISKTYP    BETA1    ALPHA1     BETA2       ALPHA2   ISPRNIT
      0        -2    1.000     1.100     2.000         0.05         2
 CONTROL  POINT INDEX INPUT
MOVING GRID DATA - MULTI-MOTION COUPLING
NCOUPL
      0
SLAVE   MASTER   XORIG  YORIG  ZORIG
```

These parameters define the control point motion with the Exponential Decay Method

Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation using exponential decay method

The control points that are code selected appear in the 'cfl3d.out' file:

```
                              .
                              .
moving grid data - data for field/multiblock mesh movement
 nskip    isktyp    beta1    alpha1    beta2    alpha2  nsprngit
   4        -2  1.000000  1.100000  2.000000  0.050000        2
   ng      nipt      njpt      nkpt
    1         2        11         2
 control point i-indices for grid levels   1  2  3
    1         1         1
    2         1         1
 control point j-indices for grid levels   1  2  3
    1         1         1
   49        25        13
   50        25        13
  137        69        35
  273       137        69
  317       159        80
  473       237       119
  609       305       153
  696       348       174
  697       349       175
  745       373       187
 control point k-indices for grid levels   1  2  3
    1         1         1
   57        29        15
```

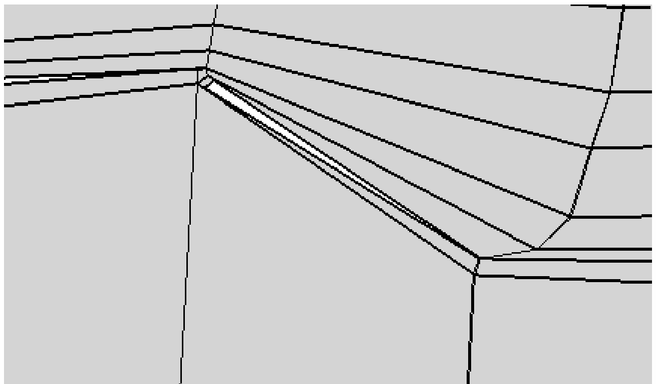
Control points at finest grid level

```
   ng      nipt      njpt      nkpt
    2         2        11         2
 control point i-indices for grid levels   4  5  6
    1         1         1
    2         1         1
 control point j-indices for grid levels   4  5  6
    1         1         1
   49        25        13
   50        25        13
  137        69        35
  145        73        37
  281       141        71
  325       163        82
  461       231       116
  548       274       137
  549       275       138
  597       299       150
 control point k-indices for grid levels   4  5  6
    1         1         1
   89        45        23
                        .
                        .
                        .
```

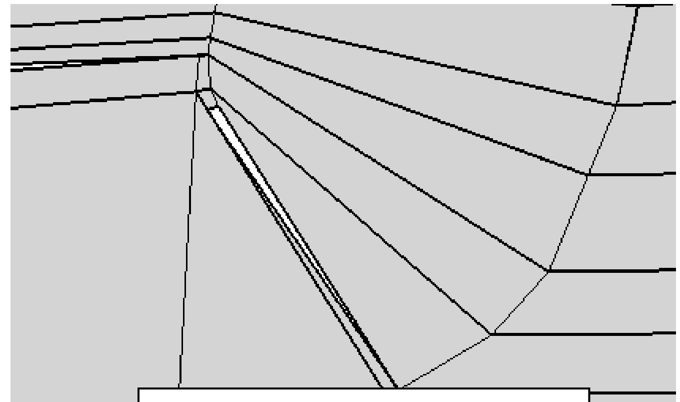
The resulting mesh movement is shown in the next slide.

Surface Motion - Deforming Mesh

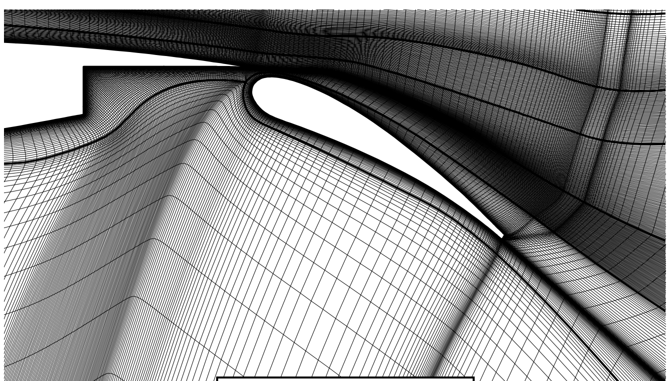
Example 2 : 2D Flap rotation using exponential decay method



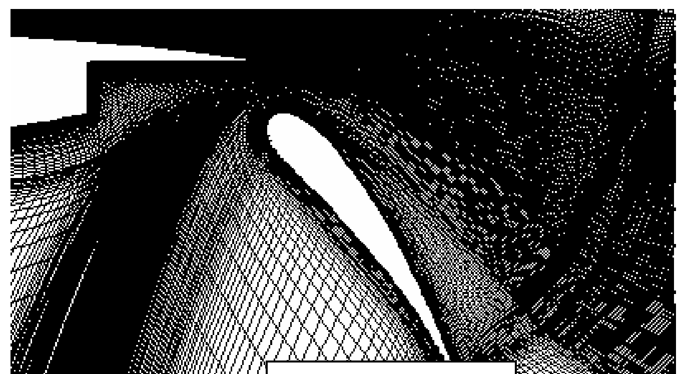
Initial macro-elements



Final macro-elements



Initial mesh



Final mesh

Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation

- The mesh movement shown in the previous slides is robust (no negative volumes) through the entire range of motion shown, however mesh quality aft of the flap is somewhat degraded after deflection.
- If β_2 is set to 1.0 or if the finite macro-element method is used with the code selected minimum number of control points (as was shown), negative volumes are the result.
- There is a simple way to fix this problem, demonstrated next. In the process an option for running the code will be demonstrated in which only the mesh motion and mesh calculations (e.g. metric and volume calculations) are performed in the code. This option greatly speeds up the code when the mesh motion is being debugged.
- The 'Mesh only' run option is invoked by using the keyword input, *meshdef 1* . Keyword input will be discussed in detail later in the course. Note spelling and capitalization are important.
- The input to accomplish this is as follows:

```
                                   .
                                   .
  cfl3d.out20
  ovrlp.bin
  patch.bin
  restart.bin
 >
 meshdef 1
 negvol 1
 <
 3 Element Airfoil case
     Mach     alpha     beta      ReUe     Tinf,dR     ialph   ihstry
                                   .
                                   .
```

Keyword input

Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation

- Setting the keyword *meshdef* to *1* also causes the control points to be output in a Tecplot file in point wise data format. Other auxiliary data are also printed out in other files.
- If one processor is used all block control points are output into the file Tecplot data file '*fort.4000*'. Data included in this file are *x,y,z* locations of control points, *x,y,z* deflections per time step, node number, and node number of the nearest surface point.
- If multiple processors are used, the control points from the blocks processed on each processor are put in the successive files '*fort.4001, fort.4002,* ...'
- Note that if the option *movie = inc* is used, the control points at every *inc* time steps will be output. If *movie = 0*, only control points at the final time step will be output.
- Once the control points are plotted it is possible to better visualize where added control points need to be placed.
- This is the option that was used to create the plots of control points shown in this presentation.

Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation

- Returning to the flap rotation example above, say we want to run it using control point option 1 (*nskip = 0, isktyp = -2,2*) but now using the finite macro-element method (*isktyp = 2*)
- The input parameters used are: $\beta_1 = 1.0$, $\alpha_1 = 0.9$.
- Keywords '*meshdef 1*' and '*negvol 1*' are set. When the keyword '*negvol 1*' is used, the code continues executing and prints a diagnostic message in 'cfl3d.out' indicating where the negative volume occurred.
- The code encounters negative volumes, with the following messages appearing in the 'cfl3d.out' file:

```
                         .
                         .
WARNING ... negative volume at i,j,k=   1  514   2 block   1 not stopping!
WARNING ... negative volume at i,j,k=   1  515   2 block   1 not stopping!
                         .
                         .
```

- The majority of negative volumes appear to be in block 1. By plotting the control point output it is clear that elements around the leading edge slat are not well defined, and probably causing poorly defined (singular) macro-elements in that region.

Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation

- The first step in solving this problem is to observe that the file 'meshdef.inp' has been created.
- This file contains the control points that were created by the code.
- Contents of this file can be pasted into the input and customized as needed.
- Since negative volumes occurred in block 1 we will add to the control points in that block.

Contents of 'meshdef.inp':

```
   GRID  NIND  NJND  NKND
      1     2    11     2
****************************** I NODE INDICES ***********************************
      1     2
****************************** J NODE INDICES ***********************************
      1    49    50   137   273   317   473   609   696   697
    745
****************************** K NODE INDICES ***********************************
      1    57
   GRID  NIND  NJND  NKND
      2     2    11     2
****************************** I NODE INDICES ***********************************
      1     2
****************************** J NODE INDICES ***********************************
      1    49    50   137   145   281   325   461   548   549
    597
****************************** K NODE INDICES ***********************************
      1    89
   GRID  NIND  NJND  NKND
      3     2     8     2
****************************** I NODE INDICES ***********************************
      1     2
****************************** J NODE INDICES ***********************************
      1    49    50   121   129   216   217   265
****************************** K NODE INDICES *********** **********************
      1    65
   GRID  NIND  NJND  NKND
      4     2    10     2
****************************** I NODE INDICES ***********************************
      1     2
****************************** J NODE INDICES ***********************************
      1    49    50   121   257   413   549   636   637   685
****************************** K NODE INDICES ***********************************
      1    33
```

Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation

- These additional points have been chosen simply to fill in gaps in the control point distribution.
- This customized input is pasted into the input file, and *nskip* set to *4*.

Contents of 'meshdef.inp' customized:

```
  GRID  NIND  NJND  NKND
     1     2    18     2
******************************** I NODE INDICES ************************************
     1     2
******************************** J NODE INDICES ************************************
     1    49    50   103   137   173   223   273   297   317
   373   423   473   543   609   696   697   745
******************************** K NODE INDICES ************************************
     1    57
  GRID  NIND  NJND  NKND
     2     2    11     2
******************************** I NODE INDICES ************************************
     1     2
******************************** J NODE INDICES ************************************
     1    49    50   137   145   281   325   461   548   549
   597
******************************** K NODE INDICES ************************************
     1    89
  GRID  NIND  NJND  NKND
     3     2    12     2
******************************** I NODE INDICES ************************************
     1     2
******************************** J NODE INDICES ************************************
     1    49    50    73   101   121   129   137   157   216
   217   265
******************************** K NODE INDICES *********** ************************
     1    65
  GRID  NIND  NJND  NKND
     4     2    10     4
******************************** I NODE INDICES ************************************
     1     2
******************************** J NODE INDICES ************************************
     1    49    50   121   257   413   549   636   637   685
******************************** K NODE INDICES ************************************
     1    10    17    33
```

Points added that remove the negative volumes in block 1

Points added to better define the flap region

Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation

Control point indices the code actually uses:

This is the data output into the new file ‘meshdef.inp’ after the code is rerun. This file is printed out because new points have been added by the code in addition to points added by the user.

```
  GRID  NIND  NJND  NKND
     1     2    26     2
********************************** I NODE INDICES ************************************
     1     2
********************************** J NODE INDICES ************************************
     1    49    50   103   109   129   137   173   203   223
   273   297   317   373   423   473   523   543   573   609
   617   637   643   696   697   745
********************************** K NODE INDICES ************************************
     1    57
  GRID  NIND  NJND  NKND
                              .
                              .
                              .
                              .
  GRID  NIND  NJND  NKND
     3     2    13     2
********************************** I NODE INDICES ************************************
     1     2
********************************** J NODE INDICES ************************************
     1    49    50    73   101   121   129   137   157   163
   216   217   265
********************************** K NODE INDICES ************************************
     1    65
  GRID  NIND  NJND  NKND
     4     2    20     4
********************************** I NODE INDICES ************************************
     1     2
********************************** J NODE INDICES ************************************
     1    49    50    73   101   121   257   313   363   413
   463   483   513   549   557   577   583   636   637   685
********************************** K NODE INDICES ************************************
     1    10    17    33
```

Control points added by user

Control points added by the code to maintain 1-1 blocking interface continuity

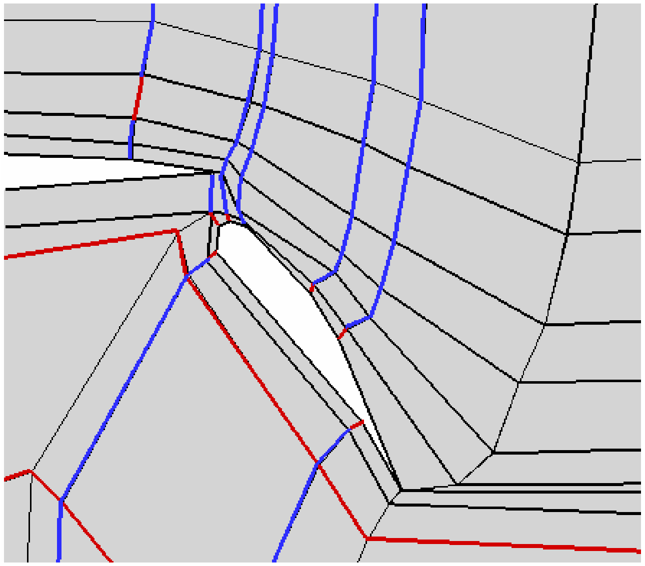
Surface Motion - Deforming Mesh

Example 2 : 2D Flap rotation

Control point lines added by the user

Control point lines added by the code to maintain continuity at 1-1 blocking interfaces

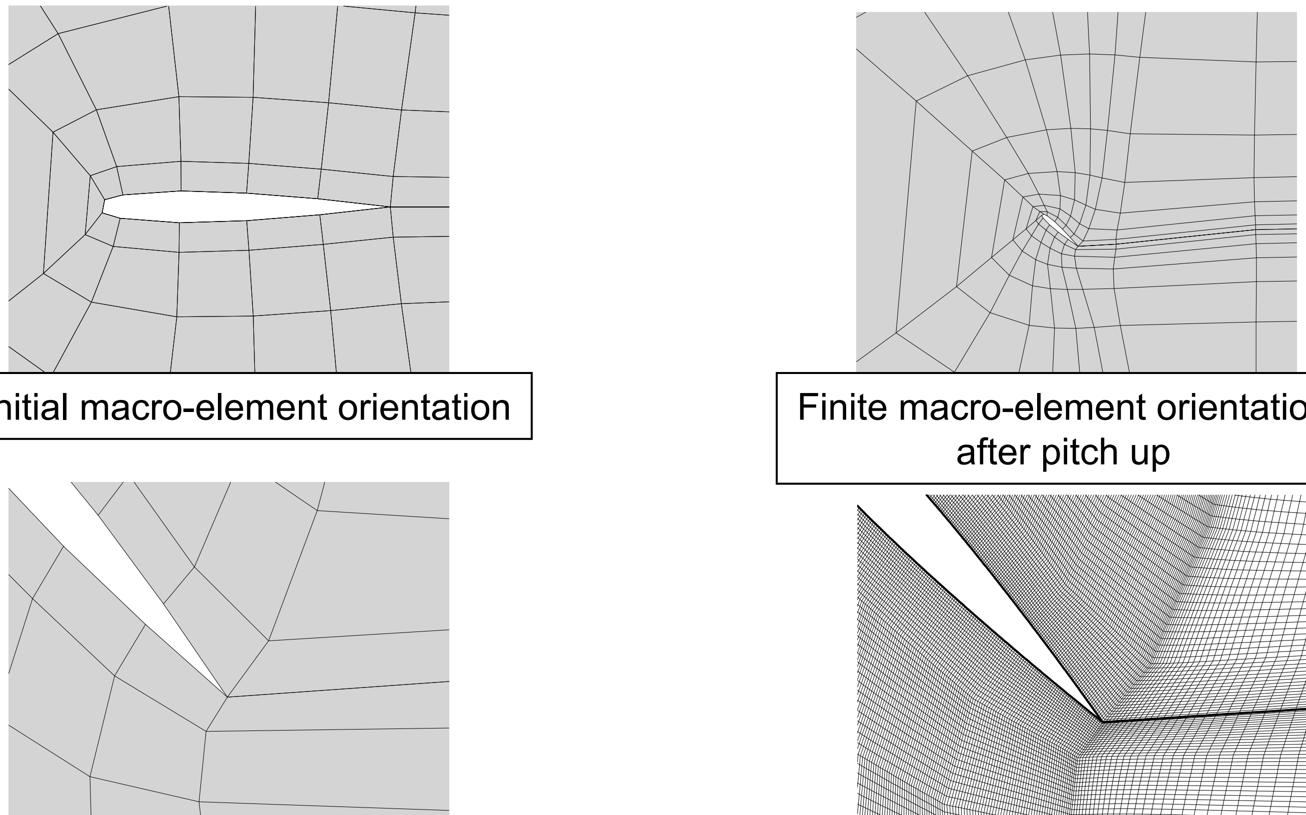
Control point indices the code actually uses:



With these new control points, the code runs robustly with no negative volumes for both the exponential decay and finite macro-element methods for a range of parameter values. Note that the region just aft of the flap retains grid quality better using the finite macro-element method than did the original.

Surface Motion - Deforming Mesh

Example 3 : 2D airfoil rotation with finite macro-element method



Initial macro-element orientation

Finite macro-element orientation after pitch up

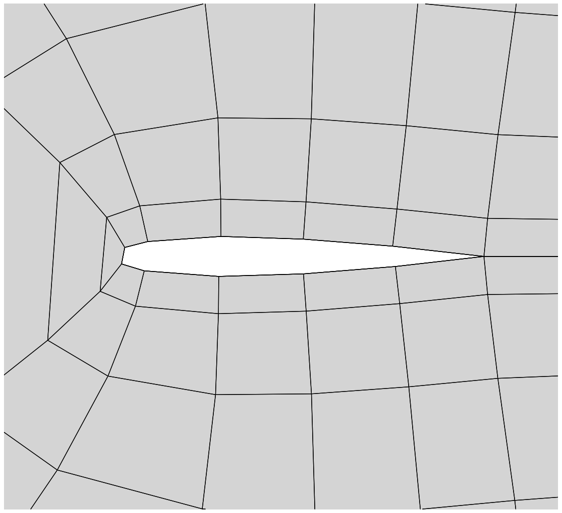
Trailing edge detail of macro-element Orientation – note orthogonality

Trailing edge detail of mesh orientation

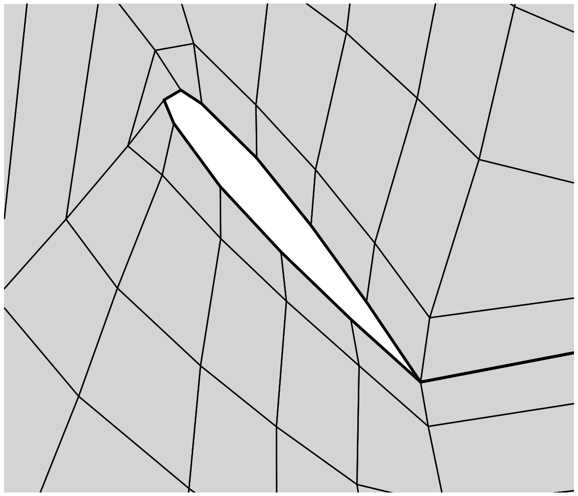
From Bartels, R. E., "Finite Macro-Element Mesh Deformation in a Structured Multi-Block Navier-Stokes Code," NASA/TM-2005-213789, July 2005.

Surface Motion - Deforming Mesh

Example 3 : 2D airfoil rotation with exponential decay method



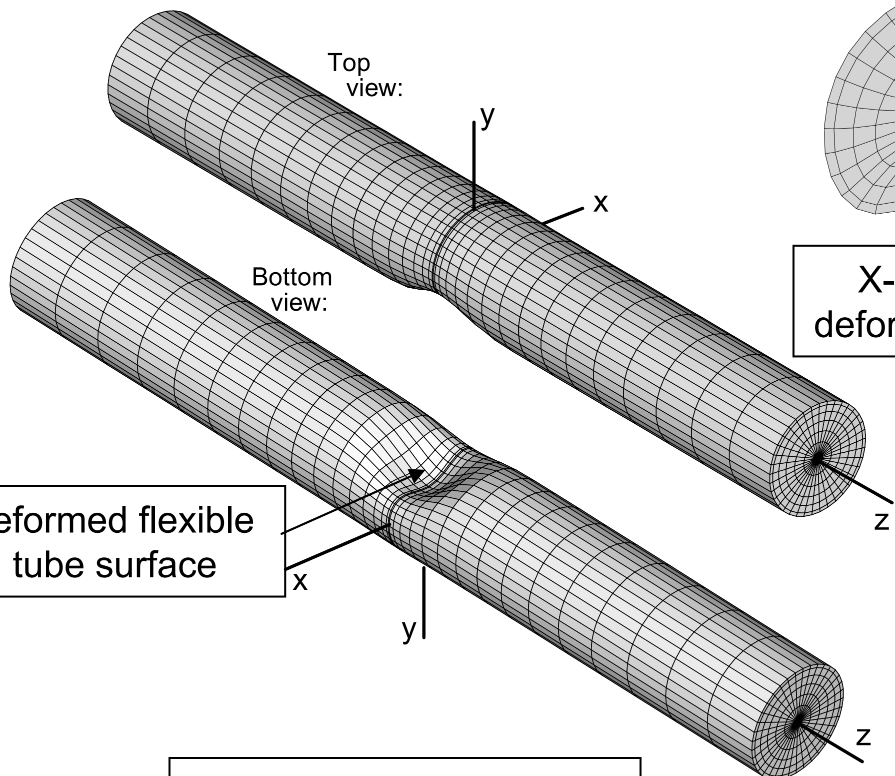
Initial control point orientation



Control point orientation
after pitch up, $\beta_2 = 2,\ \alpha_2 = .005$

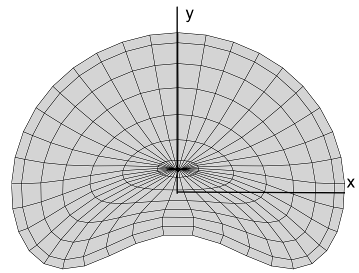
Surface Motion - Deforming Mesh

Example 4 : Internal flow through a flexible tube using the finite macro-element method

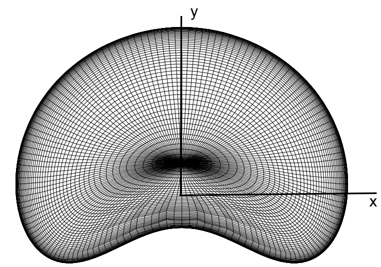


Deformed flexible tube surface

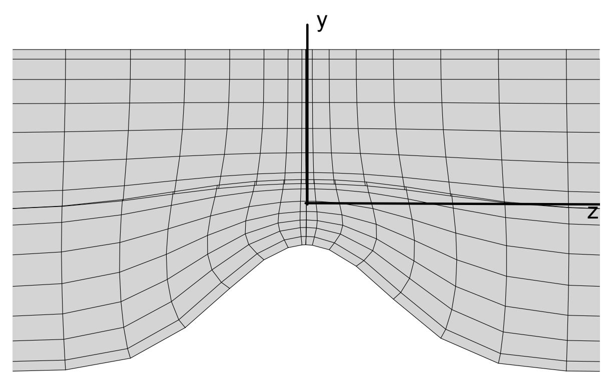
Control points for motion of internal flow field mesh



X-Y plane view of deformed control points



X-Y plane view of deformed mesh points

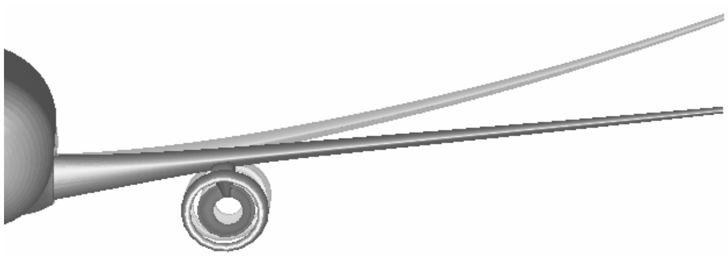


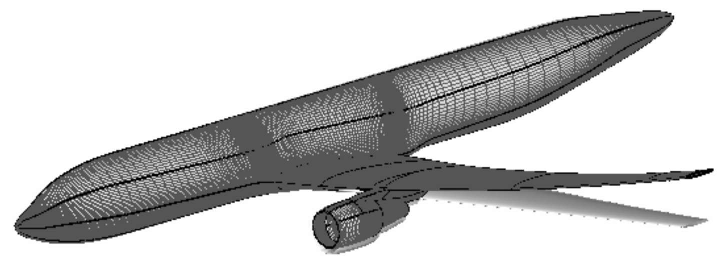
Y-Z plane view of deformed control points

Surface Motion - Deforming Mesh

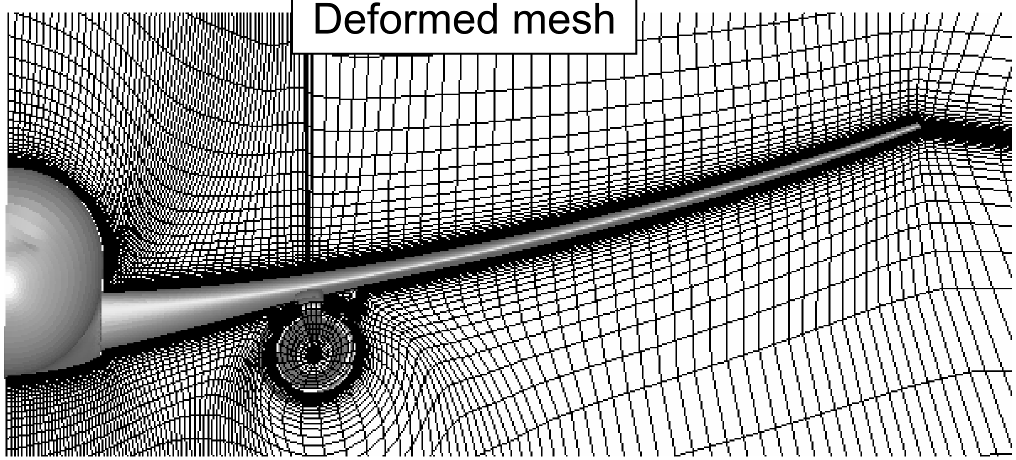
Example 5 : Transport wing bending using the Exponential Decay Method

Initial and deformed geometry





Deformed mesh



This example used mesh movement Option 2 (*isktyp = -1, nskip = 0*)

Surface Motion - Deforming Mesh

Geometric conservation law

In general the equations computed are

$$\frac{1}{J}\frac{\partial Q}{\partial t} = R(Q)$$

where

Q	- solution vector
J	- Jacobian of the grid transformation
$R(Q)$	- right hand side composed of spatial flux terms

For steady and unsteady computations:

$$R(Q) = -\left[\frac{\partial(F-F_v)}{\partial\xi} + \frac{\partial(G-G_v)}{\partial\eta} + \frac{\partial(H-H_v)}{\partial\zeta}\right]$$

where

F,G,H	- inviscid fluxes
F_v,G_v,H_v	- viscous fluxes

Surface Motion - Deforming Mesh

Geometric conservation law

For unsteady deforming mesh computations there is an additional term:

$$R(Q) = -\left[\frac{\partial(F - F_v)}{\partial \xi} + \frac{\partial(G - G_v)}{\partial \eta} + \frac{\partial(H - H_v)}{\partial \zeta}\right]$$

$$\boxed{+Q\left[\frac{\partial}{\partial t}\left(\frac{1}{J}\right) + \frac{\partial}{\partial \xi}\left(\frac{\xi_t}{J}\right) + \frac{\partial}{\partial \eta}\left(\frac{\eta_t}{J}\right) + \frac{\partial}{\partial \zeta}\left(\frac{\zeta_t}{J}\right)\right]}$$

Geometric Conservation Law (GCL), due to grid volume change

The implication of this is that a computation using rigid grid motion *may* perform somewhat differently than a deforming grid solution with the same time step size, number of sub-iterations and CFL number. However, the two *fully converged* solutions will be the same. See Bartels, R. E., “Mesh and Solution Strategies and the Accurate Computation of Unsteady Spoiler and Aeroelastic Problems,” *Journal of Aircraft*, Vol. 37, No. 3, May 2000, pp. 521-529.

Surface Motion - Deforming Mesh

Multiple types of coupled motion

Consider the example of wing plunge combined with control surface rotation. Since the control surface rotation is about a point fixed on the larger moving wing surface, coupling of the two motions will be required. There are two ways to perform this coupled motion:

1. Coupling control surface rotation and wing translation combined using mesh deformation.

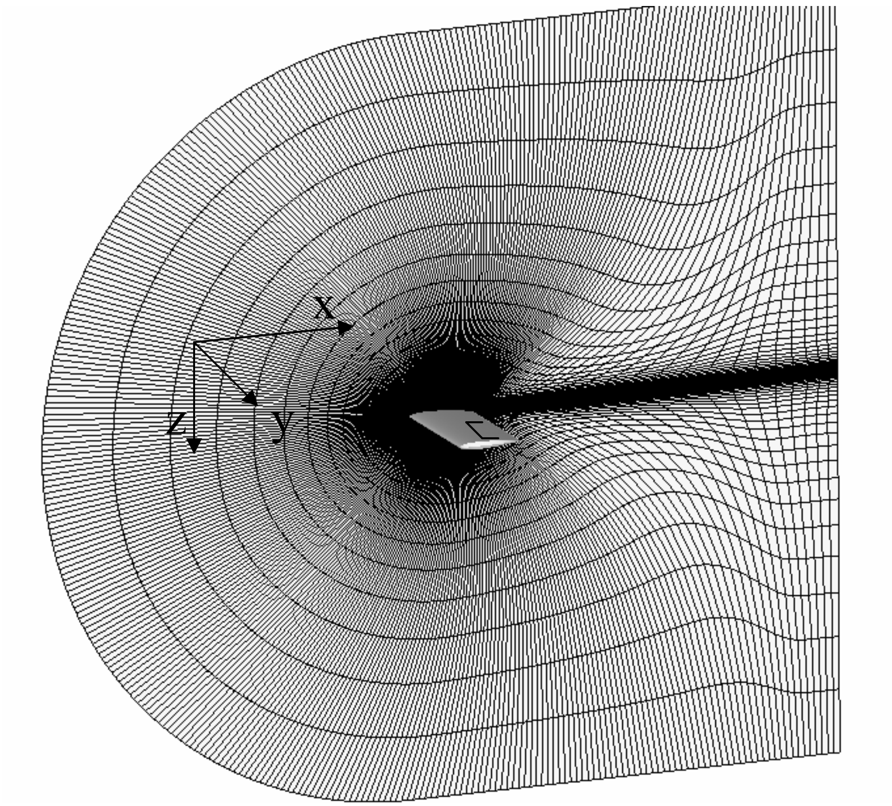
2. Coupling control surface rotation using mesh deformation with rigid grid translation.

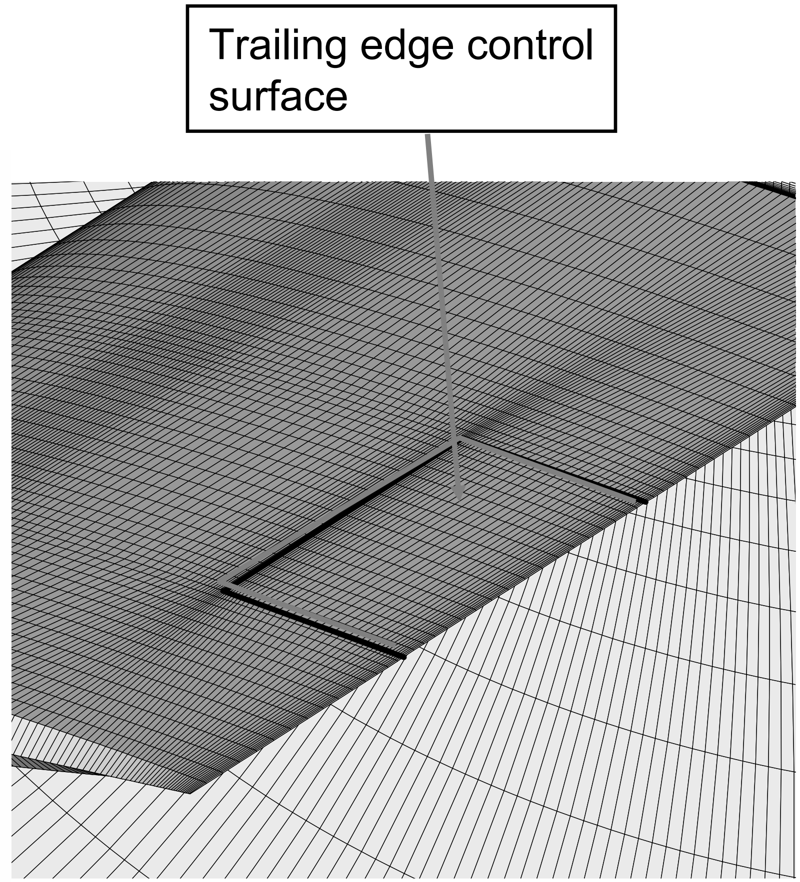
Although these two approaches result in identical wing surface motion, off body grid motion will be much different.

Surface Motion - Deforming Mesh

Example: Control surface rotation plus wing plunging

As an example consider the wing shown having both wing plunge plus control surface rotation:





Surface Motion - Deforming Mesh

Example: Multi-motion using deforming mesh

The following unsteady input file performs the wing plunging with control surface rotation using deforming mesh:

```
input/output files:
 wbgrid.cfl
 plot3dg.bin
 plot3dq.bin
 cfl3d.out
 cfl3d.res
 cfl3d.turres
 cfl3d.blomax
 cfl3d.out15
 cfl3d.prout
 cfl3d.out20
 ovrlp.bin
 patch.bin
 restart.bin
 NASA Langley BACT Model: NACA 0012 af, AR=1.5 wing,.75TE Flap
     Mach     alpha      beta        ReUe    Tinf,dR     ialph    ihstry
  0.82000   0.00000   0.00000   0.236E+07     486.00         1         0
     sref      cref      bref         xmc        ymc       zmc
    1.000   1.00000   1.00000     0.25000    0.00000   0.00000
       dt     irest   iflagts        fmax      iunst   cfl_tau
  0.04000         0      3000     1.00000          2   2.00000
    ngrid   nplot3d    nprint      nwrest       ichk       i2d    ntstep       ita
        1         1         1        1000          0         0         1        -2
```

Note that *iunst = 2* since deforming mesh is used

Surface Motion - Deforming Mesh

Example: Multi-motion using deforming mesh

```
           ncg        iem  iadvance    iforce   ivisc(i)   ivisc(j)   ivisc(k)
             2          0         0         1          5          5          5
          idim       jdim      kdim
            73        345        73
        ilamlo     ilamhi    jlamlo    jlamhi     klamlo     klamhi
             0          0         0         0          0          0
         inewg     igridc        is        js         ks         ie         je         ke
             0          0         0         0          0          0          0          0
      idiag(i)   idiag(j)  idiag(k)  iflim(i)   iflim(j)   iflim(k)
             1          1         1         3          3          3
       ifds(i)     fds(j)   ifds(k)  rkap0(i)   rkap0(j)   rkap0(k)
             1          1         1    0.3333     0.3333     0.3333
          grid      nbci0   nbcidim     nbcj0    nbcjdim      nbck0    nbckdim     iovrlp
             1          1         1         1          1          5          1          0
i0:       grid    segment    bctype      jsta       jend       ksta       kend      ndata
             1          1      1001         1        345          1         73          0
idim:     grid    segment    bctype      jsta       jend       ksta       kend      ndata
             1          1      1002         1        345          1         73          0
j0:       grid    segment    bctype      ista       iend       ksta       kend      ndata
             1          1      1003         1         73          1         73          0
jdim:     grid    segment    bctype      ista        end       ksta       kend      ndata
             1          1      1003         1         73          1         73          0
k0:       grid    segment    bctype      ista       iend       jsta       jend      ndata
             1          1         0         1         49          1         33          0
             1          2      2004         1         49         33        313          2
       tw/tinf         cq
       0.00000    0.00000
             1          3         0         1         49        313        345          0
             1          4         0        49         73          1        173          0
             1          5         0        49         73        173        345          0
kdim:     grid    segment    bctype      ista       iend       jsta       jend      ndata
             1          1      1003         1         73          1        345          0
```

Surface Motion - Deforming Mesh

Example: Multi-motion using deforming mesh

```
  mseq     mgflag     iconsf        mtt   ngam
       1          2          1          0        2
    issc epsssc(1) epsssc(2) epsssc(3)    issr epsssr(1) epsssr(2) epsssr(3)
       0    0.3000      0.3000     0.3000        0    0.3000     0.3000    0.3000
    ncyc   mglevg      nemgl         nitfo
       8          3          0          0
     mit1       mit2        mit3        mit4      mit5  ...
       1          1          1
1-1 blocking data:
      nbli
       2
number       grid        ista         jsta   ksta     iend       jend      kend    isva1   isva2
       1          1          1          1       1        49         33         1        1        2
       2          1         49          1       1        73        173         1        1        2
number       grid        ista         jsta   ksta     iend       jend      kend    isva1   isva2
       1          1          1        345       1        49        313         1        1        2
       2          1         49        345       1        73        173         1        1        2
patch interface data:
   ninter
       0
plot3d output:
    grid       iptyp        ista        iend     iinc      jsta       jend       jinc     ksta    kend    kinc
       1          0          1         49       1         1        345          1        1       1       1
 movie
   0
print out:
    grid       iptyp        ista        iend     iinc      jsta       jend       jinc     ksta    kend    kinc
       1          0          1         49       1         1        345          1        1       1       1
```

Surface Motion - Deforming Mesh

Example: Multi-motion using deforming mesh

```
Control Surfaces:
ncs
0
   Grid      ista     iend       jsta      jend      ksta      kend    iwall    inorm
Moving grid data – deforming surface (forced motion):
ndefrm
      3
     lref
     1.0
   Grid    idefrm    rfreq   u/omegax  v/omegay  w/omegaz     xorig    yorig    zorig
      1         1     0.10       0.00      0.00      0.20      0.00     0.00     0.00
      1         2     0.05       0.00     10.00      0.00      0.75     0.00     0.00
      1         2     0.05       0.00     10.00      0.00      0.75     0.00     0.00
   Grid      icsi     icsf       jcsi      jcsf      kcsi      kcsf
      1         1       49         33       313         1         1
      1        25       37         33        65         1         1
      1        25       37        281       313         1         1
Moving grid data – aeroelastic surface (aeroelastic motion):
 naesrf
      0
  Iaesrf    ngrid    grefl       uinf      qinf    nmodes  iskyhook
   Freq    gmass     damp    x0(2n-1)    xo(2n)    gf0(2n)
 Moddfl      amp     freq         t0
   Grid     iaei     iaef       jaei      jaef      kaei      kaef
Moving grid data – data for field/multiblock mesh movement
  nskip    isktyp    beta1    alpha1    beta2    alpha2   nsprgit
      0       -2      1.0       1.1      1.0     0.005        0
 Control point index input
Moving grid data – multi-motion coupling
ncoupl
      1
Slave    master    xorig        yorig        zorig
      1        1     0.75         0.00         0.00
```

User specified surface motion data now includes both translation and rotation

Multi-motion coupling data now included

Surface Motion - Deforming Mesh

Example: Multi-motion using deforming mesh

Focusing on the user specified motion input:

.

.

Moving grid data – deforming surface (forced motion):

ndefrm

3

lref

1.0

Grid	idefrm	rfreq	u/omegax	v/omegay	w/omegaz	xorig	yorig	zorig
1	1	0.10	0.00	0.00	0.20	0.00	0.00	0.00
1	2	0.05	0.00	10.00	0.00	0.75	0.00	0.00
1	2	0.05	0.00	10.00	0.00	0.75	0.00	0.00

Grid	icsi	icsf	jcsi	jcsf	kcsi	kcsf
1	1	49	33	313	1	1
1	25	37	33	65	1	1
1	25	37	281	313	1	1

.

.

The new lines prescribe the motion of the wing surface

Note that *idefrm = 1,* which corresponds to translational motion.

Surface Motion - Deforming Mesh

Example: Multi-motion using deforming plus rigid grid motion

The following unsteady input file performs the wing plunging using rigid grid translation and control surface rotation using deforming mesh:

```
input/output files:
 wbgrid.cfl
 plot3dg.bin
 plot3dq.bin
 cfl3d.out
 cfl3d.res
 cfl3d.turres
 cfl3d.blomax
 cfl3d.out15
 cfl3d.prout
 cfl3d.out20
 ovrlp.bin
 patch.bin
 restart.bin
 NASA Langley BACT Model: NACA 0012 af, AR=1.5 wing,.75TE Flap
    Mach     alpha      beta        ReUe    Tinf,dR     ialph    ihstry
  0.82000  0.00000  0.00000  0.236E+07    486.00         1         0
     sref     cref      bref         xmc        ymc        zmc
    1.000  1.00000  1.00000    0.25000   0.00000    0.00000
       dt    irest   iflagts        fmax      iunst    cfl_tau
  0.04000        0      3000     1.00000          3    2.00000
    ngrid  nplot3d    nprint      nwrest       ichk        i2d    ntstep    ita
        1        1         1        1000          0          0         1     -2
```

Note that *iunst = 3*, for deforming mesh plus rigid grid motion

Surface Motion - Deforming Mesh

Example: Multi-motion using deforming plus rigid grid motion

	ncg	iem	iadvance	iforce	ivisc(i)	ivisc(j)	ivisc(k)	
	2	0	0	1	5	5	5	
	idim	jdim	kdim					
	73	345	73					
	ilamlo	ilamhi	jlamlo	jlamhi	klamlo	klamhi		
	0	0	0	0	0	0		
	inewg	igridc	is	js	ks	ie	je	ke
	0	0	0	0	0	0	0	0
	idiag(i)	idiag(j)	idiag(k)	iflim(i)	iflim(j)	iflim(k)		
	1	1	1	3	3	3		
	ifds(i)	fds(j)	ifds(k)	rkap0(i)	rkap0(j)	rkap0(k)		
	1	1	1	0.3333	0.3333	0.3333		
	grid	nbci0	nbcidim	nbcj0	nbcjdim	nbck0	nbckdim	iovrlp
	1	1	1	1	1	5	1	0
i0:	grid	segment	bctype	jsta	jend	ksta	kend	ndata
	1	1	1001	1	345	1	73	0
idim:	grid	segment	bctype	jsta	jend	ksta	kend	ndata
	1	1	1002	1	345	1	73	0
j0:	grid	segment	bctype	ista	iend	ksta	kend	ndata
	1	1	1003	1	73	1	73	0
jdim:	grid	segment	bctype	ista	end	ksta	kend	ndata
	1	1	1003	1	73	1	73	0
k0:	grid	segment	bctype	ista	iend	jsta	jend	ndata
	1	1	0	1	49	1	33	0
	1	2	2004	1	49	33	313	2
	tw/tinf	cq						
	0.00000	0.00000						
	1	3	0	1	49	313	345	0
	1	4	0	49	73	1	173	0
	1	5	0	49	73	173	345	0
kdim:	grid	segment	bctype	ista	iend	jsta	jend	ndata
	1	1	1003	1	73	1	345	0

Surface Motion - Deforming Mesh

Example: Multi-motion using deforming plus rigid grid motion

```
  mseq     mgflag      iconsf          mtt   ngam
      1            2             1          0        2
    issc epsssc(1) epsssc(2) epsssc(3)     issr epsssr(1) epsssr(2) epsssr(3)
      0    0.3000      0.3000      0.3000        0    0.3000      0.3000    0.3000
    ncyc   mglevg      nemgl           nitfo
      8          3             0              0
     mit1       mit2         mit3          mit4      mit5  ...
      1          1             1
 1-1 blocking data:
       nbli
      2
 number       grid        ista          jsta   ksta      iend        jend       kend    isva1   isva2
      1          1            1              1       1         49          33            1         1        2
      2          1          49              1       1         73         173            1         1        2
 number       grid        ista          jsta   ksta      iend        jend       kend    isva1   isva2
      1          1            1          345       1         49         313            1         1        2
      2          1          49          345       1         73         173            1         1        2
 patch interface data:
    ninter
      0
 plot3d output:
    grid       iptyp        ista         iend      iinc      jsta        jend         jinc     ksta    kend    kinc
      1          0            1            49        1           1        345            1         1        1        1
  movie
   0
 print out:
    grid       iptyp        ista         iend      iinc      jsta        jend         jinc     ksta    kend    kinc
      1          0            1            49        1           1        345            1         1        1        1
```

Surface Motion - Deforming Mesh

Example: Multi-motion using deforming plus rigid grid motion

```
Control Surfaces:
ncs
0
   Grid        ista       iend            jsta          jend          ksta          kend       iwall     inorm
moving grid data - rigid translation (forced motion):
  ntrans
       1
     lref
     1.0
    grid      itrans       rfreq     utrans      vtrans    wtrans
       1           2       0.10       0.00        0.00       5.00
    grid     dxmax     dymax     dzmax
       1         10.         10.         10.
moving grid data - rigid rotation (forced motion):
  nrotat
       0
    lref
grid           irotat       rfreq       omegax      omegay       omegaz          xorig     yorig       zorig
grid        dthxmx   dthymx        dthzmx
Moving grid data – deforming surface (forced motion):
ndefrm
      2
    lref
    1.0
   Grid      idefrm       rfreq    u/omegax   v/omegay   w/omegaz          xorig      yorig       zorig
      1            2       0.05          0.00       10.00           0.00          0.75       0.00        0.00
      1            2       0.05          0.00       10.00           0.00          0.75       0.00        0.00
   Grid         icsi        icsf             jcsi           jcsf           kcsi           kcsf
      1           25          37               33             65               1               1
      1           25          37             281           313               1               1
```

Rigid grid motion input

Surface motion input

Surface Motion - Deforming Mesh

Example: Multi-motion using deforming plus rigid grid motion

```
Moving grid data – aeroelastic surface (aeroelastic motion):
 naesrf
      0
 Iaesrf    ngrid      grefl          uinf          qinf       nmodes   iskyhook
  Freq   gmass      damp    x0(2n-1)        xo(2n)       gf0(2n)
Moddfl      amp        freq               t0
   Grid      iaei        iaef            jaei            jaef             kaei          kaef
Moving grid data – data for field/multiblock mesh movement
 nskip       isktyp        beta1       alpha1       beta2     alpha2    nsprgit
      0             -2          1.0             1.1           1.0      0.005             0
 Control point index input
Moving grid data – multi-motion coupling
ncoupl
     1
Slave     master      xorig           yorig          zorig
     1            1      0.75            0.00            0.00
```

Aeroelastic header lines included

Deforming mesh input

Multi-motion coupling data included

Note: CFL3D does not allow initiating new kinds of motion upon restarts. Therefore if an initial deforming mesh computation is performed to reach an equilibrium before initiating a combined rigid and moving (deforming) control surface computation, the option $iunst = 3$ must be used from the start (that is after an initial steady state computation with $dt < 0$), with control surface motion set to zero.

Aeroelastic Analysis

Overview

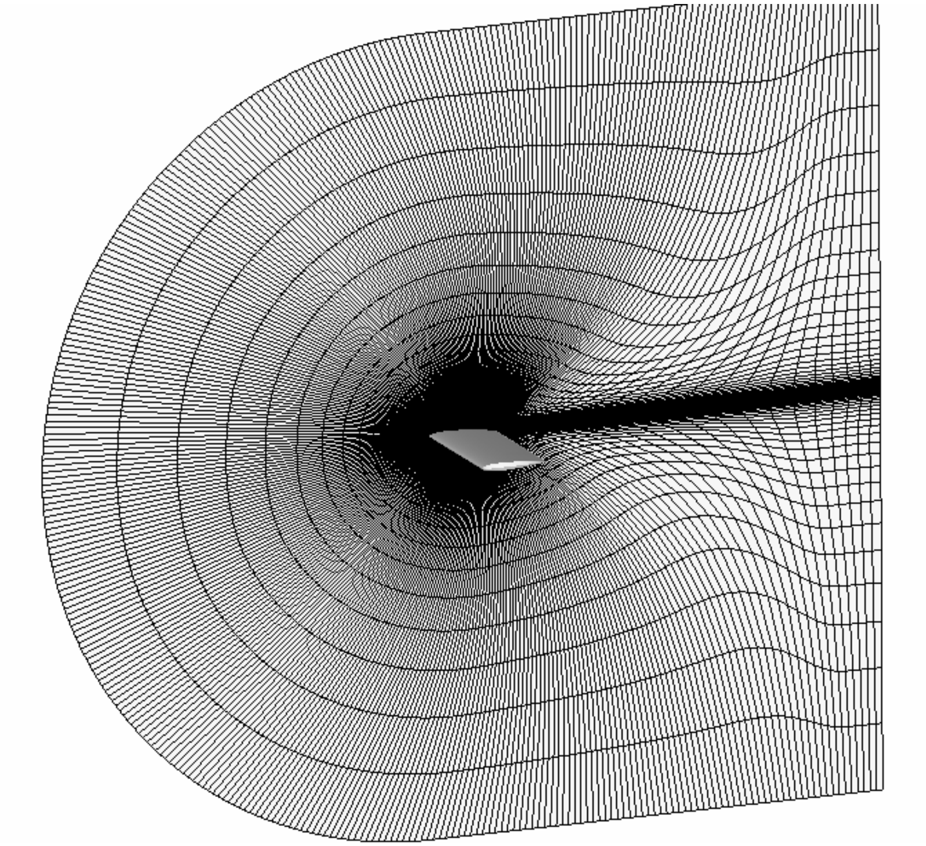
- CFL3D has the capability to perform both static and dynamic aeroelastic analysis. In this analysis the fluid and structure interact through a time marching simulation (e.g. flutter analysis, etc…)
- All aeroelastic and modal analyses are performed by running the code in unsteady mode
- CFL3D performs aeroelastic analysis for a linear structure modally and a linear and nonlinear structure when loosely coupled with either NASTRAN or ABAQUS finite element analyses.
- For modal analysis, the equations of structural dynamics must be decoupled modally
  - Eigenvalue analysis is required prior to running CFD to obtain frequencies, generalized masses and mode shapes.
  - A preprocessing step projecting the mode shapes onto the CFD surface grids is required.
  - The code reads the modal data projected onto the CFD surfaces in the file 'aesurf.dat'. This file must be contained in the directory in which the executable resides.
- CFL3D also has the capability to perform unsteady deforming body analysis using mode shapes. In this mode the user specifies modal motion (e.g. control surface rotation, wing plunge oscillation, etc…) in the aeroelastic input section

Aeroelastic Analysis

Example of an aeroelastic model

Consider the Benchmark Active Controls Technology (BACT) aeroelastic model shown. The model has pitch and plunge aeroelastic degrees of freedom. The model parameters are:

M_T	$= 5.966$	*slugs*
S_α	$= 0.01420$	*slug-ft*
I_α	$= 2.8017$	*slug-ft2*
K_h	$= 2659$	*lb/ft*
K_a	$= 2897$	*lb-ft/rad*



Aeroelastic Analysis

Example of an aeroelastic model

The coupled equations of structural dynamics are

$$\begin{bmatrix} M_T & S_\alpha \\ S_\alpha & I_\alpha \end{bmatrix} \{\ddot{\zeta}\} + \begin{bmatrix} K_h & 0 \\ 0 & K_\alpha \end{bmatrix} \{\zeta\} = \mathrm{q}_\infty \begin{Bmatrix} \iint c_p(x^*, y^*)\, dx^*\, dy^* \\ \iint c_p(x^*, y^*)(x^*_{ea} - x^*)\, dx^*\, dy^* \end{Bmatrix}$$

where ζ_1 is plunge (h) and ζ_2 is pitch (α). Eigen-analysis of this system yields the frequencies

$$\omega_h = 21.1113283\, rad/\sec\, (3.36\, Hz)$$
$$\omega_\alpha = 32.1564455\, rad/\sec\, (5.12\, Hz)$$

Aeroelastic Analysis

Example of an aeroelastic model

Using the eigenvectors

$$\phi = \begin{bmatrix} \varphi_{11} & \varphi_{12} \\ \varphi_{21} & \varphi_{22} \end{bmatrix} = \begin{bmatrix} 0.409404775 & 0.0024991919 \\ 0.001571926 & -0.5974345042 \end{bmatrix}$$

the generalized masses are obtained

$$m_1 = 1.000000000$$
$$m_2 = 1.000000000$$

Aeroelastic Analysis

Example of an aeroelastic model

… and the decoupled equations of structural dynamics

$$\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \{\ddot{q}\} + \begin{bmatrix} \omega_1^2 & 0 \\ 0 & \omega_2^2 \end{bmatrix} \{q\} = \mathrm{M}^{-1}\phi^{\mathrm{T}} \mathrm{q}_\infty \begin{Bmatrix} \iint c_p(x^*, y^*)\, dx^*\, dy^* \\ \iint c_p(x^*, y^*)(x^*_{ea} - x^*)\, dx^*\, dy^* \end{Bmatrix}$$

where

$$\zeta = \phi\, q \qquad M = \begin{bmatrix} m_1 & 0 \\ 0 & m_2 \end{bmatrix} \qquad \begin{bmatrix} m_1\omega_1^2 & 0 \\ 0 & m_2\omega_2^2 \end{bmatrix} = \phi^T \begin{bmatrix} K_h & 0 \\ 0 & K_\alpha \end{bmatrix} \phi$$

Carrying through the multiplication on the right-hand side, we have

$$\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \{\ddot{q}\} + \begin{bmatrix} \omega_1^2 & 0 \\ 0 & \omega_2^2 \end{bmatrix} \{q\} = \mathrm{M}^{-1} \mathrm{q}_\infty \begin{Bmatrix} \iint c_p(x^*, y^*) \{\varphi_{11} + \varphi_{21}(x^*_{ea} - x^*)\} dx^*\, dy^* \\ \iint c_p(x^*, y^*) \{\varphi_{12} + \varphi_{22}(x^*_{ea} - x^*)\} dx^*\, dy^* \end{Bmatrix}$$

Aeroelastic Analysis

Example of an aeroelastic model

The mode shapes that are input into CFL3D are revealed by the last equations

$$\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \{\ddot{q}\} + \begin{bmatrix} \omega_1^2 & 0 \\ 0 & \omega_2^2 \end{bmatrix} \{q\} = \mathrm{M}^{-1} \mathrm{q}_\infty \begin{Bmatrix} \iint c_p(x^*, y^*) \{\varphi_{11} + \varphi_{21}(x^*_{ea} - x^*)\} dx^* \, dy^* \\ \iint c_p(x^*, y^*) \{\varphi_{12} + \varphi_{22}(x^*_{ea} - x^*)\} dx^* \, dy^* \end{Bmatrix}$$

First mode shape, $\Phi_{z,1}$

Second mode shape, $\Phi_{z,2}$

These can be used to create the modal shape projected to each wing surface grid point for input into CFL3D. Note that x^* and y^* are in the same units as the structural model.

Aeroelastic Analysis

Aeroelastic input

```
              .
              .
              .
      dt    irest   iflagts    fmax      iunst   cfl_tau
  0.0125        1         0     1.0          2       5.0
              .
              .
              .
control surfaces:
       ncs
         0
      grid    ista    iend     jsta     jend     ksta    kend    iwall    inorm
   moving grid data - deforming surface (forced motion)
    ndefrm
         0
      lref
      grid  idefrm  rfreqi   omegax   omegay   omegaz   xorig   yorig    zorig
      grid    icsi    icsf     jcsi     jcsf     kcsi    kcsf
```

iunst = 2 for an aeroelastic simulation

User specified deforming surface Input, header lines only

Aeroelastic Analysis

Aeroelastic input

```
   moving grid data - aeroelastic surface (aeroelastic motion)
     naesrf
          1
     iaesrf      ngrid      grefl        uinf        qinf   nmodes     iskyhk
          1         -1    0.08333        730.       1000.        2          0
       freq      gmass       damp   x0(2*n-1)     x0(2*n)  gf0(2*n)
 21.1113283     1.0000       0.00         0.0         0.0        0.
 32.1564454     1.0000       0.00         0.0         0.0        0.
     moddfl        amp       freq          t0
          0      0.000       0.00        0.00
          0      0.000       0.00        0.00
       grid       iaei       iaef        jaei        jaef      kaei       kaef
          1          0          0           0           0         0          0
   moving grid data - skip data for field/multiblock mesh movement
    nskip    isktyp    beta1    alpha1    beta2    alpha2    nsprgit
    0        -2        1.0      1.1       1.0      0.005     0
   Control point index input
   moving grid data - multi-motion coupling
     ncoupl
          0
      slave     master      xorig       yorig       zorig
```

Aeroelastic input

Mesh deformation input

Aeroelastic Analysis

Aeroelastic input

Focusing on the aeroelastic input section:

Number of aeroelastic surfaces

```
moving grid data - aeroelastic surface (aeroelastic motion)
      naesrf
           1
      iaesrf     ngrid      grefl       uinf       qinf  nmodes     iskyhk
           1        -1    0.08333       730.      1000.       2          0
        freq     gmass       damp  x0(2*n-1)    x0(2*n)  gf0(2*n)
  21.1113283    1.0000       0.00        0.0        0.0       0.
  32.1564454    1.0000       0.00        0.0        0.0       0.
     moddfl        amp       freq         t0
           0     0.000       0.00       0.00
           0     0.000       0.00       0.00
        grid      iaei       iaef       jaei       jaef     kaei      kaef
           1         0          0          0          0        0         0
```

naesrf lines

nmodes lines

one line only when *ngrid = -1* (Currently this Is the only option)

iaesrf - Identifier of the aeroelastic surface for which data is being supplied
ngrid - Number of surface segments that make up this aeroelastic surface
nmodes - Number of modes to be modeled in CFL3D
iskyhk - Not currently used, any value will serve as a placeholder
uinf - Free-stream velocity, in the same units as the equations of structural dynamics
qinf - Dynamic pressure, in the same units as the equations of structural dynamics
grefl - Conversion from CFD grid units to structural equation units.

Aeroelastic Analysis

Aeroelastic input

Regarding the input parameter *grefl*, consider the equations of structural dynamics for the pitch/plunge example:

$$\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \{\ddot{q}\} + \begin{bmatrix} \omega_1^2 & 0 \\ 0 & \omega_2^2 \end{bmatrix} \{q\} = \mathrm{M}^{-1} \mathrm{q}_\infty \begin{Bmatrix} \iint c_p(x^*, y^*)\,\Phi_{z,1}\, dx^*\, dy^* \\ \iint c_p(x^*, y^*)\,\Phi_{z,2}\, \underbrace{dx^*\, dy^*}_{\text{Lengths in structural model units}} \end{Bmatrix}$$

The actual equations solved in CFL3D are:

$$\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \{\ddot{q}\} + \begin{bmatrix} \omega_1^2 & 0 \\ 0 & \omega_2^2 \end{bmatrix} \{q\} = \mathrm{grefl}^2\, \mathrm{M}^{-1}\, \mathrm{q}_\infty \begin{Bmatrix} \iint c_p(x, y)\,\Phi_{z,1}\, dx\, dy \\ \iint c_p(x, y)\,\Phi_{z,1}\, \underbrace{dx\, dy}_{\text{Lengths in CFD grid units}} \end{Bmatrix}$$

By definition: $grefl = \sqrt{S_{AE} / S_{CFD}}$

Aeroelastic Analysis

Aeroelastic input

In the present example the structural equations are in units of feet, while the CFD grid is in units of inches. **Note that the aspect ratios of the original and rescaled model must be identical**. Conversion for the present example can be obtained from

$$grefl = \sqrt{S_{AE} / S_{CFD}} = \sqrt{\frac{1}{144}} \approx 0.08333\, ft / grid\ unit$$

This is the *grefl* parameter that would be entered in the aeroelastic input section.

Aeroelastic Analysis

Modal form of the equations

Consider the decoupled equations of structural dynamics for N (or $nmodes$ in the input) modes

$$\begin{bmatrix} 1 & 0 & 0 \\ 0 & \ddots & 0 \\ 0 & 0 & 1 \end{bmatrix} \{\ddot{q}\} + \begin{bmatrix} 2\omega_1\zeta_1 & 0 & 0 \\ 0 & \ddots & 0 \\ 0 & 0 & 2\omega_N\zeta_N \end{bmatrix} \{\dot{q}\} + \begin{bmatrix} \omega_1^2 & 0 & 0 \\ 0 & \ddots & 0 \\ 0 & 0 & \omega_N^2 \end{bmatrix} \{q\}$$

$$= \begin{bmatrix} \mathrm{m}_1^{-1} & 0 & 0 \\ 0 & \ddots & 0 \\ 0 & 0 & \mathrm{m}_N^{-1} \end{bmatrix} \{\mathrm{Q}\}$$

where q is the modal variable vector and Q is the generalized force vector, each of length N. ω_1 ,…, ω_N are the natural frequencies of each structural mode in radians, and m_1 ,…, m_N are the generalized masses.

Aeroelastic Analysis

Modal form of the equations

CFL3D input definitions as they relate to the modal equations of structural dynamics are as follows:

$$gmass(1) = m_1, \quad \cdots, \quad gmass(N) = m_N$$

$$freq(1) = \omega_1, \quad \cdots, \quad freq(N) = \omega_N$$

$$damping(1) = \zeta_1 \quad , \cdots, \; damping(N) = \zeta_N$$

$$x0(1) = q_{1\,init}, \cdots, \; x0(2*N-1) = q_{N\,init}$$

$$x0(2) = \dot{q}_{1\,init}, \cdots, \quad x0(2*N) = \dot{q}_{N\,init}$$

$$gf0(2) = Q_{1\,init}, \cdots, \quad gf0(2*N) = Q_{N\,init}$$

Units for frequency is radians/time (usually time scale is seconds for the structural dynamics equations).

Aeroelastic Analysis

Aeroelastic input

- ***x0(2*n-1)*** is the initial generalized displacement of the mode; will override the value in the restart file (if restarting) when *x0(2*n-1)* is nonzero. Otherwise, it will not override the restart value. This allows the mode to be perturbed for excitation of aeroelastic dynamic response after a static aeroelastic starting solution has been performed.

- ***x0(2*n)*** is the initial generalized velocity of the mode; will override the value in the restart file (if restarting) when *x0(2*n)* is nonzero. Otherwise, it will not override the restart value. This allows the mode to be perturbed for excitation of aeroelastic dynamic response after a static aeroelastic starting solution has been performed.

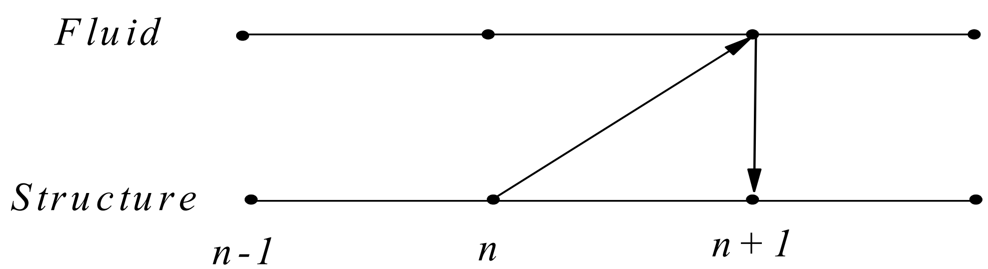
- ***gf0(2*n)*** is the generalized force offset to include for the mode. This value is included in CFL3D computation of generalized force in the following way for mode $n = 1$ to *nmodes*:

$$Q_n = q_\infty \; grefl^2 \left\{ \iint c_p \, \vec{\Phi}_n \cdot d\vec{s} \right\} - \underbrace{gf0(2*n)}_{\text{Value from input}}$$

Aeroelastic Analysis

Method of integrating the fluid/structure coupling based on the state transition matrix solution

The structural dynamic equations are written in state space form and solved using the state transition matrix solution and a predictor/corrector scheme. The fluid/structure cycling is shown below:



Predictor step:

$$\tilde{x}^{n+1} = \Theta x^n + \tfrac{1}{2}\Theta_i\left(3\overline{Q}^{\,n} - \overline{Q}^{\,n-1}\right)$$

After the predictor step, the structure surface is moved using the predicted modal solution $\tilde{x}^{n+1}$, the flow field is converged to the solution at time step $n+1$, and the new generalized force $\overline{Q}^{\,n+1}$ is computed.

Corrector step:

$$x^{n+1} = \Theta x^n + \tfrac{1}{2}\Theta_i\left(\overline{Q}^{\,n+1} + \overline{Q}^{\,n}\right)$$

Aeroelastic Analysis

Method of integration and fluid/structure coupling

The parameter Θ is the state transition matrix. The parameter Θ_i is the discrete integration of the state input, and $\overline{Q}^{\,n}$ is the state vector containing the generalized aerodynamic force at time step n. The solution and generalized force state vectors are

$$x = \begin{Bmatrix} q_1 \\ \dot{q}_1 \\ \vdots \\ q_N \\ \dot{q}_N \end{Bmatrix} \quad , \quad \overline{Q} = \begin{Bmatrix} 0 \\ Q_1 \\ \vdots \\ 0 \\ Q_N \end{Bmatrix}$$

The value $\widetilde{x}$ is the intermediate state solution used to update the mesh.

This method uses a second order backward differencing of the fluid/structure coupling. Combined with second order backward differenced fluid dynamics solver and the second order backward differencing of the mesh time metrics, the overall scheme is second order accurate. For the original method see: Edwards, J. W., Bennett, R. M., Whitlow Jr., W., Seidel, D. A.,"Time-Marching Transonic Flutter Solutions Including Angle-of-Attack Effects," *Journal of Aircraft*, 20 (1983), pg. 899-906.

Aeroelastic Analysis

Modal surface input

- Currently CFL3D assumes that the aeroelastic surface comprises all boundary segments with the boundary condition types 1005, 1006, 2004, 2014 or 2016.
- Note that the boundary condition 1001 is not considered an aeroelastic surface. Therefore, if a symmetry plane is required to deform with a pitching wing, it must be treated as an inviscid wall boundary (1005 or 1006)
- The modal input file *aesurf.dat* must have modal data for a given surface point in free field ascii format (no commas) with $\Phi_{x,n}$, $\Phi_{y,n}$, $\Phi_{z,n}$ modal deflections at each surface point for each mode n.

Aeroelastic Analysis

Format of the modal surface input

Segment limits defined in boundary condition input

The following ordering is required:

j = 1 **surface:**

$\Phi_{x,n}(i,j,k)$ $\Phi_{y,n}(i,j,k)$ $\Phi_{z,n}(i,j,k)$, $k = ksta$ to $kend$, $i = ista$ to $iend$, repeat *nseg* times

j = jdim **surface:**

$\Phi_{x,n}(i,j,k)$ $\Phi_{y,n}(i,j,k)$ $\Phi_{z,n}(i,j,k)$, $k = ksta$ to $kend$, $i = ista$ to $iend$, repeat *nseg* times

k = 1 **surface:**

$\Phi_{x,n}(i,j,k)$ $\Phi_{y,n}(i,j,k)$ $\Phi_{z,n}(i,j,k)$, $j = jsta$ to $jend$, $i = ista$ to $iend$, repeat *nseg* times

k = kdim **surface:**

$\Phi_{x,n}(i,j,k)$ $\Phi_{y,n}(i,j,k)$ $\Phi_{z,n}(i,j,k)$, $j = jsta$ to $jend$, $i = ista$ to $iend$, repeat *nseg* times

i = 1 **surface:**

$\Phi_{x,n}(i,j,k)$ $\Phi_{y,n}(i,j,k)$ $\Phi_{z,n}(i,j,k)$, $j = jsta$ to $jend$, $k = ksta$ to $kend$, repeat *nseg* times

i = idim **surface:**

$\Phi_{x,n}(i,j,k)$ $\Phi_{y,n}(i,j,k)$ $\Phi_{z,n}(i,j,k)$, $j = jsta$ to $jend$, $k = ksta$ to $kend$, repeat *nseg* times,

Repeat all of the above input for $n = 1$ to *nmodes*, repeat *ngrid* times, repeat *naesrf* times.

Aeroelastic Analysis

Format of the modal surface input

- The ordering of the aeroelastic surface points *must* correspond to the order of the points in the CFD grid file read by CFL3D.
- Aeroelastic segments must be input in the same block order as the grid file, and segments must be input in order of ascending indices.
- When creating a multi zonal grid using the utility 'splitter', be aware that the final ordering will generally *not* correspond to the ordering of the unsplit grid. Ordering of the split grid zones can be found in the '*splitter.out'* file, from which can be found the required order of the surface grid points for the '*aesurf.dat'* file.

Example: Consider a block face that has dimensions *kdim = 49*, *idim = 49* with several aeroelastic segments. If segment 1 has indices *k = 33* to *49*, *i = 13* to *33*, and segment 2 has indices *k = 1* to *33*, *i = 1* to *33*, then segment 2 must be input first.

Aeroelastic Analysis

Aeroelastic output

- Aeroelastic time history output is in the file ‘*genforce.dat*’.
- This file is generated if $iunst = 2$ and aeroelastic surfaces are defined in the input file *($naesrf \neq 0$).*
- After header information, modal response data for each mode is written sequentially.
- Unlike output data in the ‘*cfl3d.subit_res’* file, a complete time history of this data for the entire simulation is retained and written/read to/from restart files and subsequently output to the ‘*genforce.dat’* file.

Aeroelastic Analysis

Aeroelastic output

Consider the example output contained in the 'genforce.dat' file:

```
NASA Langley BACT Model: NACA 0012 af, AR=1.5 wing,.75TE Flap
 Mach=  0.7700E+00, alpha=  0.0000E+00, ReUe=  0.3860E+07
 Number of aeroelastic surfaces =  1
 Data for aeroelastic surface   1
   mode number   1
          it          time            xs(2*n-1)          xs(2*n)        gforcn(2*n)
           1     0.3125000E-01  0.0000000E+00  0.0000000E+00 -0.3471162E-05
           2     0.6250000E-01  0.0000000E+00  0.0000000E+00 -0.3214494E-05
           3     0.9375000E-01  0.0000000E+00  0.0000000E+00 -0.2996337E-05
           4     0.1250000E+00  0.0000000E+00  0.0000000E+00 -0.2789857E-05
 mode number   2
          it          time            xs(2*n-1)          xs(2*n)        gforcn(2*n)
           1     0.3125000E-01  0.2980232E-09  0.3442899E-09  0.6291896E-05
           2     0.6250000E-01  0.3089730E-09  0.3565678E-09  0.6644112E-05
           3     0.9375000E-01  0.3203131E-09  0.3692693E-09  0.6907312E-05
           4     0.1250000E+00  0.3320569E-09  0.3824084E-09  0.7143990E-05
```

Title line from the input file

Data from the input file

Mode 1 time history from starting run

Mode 2 time history from starting run

Time	- Non-dimensional time (CFL3D non-dimensionalization)
*xs(2*n-1)*	- Modal or generalized variable output
*xs(2*n)*	- Modal velocity output
*gforcn(2*n)*	- Modal or generalized force output

Aeroelastic Analysis

Strategy for aeroelastic computations

The following strategies may be used for performing static or dynamic aeroelastic simulations

- Static aeroelastic computations can be performed by:
  - Start either from scratch (*irest = 0*), or restart, after a steady state computation (in which *dt < 0*, *iunst = 0*). Starting from scratch is not recommended.
  - Set *iunst = 2* , *dt > 0* and *damp = .99999…* and perform the computation in a time marching manner to convergence.
- Flutter onset computations can be performed by:
  - Converging a static solution as outlined above.
  - Setting *damp* to the correct value for the elastic system being modeled.
  - Setting an initial perturbation *x0(2*n)* or *x0(2*n-1)* in the desired mode.*

* If a restart in the middle of a flutter computation is performed, the initial perturbation values from the previous run must be reset to zero at the restart of the new run.

Aeroelastic Analysis

User specified modal motion

The user may specify modal motion within the aeroelastic input (e.g. control surface rotation, wing plunge oscillation, impulse for frequency response, etc…) The following modifications to the aeroelastic input specifies modal motion:

```
                                .
                                .
moving grid data - aeroelastic surface (aeroelastic motion)
      naesrf
           1
      iaesrf     ngrid     grefl       uinf      qinf  nmodes    iskyhk
           1        -1   0.08333       730.     1000.       2         0
        freq     gmass      damp  x0(2*n-1)   x0(2*n)  gf0(2*n)
  21.1113283    1.0000      0.00        0.0       0.0        0.
  32.1564454    1.0000      0.00        0.0       0.0        0.
     moddfl        amp      freq         t0
          1      0.005      0.20       0.00
          0      0.000      0.00       0.00
        grid      iaei      iaef       jaei      jaef      kaei      kaef
           1         0         0          0         0         0         0
                                .
                                .
```

This line specifies motion for mode 1

Aeroelastic Analysis

User specified modal motion

moddfl

type of time-varying modal perturbation desired:

< 0, mode displacement and velocity set to zero

= 0, no perturbation (solution via the dynamic modal equations)

= 1, harmonic (sinusoidal) perturbation

= 2, Gaussian pulse

= 3, step pulse

A *(amp)*

amplitude of modal perturbation.

ω_r *(freq)*

reduced frequency of modal perturbation if moddfl = 1

half-width of Gaussian pulse if moddfl = 2

use any value as a placeholder for moddfl = 0

t_0 *(t0)*

time about which Gaussian pulse is centered if moddfl = 2

time of the step pulse if moddfl = 3

use any value as a placeholder for moddfl = 0

Aeroelastic Analysis

User specified modal motion

For harmonic perturbation the modal displacement and velocities for mode n are computed in the following way:

$$q_n = A\sin(\omega_r t^*) \quad , \quad \dot{q}_n = Ak_r \cos(\omega_r t^*)$$

where $A = amp$, $\omega_r = freq$ in radians per dimensional time, and t^* is dimensional time,

$$t^* = t\, grefl / a_\infty \quad , \quad a_\infty = U_\infty / M_\infty$$

U_∞ (uinf) is in the aeroelastic input section and M_∞ is from the main aerodynamic input section. t is CFL3D non-dimensional time.

For a Gaussian pulse the displacement and velocity for mode n are computed with

$$q_n = Ae^{-C\left[t^* - t_0\right]^2}, \dot{q}_n = -2CAe^{-C\left[t^* - t_0\right]^2}$$

$$where \quad C = \log(2)/\omega_r^2$$

Aeroelastic Analysis

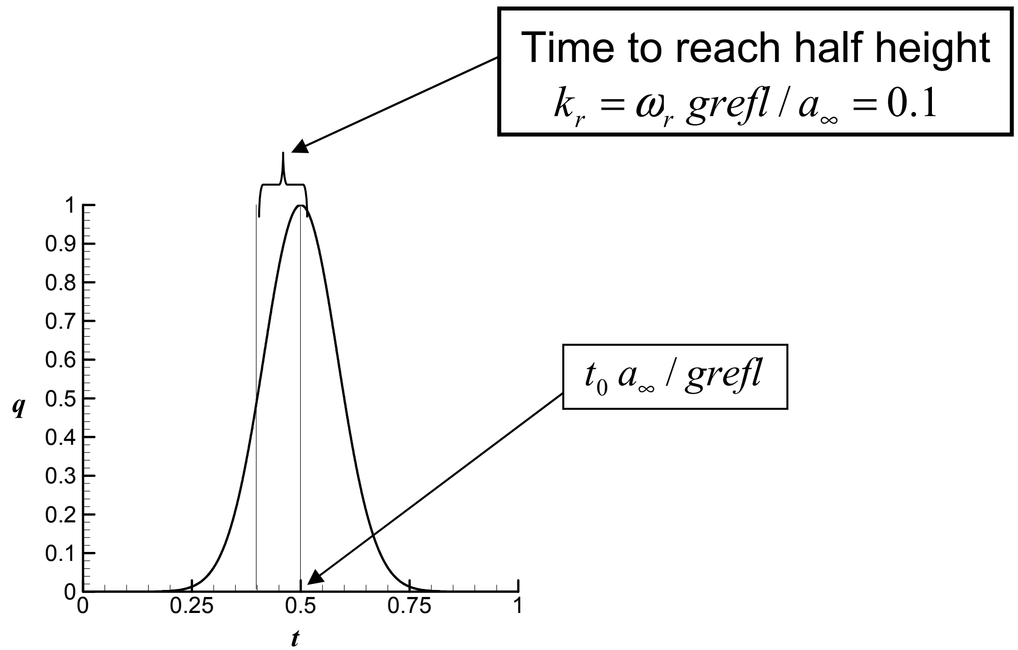
User specified modal motion

For step pulse the modal displacement and velocities for mode n are computed in the following way:

$$
\begin{aligned}
&if \quad t^* < t_0 - \frac{\Delta t^*}{2} t_0 && then \quad q_n = 0, \quad \dot{q}_n = 0 \\
&if \quad t_0 - \frac{\Delta t^*}{2} < t^* < t_0 + \frac{\Delta t^*}{2} && then \quad q_n = A, \quad \dot{q}_n = \frac{A}{\Delta t^*} \\
&if \quad t^* > t_0 + \frac{\Delta t^*}{2} t_0 && then \quad q_n = A, \quad \dot{q}_n = 0
\end{aligned}
$$

Aeroelastic Analysis

Example: Gaussian modal pulse and time step sizing



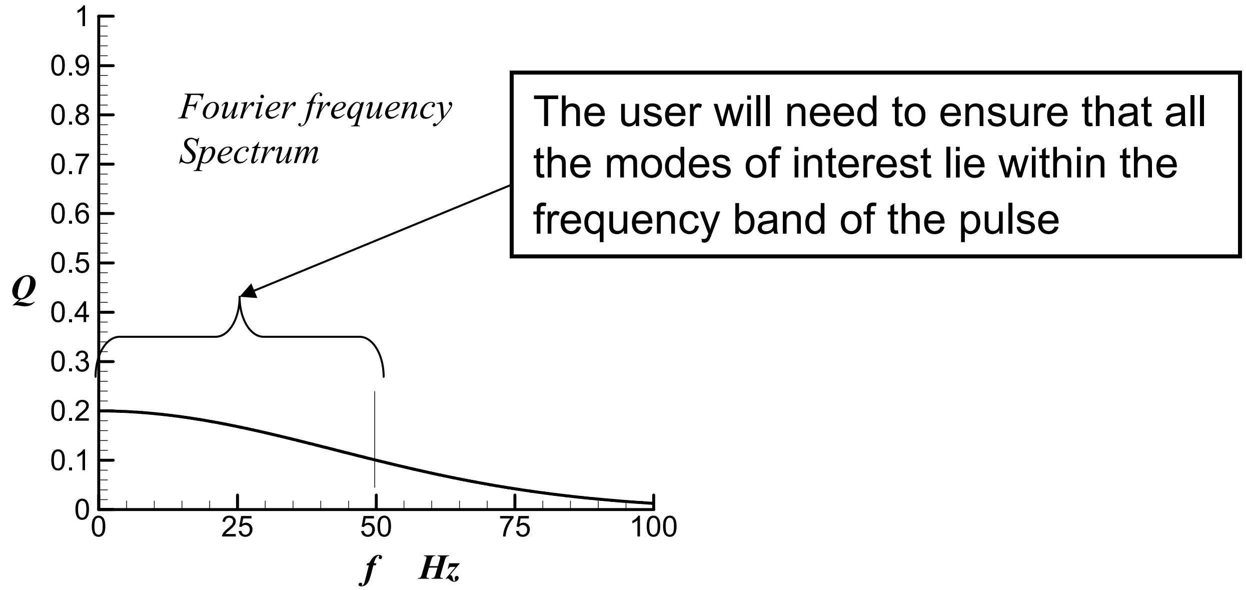
For this example: $A = 1.0 \quad , \quad k_r = 0.1 \quad , \quad t_0 = 0.5\, grefl / a_\infty$

$$C = \log(2) / \omega_r^2 \qquad t^* = t\, grefl / a_\infty \quad , \quad a_\infty = U_\infty / M_\infty$$

Recommend sizing time step so that there are an absolute *minimum* of 25 time steps within the half life of the pulse ($\Delta t = k_r/25$). In this case we would have $\Delta t = 0.004$.

Aeroelastic Analysis

Example: Shaping and sizing the Gaussian modal pulse



- For a linear response, we will usually want the amplitude as small as possible while staying significantly (say several orders of magnitude) above numerical round off errors.
- Low frequency responses will be very sensitive to the steady convergence of a solution. Therefore, great care must be exercised in adequately converging the steady state if an FRF is the desired outcome.
- The solution is very sensitive to sub-iterative convergence at each time step. A strategy of multiple restarts with different numbers of sub-iterations through the pulse region can reduce the overall run time.

Keyword Input

Overview

- There is additional input in CFL3D version 6 that does not fit into an input format consistent with earlier versions of the code. These input parameters have been included as keyword input.
- Keyword input is an optional input specified by lines started by a line with ‘>’ and ended with a line containing ‘<‘.
- The following example illustrates how keyword input is included:

```
 cfl3d.out20
 ovrlp.bin
 patch.bin
 restart.bin
>
gamma 1.32
negvol 1
<
 NASA Langley BACT Model: NACA 0012 af, AR=1.5 wing,.75TE Flap
     Mach      alpha       beta         ReUe      Tinf,dR      ialph     ihstry
  0.82000   0.00000   0.00000  0.236E+07     486.00           1          0
```

Keyword input included at the end of file specification and before the title line.

Keyword Input

Valid Keywords

Physical Properties

Name	Description	Default Value
cbar	Ref. temp. for Sutherland Law	198.6
gamma	Ratio of specific heats	1.4
pr	Prandtl number	0.72
prt	Turbulent Prandtl number	0.90

Limiters

Name	Description	Default Value
atol	Tolerance for detecting singular lines	10^{-7}
epsa_r	Eigenvalue limiter (entropy fix for high Mach flows)	0.0

Keyword Input

Valid Keywords

Preconditioning

Name	Description	Default Value
avn	Factor multiplying uref for preconditioning	1.0
cprec	Relative amount of preconditioning	0.0
uref	Limiting velocity for preconditioning	xmach

Specified CL

Name	Description	Default Value
cltarg	Target Cl	99999.
dalim	Limit of alpha change (deg) per update	0.2
icycupdt	Number of cycles between alpha updates (if > 0; if < 0, alpha is never updated)	1
rlxalph	Relaxation factor used to update angle of attack	1.0

Keyword Input

Valid Keywords

Turbulence models

Name	Description	Default Value
cflturb	Cfl no. for turbl eqns. = cflturb x abs(dt) If cflturb > 0	0 (model dependent default)
edvislim	Limiter for eddy viscosity in 2-equation turb models; eddy viscosity limited to edvislim times the laminar viscosity	100000.
ibeta8kzeta	flag (0/1) to set beta8 term when using k-enstrophy turbulence model (ivisc=15); 0 = use beta8=0.0 (helps avoid numerical problems); 1 = use beta8=2.3 (available *after* V6.3)	0
ides	flag (0/1) to perform DES with turbulence model (1) or not (0)	0
cdes	constant associated with DES	0.65
ieasmcc2d	flag (0/1) to turn on 2-D curvature correction when using EASM models (ivisc=8,9,11,12,13,14) (1) or not (0) (available *after* V6.3)	0
isarc2d	flag (0/1) to turn on 2-D curvature correction when using SA model (ivisc=5) (1) or not (0) (available *after* V6.3)	0

Keyword Input

Valid Keywords

Turbulence models

Name	Description	Default Value
sarccr3	value of cr3 parameter in SARC model (available *after* V6.3)	0.6
ikoprod	flag: 0=use approximate (vorticity-based) turb production term (-2*mut*WijWji) for turb models 6, 7, 10, or 15; 1=use strain-rate based term (2*mut*SijSij); 2=use full production term (ivisc=15 only) (available *after* V6.3)	0 (vorticity-based production)
isstdenom	flag (0/1): 0=use vorticity term in denominator of eddy viscosity in SST model (#7); 1=use strain term (available *after* V6.3)	0 (vorticity term)
itaturb	flag (0/1) to control time accuracy of turb. model; 0 for 1st order in time regardless of parameter "ita" for the mean flow; 1 for same order as set by ita	1 (turb. Time accuracy same as mean flow, set via ita)
iturbord	flag controls whether turbulence model advection terms are 1st or 2nd order upwind on RHS (1=1st, 2=2nd) (note: LHS uses 1st order in both cases) (available *after* V6.3)	1 (1st order)

Keyword Input

Valid Keywords

Turbulence models

Name	Description	Default Value
iturbprod	flag: 0=use strain-rate based turb production term (2*mut*SijSij) for EASM turb models 8, 9, 13, or 14; 1=use full production term	0 (strain-rate based term)
nfreeze	Freeze turb. model for nfreeze cycles	0 (not frozen)
nsubturb	Number of iterations of turb model per cycle	1
pklimterm	factor used to limit production of k in 2-eqn turb models (chooses min of Pk and pklimterm*Dk); make this term large for no limiting (available *after* V6.3)	20.0
tur10 & tur20	turbulent quantity freestream levels < 0 use default value (different for each turb model, see manual Appendix H) =0 use this number as the specified user input value	-1
tur1cut	value that nondimensional epsilon (or omega or enstrophy) is reset to when it tries to drop equal to or below tur1cutlev; if <=0 then no update occurs when value tries to drop equal to or below tur1cutlev (available *after* V6.3)	1.e-20 for all models except -1 for ivisc=15

Keyword Input

Valid Keywords

Turbulence models

Name	Description	Default Value
tur2cut	value that nondimensional k is reset to when it tries to drop equal to or below tur2cutlev; if <=0 then no update occurs when value tries to drop equal to or below tur2cutlev (available *after* V6.3)	1.e-20
tur1cutlev & tur2cutlev	lower levels of nondimensional epsilon (or omega or enstrophy) and k which, when reached, cause the turb quantities to be reset to tur1cut or tur2cut (available *after* V6.3)	0

Keyword Input

Valid Keywords

Deformation/grid motion

Name	Description	Default Value
idef_ss	flag (0/1) to deform volume grid to surface in file newsurf.p3d	0 (don't deform)
meshdef	flag (0/1) to bypass flow solution while still computing grid operations such as metrics and volumes; 0 = normal operation; 1 = bypass flow solution (available *after* V6.3)	0
negvol	flag (0/1) to enable/disable stop if neg. volumes/bad metrics are detected	0 (stop for negative volumes)

Input/output control

Name	Description	Default Value
ibin	flag (0/1) for formatted/unformatted output plot3d files	1 (unformatted)
iblnk	flag (0/1) for un-iblanked/iblanked output plot3d files	1 (iblanked)

Keyword Input

Valid Keywords

Input/output control

Name	Description	Default Value
iblnkfr	flag (0/1) for un-iblanked/iblanked fringe points in plot3d files (overset grids only)	1 (iblanked)
icgns	flag (0/1) to not use/use CGNS files*	0 (don't use CGNS files)
ip3dgrad	flag (0/1) for solution/derivative data output to plot3d q file (complex code only)	0 (solution to q file)
irghost	flag to read ghost-cell data from restart file (1) or not (0); V5 restart files and Beta V6 restart files do not contain ghost-cell data; newer V6 restart files do	1 (read ghost-cell data)
iwghost	flag to write ghost-cell data to restart file (1) or not (0); V5 restart files and Beta V6 restart files do not contain ghost-cell data; newer V6 restart files do	1 (write ghost-cell data)

Keyword Input

Valid Keywords

Input/output control

Name	Description	Default Value
itime2read	flag (0/1) to skip/read 2nd order (in time) turbulence terms and dt in restart file: need to skip if using an older time-accurate-with-2nd-order-time restart file	1 (read 2nd order time turbulence terms and dt)
iteravg	flag to store iteration-averaged conserved variables in PLOT3D files: 0 = no averaging or storage 1 = start averaging now 2 = continue averaging from previous run	0

Memory management

Name	Description	Default Value
memadd	additional memory (in words) added to work array (in case sizer underestimates)	0 (no addition to work)
memaddi	additional memory (in words) added to iwork array (in case sizer underestimates)	0 (no addition to iwork)

Keyword Input

Valid Keywords

Reference frame

Name	Description	Default Value
noninflag	flag (0/1) to indicate whether to use inertial (0) or noninertial (1) reference frame for governing equations; noninertial frames allow for steady state solutions if the rotation rate is constant	0 (inertial reference frame)
xcentrot	rotation center x-coordinate for non-inertial reference frame (also used for roll-angle input)	0.0
ycentrot	rotation center y-coordinate for non-inertial reference frame (also used for roll-angle input)	0.0
zcentrot	rotation center z-coordinate for non-inertial reference frame (also used for roll-angle input)	0.0
xrotate	rotation rate about x-axis for non-inertial reference frame (non-dimensionalized the same way as omegax for rotating grids - see manual)	0.0
yrotate	rotation rate about y-axis for non-inertial reference frame (non-dimensionalized the same way as omegay for rotating grids - see manual)	0.0
zrotate	rotation rate about z-axis for non-inertial reference frame (non-dimensionalized the same way as omegaz for rotating grids - see manual)	0.0

Keyword Input

Valid Keywords

Reference frame

Name	Description	Default Value
xrotrate_img	complex perturbation to rotation rate about x-axis for non-inertial reference frame, for computing rate derivatives	0.0
yrotrate_img	complex perturbation to rotation rate about y-axis for non-inertial reference frame, for computing rate derivatives	0.0
zrotrate_img	complex perturbation to rotation rate about z-axis for non-inertial reference frame, for computing rate derivatives	0.0

Other

Name	Description	Default Value
alpha_img	Imaginary perturbation to alpha	0.0
beta_img	Imaginary perturbation to beta	0.0
geom_img	Imaginary perturbation to grid	0.0

Keyword Input

Valid Keywords

Other

Name	Description	Default Value
reue_img	Imaginary perturbation to unit Re	0.0
surf_img	Imaginary perturbation to surface grid	0.0
tinf_img	Imaginary perturbation to Tinf	0.0
xmach_img	Imaginary perturbation to Mach no.	0.0
iaxi2plane	flag for use with particular axisymmetric cases (for which i2d=0 and idim=2); if iaxi2plane = 1, the time step based on CFL number is modified so it does not depend on the i-direction metrics (available *after* V6.3)	0 (no mods to time step)
ifullns	flag (0/1) to specify inclusion of cross-derivative terms; 0 = thin-layer N-S; 1 = full N-S (available *after* V6.3)	0
ivolint	flag (0/1) to use approximate/exact one-to-one boundary volumes (0 emulates V5.0)	1 (exact volumes)
roll_angle	x-axis roll angle (deg) "+" is clockwise viewed from "- x" (left roll to pilot) (grid is rotated to this angle)	0.0

Block Splitting and MPI

Overview

- Message Passing Interface (MPI) protocol is used for parallelization of CFL3D
- MPI parallelizes by parceling out grid blocks to different processors
- For MPI to be useful, at least two or more blocks and at least three processors will be required.
- Often grids will arrive as multiple block grids. However, there are several reasons that additional block splitting will be required:
  - If the original mesh is not split into a sufficient number of blocks to efficiently use the processors available.
  - If the blocks are of disparate sizes, so that load balancing will be difficult.

Block Splitting and MPI

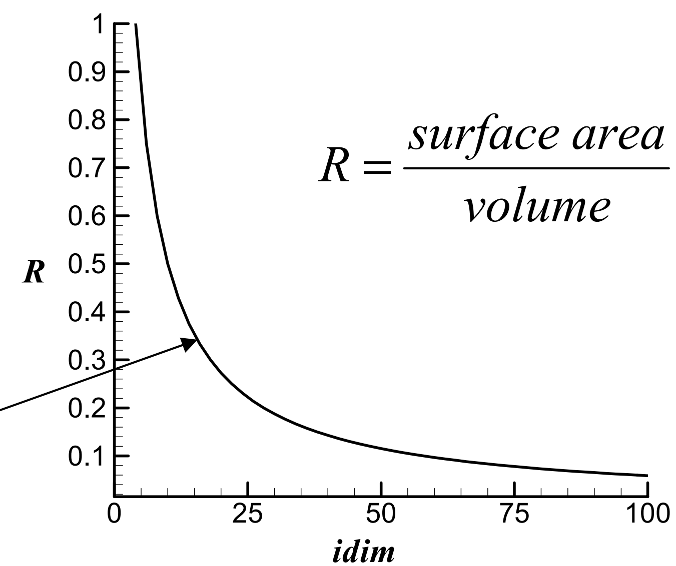
Overview

- Note, however, that there is a limit on the number of blocks for a given overall grid size for which efficient parallelization can take place.
  - Problem of growing communications between processors compared to processing per block (communication time).
  - Because CFL3D treats block boundaries explicitly, splitting into an ever increasing number of blocks amounts to making the code explicit. An increasing number of blocks means that an increasing number of sub-iterations will be required.
- The following illustrates the increasing communications with decreasing block sizes….

Block Splitting and MPI

Problem of the humming bird versus the elephant

Consider the ratio of number of surface points to the total number of grid points as grid size diminishes. These results are based on a grid having equal idim, jdim, kdim dimensions.



At an average dimension of 10x10x10, boundary data takes a third of the total memory. (Which is not a problem for MPI, but… *communication becomes a growing percentage of the computation time.)*

Block Splitting and MPI

Overview

With the issues clearly in mind, there are times when splitting is useful…

- The tool ‘splitter’ is available with CFL3D for use in splitting blocks.
- It is created by performing the following command in the ‘build’ directory:

  make splitter

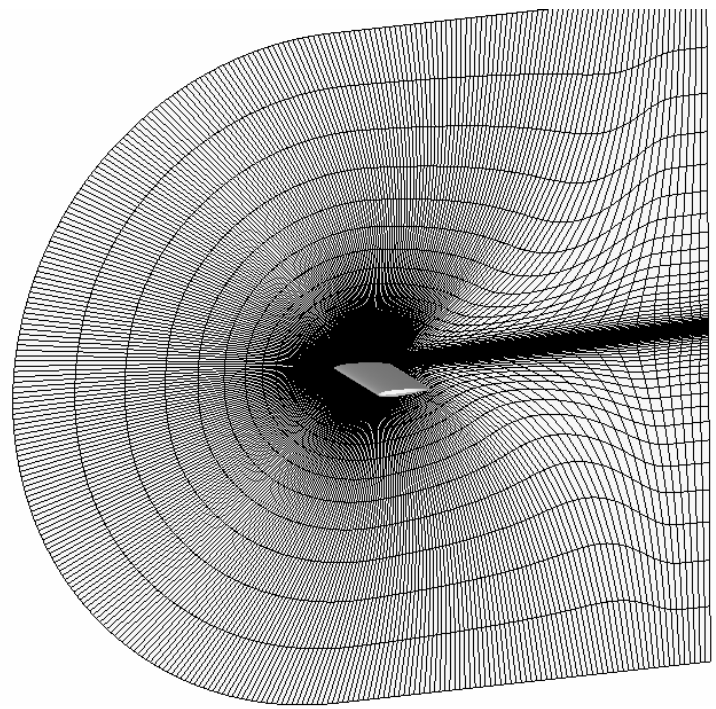
- The executable will be in the directory ‘~/cfl3dv6/build/split/seq/’.
- An example input can be found in the CFL3D version 6 web page.

Block Splitting and MPI

Example: Splitting a single C-H grid

Lets consider again the BACT wing we have looked at previously. This grid has i,j,k dimensions 73 (spanwise) x 345 (streamwise) x 73 (normal to wing).

Suppose a 32 processor PC cluster is available for this problem. It would be useful to split this block into at least 24 blocks. However consideration must also be given to how many times each dimension can be split and still retain the proper dimensions to perform multi-grid computations.



Block Splitting and MPI

Example: Splitting a single C-H grid

An acceptable block split can be obtained by requiring M, the number of split blocks, in the following computation

$$M = \frac{D-1}{d-1}$$

be an integer. D is the overall dimension of the un-split grid, and d is the proposed dimension of the split grid. For the current example, the j-dimension can be split with blocks having dimension of 9, 87 or 173.

$$M = \frac{345-1}{9-1} = 43 \quad , \quad M = \frac{345-1}{87-1} = 4 \quad , \quad M = \frac{345-1}{173-1} = 2$$

Block Splitting and MPI

Example: Splitting a single C-H grid

Note that block dimensions of *87* or *173* will allow only *3* levels of multigrid, a dimension of *9* allows *4*. We will chose a dimension of *87*.

Similar computations for the $idim = 73$ and $kdim = 73$ lead us to chose *6* blocks in those directions with dimension of *13*. This will result in a total of *144* blocks. This number of blocks will allow us to use *4, 24, 48* or *144* processors efficiently.

These computations result in *3* splits in the *j*-direction, *5* splits in the *i*-direction and *5* splits in the *k*-direction for a total of *13* splits. The input that performs these splits is shown in the next slide.

Block Splitting and MPI

Example: Splitting a single C-H grid

The splitter input file for this grid is shown:

```
INPUT (UNSPLIT) FILES
cfl3d.inp
ronnie.inp
grid.unf
sd_grid.unf
ICFLVER     IRONVER     IGRDFMT     ISDFMT
      5           1           1          1
OUTPUT (SPLIT) FILES
cfl3d.inp_split
ronnie.inp_split
grid_split.unf
sd_grid_split.unf
ICFLVER     IRONVER     IGRDFMT     ISDFMT
      5           1           1          1
NSPLITS
 13
 1
 2
 87
 1
 2
 173
 1
 2
 259
```

```
1
1
13
1
1
25
1
1
37
1
1
49
1
1
61
1
3
13
1
3
25
1
3
37
1
3
49
1
3
61
```

Block Splitting and MPI

Example: Splitting a single C-H grid

```
INPUT (UNSPLIT) FILES
cfl3d.inp
ronnie.inp
grid.unf
sd_grid.unf
ICFLVER    IRONVER    IGRDFMT    ISDFMT
      5          1          1         1
OUTPUT (SPLIT) FILES
cfl3d.inp_split
ronnie.inp_split
grid_split.unf
sd_grid_split.unf
ICFLVER    IRONVER    IGRDFMT    ISDFMT
      5          1          1         1
```

cfl3d.inp - cfl3d input file for the unsplit grid

ronnie.inp - ronnie input file for the unsplit grid, if not a patched case, enter the word **null**

grid.unf - grid file for the unsplit grid; can be formatted or unformatted

sd_grid.unf - sensitivity file for the unsplit grid NOTE: Currently not supported in Version 6; the same functionality is now handled via complex variables and a complex-valued grid file; enter the word **null**

Block Splitting and MPI

Example: Splitting a single C-H grid

INPUT (UNSPLIT) FILES
cfl3d.inp
ronnie.inp
grid.unf
sd_grid.unf

ICFLVER	IRONVER	IGRDFMT	ISDFMT
5	1	1	1

OUTPUT (SPLIT) FILES
cfl3d.inp_split
ronnie.inp_split
grid_split.unf
sd_grid_split.unf

ICFLVER	IRONVER	IGRDFMT	ISDFMT
5	1	1	1

cfl3d.inp_split - cfl3d input file for the split grid

ronnie.inp_split - ronnie input file for the split grid, if not a patched case, enter the word **null**

grid_split.unf - grid file for the split grid; can be formatted or unformatted

sd_grid_split.unf - sensitivity file for the split grid NOTE: Currently not supported in Version 6; the same functionality is now handled via complex variables and a complex-valued grid file; enter the word **null**

Block Splitting and MPI

Example: Splitting a single C-H grid

```
INPUT (UNSPLIT) FILES
cfl3d.inp
ronnie.inp
grid.unf
sd_grid.unf
ICFLVER     IRONVER     IGRDFMT     ISDFMT
      5           1           1          1
OUTPUT (SPLIT) FILES
cfl3d.inp_split
ronnie.inp_split
grid_split.unf
sd_grid_split.unf
ICFLVER     IRONVER     IGRDFMT     ISDFMT
      5           1           1          1
```

icflver

= 4 the cfl3d input file is a version 4.1 type

= -4 the cfl3d input file is a version 4.1hp type

= 5 the cfl3d input file is a version 5/6 type

ironver

= 0 ronnie input file is the old style, with all "from" blocks listed on one line

= 1 ronnie input file is the new style, with each "from" block having it's own line

NOTE: a value for ironver must always be entered, even if the case does not involve patched grids.

Block Splitting and MPI

Example: Splitting a single C-H grid

```
INPUT (UNSPLIT) FILES
cfl3d.inp
ronnie.inp
grid.unf
sd_grid.unf
ICFLVER    IRONVER    IGRDFMT    ISDFMT
      5          1          1         1
OUTPUT (SPLIT) FILES
cfl3d.inp_split
ronnie.inp_split
grid_split.unf
sd_grid_split.unf
ICFLVER    IRONVER    IGRDFMT    ISDFMT
      5          1          1         1
```

igrdfmt

= 0 grid file is formatted

= 1 grid file is unformatted

isdfmt

= 0 sensitivity file is formatted

= 1 sensitivity file is unformatted

NOTE: Currently not supported in Version 6; the same functionality is now handled via complex variables and a complex-valued grid file; however a value is still required - use 0 or 1

Block Splitting and MPI

Example: Splitting a single C-H grid

```
 .
 .
NSPLITS
13
1
2
87
1
2
173
1
2
259
 .
 .
```

nsplits

nsplits - number of grid splits to perform (can be 0 in order to convert grid from formatted to unformatted or vice versa). Following the value of *nsplits*, *nsplits* triplets of integers must appear, one integer of the triplet per line….

Block Splitting and MPI

Example: Splitting a single C-H grid

```
.
.
NSPLITS
 13
 1
 2
 87
 1
 2
 173
 1
 2
 259
  .
  .
```

iblk

ldir

index

Same triplet repeated 13 times

iblk - block number of the block to be split. NOTE: *iblk* always refers to the original, unsplit block number

ldir

= 1 split in the i-direction

= 2 split in the j-direction

= 3 split in the k-direction

index - split the block in the *ldir* direction at this value of the index

Block Splitting and MPI

Example: Splitter output

```
**************************************************
**                                              **
**   SPLITTER - CFL3D BLOCK AND INPUT FILE SPLITTER   **
**                                              **
** VERSION 6.X :  Computational Fluids Lab, Mail Stop 128,  **
**            NASA Langley Research Center, Hampton, VA     **
**            Release Date:    MMM DD, YYYY.                **
**                                              **
**************************************************
**************************************************

 memory allocation:   431.046108 Mbytes, double precision

input (unsplit) files
 cfl3d.inp
 null
 wbgrid.cfl
 null
icflver   ironver   igrdfmt   isdfmt
    5        1         1        1
output (split) files
 cfl3d.inp_split
 null
 wbgrid_split.cfl
 null
icflver   ironver   igrdfmt   isdfmt
    5        1         1        1
```

Block Splitting and MPI

Example: Splitter output

```
converting unsplit cfl3d input file to tlns3d map file

checking dimensions...

reading grid...
grid: wbgrid.cfl

block #   1:  il=  73,   jl= 345,   kl=  73

number of splits =  13
  split  block  coord  index
     1      1      J      87
     2      1      J     173
     3      1      J     259
     4      1      I      13
     5      1      I      25
     6      1      I      37
     7      1      I      49
     8      1      I      61
     9      1      K      13
    10      1      K      25
    11      1      K      37
    12      1      K      49
    13      1      K      61
```

new block	old block	i0	i1	j0	j1	k0	k1
1	1	1	13	1	87	61	73
2	1	1	13	87	173	61	73
3	1	1	13	173	259	61	73
4	1	1	13	259	345	61	73
5	1	13	25	259	345	61	73
6	1	13	25	173	259	61	73
7	1	13	25	87	173	61	73
8	1	13	25	1	87	61	73
9	1	25	37	1	87	61	73
10	1	25	37	87	173	61	73
11	1	25	37	173	259	61	73
12	1	25	37	259	345	61	73
13	1	37	49	259	345	61	73
14	1	37	49	173	259	61	73
15	1	37	49	87	173	61	73
16	1	37	49	1	87	61	73
17	1	49	61	1	87	61	73
18	1	49	61	87	173	61	73
19	1	49	61	173	259	61	73
20	1	49	61	259	345	61	73
21	1	61	73	259	345	61	73
22	1	61	73	173	259	61	73
23	1	61	73	87	173	61	73
24	1	61	73	1	87	61	73
25	1	61	73	1	87	49	61
26	1	61	73	87	173	49	61
27	1	61	73	173	259	49	61
28	1	61	73	259	345	49	61

Block Splitting and MPI

Example: Splitter output

29	1	49	61	259	345	49	61
30	1	49	61	173	259	49	61
31	1	49	61	87	173	49	61
32	1	49	61	1	87	49	61
33	1	37	49	1	87	49	61
34	1	37	49	87	173	49	61
35	1	37	49	173	259	49	61
36	1	37	49	259	345	49	61
37	1	25	37	259	345	49	61
38	1	25	37	173	259	49	61
39	1	25	37	87	173	49	61
40	1	25	37	1	87	49	61
41	1	13	25	1	87	49	61
42	1	13	25	87	173	49	61
43	1	13	25	173	259	49	61
44	1	13	25	259	345	49	61
45	1	1	13	259	345	49	61
46	1	1	13	173	259	49	61
47	1	1	13	87	173	49	61
48	1	1	13	1	87	49	61

.
.
.

.
.
.

121	1	61	73	1	87	1	13
122	1	61	73	87	173	1	13
123	1	61	73	173	259	1	13
124	1	61	73	259	345	1	13
125	1	49	61	259	345	1	13
126	1	49	61	173	259	1	13
127	1	49	61	87	173	1	13
128	1	49	61	1	87	1	13
129	1	37	49	1	87	1	13
130	1	37	49	87	173	1	13
131	1	37	49	173	259	1	13
132	1	37	49	259	345	1	13
133	1	25	37	259	345	1	13
134	1	25	37	173	259	1	13
135	1	25	37	87	173	1	13
136	1	25	37	1	87	1	13
137	1	13	25	1	87	1	13
138	1	13	25	87	173	1	13
139	1	13	25	173	259	1	13
140	1	13	25	259	345	1	13
141	1	1	13	259	345	1	13
142	1	1	13	173	259	1	13
143	1	1	13	87	173	1	13
144	1	1	13	1	87	1	13

split-grid basic dimensions are multigridable to ncg = 1

Input points: 1838505
Ouput points: 2117232

Block Splitting and MPI

Notes regarding use:

- IF A LIMITER IS DESIRED, USE IFLIM=4. This will allow for consistent results with block splitting; iflim=3 is not recommended - iflim=4 is basically a correct implementation of iflim=3 for multiple blocks, and should now be viewed as the recommended limiter for any case that needs one.
- Also, for exact consistency between split and unsplit grids, version 5 emulation (i.e. "Install -v5) should not be used. Version 5 (and earlier versions) made an approximation for cell volumes at 1-1 block interfaces that has been eliminated in version 6 in favor of the exact treatment.
- The input file part of the splitter works by first converting the unsplit CFL3D input file to a TLNS3D map file, splitting the TLNS3D map file, then converting the split TLNS3D map file back to a CFL3D input file.

Block Splitting and MPI

Notes (...continued):

- Caveats: The conversions from the CFL3D input file to a TLNS3D map file are not perfect! The user is urged check the resulting split CFL3D input (and patch) files.
  - A useful check before actually splitting the files is to run this splitter with the number of splittings = 0, and the output grid file as null. Running splitter in this way will cause to code to go through the translations, but the "split" files will have the same numbers of blocks, and the "split" grid will not be output.
  - A "diff" or "gdiff" will point to translation-induced differences that should be easier to sort out than when coupled with true splitting. Note that the 2-step process almost always results in a *reordering* of some boundary condition segments.

Running CFL3D in MPI mode

- MPI requires one processor for overhead. For example if a 32 processor cluster is employed, and there are 28 blocks to be computed on 28 processors, then the command line will read:

  mpirun –np 29 cfl3d_mpi < cfl3d.inp &

- You may want to verify the correct procedure for running mpi code on your platform (e.g. some mpp's use **-n** instead of **-np**)

Running CFL3D in MPI mode

- Because version 6 has dynamic memory allocation, there is no ***requirement*** to run precfl3d before you can run cfl3d. However, you may still find it useful to do so in order to assess how much memory will be required to run the case at hand, allowing you to determine whether a particular problem can fit within the memory of the machine, or to determine the appropriate queue in which to submit the job.
- The usage of precfl3d has changed slightly from previous versions: you must now specify the number of processors in addition to the input file, for example:

  precfl3d -np *num_procs* < cfl3d.inp &

  where ***num_procs*** is the total number of processors, including the host. When running on a single processor, that processor is the host, so num_procs=1 will suffice to assess the memory requirements for the sequential version of the code.
- An important reason why you may want to run precfl3d before running the parallel version of the code is that for ***num_procs*** > 1, precfl3d will output an auxiliary file called **ideal_speedup.dat**. This file will list the best possible speedup you could hope to achieve for the current case, using various numbers of compute processors, ranging from 1 to the number of zones in your grid.

Running CFL3D in MPI mode

The BACT case with 144 blocks was run on 24 processors (-np 25). In the 'precfl3d.out' file the following information is contained:

```
BLOCK TO NODE MAPPING
no. of blocks =  288
no. of  nodes =   24
block   node
  1      1
  2      1
  3      2
  4      2
  5      3
  6      3
  7      4
  8      4
  9      5
 10      5
 11       6
 12       6
 13       7
 14       7
     .
     .
     .
265     13
266     13
267     14
268     14
269     15
270     15
271     16
272     16
273     17
274     17
275     18
276     18
277     19
278     19
279     20
280     20
281     21
282     21
283     22
284     22
285     23
286     23
287     24
288     24
```

Running CFL3D in MPI mode

```
                    .
                    .
***************************************************************

  SUMMARY OF STORAGE REQUIREMENTS - W + WK ARRAYS

 sequential version:

     permanent array w   requires  131825665 (words)
     temporary array wk  requires    2681342 (words)
     temporary array iwk requires     187820 (words)

 parallel version, per node:

     permanent array w   requires    5506908 (words)
     temporary array wk  requires    1500235 (words)
     temporary array iwk requires     187820 (words)


>>> Estimate for mwork      (sequential)     =  134507007 <<<

>>> Estimate for mworki     (sequential)     =     187820 <<<

>>> Estimate for mwork  (per node, parallel) =    7007143 <<<

>>> Estimate for mworki (per node, parallel) =     187820 <<<

>>> Parallel code sized for  24 nodes, min. (+host)       <<<

***************************************************************
```

Running CFL3D in MPI mode

In the ‘cfl3d.out’ file the same information is found plus the following contained at the end:

.

.

computational rate by mesh sequence (based on wall time):
iseq 1 181.13 microseconds/cell/time step
        90.56 microseconds/cell/subiteration

timing for complete run - time in seconds

node	user	system	total	wall clock
0	10.15	17.60	27.75	325.00
1	3.64	0.55	4.19	228.00
2	5.37	0.92	6.29	325.00
3	3.90	0.52	4.42	228.00
4	5.36	0.87	6.23	325.00
5	5.85	1.14	6.99	324.00
6	4.54	0.89	5.43	228.00
7	4.38	0.83	5.21	227.00
8	4.03	0.79	4.82	226.00
9	4.31	0.70	5.01	228.00
10	6.08	1.00	7.08	325.00
11	4.40	0.77	5.17	227.00
12	4.19	0.65	4.84	227.00
13	4.20	0.74	4.94	226.00
14	4.42	0.66	5.08	225.00
15	4.25	0.81	5.06	226.00
16	4.35	0.68	5.03	225.00
17	4.08	0.83	4.91	225.00
18	4.22	0.87	5.09	225.00
19	4.35	0.66	5.01	225.00
20	4.17	0.66	4.83	225.00
21	3.78	0.55	4.33	224.00
22	3.59	0.49	4.08	225.00
23	3.58	0.51	4.09	224.00
24	3.40	0.40	3.80	224.00

----

total: 114.59   35.09   149.68

total run (wall) time =   0 hours   3 minutes   44 seconds

memory for cfl3d has been deallocated

Flow Field Visualization

Plot3D output

CFL3D is capable of creating Plot3D files of the grid and flow field. Specification of the region of the flow field for output is found in the following input lines:

```
                    .
                    .
   dt     irest    iflagts     fmax     iunst  cfl_tau
  -2.0      0         0         1.0       0      5.0
 ngrid   nplot3d     nprint    nwrest     ichk       i2d    ntstep      ita
    1          1          1     1000         0         1         1       -2
  ncg       iem  iadvance     iforce   ivisc(i)   ivisc(j)  ivisc(k)
    2         0          0         1         0          0         5
                    .
                    .
plot3d output:
   grid      iptyp      ista      iend      iinc      jsta      jend      jinc   ksta   kend   kinc
      1          0         1         1         1         1       999         1      1    999      1
 movie
     0
```

nplot3d specifies the number of blocks to output

Input *nplot3d* lines

If $nplot3d < 0$, then the Plot3D files are automatically set to include all solid Surfaces (no field points) for 3D cases or all field points for 2D cases

Flow Field Visualization

Plot3D output

.

.

plot3d output:

grid	iptyp	ista	iend	iinc	jsta	jend	jinc	ksta	kend	kinc
1	0	1	1	1	1	999	1	1	999	1

movie
0

Grid - Designated grid number to be output

iptyp = 0 - grid point type – grid file and Q file output

= 1 - cell center type – grid file and Q file output

= 2 - cell center type - grid file and turbulence file output ($ivisc > 1$ only)

> 2 - cell center type – grid file and function file output ($iptype = 3$ – minimum distance to nearest viscous wall or directed distance ($ivisc > 1$ only), $iptype = 4$ – eddy viscosity ($ivisc > 1$ only)

ista, jsta, ksta - starting indices in the *i,j,k* directions

iend,jend,kend - ending indices in the *i,j,k* directions (note that if these values are set higher than *idim, jdim,kdim*, the code will reset them to the block dimensions)

iinc,jinc,kinc - increment in the *i,j,k* directions

Note: Setting $ista = iend = iinc = 0$, etc… is a short hand way of specifying the entire range.

Flow Field Visualization

Movie output

.

.

plot3d output:

grid	iptyp	ista	iend	iinc	jsta	jend	jinc	ksta	kend	kinc
1	0	1	1	1	1	999	1	1	999	1

movie

10

Flag to append Plot3D solution output every 10 time steps

Note that one gird file and one solutions file are generated.

Movie = 0 no output of intermediate solutions (if nplot3d > 0), then a single solution is written at the end of the run.

Movie > 0 output of additional solutions every *movie* iterations (time steps)

Movie < 0 output of the initial flow field at the beginning of the run and output of additional solutions every *movie* iterations (time steps)

Caution: Use with care. Plot3D file will get very large very quickly.

The tool 'moovmaker' will read the plot3D solution and grid file and create a movie for a 2D flow field in which the 3rd dimension will be time. The grid output by 'moovmaker will be called 'g.bin' and the solution file will be called 'q.bin'. Creating these files will allow animating the 3rd dimension (time) to produce a movie of the flow field.

Useful CFL3D Tools

- Get_FD.F
  - This program reads two CFL3D restart files and calculate finite differences of force and moment coefficients; it is used to validate complex-variable approach for determining solution derivatives.
- INGRID_to_p3d.F
  - This program converts PEGSUS 4.x INGRID file to a PLOT3D file that can be used in CFL3D. Note that the INGRID file must correspond to grid points rather than "augmented" cell centers.
- XINTOUT_to_ovrlp.F
  - This program converts the XINTOUT overset grid interpolation file from PEGSUS to the ovrlp.bin file used by CFL3D.
- cfl3d_to_pegbc.F
  - This program creates a peg.bc.raw file for use with PEGSUS 5.x.
- cgns_to_cfl3dinput.F
  - This program reads a CGNS file and creates a PLOT3D-type grid as well as a best-guess for a CFL3D input file.

Useful CFL3D Tools

- everyother_xyz.F
  - This program reads a grid and creates an every-other-point grid. This can be useful in combination with the program v6inpdoubhalf.F, in order to reduce the required CFL3D run-time memory when you are only running on a coarser-level grid (and not taking it up to the finer level(s).
- grid_perturb.F
  - This program generates a real-valued grid (PLOT3D multiblock form) by reading in a real-valued grid (PLOT3D multiblock form) and a corresponding real-valued matrix of grid-sensitivity derivatives (PLOT3D multiblock function file form, with 3*ndv variables for the x,y,z components of the ndv design variables). The code Get_FD.F may be used with the two restart files to determine d(Cl)/d(DV), d(Cd)/d(DV), etc.
- grid_perturb_cmplx.F
  - This program generates a complex-valued grid (PLOT3D multiblock form) by reading in a real-valued grid (PLOT3D multiblock form) and a corresponding real-valued matrix of grid-sensitivity derivatives (PLOT3D multiblock function file form, with 3*ndv variables for the x,y,z components of the ndv design variables). The output grid may be read into the complex version of CFL3D (cfl3dcmplx_mpi or cfl3dcmplx_seq) to determine the solution derivatives with respect to the chosen design variable.

Useful CFL3D Tools

- initialize_field.F
  - This program creates a restart.bin restart file in which you can specify specific initial conditions, region by region. This can be useful when "freestream everywhere" is not a desirable initial condition.
- moovmaker.F
  - This program reads the PLOT3D files output by CFL3D when the MOVIE parameter is used for 2-D datasets (or 3-D datasets surface-only), and creates new PLOT3D files with time as the third (k) direction.
- p3d_to_INGRID.F
  - This program converts either PLOT3D or CFL3D type grids into either INGRID type grids that can be used with PEGSUS 4.x, or PLOT3D type grids that can be used with PEGSUS 5.x. The converted grids can contain either the grid points as given in the input grids, or "augmented" cell centers of the input grids.
- p3d_to_cfl3drst.F
  - This program reads PLOT3D files and creates an approximate restart.bin restart file. This can be useful if: (1) you are given a PLOT3D Q-file from another code, and you wish to use it as a basis for starting CFL3D, or (2) you have lost the CFL3D restart file, but you still have the PLOT3D Q-file.

Useful CFL3D Tools

- plot3dg_to_cgns.F
  - This program reads a PLOT3D grid file and a CFL3D input file and creates a CGNS file (with grid, BC, and 1-to-1 connectivity information in it).
- v6_restart_mod.F
  - This program reads a restart.bin restart file and manipulates it. It can switch between unformatted and formatted (which is useful if you need to transfer the restart file to a machine of different architecture). It can also write out the restart file either the same size, half the size, or double the size. Going to half size is useful if one wishes to restart from a fine grid solution and run on a coarser level. User can choose to coarsen/refine only particular index directions, if desired. The program cannot both coarsen and refine different directions simultaneously.
- v6inpdoubhalf.F
  - This program reads a CFL3D input file and creates a new input file appropriate for a grid of either half or double the size. This can be useful in combination with the program everyother_xyz.F when running on coarser grid levels, and you wish to reduce the run-time memory required.

References

Edwards, J. W., Bennett, R. M., Whitlow Jr., W., Seidel, D. A., "Time-Marching Transonic Flutter Solutions Including Angle-of-Attack Effects," *Journal of Aircraft*, 20 (1983), pp. 899-906.

Lee-Rausch, E. M., Batina, J. T., "Wing flutter boundary prediction using unsteady Euler method," *Journal of Aircraft,* 32 (1995), pp. 416-422.

Krist, S. L., "CFL3D User's Manual (Version 5.0)," NASA/TM-1998-208444, June 1998.

Bartels, R. E., "Mesh Strategies for Accurate Computation of Unsteady Spoiler and Aeroelastic Problems," *Journal of Aircraft*, 37 (2000), pp. 521-525.

CFL3D version 6.0 web site: http://cfl3d.larc.nasa.gov/Cfl3dv6/cfl3dv6.html.

Bartels, R. E., "Finite Macro-Element Mesh Deformation in a Structured Multi-Block Navier-Stokes Code," NASA/TM-2005-213789, July 2005.

Summary

- CFL3D is a general purpose production-level CFD code for fluid dynamics, with many capabilities and options.
- This tutorial has summarized many of the newest features of the code, and also has explained in detail how to set up and run it for general cases.
- Particular focus has been given to CFL3D's upgraded deforming mesh and aeroelastic analysis capabilities.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. REPORT DATE *(DD-MM-YYYY)*
01- 04 - 2006

2. REPORT TYPE
Technical Memorandum

3. DATES COVERED *(From - To)*

4. TITLE AND SUBTITLE
CFL3D Version 6.4—General Usage and Aeroelastic Analysis

5a. CONTRACT NUMBER

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)
Bartels, Robert E.; Rumsey, Christopher L; and Biedron, Robert T.

5d. PROJECT NUMBER

5e. TASK NUMBER

5f. WORK UNIT NUMBER
984754

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
NASA Langley Research Center
Hampton, VA 23681-2199

8. PERFORMING ORGANIZATION REPORT NUMBER
L-19247

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, DC 20546-0001

10. SPONSOR/MONITOR'S ACRONYM(S)
NASA

11. SPONSOR/MONITOR'S REPORT NUMBER(S)
NASA/TM-2006-214301

12. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified - Unlimited
Subject Category 01
Availability: NASA CASI (301) 621-0390

13. SUPPLEMENTARY NOTES
An electronic version can be found at http://ntrs.nasa.gov

14. ABSTRACT

This document contains the course notes on the computational fluid dynamics code CFL3D version 6.4. It is intended to provide from basic to advanced users the information necessary to successfully use the code for a broad range of cases. Much of the course covers capability that has been a part of previous versions of the code, with material compiled from a CFL3D v5.0 manual and from the CFL3D v6 web site prior to the current release. This part of the material is presented to users of the code not familiar with computational fluid dynamics. There is new capability in CFL3D version 6.4 presented here that has not previously been published. There are also outdated features no longer used or recommended in recent releases of the code. The information offered here supersedes earlier manuals and updates outdated usage. Where current usage supersedes older versions, notation of that is made. These course notes also provides hints for usage, code installation and examples not found elsewhere.

15. SUBJECT TERMS
Aeroelastic analysis; CFL3D; CFL3D version 6.4; Computational fluid dynamics code

16. SECURITY CLASSIFICATION OF: a. REPORT	b. ABSTRACT	c. THIS PAGE	17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON / 19b. TELEPHONE NUMBER *(Include area code)*
U	U	U	UU	269	STI Help Desk (email: help@sti.nasa.gov) / (301) 621-0390

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std. Z39.18