NASA STI Program . . . 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 plays a key part in helping NASA maintain this important role.

The NASA STI program operates under the auspices of the Agency Chief Information Officer. It collects, organizes, provides for archiving, and disseminates NASA’s STI. The NASA STI program provides access to the NASA Aeronautics and Space Database and its public interface, the NASA Technical Report Server, thus providing one of the largest collections of aeronautical and space science STI in the world. Results are published in both non-NASA channels and 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 also include organizing and publishing research results, distributing specialized research announcements and feeds, providing information desk and personal search support, and enabling data exchange services.

For more information about the NASA STI program, see the following:

- Access the NASA STI program home page at [http://www.sti.nasa.gov](http://www.sti.nasa.gov)

- E-mail your question to [help@sti.nasa.gov](mailto:help@sti.nasa.gov)

- Fax your question to the NASA STI Information Desk at 443-757-5803

- Phone the NASA STI Information Desk at 443-757-5802

- Write to:
  STI Information Desk
  NASA Center for AeroSpace Information
  7115 Standard Drive
  Hanover, MD 21076-1320
The use of trademarks or names of manufacturers in this report is for accurate reporting and does not constitute an official endorsement, either expressed or implied, of such products or manufacturers by the National Aeronautics and Space Administration.
Abstract

This manual describes the installation and execution of FUN3D version 12.4, including optional dependent packages. FUN3D is a suite of computational fluid dynamics simulation and design tools that uses mixed-element unstructured grids in a large number of formats, including structured multiblock and overset grid systems. A discretely-exact adjoint solver enables efficient gradient-based design and grid adaptation to reduce estimated discretization error. FUN3D is available with and without a reacting, real-gas capability. This generic gas option is available only for those persons that qualify for its beta release status.
Contents

About this Document .................................................. 9
Acknowledgments ......................................................... 10
Quick Start ............................................................... 11

1 Introduction .......................................................... 15
  1.1 Primary Capabilities and Features .......................... 15
  1.2 Requirements ................................................... 16
  1.3 Grid Generation ................................................. 16
  1.4 Obtaining Fun3D ............................................... 17

2 Conventions .......................................................... 18
  2.1 Compressible Equations ...................................... 19
  2.2 Incompressible Equations .................................... 21
  2.3 Generic Gas Equations ....................................... 22
  2.4 Unsteady Flows ................................................ 23

3 Boundary Conditions ................................................ 24

4 Grids ........................................................................ 26
  4.1 File Endianness ................................................... 26
  4.2 Supported Grid Formats ....................................... 26
    4.2.1 AFLR3 Grids ................................................ 27
    4.2.2 FAST Grids .................................................. 27
    4.2.3 VGRID Grids ................................................. 28
    4.2.4 FieldView Grids .......................................... 28
    4.2.5 FELISA Grids ............................................... 29
    4.2.6 Fun2D Grids ................................................. 29
  4.3 Translation of Additional Grid Formats ....................... 30
    4.3.1 PLOT3D Grids .............................................. 30
    4.3.2 CGNS Grids ................................................ 31

5 Flow Solver, nodet .................................................... 33
  5.1 Flow Solver Execution .......................................... 33
  5.2 Command Line Options ......................................... 33
  5.3 Output Files ..................................................... 34
    5.3.1 Flow Visualization ....................................... 34
    5.3.2 Flow Visualization Output From Existing Solution ... 35
6 Adjoint Solver, DUAL 36
6.1 Convergence of the Linear Adjoint Equations 36
6.2 Required Directory Hierarchy and Executing DUAL 36
6.3 rubber.data 37
6.4 Output Files 38

7 Grid Adaptation 39
7.1 Geometry Specification and Grid Freezing for REFINE 39
7.1.1 No geometry, where the surface nodes are frozen 39
7.1.2 FAUXGeom for Planar Boundaries 40
7.2 Performing Feature-Based Adaptation 40
7.3 Performing Adjoint-Based Adaptation 40
7.4 Scripting Grid Adaptation 41
7.4.1 Input File case specifics for f3d Script 41

8 Design Optimization 44
8.1 Objective/Constraint Functions 45
8.1.1 Terminology 45
8.1.2 Functional Form 46
8.2 Some Details on Specific Objective/Constraint Functions 48
8.2.1 Lift-to-Drag Ratio (Keyword: c1cd) 48
8.2.2 Rotorcraft Figure of Merit (Keyword: fom) 48
8.2.3 Rotorcraft Propulsive Efficiency (Keyword: propeff) 49
8.2.4 RMS of Stagnation Pressure (Keyword: pstag) 49
8.2.5 Near-field Target p/p∞ (Keyword: boom targ) 49
8.2.6 Coupled sBOOM Ground-Based Signatures, Noise Metrics, and Equivalent Areas (Keyword: sboom) 51
8.2.7 Supersonic Mach Cut Equivalent Area Distribution (Keyword: ae) 52
8.2.8 Target Pressure Distributions (Keyword: cpstar) 53
8.3 Geometry Parameterizations 53
8.3.1 Surface Grid Extraction 54
8.3.2 Access to Executables 54
8.3.3 Notes on Using Sculptor™ 55
8.3.4 Using Other Parameterization Packages 55
8.4 Design Optimization Directory Structure 56
8.5 Contents of the ammo Directory 56
8.5.1 ammo.input 57
8.6 Contents of the description.i Directory 59
8.6.1 Geometry Parameterization Files 60
8.6.2 rubber.data 62
8.6.3 ae_target.dat (optional) 68
8.6.4 body_grouping.data (optional) ........................................... 68
8.6.5 command_line.options (optional) ........................................ 69
8.6.6 cpstar.data.j (optional) .................................................. 70
8.6.7 machinefile (optional) .................................................... 70
8.6.8 pressure_target.dat (optional) .......................................... 70
8.6.9 transforms.i (optional) .................................................... 70
8.7 Contents of the model.i Directory ......................................... 70
8.8 Running the Optimization .................................................... 71
8.8.1 Filesystem Latency Problems ............................................ 72
8.9 Multi-objective Design ....................................................... 72
8.9.1 KSOPT ........................................................................... 73
8.9.2 PORT, SNOPT™ .............................................................. 73
8.10 Multi-point Design ............................................................. 73
8.10.1 Linear Combination of Objective Functions ......................... 74
8.10.2 Combination of Objective Functions using the Kreisselmeier-
Steinhauser Function ................................................................. 75
8.10.3 Single-Point Objective Function with Off-Design Con-
straint Functions .................................................................. 75
8.11 Optimization of Two-Dimensional Geometries ......................... 75
8.12 Using a Different Optimization Package .................................. 76
8.13 Implementing New Cost Functions/Constraints ......................... 77
8.14 Forward Mode Differentiation Using Complex
Variables .................................................................................. 77

References ................................................................................. 79

A Installation ............................................................................. 85
A.1 Extracting Files ................................................................... 85
A.2 Configure Introduction ....................................................... 85
A.3 Alternative Installation Path ................................................ 86
A.4 Fortran Compiler Option Tuning (FTune) ............................... 86
A.5 Complex Variable Version ................................................... 87
A.6 Internal Libraries ............................................................... 87
A.6.1 KNIFE ............................................................................ 87
A.6.2 REFINE .......................................................................... 87
A.7 External Libraries ............................................................. 87
A.7.1 MPI ................................................................................. 88
A.7.2 ParMETIS ......................................................................... 89
A.7.3 SUGGAR++1.0.10 or Higher .............................................. 90
A.7.4 DiRTlib v1.40 or higher .................................................... 90
A.7.5 6-DOF ............................................................................ 91
A.7.6 KSOPT ............................................................................ 91
A.7.7 PORT ............................................................................. 91
A.7.8 SNOPT™ ........................................... 91
A.7.9 NPSOL™ ............................................ 92
A.7.10 DOT/BIGDOT™ .................................... 92
A.7.11 Tecplot™ .......................................... 92
A.7.12 CGNS ............................................. 93
A.7.13 sBOOM ........................................... 93

B Fun3D Input Files ........................................... 94
B.1 stop.dat ............................................. 94
B.2 [project_rootname].flow .............................. 95
B.3 remove_boundaries_from_force_totals .............. 95
B.4 fun3d.nml ............................................ 95
  B.4.1 &project ........................................ 97
  B.4.2 &raw_grid ....................................... 98
  B.4.3 &force_moment_integ_properties ................. 100
  B.4.4 &governing_equations ............................ 101
  B.4.5 &reference_physical_properties ................ 104
  B.4.6 &inviscid_flux_method ........................... 106
  B.4.7 &turbulent_diffusion_models .................... 110
  B.4.8 &spalart ........................................ 112
  B.4.9 &code_run_control ............................... 113
  B.4.10 &nonlinear_solver_parameters .................. 116
  B.4.11 &linear_solver_parameters ..................... 119
  B.4.12 &boundary_conditions ......................... 120
  B.4.13 &component_parameters ......................... 133
  B.4.14 &two_d_trans .................................. 135
  B.4.15 &three_d_trans ................................ 137
  B.4.16 &special_parameters ............................ 138
  B.4.17 &flow_initialization ............................ 139
  B.4.18 &time_avg_params ............................... 141
  B.4.19 &global ....................................... 142
  B.4.20 &volume_output_variables ...................... 144
  B.4.21 &boundary_output_variables .................... 154
  B.4.22 &sampling_output_variables .................... 166
  B.4.23 &sampling_parameters ........................... 177
  B.4.24 &slice_data ..................................... 186
  B.4.25 &massoud_output ................................ 196
  B.4.26 &overset_data ................................... 199
  B.4.27 &rotor_data ..................................... 201
  B.4.28 &adapt_metric_construction .................... 202
  B.4.29 &adapt_mechanics ............................... 207
  B.4.30 &sonicBoom ..................................... 212
  B.4.31 &sboom .......................................... 214
B.4.32 &equivalent_area ........................................ 222
B.4.33 &noninertial_reference_frame .......................... 223
B.5 moving_body.input ........................................ 225
  B.5.1 &body_definitions ...................................... 226
  B.5.2 &forced_motion ......................................... 230
  B.5.3 &observer_motion ....................................... 234
  B.5.4 &motion_from_file ..................................... 237
  B.5.5 &surface_motion_from_file .............................. 239
  B.5.6 &sixdof_motion ......................................... 240
  B.5.7 &aeroelastic_modal_data ................................. 243
  B.5.8 &composite_overset_mesh ............................... 246
  B.5.9 &grid_transform ........................................ 247
B.6 rotor.input ................................................ 249
  B.6.1 Header .................................................. 251
  B.6.2 Actuator Surface Model ................................. 251
  B.6.3 Rotor Reference System ................................. 252
  B.6.4 Rotor Loading .......................................... 253
  B.6.5 Blade Parameters ........................................ 254
  B.6.6 Blade Element Parameters for Load Type=3 ......... 254
  B.6.7 Pitch Control Parameters for Load Type=3 .......... 255
  B.6.8 Prescribed Flap Parameters ............................ 256
  B.6.9 Prescribed Lag Parameters ............................. 256
B.7 tdata ...................................................... 257
  B.7.1 perfect_gas Keyword .................................... 257
  B.7.2 equilibrium_air_t Keyword ............................. 257
  B.7.3 equilibrium_air_r Keyword ............................. 258
  B.7.4 one Keyword ............................................ 258
  B.7.5 two Keyword ............................................ 258
  B.7.6 FEM Keyword ............................................ 259
B.8 species thermo_data ....................................... 259
B.9 kinetic_data ............................................... 262
B.10 species transp_data ...................................... 263
B.11 species transp_data_0 .................................... 264
B.12 hara namelist data ........................................ 265

C Troubleshooting .............................................. 272
  C.1 What if the solver has trouble starting or reports NaNs? ... 272
  C.2 What if the forces and moments aren’t steady or residuals don’t converge to steady-state? ..................... 272
  C.3 What if the solver dies unexpectedly? ..................... 273
  C.4 What if a segmentation fault occurs after “Calling ParMetis”? 273
C.5 What if the solver dies with an error like “input statement requires too much data” after echoing the wrong number of elements or nodes? ................................................................. 273
C.6 What if the solver dies with an error like “input statement requires too much data” after echoing the number of tetrahedra and nodes for a VGRID mesh? ................................. 273
C.7 What if the solver reports that the Euler numbers differ? ... 273
   C.7.1 Euler Number Description ............................................. 274
   C.7.2 Determining the Impact of an Euler Number Mismatch 275
About this Document

This manual is intended to guide an application engineer through configuration, compiling, installing, and executing the Fun3D simulation package. The focus is on the most commonly exercised capabilities. Therefore, some of the immature or rarely exercised capabilities are intentionally omitted in the interest of clarity. An accompanying document that provides example cases is under development.

Release of the generic gas capability is restricted because of International Traffic in Arms Regulations (ITAR), so Fun3D usually distributed with the generic gas capability disabled. See section 1.4 for details. This manual describes Fun3D with and without the generic gas capability, denoted eqn\_type= ’generic’. Features that are specific to an eqn\_type are explicitly indicated.

This document is updated and released with each subsequent version of Fun3D. In fact, a significant portion is automatically extracted from the Fun3D source code. If you have difficulties, find any errors, or have any suggestions for improvement please contact the authors at

Fun3D-Support@lists.nasa.gov

We would like to hear from you.
Acknowledgments

A large portion of this content originated as a web-based manual, which contained contributions from Eric Nielsen, Elizabeth Lee-Rausch, Jan-Renee Carlson, Karen Bibb, Bill Jones, Jeff White, Eric Lynch, Dave O’Brien, Marilyn Smith, and Clara Helm. Sriram Rallabhandi provided the description of sBOOM. The namelist documentation is parsed from FUN3D source code, which is written by the developers. Contributors to FUN3D include Ponnam-palam (Bala) Balakumar, Karen Bibb, Bob Biedron, Jan Carlson, Mark Carpenter, Peter Gnooffo, Dana Hammond, Bill Jones, Bil Kleb, Beth Lee-Rausch, Eric Nielsen, Hiroaki Nishikawa, Mike Park, Chris Rumsey, Jim Thomas, Veer Vatsa, Jeff White, Alejandro Campos, Rajiv Shenoy, Marilyn Smith, Joe Derlaga, Natalia Alexandrov, W. Kyle Anderson, Harold Atkins, Bill Wood, Austen Duffy, Clara Helm, Chris Cordell, Kyle Thompson, Hicham Alkandry, Shelly Jiang, Eric Lynch, Jennifer Abras, Nicholas Burgess, Dave O’Brien, Tommy Lambert, Ved Vyas, Shatra Reehal, Kan Yang, Andrew Sweeney, Brad Neff, Genny Pang, Gregory Bluvshteyn, Dan Gerstenhaber, Geoff Parsons, and Rena Rudavsky.
Quick Start

This section takes you from source code tarball to a rudimentary flow solution using single processor execution on a typical Unix-style environment (e.g. Linux, Mac® OS) with a Fortran compiler and the GNU Make utility. FUN3D is most commonly executed in parallel, but the intent here is to provide the most basic installation, setup, and execution of the FUN3D flow solver without the complexity of any third-party libraries or packages.

See section 1.4 for instructions on obtaining the FUN3D source code tarball. Once you have it, unpack the source code tarball, configure it for your system (section A), compile it, and add the executables directory to your search path. For C Shell, e.g.,

```
tar zxf fun3d-12.4.tar.gz
cd fun3d-12.4
mkdir _seq
cd _seq
./configure --prefix=${PWD}
make install
setenv PATH ${PWD}/bin:${PATH}
cd ..
```

For Bourne Shell, the `setenv` command is `export PATH=${PWD}/bin:${PATH}`. The change to the `PATH` environment variable can be made permanently by adding the `setenv` or `export` command to your shell start up file. Next, move to the `doc/quick_start` directory,

```
  cd doc/quick_start
```

where you will find a very coarse 3D wing grid (`inv_wing.fgrid`) intended for inviscid flow simulation (section 4). Also in this directory are the associated boundary conditions file `inv_wing.mapbc` (section 3) and a FUN3D input file `fun3d.nml` in Fortran namelist format (section B.4).

Execute the flow solver (section 5.1) by running the command `nodet` This should produce screen output similar to

```
1  FUN3D 12.4-70107 Flow started 03/31/2014 at 10:33:12 with 1 processes
2  Contents of fun3d.nml file below------------------------
3  &project
4    project_rootname = 'inv_wing'
5  /
6  &raw_grid
7    grid_format = 'fast'
8    data_format = 'ascii'
9  /
10  &governing_equations
11    viscous_terms = 'inviscid'
```
If Fun3D completed successfully, a Mach 0.7 inviscid flow over a very coarse representation of an ONERA M6 semi-span wing [1] at two degrees angle of attack is available. If not, please refer to Troubleshooting on page 272.

With visualization software capable of reading Tecplot™ files, you can visualize various surface quantities with inv_wing.tec_boundary.dat as shown by the pressure contours in Fig. 1. Iterative convergence history can be plotted from inv_wing.hist.dat as shown in Fig. 2. Histories of all five conservation equation residual norms are denoted $R_1$-$R_5$, and the lift coefficient convergence history is denoted $C_L$. 

```plaintext

12 / 
13 $reference_physical_properties
14       mach_number  = 0.7
15 / 
16 $code_run_control
17       restart_read   = 'off'
18       steps           = 150
19       stopping_tolerance = 1.0e-12
20 / 
21 $global
22       boundary_animation_freq = -1
23 / 
24 Contents of fun3d.nml file above------------------------
25 rotor.input not found
26 moving_body.input not found
27 moving_body.input not found
28 ... opening inv_wing.fgrid
29 ... nnodesg: 6309 ntet: 35880 ntface: 1392
30 ... cell statistics: type, min volume, max volume, max face angle
31 cell statistics: tet, 0.38305628E-05, 0.14174467E+02, 143.526944837
32 cell statistics: all, 0.38305628E-05, 0.14174467E+02, 143.526944837
33 ... Constructing partition node sets for level-0...  35880 T
34 ... Edge Partitioning ....
35 ... Boundary partitioning....
36 ... Reordering for cache efficiency....
37 ... Write global grid information to inv_wing.grid_info
38 ... Time after preprocess TIME/Mem(MB):  0.71  100.15  100.15
39
40 Writing boundary output: inv_wing.tec_boundary.dat
41 Time step: 79, ntt: 79, Prior iterations: 0
42 Writing inv_wing.flow (version 11.8) lmpi_io 2
43 inserting current history iterations 79
44 Time for write: 0.0 s
45 Done.

```

197 Lift 0.809996568758631E-01 Drag 0.107734796873048E-01
198  77  0.347515633256081E-12  0.16460E-10  0.26803E+01  0.00000E+00 -0.62327E+00
199 Lift 0.809996568770187E-01 Drag 0.107734796873170E-01
200  78  0.264078581490539E-12  0.12961E-10  0.26803E+01  0.00000E+00 -0.62327E+00
201 Lift 0.809996568779106E-01 Drag 0.107734796873263E-01
202  79  0.201939553281957E-12  0.10194E-10  0.26803E+01  0.00000E+00 -0.62327E+00
203 Lift 0.809996568786974E-01 Drag 0.107734796873335E-01
204
205 Writing boundary output: inv_wing.tec_boundary.dat
206 Time step: 79, ntt: 79, Prior iterations: 0
207
208 Writing inv_wing.flow (version 11.8) lmpi_io 2
209 inserting current history iterations 79
210 Time for write: 0.0 s
211 Done.

If Fun3D completed successfully, a Mach 0.7 inviscid flow over a very coarse representation of an ONERA M6 semi-span wing [1] at two degrees angle of attack is available. If not, please refer to Troubleshooting on page 272. 

With visualization software capable of reading Tecplot™ files, you can visualize various surface quantities with inv_wing.tec_boundary.dat as shown by the pressure contours in Fig. 1. Iterative convergence history can be plotted from inv_wing.hist.dat as shown in Fig. 2. Histories of all five conservation equation residual norms are denoted $R_1$-$R_5$, and the lift coefficient convergence history is denoted $C_L$. 

```
Figure 1: Mach 0.7 flow about a coarse ONERA M6 semi-span wing at 2 degrees angle of attack.
Figure 2: Iterative convergence history for coarse ONERA M6 wing.
1 Introduction

Fun3D began as a research code in the late 1980s. [2] The code was created to develop new algorithms for unstructured-grid fluid dynamic simulations of incompressible and compressible transonic flows. The project has since grown into a suite of codes that cover not only flow analysis, but adjoint-based error estimation, mesh adaptation, and design optimization of fluid dynamic problems extending into the hypersonic regime. [3]

Fun3D is currently used as a production flow analysis and design tool to support NASA programs. Continued research efforts have also benefited by the improvements to stability, ease of use, portability, and performance that this shift to simultaneous support of development and production environments has required. These benefits also include the rapid evaluation of new techniques on realistic simulations and a rapid maturation of experimental techniques to production-level capabilities.

1.1 Primary Capabilities and Features

The primary capabilities of Fun3D are:

- Parallel domain decomposition with Message Passing Interface (MPI) communication for distributed computing
- Two-dimensional (2D) and Three-dimensional (3D) node-based, finite-volume discretization
- Thermodynamic models: perfect gas (compressible and incompressible) and thermochemical equilibrium, and non-equilibrium
- Time-accurate options from first- to fourth-order with temporal error controllers
- Upwind flux functions: flux difference splitting, flux vector splitting, artificially upstream flux vector splitting, Harten-Lax-van Leer contact, low dissipation flux splitting scheme, and others
- Turbulence models: Spalart-Allmaras, Menter k-omega SST, Wilcox k-omega, detached eddy simulation, and others, including specified or predicted transition
- Multigrid with implicit time stepping where the linear system is solved using either point-implicit, line-implicit, or Newton-Krylov
- Propulsion simulation including inlets, nozzles, and system performance

\[1^\text{The multi-species, thermochemical non-equilibrium capability requires the high-energy physics library, which is only made available upon specific request and under certain conditions, see section 1.4 for details.}\]
- Grid motion: time-varying translation, rotation, and deformation including overset meshes and six degrees of freedom trajectory computations
- Adjoint- and feature-based grid adaptation
- Gradient based sensitivity analysis and design optimization via hand-coded discrete adjoint for reverse mode differentiation and automated complex variables for forward mode differentiation

Before exploring more advanced applications (e.g., grid adaptation, moving grids, overset grids, design optimization), the user should become familiar with Fun3D’s basic flow solving capabilities and have appropriate computational capability available as indicated in the next section.

1.2 Requirements

The Fun3D development team’s typical computing platform is Linux clusters; so this is the most thoroughly tested environment for the software. A number of users also run on other UNIX-like environments including Mac OS X™; these platforms are supported as well. Users have also run on other architectures such as Microsoft Windows™-based PC’s; however, the team cannot provide explicit support for these environments.

The user will need GNU Make and a Fortran compiler that supports at least the Fortran 95 standard. During configuration, the Fortran compiler is tested, and any newer Fortran features or extensions are detected are used to the greatest extent possible. A large number of compilers are tested by an automated build framework, including Intel®, Portland Group®, NAG®, Lahey/Fujitsu®, Cray®, Absoft®, IBM®, GFortran, and G95.

While the code can be compiled to run on only a single processor, as demonstrated in the Quick Start section, most applications will require compiling against an MPI implementation and the ParMETIS domain decomposition library to allow parallel execution.

The flow solver uses approximately 2.4 kilobytes of memory per grid point for a perfect gas RANS simulation with a loosely-coupled turbulence model. For example, a grid with one million mesh points would require approximately 2.4 gigabytes of memory. Memory usage will increase slightly with the increase in the number of processors because of the increasing boundary data exchanged. Different solution algorithms and co-visualization options will also require additional memory. Typically, one CPU core per 50,000 grid points is suggested, where a 3D mesh of 20 million grid points would require 400 cores.

1.3 Grid Generation

Fun3D has no grid generation capability. For internal development at NASA, the most common sources of 3D grids are VGRID (ViGYAN, Inc. and NASA
Langley), SolidMesh/AFLR3 (Mississippi State), Pointwise (Pointwise, Inc.), and GridEx (NASA Langley).

For 2D grids, the development team normally uses the AFLR2 software written by Dave Marcum et al. at Mississippi State University. Scripts are available to facilitate the use of this grid generator, but the generator itself must be obtained from Marcum. BAMG is also used for 2D grid generation and adaptation.

1.4 Obtaining Fun3D

Fun3D is export restricted and can only be given to a “U.S. Person,” which is a citizen of the United States, a lawful permanent resident alien of the U.S., or someone in the U.S. as a protected political asylee or under amnesty. The word “person” includes U.S. organizations and entities, such as companies or universities, see 22 CFR §120.15 for the full legal definition. Release of the high-energy, real-gas capability is further restricted because of International Traffic in Arms Regulations (ITAR).

To request the Fun3D software suite, which will include the REFINE grid adaptation and mesh untangling library and the KNIFE cut-cell library, please use the website request form available at http://fun3d.larc.nasa.gov/chapter-1.html#request_fun3d or send an email to James.W.Godsey@nasa.gov containing the following information:

- “U.S. person” to put on agreement form, i.e., an institution or individual
- Point of contact (if “U.S. Person” is not an individual)
- Point of contact email address
- Phone number, extension
- FAX number (if available)
- Address (PO boxes not allowed)
- Proposed application\(^2\) (optional)
- How did you discover Fun3D? (optional)

After some background checks to verify that you qualify as a “U.S. Person,” you will be sent a software usage agreement form. Once a completed usage agreement form is received and the Fun3D support team is notified, the Fun3D support team will make arrangements for transfer of the Fun3D software suite.

\(^2\)The high-energy physics library that allows multiple species and non-equilibrium chemistry are only included upon specific request—be sure to note that you desire access to this beta functionality as part of your application.
2 Conventions

This chapter discusses the coordinate system orientation and nondimensionalization used by FUN3D. The nomenclature for this section is

\[ a = \text{Speed of sound} \]
\[ C = \text{Sutherland constant} \]
\[ e = \text{Energy per unit mass} \]
\[ f = \text{Frequency} \]
\[ h = \text{Enthalpy per unit mass} \]
\[ k = \text{Thermal conductivity} \]
\[ L = \text{Length} \]
\[ M = \text{Mach number} \]
\[ p = \text{Pressure} \]
\[ R = \text{Gas constant} \]
\[ Re = \text{Reynolds number} \]
\[ t = \text{Time} \]
\[ T = \text{Temperature} \]
\[ u, v, w = \text{Cartesian components of velocity} \]
\[ x, y, z = \text{Cartesian directions} \]
\[ \alpha = \text{Angle of attack} \]
\[ \beta = \text{Angle of sideslip} \]
\[ \gamma = \text{Heat capacity ratio} \]
\[ \mu = \text{Viscosity} \]
\[ \rho = \text{Density} \]

where an asterisk (*) denotes a dimensional quantity. A subscript \( \text{ref} \) denotes a reference quantity. For fluid variables, such as pressure, \( \text{ref} \) usually corresponds to the value ‘at \( \infty \)’ for external flows or another condition for internal flows. The units of various reference quantities must be consistent. For example, if the reference speed of sound is defined in feet/sec, then the dimensional reference length, \( L_{\text{ref}}^* \), must be in feet. In what follows, \( L_{\text{ref}}^* \) is the length in the grid that corresponds to the dimensional reference length; \( L_{\text{ref}} \) is considered dimensionless.

FUN3D’s angle of attack, sideslip angle, and associated force coefficients are based on a body-fixed coordinate system:

- positive \( x \) is toward the back of the vehicle;
- positive \( y \) is toward the right of the vehicle; and
• positive $z$ is upward

as shown in Fig. 3. This differs from the standard wind coordinate system by a 180 degree rotation about the $y$ axis. The $\alpha$ and $\beta$ flow angle conventions are shown in Fig. 4.

Figure 3: Fun3D body coordinate system.

![Fun3D body coordinate system](image)

Figure 4: Fun3D freestream flow angle definition.

```
\begin{align*}
  x &= x^*/(L^*_{ref}/L_{ref}) \\
  y &= y^*/(L^*_{ref}/L_{ref}) \\
  z &= z^*/(L^*_{ref}/L_{ref}) \\
  t &= t^*a^*_{ref}/(L^*_{ref}/L_{ref}) \\
  \rho &= \rho^*/\rho^*_{ref} \\
\end{align*}
```

$\rho_{ref} = 1$
\[ |V| = |V^*|/a_{ref}^* \quad |V|_{ref} = M_{ref} \]
\[ u = u^*/a_{ref}^* \quad u_{ref} = M_{ref} \cos \alpha \cos \beta \]
\[ v = v^*/a_{ref}^* \quad v_{ref} = -M_{ref} \sin \beta \]
\[ w = w^*/a_{ref}^* \quad w_{ref} = M_{ref} \sin \alpha \cos \beta \]
\[ p = p^*/(\rho_{ref}^* a_{ref}^* \alpha^2) \quad p_{ref} = 1/\gamma \]
\[ a = a^*/a_{ref}^* \quad a_{ref} = 1 \]
\[ T = T^*/T_{ref}^* \quad T_{ref} = 1 \]
\[ e = e^*/(\rho_{ref}^* a_{ref}^* \alpha^2) \quad e_{ref} = 1/(\gamma(\gamma - 1)) + M_{ref}^2/2 \]

To see how the nondimensional Navier-Stokes equations that are solved in FUN3D are obtained from their dimensional counterparts, it is sufficient to look at the unsteady, one-dimensional equations for conservation of mass, momentum, and energy:

\[ \frac{\partial \rho^*}{\partial t^*} + \frac{\partial (\rho^* u^*)}{\partial x^*} = 0 \]
\[ \frac{\partial (\rho^* u^*)}{\partial t^*} + \frac{\partial}{\partial x^*} \left[ \rho^* u^* u^* + p^* - \frac{4}{3} \mu^* \frac{\partial u^*}{\partial x^*} \right] = 0 \]
\[ \frac{\partial e^*}{\partial t^*} + \frac{\partial}{\partial x^*} \left[ (e^* + p^*) u^* - \frac{4}{3} \mu^* u^* \frac{\partial u^*}{\partial x^*} - k^* \frac{\partial T^*}{\partial x^*} \right] = 0 \]

where \( k^* \) is the thermal conductivity. For a thermally and calorically perfect gas, we also have the equation of state, the definition of the speed of sound, and the specific heat relation:

\[ T^* = \frac{p^*}{\rho^* R^*} \]
\[ a^* = \sqrt{\gamma R^* T^*} \quad (\gamma = c_{p^*}/c_{v^*}) \]
\[ c_{p^*} + c_{v^*} = R^* \quad R^*/c_{p^*} = (\gamma - 1)/\gamma \]

The laminar viscosity is related to the temperature via Sutherland’s law

\[ \mu^* = \mu_{ref}^* \frac{T_{ref}^*}{T^*} + C^* \left( \frac{T^*}{T_{ref}^*} \right)^{3/2} \]

where \( C^* = 198.6^\circ R \) for air.

Substitution of the nondimensional variables defined above into the equation of state and the definition of the speed of sound gives:

\[ T = \frac{\gamma p}{\rho} = a^2 \]
Sutherland’s law in nondimensional terms is given by
\[ \mu = \frac{1 + C}{T + C} T^{3/2} \]
where \( C = 198.6^\circ R/T^*_{ref} \) and where \( T^*_{ref} \) is in degrees Rankine.

Substitution of the dimensionless variables into the conservation equations gives, after some rearrangement,
\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} = 0
\]
\[
\frac{\partial (\rho u)}{\partial t} + \frac{\partial}{\partial x} \left[ \rho u^2 + p - \frac{4}{3} \frac{M_{ref}}{Re_{Lref}} \mu \frac{\partial u}{\partial x} \right] = 0
\]
\[
\frac{\partial e}{\partial t} + \frac{\partial}{\partial x} \left[ (e + p) u - \frac{4}{3} \frac{M_{ref}}{Re_{Lref}} \mu u \frac{\partial u}{\partial x} - \frac{M_{ref}}{Re_{Lref} P_r (\gamma - 1)} \mu \frac{\partial T}{\partial x} \right] = 0
\]
where \( P_r \) is the Prandtl number (generally assumed to be 0.72 for air)
\[ P_r = \frac{c_p \mu^*}{k^*} \]
and where \( Re_{Lref} \), the Reynolds number per unit length in the grid, corresponds to the input variable `reynolds_number` in the `fun3d.nml` file. \( Re_{Lref} \) is related to the Reynolds number characterizing the physical problem, \( Re_{L^*_{ref}} \) by
\[ Re_{L_{ref}} = \frac{\rho^*_{ref} |V^*|_{ref} (L^*_{ref}/L_{ref})}{\mu^*_{ref}} = \frac{\rho^*_{ref} |V^*|_{ref} L^*_{ref} \mu^*_{ref}}{L_{ref}} = \frac{Re_{L^*_{ref}}}{L_{ref}} \]

### 2.2 Incompressible Equations

\[
\begin{align*}
x &= x^* / (L^*_{ref}/L_{ref}) \\
y &= y^* / (L^*_{ref}/L_{ref}) \\
z &= z^* / (L^*_{ref}/L_{ref}) \\
t &= t^* |V^*|_{ref} / (L^*_{ref}/L_{ref})
\end{align*}
\]

\[
\begin{align*}
|V| &= |V^*| / |V^*|_{ref} \\
u &= u^* / |V^*|_{ref} & u_{ref} &= \cos \alpha \cos \beta \\
v &= v^* / |V^*|_{ref} & v_{ref} &= -\sin \beta \\
w &= w^* / |V^*|_{ref} & w_{ref} &= \sin \alpha \cos \beta \\
p &= p^* / (\rho^*_{ref} |V^*|_{ref}^2) & p_{ref} &= 1
\end{align*}
\]
For incompressible flows, Fun3D does not model any heat sources. The temperature $T^*$ is constant and so is the viscosity $\mu^*$. After dividing through by a constant reference density, the one-dimensional continuity and momentum equations are:

$$\frac{\partial u^*}{\partial x^*} = 0$$

$$\frac{\partial u^*}{\partial t^*} + \frac{\partial}{\partial x^*} \left[ u^* \frac{p^*}{\rho^*_{ref}} - \frac{4}{3} \frac{\mu^*_{ref}}{\rho^*_{ref}} \frac{\rho^*}{\rho^*_{ref}} \right] = 0$$

The fundamental difference between the nondimensionalization of the compressible equations and the incompressible equations is that the sound speed is used in the former and the flow speed in the latter. Substitution of the dimensionless variables defined above into the conservation equations gives, after some rearrangement,

$$\frac{\partial u}{\partial x} = 0$$

$$\frac{\partial u}{\partial t} + \frac{\partial}{\partial x} \left[ u^2 + p - \frac{4}{3} \frac{1}{Re_{L_{ref}}} \frac{\rho^*_{ref}}{\rho^*} \frac{\mu^*_{ref}}{\rho^*_{ref}} \right] = 0$$

where, exactly the same as in the compressible-flow path, the Reynolds number per unit length in the grid is

$$Re_{L_{ref}} = \frac{\rho^*_{ref} |V^*|_{ref} L^*_{ref}}{\mu^*_{ref}} = \frac{Re^*_{L_{ref}}}{L_{ref}}$$

### 2.3 Generic Gas Equations

The generic gas path requires all reference quantities (velocity, density, temperature) be entered in the meter-kilogram-second (MKS) system. The transport property nondimensionalization includes the effects of rescaling using the grid length conversion factor. The nondimensionalization of other flow variables follows the practice used to derive the Mach number independence principle. Neither Mach number nor Reynolds number can be used to define reference conditions; these are derived from the fundamental reference quantities. The derived Reynolds number is relative to a one meter reference length. Temperature is never non-dimensionalized; it always appears in units of degrees Kelvin.

\[
\begin{align*}
\rho &= \frac{\rho^*}{\rho^*_{ref}} & \rho^*_{ref} \quad [\text{kg/m}^3] \\
u &= \frac{u^*}{V^*_{ref}} & V^*_{ref} \quad [\text{m/s}] \\
v &= \frac{v^*}{V^*_{ref}} & T^*_{ref} \quad [\text{K}] \\
w &= \frac{w^*}{V^*_{ref}} & L^*_{ref} \quad [\text{m}]
\end{align*}
\]
\[ a = \frac{a^*}{V_{ref}^*} \]
\[ p = \frac{p^*}{(\rho_{ref}^* V_{ref}^2)} \]
\[ e = \frac{e^*}{V_{ref}^2} \]
\[ h = \frac{h^*}{V_{ref}^2} \]
\[ \mu = \frac{\mu^* (T^*)}{\rho_{ref}^* V_{ref}^2 L_{ref}^*} \]

### 2.4 Unsteady Flows

One of the challenges in unsteady flow simulation is determining the non-dimensional time step \( \Delta t \). The number of time steps at that \( \Delta t \) necessary to resolve the lowest frequency of interest will impact the cost of the simulation and too large a \( \Delta t \) will corrupt the results with temporal errors. Time is non-dimensionalized within FUN3D by

\[
    t = \frac{t^* a_{ref}^*}{(L_{ref}^*/L_{ref})} \quad \text{(compressible)}
\]
\[
    t = \frac{t^* |V_{ref}^*|}{(L_{ref}^*/L_{ref})} \quad \text{(incompressible)}
\]

where, as in the previous sections, quantities denoted with * are dimensional.

In all unsteady flows, one or more characteristic times \( t_{chr}^* \) may be identified. In a flow with a known natural frequency of oscillation (e.g., vortex shedding from a cylinder), or in situations where a forced oscillation is imposed (e.g., a pitching wing), a dominant characteristic time is readily apparent. In such cases, if the characteristic frequency in Hz (cycles/sec) is \( f_{chr}^* \), then

\[
    t_{chr}^* = \frac{1}{f_{chr}^*}
\]

In other situations, no oscillatory frequency may be apparent (or not known a priori). In such cases, the time scale associated with the time it takes for a fluid particle (traveling at a nominal speed of \( |V_{ref}^*| \)) to pass the body of reference length \( L_{ref}^* \) is often used:

\[
    t_{chr}^* = \frac{L_{ref}^*}{|V_{ref}^*|}
\]

The corresponding nondimensional characteristic time is therefore given by:

\[
    t_{chr} = \frac{t_{chr}^* a_{ref}^*}{(L_{ref}^*/L_{ref})} \quad \text{(compressible)}
\]
\[
    t_{chr} = \frac{t_{chr}^* |V_{ref}^*|}{(L_{ref}^*/L_{ref})} \quad \text{(incompressible)}
\]

Once the nondimensional characteristic \( t_{chr} \) is determined, the user must decide on an appropriate number of time steps \( N \) to be used for resolving the characteristic time. Then the nondimensional time step may be specified as:

\[
    \Delta t = t_{chr} / N
\]

The proper value of \( N \) must be determined by the user. However, a reasonable rule of thumb for second-order time integration is to take \( N = 200 \). Note that if there are multiple frequencies requiring resolution in time, the most restrictive should be used to determine \( \Delta t \).
3 Boundary Conditions

This chapter discusses the boundary conditions available in FUN3D. Table 1 lists the integers used to specify FUN3D boundary conditions with a short description. Each grid description subsection in section 4 indicates how these integers are specified. Details of the boundary condition implementation are provided by Carlson. [4] Some boundary conditions have required or optionally specified parameters defined in the &boundary_conditions namelist, see section B.4.12 for further boundary condition details.
<table>
<thead>
<tr>
<th>BC</th>
<th>Description</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>−1</td>
<td>Overlap</td>
<td>overset grid boundary</td>
</tr>
<tr>
<td>3000</td>
<td>Tangency</td>
<td>zero normal velocity, specified via fluxes</td>
</tr>
<tr>
<td>4000*</td>
<td>Viscous</td>
<td>explicitly set the no-slip condition</td>
</tr>
<tr>
<td>5000</td>
<td>Farfield</td>
<td>Riemann invariants</td>
</tr>
<tr>
<td>5026</td>
<td>Extrapolate</td>
<td>supersonic outflow, variables extrapolated from the interior</td>
</tr>
<tr>
<td>5050</td>
<td>Freestream</td>
<td>external freestream, specified via fluxes</td>
</tr>
<tr>
<td>5051*</td>
<td>Back pressure</td>
<td>specified static pressure (switches to extrapolation boundary condition in the presence of supersonic flow)</td>
</tr>
<tr>
<td>5052*</td>
<td>Mach outflow</td>
<td>static pressure outflow boundary condition set via a specified subsonic Mach number (not for boundary layer ingestion)</td>
</tr>
<tr>
<td>6021</td>
<td>Symmetry plane 1</td>
<td>symmetry enforced by replacing $x$-momentum with zero velocity normal to arbitrary boundary plane.</td>
</tr>
<tr>
<td>6022</td>
<td>Symmetry plane 2</td>
<td>symmetry enforced by replacing $y$-momentum with zero velocity normal to arbitrary boundary plane.</td>
</tr>
<tr>
<td>6023</td>
<td>Symmetry plane 3</td>
<td>symmetry enforced by replacing $z$-momentum with zero velocity normal to arbitrary boundary plane.</td>
</tr>
<tr>
<td>6100</td>
<td>Periodicity</td>
<td>discrete periodicity, limited to nominally 2D grids extruded across n planes in a third dimension</td>
</tr>
<tr>
<td>6661</td>
<td>$X$-symmetry plane</td>
<td>enforces symmetry for $x$ Cartesian plane</td>
</tr>
<tr>
<td>6662</td>
<td>$Y$-symmetry plane</td>
<td>enforces symmetry for $y$ Cartesian plane</td>
</tr>
<tr>
<td>6663</td>
<td>$Z$-symmetry plane</td>
<td>enforces symmetry for $z$ Cartesian plane</td>
</tr>
<tr>
<td>7011*</td>
<td>Subsonic inflow</td>
<td>subsonic inflow ($p_{t, bc} = \frac{p_{total, plenum}}{p_{static, freestream}}$, $T_{t, bc} = \frac{T_{total, plenum}}{T_{static, freestream}}$) for nozzle or tunnel plenum ( $M_{inflow} &lt; 1$ )</td>
</tr>
<tr>
<td>7012*</td>
<td>Subsonic outflow</td>
<td>subsonic outflow ($p_{bc} = \frac{p_{static, inlet}}{p_{static, freestream}}$ for inlet flow (does not allow for reverse or supersonic flow at the outflow boundary face)</td>
</tr>
<tr>
<td>7021*</td>
<td>Reaction control jet plenum</td>
<td>models the plenum of a reaction control system (RCS) jet</td>
</tr>
<tr>
<td>7031*</td>
<td>Mass flow out</td>
<td>specification of massflow out of the control volume</td>
</tr>
<tr>
<td>7036*</td>
<td>Mass flow in</td>
<td>specification of massflow in to the control volume</td>
</tr>
<tr>
<td>7100*</td>
<td>Fixed inflow</td>
<td>fixed primitive variables in to control volume</td>
</tr>
<tr>
<td>7101*</td>
<td>Fixed inflow profile</td>
<td>specified profile</td>
</tr>
<tr>
<td>7103*</td>
<td>Pulsed supersonic inflow</td>
<td>pulsing supersonic flow</td>
</tr>
<tr>
<td>7104*</td>
<td>Ramped supersonic inflow</td>
<td>ramping supersonic flow</td>
</tr>
<tr>
<td>7105*</td>
<td>Fixed outflow</td>
<td>specified primitive outflow conditions</td>
</tr>
</tbody>
</table>

* See `&boundary_conditions` namelist in section B.4.12 to specify auxiliary information and for further descriptions.
4 Grids

This chapter explains how to supply the proper file formats to Fun3D, but does not cover how to create a mesh. See section 1.3 for grid generation guidance. Fun3D supports a direct reader for many grid formats. The format of the grid is specified in the \texttt{&raw\_grid} namelist. In addition to the directly read formats, translators are provided to convert additional grid formats into a format that can be read directly, see section 4.3.

4.1 File Endianness

The ordering of bytes within a data item is known as “endianness.” If the endianness of a file is different than the native endianness of the computer then a conversion must be performed. The endianness of each grid file format is described in section 4.2. If your compiler supports it, Fun3D will attempt to open binary files with a \texttt{open(convert=...)} keyword extension. Consult the documentation of the Fortran compiler you are using to determine if other methods are available. For example, with the Intel® Fortran compiler, the endianness of file input and output can be controlled by setting the \texttt{F\_UFMTENDIAN} environment variable to \texttt{big} or \texttt{little}.

4.2 Supported Grid Formats

Fun3D natively supports the grid formats summarized in Table 2.

<table>
<thead>
<tr>
<th>Format</th>
<th>Grid files</th>
<th>BC File</th>
</tr>
</thead>
<tbody>
<tr>
<td>AFLR3</td>
<td>.ugrid</td>
<td>.mapbc</td>
</tr>
<tr>
<td>FAST</td>
<td>.fgrid</td>
<td>.mapbc</td>
</tr>
<tr>
<td>FieldView</td>
<td>.fvgrid_fmt</td>
<td>.mapbc</td>
</tr>
<tr>
<td></td>
<td>.fvgrid_unf</td>
<td>.mapbc</td>
</tr>
<tr>
<td>FUN2D</td>
<td>.faces</td>
<td>.mapbc</td>
</tr>
<tr>
<td>VGRID</td>
<td>.cogag, .bc</td>
<td>.mapbc'</td>
</tr>
<tr>
<td>FELISA</td>
<td>.gri, .fro</td>
<td>.bco</td>
</tr>
</tbody>
</table>

* Same suffix, but GridTool format.

The standard Fun3D .mapbc file format contains the boundary condition information for the grid. The first line is an integer corresponding to the number of boundary groups contained in the grid file. Each subsequent line in this file contains two integers, the boundary face number and the Fun3D boundary condition integer; these numbers may optionally be followed by a character string that specifies a “family” name for the boundary. The family name is required if the \texttt{patch\_lumping} option (section B.4.2) is invoked to combine
patches into fewer patch families. Below is a sample .mapbc file illustrative for all grid formats except GridTool/VGRID, FELISA, and FUN2D, which are described later.

```
13
1  6662 box_ymin
2  5025 box_zmax
3  5050 box_xmin
4  5025 box_ymax
5  5025 box_zmin
6  5025 box_xmax
7  3000 wing_upper
8  3000 wing_lower
9  3000 wing_upper
10  3000 wing_upper
11  3000 wing_lower
12  3000 wing_lower
13  3000 wing_tip
```

4.2.1 AFLR3 Grids

AFLR3, SolidMesh, Pointwise, and GridEx can all produce this format and Fun3D ships with translators that convert Plot3D and CGNS grids to AFLR3 format. The format is documented online at [http://simcenter.msstate.edu/docs/solidmesh/ugridformat.html](http://simcenter.msstate.edu/docs/solidmesh/ugridformat.html)

AFLR3 grid file format types are indicated by file suffixes. The formatted (plain text) style has a .ugrid suffix while other types vary according to endianness (see section 4.1) and binary type as shown in Table 3. The boundary conditions are specified via the standard Fun3D .mapbc format.

<table>
<thead>
<tr>
<th>Type</th>
<th>Little endian</th>
<th>Big endian</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fortran Stream, C Binary</td>
<td>.lb8.ugrid</td>
<td>.b8.ugrid</td>
</tr>
<tr>
<td>Fortran Unformatted</td>
<td>.lr8.ugrid</td>
<td>.r8.ugrid</td>
</tr>
</tbody>
</table>

4.2.2 FAST Grids

The .fgrid file contains the complete grid stored in ASCII FAST format. The format is documented online at [http://simcenter.msstate.edu/docs/solidmesh/FASTformat.html](http://simcenter.msstate.edu/docs/solidmesh/FASTformat.html) The boundary conditions are specified via the standard Fun3D .mapbc format.
4.2.3 VGRID Grids

The .cogsg file contains the grid nodes and tetrahedra stored in unformatted VGRID format. The VGRID cogsg files always have big endian byte order regardless of the computer used in grid generation. See section 4.1 for instructions on specifying file endianness.

The .bc file contains the boundary information for the grid, as well as a flag for each boundary face. For viscous grids with a symmetry plane, VGRID is known to produce boundary triangles in the .bc file that are incompatible with the volume tetrahedra. Fun3D provides a repair_vgrid_mesh utility to swap the edges of these inconsistent boundary triangles. If Fun3D reports that there are boundary triangles without a matching volume tetrahedra, use this utility.

VGRID has a different .mapbc boundary condition format. For each boundary flag used in the .bc file, the .mapbc file contains the boundary type information. The VGRID boundary conditions are described at the website: http://tetruss.larc.nasa.gov/usm3d/bc.html. The Fun3D boundary condition integers can also be used in place of the VGRID boundary condition integers. Internally, Fun3D converts the VGRID boundary condition integers to the Fun3D boundary condition integers as indicated in Table 4.

<table>
<thead>
<tr>
<th>VGRID</th>
<th>FUN3D</th>
</tr>
</thead>
<tbody>
<tr>
<td>-1</td>
<td>-1</td>
</tr>
<tr>
<td>0</td>
<td>5000</td>
</tr>
<tr>
<td>1</td>
<td>6662</td>
</tr>
<tr>
<td>2</td>
<td>5005</td>
</tr>
<tr>
<td>3</td>
<td>5000</td>
</tr>
<tr>
<td>4</td>
<td>4000</td>
</tr>
<tr>
<td>5</td>
<td>3000</td>
</tr>
<tr>
<td>44</td>
<td>4000</td>
</tr>
<tr>
<td>55</td>
<td>3000</td>
</tr>
</tbody>
</table>

4.2.4 FieldView Grids

The .fvgrid_fmt file contains the complete grid stored in ASCII FieldView FV-UNS format, and the .fvgrid_unf file contains the complete grid stored in unformatted FieldView FV-UNS format. Supported FV-UNS file versions are 2.4, 2.5, and 3.0. With FV-UNS version 3.0, the support is only for the grid file in split grid and results format; the combined grid/results format is not supported. Fun3D does not support the arbitrary polyhedron elements of the FV-UNS 3.0 standard. For ASCII FV-UNS 3.0, the standard allows comment lines (line starting with !) anywhere in the file. Fun3D
only allows comments immediately after line 1. Only one grid section is allowed. The precision of the unformatted grid format should be specified by the fieldview-coordinate-precision variable in the &raw_grid namelist, see section B.4.2. The boundary conditions are specified via the standard Fun3D mapbc format.

4.2.5 FELISA Grids

The .gri file contains the grid stored in formatted FELISA format. The .fro file contains the surface mesh nodes and connectivities and associated boundary face tags for each surface triangle. This file can contain additional surface normal or tangent information (as output from GridEx or SURFACE mesh generation tools), but the additional data is not read by Fun3D. The .bco file contains a flag for each boundary face. If original FELISA boundary condition flags (1, 2, or 3) are used, they are translated to the corresponding Fun3D 4-digit boundary condition flag according to Table 5. Alternatively, Fun3D 4-digit boundary condition flags can be assigned directly in this file.

<table>
<thead>
<tr>
<th>FELISA</th>
<th>FUN3D</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3000</td>
</tr>
<tr>
<td>2</td>
<td>6662</td>
</tr>
<tr>
<td>3</td>
<td>5000</td>
</tr>
</tbody>
</table>

4.2.6 Fun2D Grids

The .faces file contains the complete grid stored in formatted Fun2D format (triangles). Internally, Fun3D will extrude the triangles into prisms in the y-direction and the 2D mode of Fun3D is automatically enabled. Output from the flow solver will include this one-cell wide extruded mesh.

Boundary conditions are contained in the Fun2D grid file as integers 0–8. The mappings to Fun3D boundary conditions are given in Table 6. If Fun3D does not detect a .mapbc, it will write a .mapbc file that contains the default Table 6 mapping. If you wish to change the boundary conditions from the defaults based on the .faces file, simply edit them in this .mapbc file and rerun Fun3D. The boundary conditions in the .mapbc file have precedence over the .faces boundary conditions. If you wish to revert to the boundary conditions in the .faces file after modifying the .mapbc, you can remove the .mapbc and rerun Fun3D.
Table 6: Boundary type mapping between Fun2D and Fun3D.

<table>
<thead>
<tr>
<th>FUN2D</th>
<th>FUN3D</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>3000</td>
</tr>
<tr>
<td>1</td>
<td>4000</td>
</tr>
<tr>
<td>2</td>
<td>5000</td>
</tr>
<tr>
<td>3</td>
<td>-1</td>
</tr>
<tr>
<td>4</td>
<td>4010</td>
</tr>
<tr>
<td>5</td>
<td>4010</td>
</tr>
<tr>
<td>6</td>
<td>5005</td>
</tr>
<tr>
<td>7</td>
<td>7011</td>
</tr>
<tr>
<td>8</td>
<td>7012</td>
</tr>
</tbody>
</table>

4.3 Translation of Additional Grid Formats

While Fun3D supports the direct read of multiple formats, utilities are provided to translate additional grid formats into a format that Fun3D can read.

4.3.1 PLOT3D Grids

The utility `plot3d_to_aflr3` converts a PLOT3D structured grid to an AFLR3-format hexahedral unstructured grid. The original structured grid must be 3D multiblock [http://www.grc.nasa.gov/WWW/wind/valid/plot3d.html](http://www.grc.nasa.gov/WWW/wind/valid/plot3d.html) (no iblanking) with the file extension `.p3d` for formatted ASCII or the file extension `.ufmt` for Fortran unformatted. Only one-to-one connectivity is allowed with this option (no patching or overset). The grid should contain no singular (degenerate) lines or points. A neutral map file with extension `.nmf` is also required. This file gives boundary conditions and connectivity information. The `.nmf` file is described at [http://geolab.larc.nasa.gov/Volume/Doc/nmf.htm](http://geolab.larc.nasa.gov/Volume/Doc/nmf.htm).

Note that the Type name in the `.nmf` file must correspond with one of Fun3D’s BC types, plus it allows the Type one-to-one. If the Type is not recognized, you will get errors like:

This may be an invalid BC index.

An example `.nmf` file is shown here for a simple single-zone airfoil C-grid 
\((5 \times 257 \times 129)\) with six exterior boundary conditions and one one-to-one patch in the wake where the C-grid attaches to itself:

```plaintext
# ===== Neutral Map File generated by the V2k software of NASA Langley's GEOLAB =====
# ===================================================================================
# Block# IDIM JDIM KDIM
# ===================================================================================
1
1 5 257 129
# Type B1 F1 S1 E1 S2 E2 B2 F2 S1 E1 S2 E2 Swap
# ===================================================================================
```

Note that the Type name in the `.nmf` file must correspond with one of Fun3D’s BC types, plus it allows the Type one-to-one. If the Type is not recognized, you will get errors like:

This may be an invalid BC index.

An example `.nmf` file is shown here for a simple single-zone airfoil C-grid 
\((5 \times 257 \times 129)\) with six exterior boundary conditions and one one-to-one patch in the wake where the C-grid attaches to itself:
FUN3D is distributed with a utility cgns_to_aflr3 that converts CGNS files to AFLR3 grids. This utility will only be built if FUN3D is configured with a CGNS library, see section A.7.12. Only the Unstructured type of CGNS files are supported. The following CGNS mixed element types are supported: PENTA_6 (prisms), HEX_8 (hexes), TETRA_4 (tets), and PYRA_5 (pyramids).

The CGNS file must include Elements_t nodes for all boundary faces (type QUAD_4 or TRI_3) to refer to the corresponding boundary elements. Otherwise, the utility cannot recognize what boundaries are present because it currently identifies boundaries via these 2D element types. The cgns_to_aflr3 utility requires that the BC elements be listed either as a range or a sequential list. It is also helpful to have separate element nodes for each boundary element of a given BC type. This way, it is easier to interpret the boundaries, i.e., body versus symmetry versus farfield. Visualization tools, such as Tecplot™, can easily distinguish the various boundary condition groups as long as each group has its own node in the CGNS tree. Under BC_t, cgns_to_aflr3 reads these BC names, but ignores additional boundary data (e.g., BCDataSet, BCData).

If the CGNS file is missing BCs (no BC_t node), cgns_to_aflr3 still tries to construct the BCs based on the boundary face Elements_t information. If these boundary element nodes have a name listed in Table 7, a .mapbc file will be written that contains the FUN3D boundary condition numbers. If the name is not recognized, you will see the message:

WARNING: BC type ... in CGNS file not recognized.

in which case you will need to fix it by by editing the .mapbc file manually. Always check the .mapbc file after the utility has run, to make sure that the BCs have all been interpreted and set correctly. If a translation problem is observed, you should edit the .mapbc file before running FUN3D.
Table 7: Boundary type mapping between CGNS and Fun3D.

<table>
<thead>
<tr>
<th>CGNS</th>
<th>FUN3D</th>
</tr>
</thead>
<tbody>
<tr>
<td>BCSymmetryPlane</td>
<td>6661, 6662, or 6663 via prompt</td>
</tr>
<tr>
<td>BCFarfield</td>
<td>5000</td>
</tr>
<tr>
<td>BCWallViscous</td>
<td>4000</td>
</tr>
<tr>
<td>BCWall</td>
<td>4000</td>
</tr>
<tr>
<td>BCWallInviscid</td>
<td>3000</td>
</tr>
<tr>
<td>BCOoutflow</td>
<td>5026</td>
</tr>
<tr>
<td>BCTunnelOutflow</td>
<td>5026</td>
</tr>
<tr>
<td>BCInflow</td>
<td>5000</td>
</tr>
<tr>
<td>BCTunnelInflow</td>
<td>5000</td>
</tr>
</tbody>
</table>
5 Flow Solver, NODET

This chapter covers what is required to run an initial flow solution, how to restart a flow solution, and how to specify what outputs the solver NODET produces.

5.1 Flow Solver Execution

The grid and flow conditions are specified in the file fun3d.nml; see section B.4 for the file description. If you configured FUN3D without MPI, the executable is named nodet. If you configured FUN3D with MPI, the executable is named nodet_mpi. Configuration and installation is explained in detail in section A. The executable nodet can be invoked directly from the command line,

    nodet [fun3d options]

but the MPI version nodet_mpi will need to be invoked within an MPI environment. The most common method is via

    [MPI run command] [MPI options] nodet_mpi [fun3d options]

The details of the MPI run command and MPI options will depend on the MPI implementation. The MPI run command is commonly mpirun or mpiexec. The MPI options may contain the number of processors -np [n], a machine file -machinefile [file], or no local -nolocal. If a queuing system is used (e.g., PBS) this command will need to be run inside an interactive job or a script. See your MPI documentation or system administrator to learn the details of your particular environment.

If you have provided a grid with boundary conditions and fun3d.nml, you will then see the solver start to execute. If an unexpected termination happens during execution, especially during grid processing or the first iteration, you may need to set your shell limits to unlimited,

    $ ulimit unlimited # for bash
    $ unlimit # for c shell

A detailed description of the output files is given below.

5.2 Command Line Options

These options are specified after the executable. The majority of the command line options are functionality under development and there is work underway to migrate command line options to namelists. Namelists are the preferred input method. Command line options should be avoided unless they are the only way to activate the functionality you require. These commands are always preceded by -- (double minus). More than one option may appear on the
command line (each option proceeded by a -- ). You can see a listing of the available command line options in any of the codes in the FUN3D suite by using the command line option --help after the executable name,

```
./nodet_mpi --help
```

The options are then listed in alphabetical order, along with a short description and a list of any auxiliary parameters that might be needed, and then the code execution stops. Specific examples of the use of command line options are found throughout this, and later, chapters.

5.3 Output Files

These are the output files produced by the flow solver, NODET.

[project_rootname].flow This file contains the binary restart information and is read by the solver for restart computations. See the restart_read namelist variable in section B.4.9 to control restart behavior.

[project_rootname].hist.dat This file contains the convergence history for the RMS residual, lift, drag, moments, and CPU time, as well as the individual pressure and viscous components of each force and moment. The file is in Tecplot™ format. See section B.4.13 for an improved method to track forces and moments.

[project_rootname].subhist.dat For time accurate computations only. This file contains the sub-iteration convergence history for the RMS residuals. The file is in Tecplot™ format.

[project_rootname].forces This file contains a breakdown of all the forces and moments acting on each individual boundary group. The totals for the entire configuration are listed at the bottom. See section B.4.13 for an improved method to track forces and moments.

5.3.1 Flow Visualization

There are four basic categories of output: boundary data, sampling data (on entities such as planes, boxes and spheres), volumetric data, and slice data controlled by the namelists in Table 8.

Each namelist has a corresponding frequency variable. A positive frequency will cause the output to be generated every frequency time step/iteration. A negative frequency will cause output to be written only at the end of a run. A zero frequency (the default) with produce no output. See the corresponding namelist descriptions for details.
Table 8: Solver output types.

<table>
<thead>
<tr>
<th>Type</th>
<th>Namelist</th>
<th>Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>domain boundaries</td>
<td>&amp;boundary_output_variables</td>
<td>B.4.21</td>
</tr>
<tr>
<td>domain volume</td>
<td>&amp;volume_output_variables</td>
<td>B.4.20</td>
</tr>
<tr>
<td>boundary slices</td>
<td>&amp;slice_data</td>
<td>B.4.24</td>
</tr>
<tr>
<td>various geometries</td>
<td>&amp;sampling_parameters and &amp;sampling_output_variables</td>
<td>B.4.23 and B.4.22</td>
</tr>
<tr>
<td>point</td>
<td>&amp;sampling_parameters and &amp;sampling_output_variables</td>
<td>B.4.23 and B.4.22</td>
</tr>
</tbody>
</table>

5.3.2 Flow Visualization Output From Existing Solution

If a FUN3D flow solution already exists, visualization files by setting `nsteps = 0` in the `&code_run_control` namelist within the `fun3d.nml` file and setting the `restart_read` variable to something other than 'off'. This will allow generation of visualization output without having to do additional timesteps or iterations.
6 Adjoint Solver, dual

This section describes how to execute the adjoint solver, dual, directly. Typically, dual is executed by scripts that manage the multiple steps required for design optimization (section 8) or grid adaptation (section 7). However, it may be necessary to run dual directly to diagnose problems or gain experience during setup including determining input parameters and termination strategies. Fun3D is configured to compile dual by default. While the adjoint method is available for most commonly used Fun3D capabilities, only a subset of Fun3D’s full capabilities are implemented in the adjoint solver.

6.1 Convergence of the Linear Adjoint Equations

The adjoint solution is dependent on the primal flow solution (and the convergence of the primal flow equations). While the primal solution may have converged enough to give acceptable force and moment results, the flow residuals might still be large, which can cause the adjoint solution scheme to diverge. This divergence issue is most common in turbulent simulations. A divergent adjoint scheme can be improved in some circumstances with the --outer_loop_krylov command line option. It is critical to run the flow solver and the adjoint solver with the same governing equations and boundary conditions.

The scaling of the adjoint residuals is different from the flow residuals and is dependent on the choice of the adjoint cost functions. The number of iterations steps and the residual tolerance stopping_tolerance will need to be adjusted, see section B.4.9. The sensitivities should converge at the same rate as your functions (i.e., lift), but an adjoint with some algebraic error may still provide reasonable sensitivities for design and grid adaptation.

6.2 Required Directory Hierarchy and Executing dual

The executable dual can be invoked directly from the command line,

    dual [fun3d options]

but the MPI version dual_mpi will need to be invoked within an MPI environment. The most common method is via

    [MPI run command] [MPI options] dual_mpi [fun3d options]

Any [fun3d options] provided to nodet that control the flow solver residual will also be required for the adjoint solver for a consistent adjoint solution and solution scheme. See the flow solver execution instructions for more details, section 5.1.
DUAL expects the cost function description `./rubber.data` to be in the parent directory of the directory from which it is invoked. The input and flow restart files are shared with NODET in the directory `../Flow/`. The flow solver must be run to completion, to provide a flow restart file, before DUAL is invoked. See Table 9 for the required files and locations.

Table 9: Adjoint solver DUAL directory hierarchy.

<table>
<thead>
<tr>
<th>Relative Path</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>../Flow/[project_rootname].flow</code></td>
<td>Primal flow solution (restart)</td>
</tr>
<tr>
<td><code>../Flow/fun3d.nml</code></td>
<td>Main input namelist file</td>
</tr>
<tr>
<td><code>../rubber.data</code></td>
<td>Description of the adjoint cost function</td>
</tr>
</tbody>
</table>

6.3 `rubber.data`

The minimum required information for running the adjoint (and grid adaptation) is included below. See section 8.6.2 for complete details on including the information required for design. The reader for this file requires the exact number of header lines. Be very careful when editing this file. The cost function is c1 in this case. Available cost functions are discussed in section 8.1.

```
#########################################################################
######################## Design Variable Information ####################
#########################################################################
Global design variables (Mach number / angle of attack)
Index Active Value Lower Bound Upper Bound
Mach 0 0.10000000000000E+01 0.00000000000000E+00 0.50000000000000E+01
AOA 0 0.10000000000000E+01 0.00000000000000E+00 0.50000000000000E+01
Number of bodies
0

#########################################################################
######################## Function Information ##########################
#########################################################################
Number of composite functions for design problem statement
1

#########################################################################
Cost function (1) or constraint (2)
1
If constraint, lower and upper bounds
0.10000000000000E+00 0.50000000000000E+00
Number of components for function 1
1
Physical timestep interval where function is defined
1 1
Composite function weight, target, and power
1.0 0.0 1.0
Components of function 1: boundary id (0=all)/name/value/weight/target/power
```
6.4 Output Files

The adjoint solver will export visualization files in the same manner as the flow solver when requested, see section 5.3.1.

**[project_rootname].adjoint** This file contains the binary restart information and is read by the and adjoint solver for restart computations.

**[project_rootname].hist.tec** This file contains the convergence history for the RMS residual of the adjoint equations and CPU time. The file is in the same Tecplot™ format as the flow solver produces. History information is truncated when the adjoint solver is restarted.
7 Grid Adaptation

Fun3D implements metric-based adaptation, where grid adaptation is separated into two tasks. The first step is to construct a metric that describes the desired size and anisotropy of the adapted grid elements. The second step is to produce an adapted grid that is based on this metric.

Feature-based adaptation constructs the metric based on properties of the flow solution. Adjoint-based adaptation constructs the metric from the flow and adjoint solutions to reduce estimated errors in a specified output function. The namelist &adapt_metric_construction (section B.4.28) for specifying details of the metric.

Fun3D supports a number of grid adaptation libraries. The namelist &adapt_mechanics (section B.4.29) specifies the grid adaptation library and its options. The REFINE library is distributed and installed with Fun3D by default.

7.1 Geometry Specification and Grid Freezing for REFINE

When adapting a grid with REFINE, all boundary faces must be specified as frozen or a geometry definition must be provided via FAUXGeom. Use the default patch_lumping='none' in the &raw_grid namelist, as lumping will change boundary patch indexes making it more difficult to specify geometry.

7.1.1 No geometry, where the surface nodes are frozen.

REFINE cannot preserve the high aspect ratio structures within viscous layers, and so viscous layers must be frozen for a specified distance away from the surface to maintain grid quality. This is invoked with the adapt_freezebl command within the &adaptation_mechanics namelist, see section B.4.29 for more details.

Additionally, specific surfaces that do not have a viscous boundary condition can be frozen by listing the surface numbers (one per line) in a file named [project_rootname].freeze. For example, [project_rootname].freeze that contains

```
5
7
```

will freeze points on boundary patches 5 and 7. This is also useful for boundary surfaces that do not have an analytical definition handled by FAUXGeom.
7.1.2 FAUXGeom for Planar Boundaries

For viscous problems, where the mesh on the complex geometry of the body is frozen, FAUXGeom can be used to provide an analytical definition of the farfield boundary surfaces. This allows adaptation to occur on the planar surfaces of the mesh, even when the boundary layer mesh is frozen. This is a particularly important capability for symmetry planes. At present, FAUXGeom can only handle planar surfaces.

FAUXGeom reads the file `faux_input`. Here is an example file:

```
4
5 xplane -5.0
3 yplane -1.0
1 zplane 1.0
16 general_plane 2.0
 0.707 0.707 0.0
```

The first line is how many faux surfaces are being defined. The subsequent lines have a face number, type of face, and a distance associated with the particular geometry. In this example, the first faux face defined corresponds to surface 5 in the mesh and is a \( x = -5.0 \) constant plane. Faux faces are similarly defined for the \( z \) and \( y \) planes of surfaces 3 and 1. Surface 16 is a plane perpendicular to a \((0.707, 0.707, 0.0)\) normal that is located 2.0 away from the origin in the direction of the normal; the plane passes through the point \((1.414, 1.414, 0.0)\).

7.2 Performing Feature-Based Adaptation

The `&adapt_metric_construction` variable `adapt_feature_scalar_form` defines the operator that is applied to the `adapt_feature_scalar_key` to compute an adaptation intensity. This intensity is raised to the `adapt_exponent` power to produce a scaling of an isotropic element size estimate on the current grid. The anisotropy of the metric is introduced by the Hessian of the `adapt_hessian_key` variable.

Set `restart_read='on'` in section B.4.9 to read the flow solution. Run `NODET` with the `--adapt` command line option in the directory with the flow restart. The result will be a new grid and interpolated solution file with the `adapt_project` project name. After adaptation, the flow solver can now be restarted with this new grid and interpolated solution by changing the `project_rootname`.

7.3 Performing Adjoint-Based Adaptation

Adjoint-based adaptation requires that a flow solution be calculated in the `Flow` directory and an adjoint solution be calculated in the `Adjoint` directory.
See section 6 for more information on obtaining an adjoint solution. The adjoint solution is based on the functional defined in rubber.data and this is the same functional targeted for grid adaptation.

Adaptation is performed by executing dual with the command line options --rad --adapt. The adjoint solver reads the fun3d.nml in the ../Flow directory), so this is the place to specify &adapt_metric_construction and &adapt_mechanics options. The freeze and FAUXGeom files are read in the current directory, Adjoint.

The result will be a new grid and interpolated solution restart file in the ../Flow directory and an interpolated adjoint restart in the Adjoint directory. The project name of these new files is adapt_project.

### 7.4 Scripting Grid Adaptation

The Fun3D installation includes the f3d script. To find the other components of the Fun3D suite, the f3d script expects to be in the bin directory of the Fun3D installation. Don’t copy or link f3d from the bin directory. The input file case_specifics is described in section 7.4.1.

Execute the f3d script in a directory that contains all of the the input files (e.g., grid, fun3d.nml, case_specifics). The script will create the required Flow and Adjoint directories to run the case. It has the following commands,

```
usage: f3d <command>

<command> description
---------- ----------
start     Start adaptation
view      Echo a single snapshot of stdout
watch     Watch the result of view
shutdown  Kill all running fun3d and ruby processes
clean     Remove output and sub directories
```

The command start begins adaptation by launching a background job. The commands view and watch allow the adaptation progress to be monitored. (Use Ctrl-C to escape the watch command.) The shutdown command kills all ruby (f3d) and Fun3D jobs. The clean command removes the Flow and Adjoint subdirectories and the output log file.

#### 7.4.1 Input File case_specifics for f3d Script

The f3d script has one input file, named case_specifics. Here is an example

```
root_project "
number_of_processors 2
mpirun_command 'mpiexec'
```
where the defaults are listed. Adaptation will be performed from the first grid adaptation iteration 1 to the last grid adaptation iteration 10. The string in quotes next to root.project is the project root name. A two digit iteration number will be appended to it. The project name for the first adaptation will be [root.project]01 and the last will be [root.project]10. All the files required to run NODET and DUAL should be provided in the current directory and the grid filename should include the root project name and iteration number, [root.project]01. Flow and Adjoint subdirectories are created by the script during execution, and the input files are placed in their correct location by the script.

Command line options can be passed to the codes via,

```
all_cl ' ' 
flo_cl ' ' 
adj_cl ' ' 
rad_cl ' ' 
```

where all_cl is provided to all codes, flo_cl is provided to NODET, adj_cl is provided to DUAL during the adjoint solve, and rad_cl is provided to DUAL during error estimation and adaptation. For example, the line

```
adj_cl ' --outer_loop_krylov ' 
```

turns on Krylov projection wrapping to stabilize the adjoint solve.

The main input file fun3d.nml provided in the current directory can be modified by the following commands

```
all_nl['variable']= value 
flo_nl['variable']= value 
adj_nl['variable']= value 
rad_nl['variable']= value 
```

where all_nl changes fun3d.nml for all codes, flo_nl for NODET, adj_nl for DUAL during the adjoint solve, and rad_nl for DUAL during error estimation and adaptation. An example is

```
adj_nl['steps']=500 
adj_nl['stopping_tolerance']=1.0e-12 
```

where the termination criteria of the adjoint solver can be specified separately than the flow solver.

The case specifics is actually executable Ruby code. This allows values to be computed or conditionally executed, but also require nested quotes for character strings,
rad_nl["adapt_complexity"] = 5000*iteration
number_of_processors 128 if (iteration>5)
all_nl["flux_construction"] = "vanleer"
8 Design Optimization

The Fun3D design framework uses a gradient-based optimization procedure. One potential approach to obtaining the required sensitivity derivatives is a conventional forward mode of differentiation, such as finite-differencing, complex-variable formulations, operator overloading, or direct differentiation. Since the cost of these techniques scales directly with the number of input parameters, these methods are most efficient for problems where the number of outputs is considerably larger than the number of inputs. For such problems, Fun3D provides a complex variable formulation as described in section 8.14. However, for most aerodynamic design problems, the converse is true; the number of design variables is typically much larger than the number of objective functions and/or constraints. In this context, an adjoint, or reverse mode of differentiation is preferred.

Fun3D provides a discrete adjoint capability to efficiently determine the sensitivities required by a gradient-based design procedure. The adjoint approach enables the user to compute sensitivity derivatives of an output function with respect to an unlimited number of design variables at a cost equivalent to a single additional flow solution. For a general review of sensitivity analysis techniques, see [6] and [7].

The adjoint approach used in Fun3D relies on discrete linearizations of the relevant components of the flow solver. Most of Fun3D’s compressible perfect gas and incompressible capabilities are accounted for within the adjoint-based framework. Discretely consistent sensitivities have been demonstrated for both steady and unsteady inviscid, laminar, and turbulent flows based on the one-equation model of Spalart and Allmaras. Grid topologies may contain any combination of element types and may also contain overset grid discretizations. Grids may be static, non-inertial, or may contain any combination of static, rigidly-moving, or deforming overset component grids. Both compressible and incompressible formulations are available. The most commonly-used boundary conditions are implemented in the adjoint framework, and a broad range of objective/constraint functions is also available. However, the user is encouraged to review the latest release notes or contact Fun3D-Support@lists.nasa.gov to determine if a specific analysis capability is currently supported by the adjoint implementation. For a detailed overview of the adjoint-based procedure used in Fun3D and examples of its use for design optimization, see [8] and the references contained therein.

Users are encouraged to gain extensive experience using Fun3D for analysis purposes before attempting design optimization. This experience will aid in properly setting up optimization cases, understanding the steps involved, and interpreting the results.

The adjoint-based algorithms are very efficient, but a typical optimization will still require the equivalent of $O(20)$ typical analyses. Therefore, secur-
ing sufficient computational resources is critical to performing realistic high-fidelity design. It should also be noted that the various optimization packages supported by Fun3D may behave very differently for a given design problem; moreover, the optimal algorithm is generally problem-dependent.

At this time, the documentation provided here is aimed at design optimization of steady flows. The extension to simulations involving unsteady flows is available for general use (e.g., see [9]), but is not currently covered here. Please contact Fun3D-Support@lists.nasa.gov if interested in using this capability.

8.1 Objective/Constraint Functions

To perform a gradient-based optimization, the user must specify at least one objective function to quantify the merit of the configuration. In the Fun3D design infrastructure, such objective functions may take a very general form as described here. Note that all of the supported optimization packages always seek to minimize the chosen objective function. Care should be taken to pose the objective function accordingly. Multiple outputs may be accounted for in a variety of ways. Constraints may be included implicitly within the objective function(s) as penalty terms. Explicit constraint functions may also be posed, as either equality or inequality constraints.

Note that the primary limitation in posing the problem statement is the general ability of the chosen optimization package to handle the design problem posed by the user. For example, the PORT optimization software does not support the use of explicit constraints. KSOPT is the only supported optimization package that supports the use of more than one objective function; however, Fun3D offers several approaches to scalarize multiple objectives for other packages. Multi-point design is also supported in several forms. See section 8.9 and section 8.10 for specific details on these capabilities.

The Fun3D flow and adjoint solvers do not distinguish between objective functions and constraints. The solvers themselves merely provide function values and their sensitivities for use during the optimization procedure. The actual optimization packages are the only components in the design framework that make a distinction between objective functions and constraint functions.

8.1.1 Terminology

It is useful to establish some basic terminology when composing the design problem statement. Within the Fun3D design infrastructure, the user specifies one or more component functions based on typical solver outputs. These component functions are then combined to form a single composite function. Multiple component functions may be used to form composite functions, and in turn, multiple composite functions may ultimately be specified. The user then classifies each composite function, designating it either an objective function
or a constraint function. Again, this distinction is solely for the optimization algorithm; Fun3D simply evaluates and linearizes each of the composite functions in a generic sense and provides them to the optimization scheme.

The adjoint formulation requires a separate adjoint solution for each composite function. For example, a drag-minimization problem with an explicit lift constraint will generally require two adjoint solutions at each step of the design procedure (one based on drag and one based on lift). Rather than performing separate adjoint executions for each function, Fun3D’s adjoint solver is implemented such that multiple adjoint solutions may be computed simultaneously by cycling through a series of right-hand side vectors. In this manner, much of the computational overhead associated with discretizing the adjoint system is amortized over the collection of specified functions, and each additional function only increases the overall computational cost by approximately 40%. See [10] for further details on this aspect of the implementation.

### 8.1.2 Functional Form

Composite functions take the following general form in Fun3D:

\[ f_i = \sum_{j=1}^{J_i} \omega_j (C_j - C_0^*)^{p_j} \]  

(1)

Here, the index \( J_i \) corresponds to the number of individual component functions comprising composite function \( i \). The factor \( \omega_j \) represents a user-specified weighting coefficient in the summation; \( C_j \) is a Fun3D scalar output quantity, \( C_0^* \) is a user-specified target value for that output quantity, and \( p_j \) is a user-specified exponent. The currently available Fun3D output functions that may be posed as \( C_j \) are listed in Table 10. Though not explicitly represented in Eq. 1, the implementation also allows the user to only use specific boundary contributions to \( C_j \) and not all boundaries if desired. This could be used to focus the optimization function on forces acting on the wing or tail only. Note that when composing an objective or constraint function it is often helpful to scale the expected value to an \( O(1) \) quantity. This can be readily done using the \( \omega_j \) factor. Additional details relevant to more complex functions are covered in section 8.2. The specific input mechanism for providing each of the component/composite function parameters will be discussed at length in section 8.6.2.

To demonstrate the use of the general functional form given by Eq. 1, several examples are given here:

**Unconstrained Drag Minimization** For an unconstrained problem in which the user wishes solely to minimize drag, one potential approach might be to specify a single composite function consisting of a single component
Table 10: Objective/constraint component function keywords.

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>cl, cd</td>
<td>Lift, drag coefficients</td>
</tr>
<tr>
<td>clp, cdp</td>
<td>Lift, drag coefficients: pressure contributions</td>
</tr>
<tr>
<td>clv, cdv</td>
<td>Lift, drag coefficients: shear contributions</td>
</tr>
<tr>
<td>cmx, cmy, cmz</td>
<td>x/y/z-axis moment coefficients</td>
</tr>
<tr>
<td>cmxp, cmyp, cmzp</td>
<td>x/y/z-axis moment coefficients: pressure contributions</td>
</tr>
<tr>
<td>cmxv, cmyv, cmzv</td>
<td>x/y/z-axis moment coefficients: shear contributions</td>
</tr>
<tr>
<td>cx, cy, cz</td>
<td>x/y/z-axis force coefficients</td>
</tr>
<tr>
<td>cxp, cyp, czp</td>
<td>x/y/z-axis force coefficients: pressure contributions</td>
</tr>
<tr>
<td>cxv, cyy, czv</td>
<td>x/y/z-axis force coefficients: shear contributions</td>
</tr>
<tr>
<td>clcd</td>
<td>Lift-to-drag ratio</td>
</tr>
<tr>
<td>fom</td>
<td>Rotorcraft figure of merit</td>
</tr>
<tr>
<td>propeff</td>
<td>Rotorcraft propulsive efficiency</td>
</tr>
<tr>
<td>pstag</td>
<td>Stagnation pressure RMS in cutting plane</td>
</tr>
<tr>
<td>boom_targ</td>
<td>Near-field target p/p∞</td>
</tr>
<tr>
<td>sboom</td>
<td>Coupled sBOOM ground-based noise metrics</td>
</tr>
<tr>
<td>ae</td>
<td>Supersonic equivalent area target distribution</td>
</tr>
<tr>
<td>cpstar</td>
<td>Target pressure distributions</td>
</tr>
</tbody>
</table>

function with $\omega_1 = 1.0$, $C_1 = cd$, $C_1^* = 0.0$, and $p_1 = 2$. In this manner, the objective function is simply

$$f = C_D^2,$$

(2)

where the quadratic form has been chosen to provide a convex function space.

**Drag Minimization with Lift Penalty** To add an interior penalty term accounting for a lift equality constraint of 0.5, one might instead use two component functions within the same single composite function where $\omega_1 = 10.0$, $\omega_2 = 1.0$, $C_1 = cd$, $C_2 = cl$, $C_1^* = 0.0$, $C_2^* = 0.5$, and $p_1 = p_2 = 2$. These parameters yield

$$f = 10C_D^2 + (C_L - 0.5)^2.$$

(3)

In this case, any deviation of the lift coefficient from its target value of 0.5 will “penalize” the objective function. The weighting parameters $\omega_j$ have been selected based on typical magnitudes of $C_D$ and $C_L$, so as to produce roughly equivalent contributions to the objective function. Note that the choice of these weighting parameters is heuristic in nature and often troublesome in practice.

**Drag Minimization with Explicit Lift Constraint** In this example, the lift constraint $C_L = 0.5$ is instead posed as an explicit constraint for the optimizer. Here, two composite functions are formed, each with a single component
function. First, an objective function is specified as in Eq. 2 with $\omega_1 = 1.0$, $C_1 = c_d$, $C_1^* = 0.0$, and $p_1 = 2$. As before, this yields

$$f_1 = C_D^2.$$  \hspace{1cm} (4)

However, an additional composite function for the lift constraint is also specified with $\omega_1 = 1.0$, $C_1 = c_l$, $C_1^* = 0.5$, and $p_1 = 1$, which gives

$$f_2 = C_L.$$  \hspace{1cm} (5)

This explicit form of the lift constraint is generally preferred in practice.

### 8.2 Some Details on Specific Objective/Constraint Functions

Many of the scalar functions shown in Table 10 and designed to be used as the term $C_j$ in Eq. 1 are straightforward. For example, the keyword $c_d$ is sufficient to characterize a drag-based component function. However, some of the scalar functions listed in Table 10 require the user to be aware of specific requirements and/or to provide additional auxiliary data. In this section, scalar functions requiring further data and/or explanation are covered.

#### 8.2.1 Lift-to-Drag Ratio (Keyword: $c_l c_d$)

This function must be specified with a 0 for its boundary index, i.e., it must be applied to the entire configuration and is not available for individual boundary patches. This function is only available for compressible flows.

#### 8.2.2 Rotorcraft Figure of Merit (Keyword: $f_{om}$)

This function is defined as

$$f = \frac{C_L^3}{2C_{Mz}^2}.$$  \hspace{1cm} (6)

Note that this functional form assumes that the rotor axis of rotation is in the $+z$ direction. The definition also represents the square of the traditional Figure of Merit function. See [11] for a motivation for this modified form. This function must be specified with a 0 for its boundary index, i.e., it must be applied to the entire configuration and is not available for individual boundary patches. This function is only available for compressible flows.
8.2.3 Rotorcraft Propulsive Efficiency (Keyword: propeff)

This function is defined as

\[ f = \frac{-C_z}{C_{M_z}}. \]  

(7)

Note that this functional form assumes that the rotor axis of rotation is in the +z direction. The minus sign has been introduced to yield a positive efficiency since \( C_{M_z} \) is negative. This function must be specified with a 0 for its boundary index, i.e., it must be applied to the entire configuration and is not available for individual boundary patches. This function is only available for compressible flows.

8.2.4 RMS of Stagnation Pressure (Keyword: pstag)

This function computes the RMS of stagnation pressure in a circular disk that passes through the grid in a specified location and orientation. The user must specify the variables in the following \&pstag_function namelist within fun3d.nml. The defaults are included below:

\[
\begin{align*}
&\text{\&pstag_function} \\
&\quad \text{slice\_orientation} = 1 \\
&\quad \text{disk\_radius} = 1.0 \\
&\quad x\_disk\_origin = 0.0 \\
&\quad y\_disk\_origin = 0.0 \\
&\quad z\_disk\_origin = 0.0
\end{align*}
\]

The scalar integer value of \texttt{slice\_orientation} represents the orientation of the cutting plane. The valid values are 1 (x-plane), 2 (y-plane), or 3 (z-plane). The scalar real value \texttt{disk\_radius} specifies the radius of the cutting plane disk in grid units. The scalar real values of \texttt{x\_disk\_origin}, \texttt{y\_disk\_origin}, and \texttt{z\_disk\_origin} provide the origin of the cutting plane disk in grid units. This function is only available for compressible flows.

8.2.5 Near-field Target \( p/p_\infty \) (Keyword: boom\_targ)

This function is only available for compressible flows and is designed to mitigate sonic boom effects by shaping off-body pressure distributions in the vicinity of an aircraft nominally oriented along the x-axis. The user specifies \( yz \)-coordinate pairs through which rays are passed parallel to the x-axis. In the case of a nonzero angle of attack, the rays are rotated about a user-specified center of rotation to align them with the freestream direction. The user also provides the minimum and maximum x-extent for the rays. A user-specified
number of points are evenly distributed along each ray and the grid element containing each point is identified. See section B.4.30 for guidance on the required namelist inputs.

The functional form is given by

$$f = \sum_{i=1}^{N} \omega_i \left( \frac{p}{p_{\infty}} \bigg|_i - \frac{p}{p_{\infty}} \bigg|_* \right)^2$$

(8)

where $p$ is the local static pressure and. The summation takes place over all points in the rays defined by the user, and the values of $p$ are evaluated at the centroids of the enclosing elements. The values of $\omega_i$ and $\frac{p}{p_{\infty}} \bigg|_*$ are user-supplied pointwise weighting coefficients and target values of $p/p_{\infty}$, respectively, which must be provided in a file named pressure_target.dat. If this file is not present, the target values of $p/p_{\infty}$ are set to 1.0 and the weighting coefficients are set to 1.0. Note that with the above functional form, the target and exponent parameters present in Eq. 1 are usually set to 0.0 and 1, respectively.

A template for pressure_target.dat is typically generated by first extracting a set of $p/p_{\infty}$ distributions for a known configuration by running the optimization driver with Operation to perform set to 1 (“Analysis only”) in ammo.input. Note that the input value weight must be set to .true. and the desired ray extraction parameters must be specified in the &sonic_boom namelist in fun3d.nml. This operation produces a pressure_signatures.dat file which uses the same file format intended for the pressure_target.dat target input file. (Note that the file format is amenable to Tecplot™ usage.) The user may then use the pressure_signatures.dat file to develop a pressure_target.dat input file by modifying the existing pressures to reflect their target values as desired. Note that by specifying weight=.true. in the &sonic_boom namelist, a column of data representing pointwise weighting coefficients (all initially set to 1.0) will be provided in pressure_signatures.dat. This column of data is required to be present in pressure_target.dat. The individual weights may be left as 1.0, or they may be modified on an individual basis to optionally weight a specific region of the signature more or less in the final objective function. A brief example of this file format for a case involving two off-body signatures is shown below. Note that target distributions need not have the same number of locations as, nor line up with, the eventual sampling locations along the extraction rays. FUN3D will linearly interpolate between input target values to obtain values at the sampling locations.

```
VARIABLES = "x", "y", "z", "p/pinf", "weight"
zone t="Signal 1"
   -0.500E+01 0.100E-11 0.826E+00 0.110010E+01 0.100E+01
   -0.472E+01 0.100E-11 0.835E+00 0.110011E+01 0.100E+01
   -0.415E+01 0.100E-11 0.855E+00 0.110012E+01 0.100E+01
   -0.354E+01 0.100E-11 0.876E+00 0.110016E+01 0.100E+01
50
```
8.2.6 Coupled sBOOM Ground-Based Signatures, Noise Metrics, and Equivalent Areas (Keyword: sboom)

This capability uses sBOOM to inversely design ground pressure signatures, optimize a ground-based noise metric, or match equivalent area distributions. [12–14]. Fun3D must be configured and built with the sBOOM library as described in section A.7.13 to use this capability.

In the coupled Fun3D-sBOOM implementation, Fun3D is responsible for computing pressure signals in the immediate vicinity of an aircraft (typically within 10 body lengths). The sBOOM tool then propagates these disturbances to the ground using an augmented Burgers equation that considers effects such as non-linearity, thermo-viscous absorption, and any number of molecular relaxation phenomena during the propagation of waveforms through the atmosphere. In this manner, the user can directly simulate ground-based noise metrics such as A-weighted loudness or compute other loudness measures (e.g., Perceived Level) from the computed ground signatures. In a similar fashion, a coupled adjoint problem is used to determine the discrete sensitivities of the ground-based metrics with respect to any of Fun3D’s typical design parameters which may then be used to optimize the configuration.

sBOOM can generate off-track signatures based on ray theory using user input azimuthal angles. sBOOM can also predict the sonic boom signatures in the presence of wind, turn rate (changing heading angle), climb rate, climb angle, and acceleration (dMach/dt). During maneuvering flight, boom focusing is possible. The current version sBOOM in not able model focusing and will exit with an appropriate message if focusing occurs.

Equivalent area distributions are computed with reversed augmented Burgers equation (when $\text{rs} < (\text{alt} - \text{hg})$) or a direct conversion of off-body pressures (when $\text{rs} > (\text{alt} - \text{hg})$). This is different than the Mach cut equivalent area matching approach in section 8.2.7. The discrete sensitivities of the difference between a target and the computed equivalent areas are provided to Fun3D. The target area is specified with the target_dpress and target_xx variables in the &sboom namelist.

The user must provide inputs relevant to the nearfield pressure signal extraction (see section B.4.30) as well as parameters specific to the sBOOM library (see section B.4.31). Note that when the sboom keyword is used as
the component function name, the actual form of the objective/constraint component function is determined entirely within sBOOM. In this case, the values of $\omega$, $C^*$, and $p$ in Eq. 1 are ignored. This function is only available for compressible flows.

8.2.7 Supersonic Mach Cut Equivalent Area Distribution (Keyword: ae)

This function aims to match a target Mach cut equivalent area distribution for supersonic flows. The Mach cut equivalent area distribution is directly computed from surface pressures and geometry for this function. This is a different approach than the equivalent area computation of sBOOM in section 8.2.6.

The function is defined as

$$f = \sum_{i=1}^{N} \omega_i (L_i + V_i - A^*_i)^2$$

where $N$ represents the total number of longitudinal stations used to sample the solution and geometry for the current azimuth, and $L_i$ and $V_i$ are the lift and volume contributions, respectively, to the current equivalent area. The term $A^*_i$ represents the user-supplied target equivalent area distribution. The $\omega_i$ enables the user to locally weight individual segments of the distribution if desired. Note that with the above functional form, the target and exponent parameters present in Eq. 1 are usually set to 0.0 and 1, respectively. This function is only available for compressible flows, and the configuration is assumed to align with the x-axis.

Any number of desired azimuthal (centerline/off-track) locations may be specified and used as individual component functions. The user must provide the data indicated in the &equivalent_area namelist in fun3d.nml as described in section B.4.32. A centerline symmetry plane may be used to reduce computational expense; in this case, the cutting planes at each longitudinal station will be correctly accounted for on the virtual side of the aircraft. A file ae_target.dat must also be provided, which describes the (optionally weighted) target equivalent area profiles.

A template for ae_target.dat is typically generated by first extracting a set of equivalent area distributions for a known configuration by running the optimization driver with Operation to perform set to 1 (“Analysis only”) in ammo.input. This operation will produce a Tecplot™ file [project]_ae.dat which uses the same file format intended for the target input file ae_target.dat. The user may then use the [project]_ae.dat file to develop a ae_target.dat input file by modifying the existing equivalent areas to reflect their target values as desired. Note that the file [project]_ae.dat contains a column of data representing the pointwise weighting coefficients $\omega_i$ (all initially set to 1.0). This column of data is required to be present in ae_target.dat. The
individual weights may be left as 1.0, or they may be modified on an individual basis to optionally weight a specific region of the distributions more or less in the final objective function. A brief example of this file format for a case involving three azimuthal signatures is shown below. Note that target distributions need not have the same number of locations as, nor line up with, the longitudinal sampling locations. Fun3D will linearly interpolate between input target values to obtain values at the sampling locations. Also note that in the input file ae_target.dat, the second and third columns of the format are ignored.

VARIABLES = "x", "V", "L", "Ae", "weight"
zone t="Ae Function 1"
  -0.01000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.10000E+01
  0.13839E+01 0.25482E-01 -0.26289E-02 0.22853E-01 0.10000E+01
  0.27678E+01 0.47548E-01 -0.64155E-02 0.41133E-01 0.10000E+01
  0.41517E+01 0.76165E-01 -0.10361E-01 0.65804E-01 0.10000E+01
zone t="Ae Function 2"
  -0.01000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.10000E+01
  0.14018E+01 0.25700E-01 -0.26610E-02 0.23039E-01 0.10000E+01
  0.28036E+01 0.48216E-01 -0.64628E-02 0.41752E-01 0.10000E+01
  0.42054E+01 0.77379E-01 -0.10358E-01 0.67020E-01 0.10000E+01
  0.56072E+01 0.11457E+00 -0.14045E-01 0.10052E+00 0.10000E+01
  0.98126E+01 0.30728E+00 -0.25726E-01 0.28156E+00 0.10000E+01
zone t="Ae Function 3"
  -0.01000E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.10000E+01
  0.14155E+01 0.26009E-01 -0.26166E-02 0.23392E-01 0.10000E+01
  0.28310E+01 0.48902E-01 -0.62883E-02 0.42614E-01 0.10000E+01
  0.42465E+01 0.78591E-01 -0.10011E-01 0.68579E-01 0.10000E+01

Finally, the solver will also provide the user with a Tecplot™ output file [project]_ae_cuts.i.dat for the ith specified equivalent area function. These files contain the actual cross-sectional slices of the aircraft that were generated for each azimuthal function.

8.2.8 Target Pressure Distributions (Keyword: cpstar)

Fun3D has an inverse design capability where the objective function may be composed of target pressure distributions. The file containing the jth target distribution must be named cpstar.data.j. However, setup is tedious, primarily due to the difficulty in specifying pressure distributions on a three-dimensional configuration. If this capability is of interest, please contact Fun3D-Support@lists.nasa.gov for more detailed guidance.

8.3 Geometry Parameterizations

In order to perform shape optimization, Fun3D must be provided with a set of design variables describing the geometric shape of the configuration.
FUN3D is currently set up to interface directly with geometry parameterizations provided by MASSOUD [15], Bandaids [16], or Sculptor™. MASSOUD and Bandaids are software packages developed by Jamshid Samareh of NASA Langley (Jamshid.A.Samareh@nasa.gov). Users should contact him for copies of the software; tutorial information for these tools is available on the FUN3D website. These packages allow the user to parameterize completely arbitrary shapes using a free-form deformation technique. The packages are very efficient, robust, and also provide analytic Jacobians of the parameterization, which are necessary for FUN3D-based design. Sculptor™ is a popular commercial package developed by Optimal Solutions and also provides the necessary data for FUN3D-based design. Note that any combination of parameterizations based on these tools may be used within the context of a single optimization. For example, the planform of a wing or tail surface may be best treated using MASSOUD, while Bandaids or Sculptor™ may be most appropriate for a wing-body fillet region or a feature such as a fuselage protuberance.

8.3.1 Surface Grid Extraction

To parameterize a surface grid using any of the above tools, it must first be extracted to a Tecplot™ file. To do this, add a &massoud_output namelist to fun3d.nml to group all of the required boundary patches for a body to be parameterized into a single body (see also section B.4.25):

```
&massoud_output
 n_bodies = 2 ! parameterize 2 bodies: wing and tail
 nbndry(1) = 6 ! # of bounds that comprise wing
 boundary_list(1) = '3-8' ! wing bounds (account for lumping!)
 nbndry(2) = 3 ! # of bounds that comprise tail
 boundary_list(2) = '9,10,12' ! tail bounds (account for lumping!)
/
```

Note that the boundary indices shown here must reflect any patch lumping that may have been requested in the &raw_grid namelist (see also section B.4.2). A single iteration of the flow solver should now be executed with the --write_massoud_file command line option. This will generate a [project]_massoud_bndry#.dat file for each of the boundary groups present in the &massoud_output namelist. These files contain the information necessary to parameterize the surface grid using any of the aforementioned tools. See the documentation for those packages for further instructions on how to construct the actual parameterization.

8.3.2 Access to Executables

If MASSOUD or Sculptor™ is being used for parameterizations, the executable for those packages must be available in the runtime PATH, and must be named
massoud or sculptor, respectively. The optimization driver supplied with Fun3D will attempt to call these executables if such parameterization types are present. If Bandaids are being used, no additional executables must be supplied; all Bandaid evaluations are handled internally by Fun3D.

8.3.3 Notes on Using Sculptor™

If Sculptor™ is being used, Fun3D will invoke Sculptor™ in batch (non-GUI) mode during the course of the optimization. However, current versions of Sculptor™ will still attempt to communicate with an X server, even when run in this fashion. If the system does not run an X server (such as compute nodes on a cluster), then a fake X server such as Xvfb is recommended. You will need to execute the fake server prior to running the design optimization. For example, a run script may have the following commands:

```
Xvfb :1 &
export DISPLAY=:1.0 # for bash
setenv DISPLAY :1.0 # for c shell
[any command that uses Sculptor]
```

The syntax here may vary; if this does not allow the optimization driver to run Sculptor™ in batch mode successfully on the system, the user should get in touch with Sculptor™ support for assistance.

In addition, the parameterization of all bodies treated using Sculptor™ must be bookkept within a single set of Sculptor™ input files. For example, in the wing-tail example above, both bodies must be contained in a single instance of Sculptor™ files. Therefore, the &massoud_output namelist described above should group all of the desired boundaries necessary to describe the geometry(s) of interest into a single body:

```
&massoud_output
  n_bodies = 1 ! wing and tail grouped into a body
  nbndry(1) = 9 ! # of tail and wing bounds
  boundary_list(1) = '3-10,12' ! wing and tail boundaries
/
```

Each of the desired bodies may be worked on independently within Sculptor, but they must ultimately appear as a single body to Fun3D.

8.3.4 Using Other Parameterization Packages

Finally, the Fun3D interface for external geometric parameterizations has been designed to accommodate any parameterization tool desired. The tool must provide the surface mesh coordinates as a function of some vector of design variables for the body of interest. The partial derivatives of these
coordinates with respect to the design variables must also be supplied. Contact Fun3D-Support@lists.nasa.gov for further guidance in using an alternative parameterization package. See [17] for an example of such an approach.

8.4 Design Optimization Directory Structure

The optimization driver opt_driver requires a very specific directory structure. It can be established by running opt_driver in an interactive mode with the --setup_design command line option. The number of design points should be 1 for single-point design or greater than 1 for multi-point design.

\[
\text{opt_driver} \ --\text{setup-design [number of design points]}
\]

This interactive command will prompt the user for several directory paths required by the optimization, namely the paths to the FUN3D source code, the configuration directory where FUN3D was configured and built, and the path to the location where the design will be performed. Here, directories should be provided as absolute paths contained in single quotes, with trailing slashes omitted, i.e.,

'absolute/path'

At the completion of this setup procedure, a summary of the files required from the user will be echoed to the screen. The directories created in the specified run location are shown in Table 11. The i suffix in description.i

<table>
<thead>
<tr>
<th>Directory</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ammo</td>
<td>Location where optimization will be executed</td>
</tr>
<tr>
<td>description.i</td>
<td>Location of all baseline input files describing design point i</td>
</tr>
<tr>
<td>model.i</td>
<td>Location where analysis &amp; sensitivity analysis of design point i will be performed</td>
</tr>
</tbody>
</table>

and model.i represents the design point index. For single-point design, this will be 1; for multi-point design, this value will range from 1 to the number of user-specified design points. The setup procedure will populate the various directories with links to the required FUN3D executables and templates for various input files described below.

8.5 Contents of the ammo Directory

The ammo directory will contain files related to the optimization procedure itself. This includes the ammo.input input file and a link to the opt_driver executable.
8.5.1 **ammo.input**

The input parameters contained in this file control the actual optimization procedure. A template of the file will be installed during the setup procedure; an example is also provided below.

```
Optimization package
4
Base directory from which to run optimization
'/net/aamber/nielsen/TestAMMO'
Number of design points
2
Weights for each design point
1.0
1.5
Operation to perform
1
Restart the optimization
0
Maximum number of flow solves
10
Maximum number of design cycles
5
Relative convergence criterion for subproblem
1.e-5
Absolute feasibility tolerance for constraint violation
1.0
Number of bodies with spatial transforms
2
List of bodies with spatial transforms
2 3
Body grouping desired
0
Executable for running MPI programs
'mpirun'
Number of processors from which to run adjoint solver
1024
DOT method
0
```

The various inputs specified in the **ammo.input** input file are described below. Care should be taken to preserve the structure of this file when modifying its contents.

**Optimization package** This scalar integer specifies the optimizer to be used. The available choices are (1) DOT/BIGDOT™, (3) KSOPT, (4) PORT, (5) NPSOL™, and (6) SNOPT™. Note that the PORT package does not support the use of explicit constraints. Also note that Fun3D must be configured and built against the selected library.
Base directory from which to run optimization  This should be an absolute path to the design location specified during the setup procedure. Path should be enclosed in single quotes and contain no trailing slashes.

Number of design points  This scalar integer is the number of design points to be considered during the optimization. The value should be at least 1 and less than or equal to the number of design points specified during the setup procedure.

Weights for each design point  A non-negative real-valued scalar should be specified on separate lines for each design point. The value represents the weighting to be applied in the linear combination of objective functions from each individual design point as used to construct the final composite objective function (see section 8.10). For single point optimization, a single value of 1.0 should be specified.

Operation to perform  This scalar integer specifies what operation the optimization driver should perform. The valid values are (1) Analysis only, (2) Analysis and sensitivity analysis, and (3) Optimization.

Restart the optimization  This scalar integer specifies whether to start the optimization from the baseline problem description (0), or to restart the optimization from a previous optimization run already executed in this directory (1).

Maximum number of flow solves  This scalar integer is only relevant for PORT-based optimizations and sets an upper limit on the number of flow solutions allowed during the design.

Maximum number of design cycles  This scalar integer sets an upper limit on the number of design cycles the optimizer may perform.

Relative convergence criterion for subproblem  This scalar real value is only relevant for DOT/BIGDOT™- and PORT-based optimizations and specifies the relative function convergence criterion for which the optimization will terminate.

Absolute feasibility tolerance for constraint violation  This scalar real value is only relevant for optimizations based on the DOT/BIGDOT™, NPSOL™, and SNOPT™ packages. The value specifies the feasibility tolerance for constraints.
Number of bodies with spatial transforms  This scalar integer specifies the number of MASSOUD-parameterized bodies for which spatial transforms should be applied. See also section 8.6.9.

List of bodies with spatial transforms  This is a list of integers separated by spaces specifying the MASSOUD-parameterized bodies to which spatial transforms are to be applied. There should be Number of bodies with spatial transforms entries in this list. If Number of bodies with spatial transforms is zero, this line of data should not be present. See also section 8.6.9.

Body grouping desired  This scalar integer specifies whether any body grouping should be applied. A value of 0 indicates no grouping should be applied; a value of 1 indicates grouping should be applied. See also section 8.6.4.

Executable for running MPI programs  This single character string enclosed in single quotes will be used as a prefix when running MPI programs. This is usually mpirun or mpiexec, depending on the MPI implementation, or perhaps aprun on Cray® systems.

Number of processors from which to run adjoint solver  This scalar integer specifies the number of processors on which to execute the adjoint solver. Normally this is the same number of processors requested for the job and used for the flow solver. However, in the event of a split communicator in the flow solver (for Suggar++, VisIt, dedicated file I/O, etc), the adjoint solver must be run on the same number of processors that the actual flow solver was run on (does not include processors set aside for split communicator functionalities).

DOT method  This scalar integer is only used for DOT/BIGDOT™-based optimization and specifies the optimization method to be used with DOT/BIGDOT™. See the DOT/BIGDOT™ documentation for further information.

8.6 Contents of the description.i Directory

The description.i directory serves as a repository for the baseline files for the CFD model, the geometric parameterization, and several other input files related to the computational model for the i-th design point. These files must be set up by the user prior to the run and will not be modified by Fun3D during execution. During the initial setup procedure, templates for several input files will be placed in this location to aid in setting up the case. During the
actual optimization, the optimization driver will copy files from this directory into the `model.i` directory as needed.

Any files normally required by the flow solver must be present in this directory. This would typically include the grid and boundary condition files and `fun3d.nml`. If the mesh uses overset grids assembled with the Suggar++ utility, the Suggar++ DCI file must be present as well. The optional file `remove_boundaries_from_force_totals` (section B.3) may also be present, if desired.

In addition to the files normally required by the flow solver, a number of other files must also be present to perform the design optimization, some of which are optional. These are described below.

### 8.6.1 Geometry Parameterization Files

If performing shape optimization, the user must provide the relevant parameterization files for each body in the mesh to be modified. The specific set of files required for each body depends on the parameterization package(s) being used.

**MASSOUD Parameterizations** For MASSOUD parameterizations, the MASSOUD parameterization files should be named `design.gp.j`, where `j` is the index of the body to be designed. The files specifying the values of the raw MASSOUD variables should be named `design.j` for each of the bodies to be designed. For Fun3D-based design, the custom design variable linking feature of MASSOUD must be used. If the raw MASSOUD variables are intended to be used as-is, simply set the linking matrix as the identity matrix in the MASSOUD `.usd` file. These files specifying the design variable linking for each body should be named `design.usd.j`.

The MASSOUD control file specifies the names of the files outlined above for MASSOUD and must be provided as `massoud.j` for the `j`th body. The files listed in the MASSOUD control file must reflect these names. The first line of the MASSOUD control file(s) must have a positive integer equal to the number of custom design variables. If the intent is simply to use the raw MASSOUD variables as-is, this value is simply the number of raw MASSOUD variables for that body. For the in/out-of-core parameter, use in-core (0). The file name for Tecplot™ output viewing must be named `model.tec.j` for the `j`th body. The design variable grouping file specified should be named `designVariableGroups.j` for the `j`th body. The FAST output file name can be named anything the user wishes; the Fun3D tools do not use this MASSOUD output file. Finally, the user design variable file for the `j`th body should be named `customDV.j`. In summary, a `massoud.j` control file for the `j`th body should look like the following:
#MASSOUD INPUT FILE
# runOption 0-analysis, >0-sd users dvs, -1-sd massouds dvs
52
# core 0-incore solution, 1-out of core solution
0
# input parameterized file
design.gp.1
# design variable input file
design.1
# input sensitivity file - used for runOption > 0
design.usd.1
# output file grid file
newframe.fast.1
# output Tecplot file for viewing
model.tec.1
# file containing the design variables group
designVariableGroups.1
# user design variable file
customDV.1

**Bandaid Parameterizations** For Bandaid parameterizations, the input files created by the Bandaid setup tool should be named `bandaid.data.j` for the jth body. Because Bandaid parameters behave linearly, the sensitivities contained in these files are constant and this input is all that is required during the course of a design.

**Sculptor™ Parameterizations** For Sculptor™ parameterizations, the user must provide `[project].mdf`, `[project].sd1`, `[project].vol`, and `[project].stu` files. See the Sculptor™ documentation for more details on each of these files. A file named `[project].def` must also be provided. An example `[project].def` file for a simple two-body parameterization is shown below:

```
set_mdf [project].mdf
default 1 DV1-T1 0.00
default 1 DV1-T2 0.00
default 1 DV1-T3 0.00
default 1 DV1-T4 0.00
default 1 DV1-T5 0.00
default 2 DV2-T1 0.00
default 2 DV2-T2 0.00
default 2 DV2-T3 0.00
export model.tec.1
exit
```
The filename specified for the `export` command must be `model.tec.1`. The remainder of the file is dictated by the specific parameterization developed in the Sculptor™ application.

After the configuration has been parameterized using Sculptor™ and all of the appropriate files have been assembled for FUN3D-based design, a copy of the original `[project]_massoud_bndry#.dat` file must also be placed in the `description.i` directory, but it must be renamed `[project].sd1`. Sculptor™ requires this baseline file during the optimization.

Finally, prior to performing the design, the `[project].sd1` file must be read into Sculptor™ in GUI mode as “Import Mesh/CFD as Tecplot Point FE.” Following this, the Sculptor volumes need to be imported onto the `[project].sd1` file, and then the model must be saved again. Once this is done, the command `export model.tec.1` within the `[project].def` batch script will generate a `model.tec.1.sd1` file as needed for FUN3D-based design optimization.

8.6.2 rubber.data

This section describes how to set up each block of the design control file `rubber.data`. The template provided in the `Adjoint` directory of the source code distribution is installed in the `description.i` directory during setup. This file serves as the primary control file during the course of the optimization and stores all of the high-level information relevant to the design. The file is repeatedly read and updated by the various tools during the design procedure. A simple example of this file to be used for discussion purposes is shown below.

```plaintext
# Global design variables (Mach number / angle of attack)
Var Active Value Lower Bound Upper Bound
Mach 0 0.800000000000000E+00 0.000000000000000E+00 0.900000000000000E+00
AOA 1 1.000000000000000E+00 0.000000000000000E+00 5.000000000000000E+00

# Number of bodies
2

# Rigid motion design variables for 'wing'
Var Active Value Lower Bound Upper Bound
RotRate 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
RotFreq 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
RotAmpl 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
RotOrgx 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
RotOrgy 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
RotOrgz 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
RotVecx 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
RotVecy 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
RotVecz 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
TrnRate 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
TrnFreq 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
TrnAmpl 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
TrnVecx 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
TrnVecy 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00
TrnVecz 0 0.000000000000000E+00 0.000000000000000E+00 0.000000000000000E+00

Parameterization Scheme (Massoud=1 Bandaids=2 Sculptor=4)
```
Number of shape variables for 'wing'

<table>
<thead>
<tr>
<th>Index</th>
<th>Active</th>
<th>Value</th>
<th>Lower Bound</th>
<th>Upper Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
<td>2.00000000000000E+00</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>1.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
<td>2.00000000000000E+00</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>1.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
<td>2.00000000000000E+00</td>
</tr>
</tbody>
</table>

Rigid motion design variables for 'tail'

<table>
<thead>
<tr>
<th>Var</th>
<th>Active</th>
<th>Value</th>
<th>Lower Bound</th>
<th>Upper Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>RotRate</td>
<td>0</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
</tr>
<tr>
<td>RotFreq</td>
<td>0</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
</tr>
<tr>
<td>RotAmpl</td>
<td>0</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
</tr>
<tr>
<td>RotOrgx</td>
<td>0</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
</tr>
<tr>
<td>RotOrgy</td>
<td>0</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
</tr>
<tr>
<td>RotOrgz</td>
<td>0</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
</tr>
<tr>
<td>RotVecx</td>
<td>0</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
</tr>
<tr>
<td>RotVecz</td>
<td>0</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
<td>0.00000000000000E+00</td>
</tr>
</tbody>
</table>

Parameterization Scheme (Massoud=1 Bandaids=2 Sculptor=4)

Number of shape variables for 'tail'

<table>
<thead>
<tr>
<th>Index</th>
<th>Active</th>
<th>Value</th>
<th>Lower Bound</th>
<th>Upper Bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>2.00000000000000E+00</td>
<td>-1.00000000000000E+00</td>
<td>5.00000000000000E+00</td>
</tr>
</tbody>
</table>

Function Information

Number of composite functions for design problem statement

Cost function (1) or constraint (2)

If constraint, lower and upper bounds

Number of components for function 1

Physical timestep interval where function is defined

Components of function 1: boundary id (0=all)/name/value/weight/target/power

Current value of function 1

Current derivatives of function wrt global design variables

Current derivatives of function wrt rigid motion design variables of body 1
Current derivatives of function wrt shape design variables of body 1

Current derivatives of function wrt rigid motion design variables of body 2

Current derivatives of function wrt shape design variables of body 2

 ############################################################################
Cost function (1) or constraint (2)
If constraint, lower and upper bounds
-0.03 -0.01
Number of components for function 2
1
Physical timestep interval where function is defined
1 1
Composite function weight, target, and power
1.0 0.0 1.0
Components of function 2: boundary id (0=all)/name/value/weight/target/power
0 cmy 0.000000000000000 1.000 0.00000 1.000
Current value of function 2
0.000000000000000
Current derivatives of function wrt global design variables
0.000000000000000

Current derivatives of function wrt rigid motion design variables of body 1

Current derivatives of function wrt shape design variables of body 1

0.000000000000000
0.000000000000000
0.000000000000000
0.000000000000000
0.000000000000000
0.000000000000000

Current derivatives of function wrt rigid motion design variables of body 2

Current derivatives of function wrt shape design variables of body 2

64
Global Design Variable Data  This section of rubber.data lays out global
design variables for the computation. The Mach number and angle of attack
variables appear on their own row in the file and have several attributes that
must be set by the user. The first column is a dummy index and is merely
to assist the user in quickly navigating through the file. The second column
is a toggle to activate the design variable. If this value is a 1, the variable
will be allowed to change during the design. If the value is assigned a 0, this
variable will be held constant at the value specified. For incompressible flows
and mixed-element grids, the Mach number must be declared inactive.

The third column in the design variable block is the current value for
this design variable. The values of any active variables in this file will take
precedence over other input decks during design. For example, the flow solver
will run an angle of attack of 1 degree in this case, regardless of what may be
specified in fun3d.nml. Columns four and five specify the upper and lower
bounds for the current design variable.

Body-Specific Design Variable Data  The next input following the Mach
number and angle of attack entries specifies the number of bodies for which
the user has provided shape parameterizations. Note that not every body in
the grid must be included here. If the wing of an aircraft is the sole focus of
the optimization, there is no need to account for other boundaries such as the
tail or fuselage here.

Following the number of bodies, there should be two blocks of design vari-
ables for each desired body, namely a list of rigid motion variables controlling
the dynamics of the body, and a set of shape parameters controlling the shape
of the body. The columns of inputs are identical to those described above for
Mach number and angle of attack.

The bodies present in the computation may be listed in any order; how-
ever, the order of their appearance in this control file must match the integer suffix on their parameterization files that are provided in the description.i directory, as well as files such as body_grouping.data, transforms.j, etc.

The first section for the current body specifies design variables governing rigid body motion and is only applicable for time-dependent problems. For optimization of steady flows and/or static geometries, the rigid motion data is irrelevant but must be present in this file. These variables should be set as inactive in these cases.

The next block of inputs relates to the shape parameterization for the current body. First, the parameterization scheme is identified by a scalar integer. The following values are available: (1) MASSOUD, (2) Bandaids, and (4) Sculptor™. The next input specifies the number of parameterized shape variables on the current body and the subsequent lines lay out the design variable information for that body. A row of data must be provided for every variable in the parameterization, whether it is active or not. (Note however, that internally, the optimizer is only made aware of the variables marked as active.) If a parameterization contains 25 variables, then 25 rows must appear in this corresponding block of rubber.data, even if only a subset is active. If the design variable linking feature in MASSOUD or Bandaids has been used to create additional derived variables, they must also appear here. Note that the “Active” attribute for shape variables may take values of not only 0 or 1, but also −1 in certain multi-point design scenarios (see section 8.10).

Care should be taken in choosing upper and lower bounds for shape variables. Optimizers tend to fully explore the design space, which may result in infeasible shapes (or extreme shapes the mesh movement/solvers cannot handle robustly). Set these limits conservatively; one can always restart a design with less restrictive bounds.

As noted previously, when using Sculptor™ parameterizations for multiple bodies, all such design variables must appear as a single concatenated body in rubber.data.

**Cost Function/Constraint Specification** The first line following the design variable block specifies the total number of composite functions to be used as objectives or constraints for the current design point. Multiple composite objective functions may be specified in certain cases; see section 8.9. Otherwise, a single composite objective function must be specified. The example file shown here contains a single composite objective function based on the lift coefficient and a single explicit constraint based on the pitching moment. Note that explicit constraints may only be specified if the optimization package chosen in ammo.input supports them.

Following the scalar value specifying the total number of composite objectives and constraints, each composite function and/or constraint will have a block of data associated with it. Objective functions and constraints may be
The first two inputs in the composite function block specify a scalar integer indicating how the current function is to be viewed by the optimizer. The two subsequent inputs represent lower and upper bounds on the function if it is to be used as a constraint. If the function is an objective function, the first input value should be 1, and the lower and upper bounds must be present but their values are irrelevant. However, if the current function is to be used as a constraint, special attention must be paid to these inputs depending on the optimization package being used.

**Constraints Using NPSOL™ and SNOPT™** If the current function is to be used as an inequality constraint, the first input should be 2, and the lower and upper bounds should be set to their appropriate values. If the current function is to be used as an equality constraint, the first input should be 2; however, the lower and upper bounds should both be set equal to the desired constraint value.

**Constraints Using KSOPT and DOT/BIGDOT™** These optimization packages assume constraint functions of the form $f \leq 0$, such that the bound of the feasibility region is implicit in the function definition and the lower and upper bound inputs must be present but are not used. If the current function is intended as an inequality constraint, the first input should be 2. If the current function is intended as an equality constraint, the first input value should be 3. In this case, Fun3D’s design driver will provide the current function to the optimizer as an inequality constraint, but will also bookkeep an equal and opposite function as an additional inequality constraint. In this manner, an equality constraint is achieved by only allowing the intersection of the two inequality constraints as feasible.

Following the classification of the current function, the next line states how many component functions comprise the current composite function. This can be any positive integer greater than or equal to 1. Following the number of component functions, the user must specify the physical time step interval over which the function is to be applied. This input is only relevant to optimization of unsteady flows. For steady flows, the values of these two inputs are ignored but must be present.

The weight, target, and power to be applied to the current composite function are specified next. These values are only relevant when combining multiple composite objective functions into a single global objective function (see section 8.9). For all other cases, these values should be specified as 1.0, 0.0, and 1.0, respectively.

At this point, each component function that contributes to the current composite function has a line specifying several pieces of data. The first col-
umn is the boundary patch over which to apply the current component. This index corresponds directly to the boundary patches in the CFD grid, and must reflect any patch lumping that is indicated in the &raw.grid namelist in fun3d.nml (see section B.4.2). If a component function is to be used over the entire grid (total drag, for example), simply put a 0 in this column. Alternatively, if a single boundary patch is to be targeted, one might apply the component function to only that patch. Several patches may be targeted by including a component function for each. The next column is the keyword for the aerodynamic quantity to be used for the current function component. For a list of available keywords, see section 8.1. The next column contains the current value of the current function component. This is an output value during the optimization and need not be set by the user. The final three columns in the row correspond to the weight, target value, and power to be applied to the current component function in constructing the overall composite function.

**Current Function Value and Sensitivities** Following the specification of the component functions, the next line of rubber.data contains the current value of the composite function. This is an output and need not be set by the user.

The remaining lines in the current function block contain the sensitivity derivatives with respect to all of the design variables listed in the top half of the file. This section is divided into derivatives with respect to the global design variables, as well as the rigid motion and shape design variables for each of the bodies laid out in the top portion of the file. These derivatives are outputs set by Fun3D and not by the user. However, a line for each design variable (both global variables as well as body-specific variables) must be provided in each composite function block present. The values do not matter, but the solvers need positions available in the file to store the current values.

### 8.6.3 ae_target.dat (optional)

If the function keyword ae is specified anywhere in rubber.data, the file ae_target.dat must be present prior to performing the optimization. This file provides the target equivalent area distribution(s) for each of the azimuthal locations specified in the &equivalent_area namelist in fun3d.nml (in the same order). See section 8.2.7 and section B.4.32.

### 8.6.4 body_grouping.data (optional)

This file is used to specify body grouping information. For example, if the objective function is the Figure of Merit FM for a three-bladed rotor, then the three blades (each typically specified as a separate parameterized body in
rubber.data) should be associated into one group, so that sensitivity derivatives will reflect a composite $\partial (FM) / \partial D$ for all three blades. This capability requires that the bodies to be associated all have identical parameterizations (same number of design variables on each body, etc). The format of the body_grouping.data file is as follows:

```
Number of groups to create
1
Number of bodies in group, list of bodies
3
1 2 3
```

The first scalar integer specifies the number of groups to create (i.e., one rotor). The next set of inputs specifies the number of bodies in each group, followed by the bodies that comprise that group (i.e., each of the three rotor blade bodies).

**8.6.5 command_line.options (optional)**

The command_line.options file specifies any command line options to be used with the flow solver, the adjoint solver, or the MPI job launcher (mpirun, mpiexec, aprun, etc). An example of this file is shown below.

```
3
1 flow
'--rmstol 1.e-7'
1 adjoint
'--rmstol 1.e-3'
2 mpirun
'--nolocal'
'--machinefile ../machinefile'
```

The first line of the file specifies the number of programs for which command line options are being provided. The subsequent line must contain an integer followed by a keyword. The integer specifies how many command line options are being provided for the code identified by the keyword. The valid keywords are flow, adjoint, and mpirun. This line is followed by a line for each of the command line options provided for the code identified by the keyword. Each command line option should appear in single quotation marks on its own line. The specified programs and their associated command line options may appear in any order.
8.6.6 cpstar.data.j (optional)

FUN3D has an inverse design capability where the objective function may be composed of target pressure distributions. The file containing the jth target distribution must be named cpstar.data.j. However, setup is tedious, primarily due to the difficulty in specifying pressure distributions on a three-dimensional configuration. If this capability is of interest, please contact Fun3D-Support@lists.nasa.gov for more detailed guidance.

8.6.7 machinefile (optional)

If the optimization will be executed in an environment which requires an explicit list of machines on which the MPI jobs will be executed, this file must be present. It should take the format required of the particular MPI implementation being used. If the optimization will be executed in an automated queuing environment, the job scheduler normally assigns the machines to be used at runtime and this file is therefore not required.

8.6.8 pressure_target.dat (optional)

If the function keyword boom_targ is specified anywhere in rubber.data, the file pressure_target.dat must be present prior to performing the optimization. This file provides the nearfield target $p/p_{\infty}$ distribution(s) for each of the off-body locations specified in the &sonic_boom namelist in fun3d.nml (in the same order). See section 8.2.5 and section B.4.30.

8.6.9 transforms.i (optional)

Since MASSOUD uses a coordinate system specific to an assumed aircraft orientation, it is sometimes necessary to reorient a body from its physical position to a MASSOUD-aligned coordinate system and vice-versa. Examples might include a vertical tail or the various blades of a rotor system. The file describing the transform for the ith body should be included as transforms.i. The format of a typical transforms.i file is as follows:

```
ROTATE 0.0 0.0 1.0 -120.0
```

This would rotate the MASSOUD parameterization for the ith body by $-120$ degrees about a unit vector in the $+z$ direction. The commands TRANSLATE and SCALE are also available.

8.7 Contents of the model.i Directory

Just as for the description.i directories, the i in the model.i naming convention represents the design point index. This value is 1 for single point design
or the design point index for multi-point design. The `model.i` directory contains the subdirectories `Flow`, `Adjoint`, and `Rubberize`. During the course of the design procedure, Fun3D will evaluate the relevant parameterizations and perform flow and adjoint solutions within these locations. These subdirectories are populated during the initial setup procedure with links to the required executables from the user’s Fun3D installation. Files in the `model.i` subdirectories should not be modified by the user, although one may wish to observe various solver output files during the course of the optimization. All user-provided inputs are confined to files located in the `description.i` and `ammo` directories.

### 8.8 Running the Optimization

Once all of the required inputs and files have been provided, the user should first request a single function evaluation from the optimization driver. This is strongly recommended in order to identify any potential issues in the various inputs. To perform this check, set the value of `Operation to perform` in `ammo.input` to 1, and execute the optimization driver from the `ammo` subdirectory:

```bash
opt_driver > screen.output &
```

Here, the output from the optimization driver has been redirected to a file called `screen.output`. This file is very useful if a problem needs to be diagnosed with the execution. It is also good practice to include this file with help requests to Fun3D-Support@lists.nasa.gov.

At the completion of the function evaluation, the user should check for the desired/expected result. This is also a good opportunity to establish reasonable values for the number of time steps to run, the residual tolerance at which the solver should quit, and so forth. Such run control parameters may be set in `fun3d.nml` or via the `command_line.options` file.

Once the function evaluation procedure has been verified, the user should perform the same test for a sensitivity analysis by setting `Operation to perform` in `ammo.input` to 2 and re-executing the optimization driver. Similar checks on convergence parameters, etc for the adjoint solver should be noted and applied to the relevant input files.

With successful function and gradient evaluations in hand, an actual optimization may be initiated. The value of `Operation to perform` in `ammo.input` should be set to 3, and the optimization driver can be executed as before. The user should closely monitor the screen output as the process proceeds, especially during the first several design cycles when input parameters may first cause problems. The largest changes in design variables often occur early on as well, which can cause issues with mesh movement operations, solver convergence, and other aspects. It is also very useful to occasionally filter the
screen output for the current function value(s) reported at the completion of each flow solution in order to monitor overall progress:

    grep "Current value of function" screen.output

When an optimization completes, the optimizer will report the reason for the termination to the screen, which may be a local minimum, or some problem encountered during the simulation. A summary of the optimization is provided by each optimization package in the file(s) noted in Table 12. The final set of design variables and function/constraint values determined by the optimizer will be available in `model.i/rubber.data`. To track the history of the optimization, a backup of all intermediate copies of `rubber.data` are stored in the directory `model.i/Rubberize/surface_history`. Intermediate copies of the surface grids developed during the design process are also stored in this location as `model.tec.j.sd1.iter`, where `j` is the body index, and `iter` is the design iteration. These files may be used to produce animations of the surface history if desired. Using the broad range of visualization output options in the flow solver, the user has great freedom to produce customized animations during the course of the design.

<table>
<thead>
<tr>
<th>Optimization Package</th>
<th>Summary File(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>DOT/BIGDOT™</td>
<td>dot.output</td>
</tr>
<tr>
<td>KSOPT</td>
<td>ksopt.output</td>
</tr>
<tr>
<td>PORT</td>
<td>port.output</td>
</tr>
<tr>
<td>NPSOL™</td>
<td>npsol.printfile, npsol.summaryfile</td>
</tr>
<tr>
<td>SNOPT™</td>
<td>snopt.printfile, snopt.summaryfile</td>
</tr>
</tbody>
</table>

### 8.8.1 Filesystem Latency Problems

Design optimization using some cached file systems may experience problems due to the rapid execution of the various tools during the design process. In some cases, a file system lag may cause some processes to receive older/stale versions of files during execution. Specifying the `--sleep_delay [seconds]` command line option to the `opt_driver` executable will pause the optimization process with a sleep duration of `seconds` between subsequent code executions to allow the file system to perform correctly. On older systems, delays as large as 60 seconds are sometimes necessary; more recent systems seem to perform considerably better and values of 5-10 seconds are often sufficient.

### 8.9 Multi-objective Design

KSOPT, PORT, and SNOPT™ are the only packages currently supported for use with multi-objective design. Details on the usage for each package are
8.9.1 KSOPT
KSOPT is the only supported optimization package with explicit support for multiple objective functions. When using KSOPT, the user may designate any number of composite functions as objective functions in rubber.data.

8.9.2 PORT, SNOPT™
The Fun3D design driver offers a simple approach to scalarizing multiple user-specified objective functions for use with the PORT or SNOPT™ packages. If multiple composite functions are specified in rubber.data, the Fun3D design driver will combine them using the weight, target, and power values specified at the composite function level (i.e., the input values that appear just before the component function data is specified in rubber.data, see section 8.6.2). If \( N \) composite functions \( f \) are labeled as objective functions in rubber.data, the scalarized objective function \( F \) to be provided to the optimization procedure will take the form

\[
F = \alpha_1(f_1 - f_{1*})^{p_1} + \alpha_2(f_2 - f_{2*})^{p_2} + ... + \alpha_N(f_N - f_{N*})^{p_N} \tag{10}
\]

where \( \alpha_i, f_{i*} \), and \( p_i \) are the weight, target, and power values associated with each composite function in rubber.data.

8.10 Multi-point Design
The Fun3D design infrastructure offers several approaches to multi-point optimization. This refers to design problems where the user may wish to simultaneously optimize a configuration for operations at two different conditions — perhaps the beginning and end of a cruise segment for example, where the aircraft weight may be substantially different. The user may also wish to design for cruise and takeoff or landing (or all three). The various design points may be characterized by different flow conditions (i.e., speed, angle of attack, etc), or more generally, by the geometries (and therefore grids) at each point. For example, one design point may consist of a cruise geometry operating at Mach 0.8, while another design point may be a landing configuration operating at Mach 0.2 with a high-lift system deployed. For examples of Fun3D-based multi-point design in practice, see the studies in [11] and [18]. In these references, a tilt-rotor geometry has been optimized for a set of several blade collective settings as well as hover and forward flight conditions.

To perform a multi-point optimization, the user must request the desired number of design points when setting up the directory structure where the design will be performed (see section 8.4). The user must populate each of the
directories for each design point just as in the single-point design context. The order of the design points does not matter. The value of Number of design points in ammo.input should be set appropriately. Ultimately, Fun3D provides several ways to formulate the multi-point design problem. These approaches are outlined below.

In general, the optimizer will be seeking a unique set of design variables to simultaneously achieve goals at all of the design points. For this reason, a consistent set of design variables across all design points must be used. This applies to the global variables Mach number and angle of attack as well as any body-specific variables such as shape parameters. For example, if a set of 15 thickness variables is provided for a wing shape in cruise, other design points (again, perhaps a landing configuration as an example) must utilize the same set of 15 thickness variables. Moreover, the same subset of design variables must be active at each design point.

### Multi-valued Design Variables

In some situations, the user may desire different optimal values of a design variable at different design points. For example, consider power minimization for a rotor in hover at two different weight conditions, where each of the two design points may have different minimum thrust coefficients posed as constraints. In addition to other design variables that may be present, the user may have a shape parameter controlling the blade collective setting (blade pitch). However, rather than constraining the optimal blade collective to a single unique value, the user may desire separate, optimal values for each design point. As another example, consider a configuration with an ability to actively morph its outer mold line. In this case, the user may wish to determine optimal values of the shape parameters that are unique to different design points.

To accommodate such multi-valued design variables, the user may set the “Active” attribute for individual shape design variables to −1 in rubber.data (see section 8.6.2). If this is done, it must be applied consistently for that same variable across all design points. For variables with this attribute, the Fun3D design driver will internally bookkeep separate values of the variable for each design point. This feature is currently only available for use with the SNOPT package.

### 8.10.1 Linear Combination of Objective Functions

The most straightforward approach to multi-point design is to linearly combine individual objective functions $f_i$ from each of the $N$ design points $i$ into a single global objective function $f_{mp}$:

$$f_{mp} = \alpha_1 f_1 + \alpha_2 f_2 + \alpha_3 f_3 + ... + \alpha_N f_N$$

(11)
To perform the optimization in this fashion, a single composite objective function should be posed in each description.i/rubber.data file. Each of the $\alpha_i$ weighting coefficients must be specified as weights for each design point in ammo.input, in the corresponding order.

This form of multi-point design is supported by PORT, DOT/BIGDOT™, and SNOPT™. Note that PORT and SNOPT™ will also combine multiple objective functions within each design point as described in section 8.9 if desired. Explicit constraints can be posed at each design point when using DOT/BIGDOT™ or SNOPT™; such constraints are each treated individually.

### 8.10.2 Combination of Objective Functions using the Kreisselmeier-Steinhauser Function

Another alternative for performing multi-point design is the approach inherent in the KSOPT package. In this approach, all objective functions and constraints are combined using the Kreisselmeier-Steinhauser (KS) function. The user is referred to [19] for the details of this formulation. Here, the Fun3D design driver gathers any number of objective and constraint functions across all design points and provides them to KSOPT, which internally constructs its KS function for the actual optimization problem.

### 8.10.3 Single-Point Objective Function with Off-Design Constraint Functions

In this approach to multi-point design, a single objective function is provided to the optimizer, while all other functions are treated as explicit constraints. Here, the user should designate a single composite function across all of the description.i/rubber.data input files as an objective function. Any other composite functions at each design point should be designated as constraint functions. KSOPT, SNOPT™, and NPSOL™ support this form of multi-point optimization; SNOPT™ can also construct the final objective function by linearly combining multiple objective functions within the desired design point as described in section 8.9.

### 8.11 Optimization of Two-Dimensional Geometries

While the Fun3D flow solver supports a 2D mode of operation, this capability is not currently available from within the design infrastructure. Instead, the optimization must be performed as a pseudo-3D case. The user should provide a nominally two-dimensional grid, with a single layer of elements in the spanwise ($y$) direction. The mesh should consist of either prisms or hexahedra (or both), but should contain no pyramids or tetrahedra. Follow the same procedure used for 3D cases to extract the surface grid for parameterization.
The surface should be parameterized just as for a 3D simulation; however, the parameterization should allow no spanwise asymmetries in the geometry to develop. When using MASSOUD or Bandaids, this is readily accomplished by linking the raw parameters with an equal weighting across the span into a single set of design variables that operate in a chordwise fashion. Note that the sidewalls should use symmetry boundary conditions so that only in-plane mesh deformation occurs during the optimization. The design may now be executed as usual, with the 2D nature of the problem enforced implicitly through the parameterization.

8.12 Using a Different Optimization Package

In a CFD-based design context, the term “function” implies an evaluation of the geometric parameterization, mesh movement (both surface and volume), a flow solution, and an evaluation of the output function/constraint for a given set of design variables. The file manipulations and solver operations necessary to achieve this are not trivial. For users interested in using the tools as “black boxes” providing function data for an optimization package, a wrapper has been provided in the LibF90 directory of the distribution named analysis.f90. This module contains a subroutine called perform_analysis() which performs the extensive set of tasks involved with producing the final desired function output.

To obtain sensitivities, the Fun3D package relies on a discrete adjoint formulation. As with function evaluations, the low-level operations required to perform an adjoint-based sensitivity analysis are numerous. A wrapper routine called perform_sensitivity_analysis() in the LibF90/sensitivity.f90 module will perform an adjoint solution for the flow field, an adjoint solution for the mesh movement scheme, an evaluation of the linearized geometric parameterization, and finally produce the desired sensitivity derivatives.

The Fun3D design driver uses the wrappers perform_analysis() and perform_sensitivity_analysis() to greatly simplify function and gradient evaluations when connecting to off-the-shelf optimization packages. If the user wishes to implement a new optimization strategy, it is highly recommended that these wrappers be used in a similar fashion. A review of the existing modules in the Design directory of the Fun3D source code distribution, which implement the currently available optimization interfaces, is also strongly suggested. Users may contact Fun3D-Support@lists.nasa.gov for further guidance in leveraging Fun3D’s capabilities from within their own existing design framework.
8.13 Implementing New Cost Functions/Constraints

Implementation of new cost functions or constraints is not a trivial undertaking and requires extensive modification of Fun3D source code. Experience in Fortran 2003, unstructured-grid discretizations, development in a domain-decomposed/distributed-memory environment, and general CFD methods are essential. Routines to evaluate the proposed function and linearizations of the function with respect to both the flow field variables and grid are ultimately required. The complex-variable form of Fun3D (see section 8.14) is invaluable in verifying the accuracy of these linearizations. It is highly recommended that the user contact Fun3D-Support@lists.nasa.gov for guidance prior to attempting the implementation of a new cost function or constraint.

8.14 Forward Mode Differentiation Using Complex Variables

The reverse, or adjoint, mode of differentiation is primarily used for design with Fun3D. A forward mode of differentiation is also provided based on the use of complex variables [20–22]. This capability is useful for design problems containing few design variables and many cost functions or constraints. To generate and build a complex-variable Fun3D executable, see section A.5.

The complex-valued flow solver reads the usual real-valued grid files and is set up to compute derivatives of every output variable with respect to Mach number, angle of attack, shape parameters, non-inertial rotation rates, or the \( x \), \( y \), or \( z \) coordinate of a single grid point (others are trivial to add). This choice is controlled by the file perturb.input. A template for this file is provided in the FUN3D_90 directory and an example is also shown below.

```
PERTURB   EPSILON GRIDPOINT
         2   1.e-50   666
```

- 0 = No perturbation
- 1 = Mach number
- 2 = Alpha
- 3 = Shape
- 4 = \( x \)-rotation rate
- 5 = \( y \)-rotation rate
- 6 = \( z \)-rotation rate
- 7 = Grid point \( x \)
- 8 = Grid point \( y \)
- 9 = Grid point \( z \)
- 100+ = add an imaginary source term to equation

PERTURE-100 of node GRIDPOINT
(to verify the adjoint lambda value)
The value of PERTURB specifies the variable for which sensitivities will be taken with respect to. The valid integer values are as shown above. The input EPSILON specifies the magnitude of the imaginary perturbation to be applied. The recommended value is $1.0 \times 10^{-50}$. If the value of PERTURB is greater than six, the value of GRIDPOINT specifies the grid point index to perturb. The remaining lines in perturb.input are not read; they are simply reminders of the valid inputs just described. The complex-valued flow solver may then be executed in a manner similar to the real-valued flow solver:

\texttt{mpirun ./complex_nodet_mpi}

To compute derivatives with respect to a shape parameterization variable, the sensitivities of the parameterization must first be evaluated in the directory model.i/Rubberize using the relevant parameterization software. The value of PERTURB should be set to 3 in perturb.input. The complex-valued flow solver can then be executed in the following fashion:

\texttt{mpirun ./complex_nodet_mpi --dv_index [body] [dv] --snap_grid}

Here, the values of body and dv specify the body and design variable index in rubber.data to which to apply the imaginary perturbation. The \texttt{--snap_grid} argument forces the flow solver to propagate the surface sensitivities into the volume mesh using FUN3D’s elasticity-based deformation mechanics.

At the completion of the complex-valued flow solve, outputs will contain both real and imaginary parts. The imaginary part represents the sensitivity of that output with respect to the perturbation variable that was specified in perturb.input.
References


11. Nielsen, E.; Lee-Rausch, E.; and Jones, W.: Adjoint-Based Design of Rotor
tors in a Noninertial Reference Frame. *AIAA Journal of Aircraft*, vol. 47, 


13. Rallabhandi, S.; Nielsen, E.; and Diskin, B.: Sonic Boom Mitigation 
Through Aircraft Design and Adjoint Methodology. AIAA Paper 2012–
3220, 2012.

14. Rallabhandi, S.: Application of Adjoint Methodology to Supersonic Air-
craft Design Using Reversed Equivalent Areas. AIAA Paper 2013–2663, 
2013.

15. Samareh, J. A.: Novel Multidisciplinary Shape Parameterization Ap-
pp. 1015–1024.

16. Samareh, J. A.: Aerodynamic Shape Optimization Based on Free-From 

17. Nielsen, E.; and Jones, W.: Integrated Design of an Active Flow Control 
System Using a Time-Dependent Adjoint Method. *Mathematical Modeling 
of Natural Phenomena*, vol. 6, no. 3, 2011, pp. 141–165.

18. Jones, W.; Nielsen, E.; Lee-Rausch, E.; and Acree Jr., C.: Multi-point 
Adjoint-Based Design of Tilt-Rotors in a Noninertial Reference Frame. 

the Kreisselmeir-Steinhauser Function. NASA CR-4220, NASA Langley 


sitivity Analysis for the Navier-Stokes Equations on Unstructured Meshes 
See also AIAA Paper 99-3294.


Appendix A

Installation

FUN3D is distributed as gzipped archive of source code. The GNU build system is used to package and install FUN3D. The required installation steps are detailed in this section. Due to the large range of capabilities, configuring the dependent packages is the most involved step and is the focus this section.

As was illustrated in the Quick Start section, four basic steps are required:

1. Extract the source code from the gzipped tarball archive with tar
2. Configure the desired dependencies and compiler options with configure
3. Compile via make
4. Install the compiled binaries and supporting scripts via make install

If any difficulties arise with the installation process please, send the entire config.log file produced by configure and the full stdout and stderr of make to Fun3D-Support@lists.nasa.gov. The user is strongly advised against editing the configure script or any Makefile it produces. We are unable to assist users who have edited these files.

A.1 Extracting Files

After downloading the source code as a gzipped tarball, the user can unpack it with

```
tar zxf fun3d-12.4.tar.gz
```

which will create the directory fun3d-12.4. If you have do not have a GNU-compatible tar, you may have to insert a separate decompression step, i.e.,

```
gzip -d fun3d-12.4.tar.gz | tar zxf -
```

A.2 Configure Introduction

The FUN3D suite of tools is configured and built via the GNU build system and must be configured first. Change to this directory, e.g., cd fun3d-12.4, and execute

```
./configure --help
```
to see a list of all available compilation options. When configure is invoked, detailed results of all the tests it performs are written to the file config.log.

Some features of the configure step that have caused problems for users are:

- An incorrect spelling of a --enable-* or --with-* option is silently ignored. This will result in the intended option not being included in the compiled executable.
- Option values containing spaces must be quoted to be correctly interpreted by the shell (i.e., FCFLAGS='--option1 --option2').
- If the configure command is executed more than once with different options, make clean is required before the make step, so that changes to the configuration are correctly reflected in the compiled executable.

### A.3 Alternative Installation Path

The path to the installation directory is specified by the --prefix= option. The default is to install to /usr/local with executables placed in /usr/local/bin. This default location may not be available if the user does not have write permission to this directory (without root or administrator privileges).

To install to an alternative path (e.g., $HOME/local), use the --prefix= option to set the installation path

```bash
./configure --prefix=$HOME/local
```

Finally, to include the Fun3D executables in the command search path, add

```bash
setenv PATH $HOME/local/bin:$PATH
```
to the ~/.cshrc file or the equivalent for your shell.

### A.4 Fortran Compiler Option Tuning (FTune)

By default, configure will use compiler and linker options chosen by the Fun3D team. The process is referred to as “FTune.” The users PATH is searched in a predefined order until the first Fun3D-compatible compiler is found. When configured with MPI, the build will use mpif90 located in the bin directory of the given MPI installation. However, the user can explicitly specify the desired Fortran compiler via the FC environment variable.

To directly specify the compiler and linker options, use the FCFLAGS and LDFLAGS environment variables. The default behavior is to append their values to the options defined by FTune. If the --disable-ftune option is given to configure, FTune will be disabled and the values given by FCFLAGS and

---

A1 To see what the underlying compiler is, use mpif90 -show.
LDLAGS will be used explicitly. For example, to ensure that the Intel® Fortran compiler ifort is used with only the -O3, -ip, and -lm options,

```
./configure --disable-ftune \
FC=ifort \
FCFLAGS='-O3 -ip' \
LDFLAGS='-lm'
```

The order of variables and options are inconsequential, and single quotation marks (') are used to protect values with spaces from the shell. Some FTune options may be unconditionally required for a given compiler, as in the case of linking with the math library -lm above.

### A.5 Complex Variable Version

The Fun3D suite can be compiled with the real variables in the code replaced with complex variables by a source translation tool. This permits the computation of forward-mode sensitivities, see section 8.14 for details. To enable, add the --enable-complex configure option to the configure script. The complex-valued code can be compiled with make complex; and a make install will place the complex-valued executables in the bin installation directory. Enabling the complex variable version will increase the compile time.

### A.6 Internal Libraries

Fun3D has internal dependencies to libraries that are distributed with Fun3D. These libraries are automatically built and linked to Fun3D by default.

#### A.6.1 KNIFE

The KNIFE cutcell library provides cutcell capabilities. The --without-knife option will disable this library.

#### A.6.2 REFINE

The REFINE library provides access mesh adaptation and untangling capabilities. The --without-refine option will disable this library.

### A.7 External Libraries

Fun3D relies on external libraries to enable some of its advanced applications. Use Table A1 to determine which set of external libraries are necessary for your applications of interest. Discussions of each external library are found in the following sections.
Table A1: Configuration options.

<table>
<thead>
<tr>
<th>Option</th>
<th>Parallel Execution</th>
<th>Overset Motion</th>
<th>Computed Trajectories</th>
<th>Constrained Design</th>
<th>Binary Tecplot</th>
<th>CGNS</th>
<th>Sonic Boom Propagation</th>
</tr>
</thead>
<tbody>
<tr>
<td>MPI</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ParMETIS</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SUGGAR</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DiRTLIB</td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6-DOF</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>KSOPT</td>
<td></td>
<td>x</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PORT</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SNOPT™</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>NPSOL™</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DOT/BIGDOT™</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tecplot™</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CGNS</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>sBOOM</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>x</td>
</tr>
</tbody>
</table>

It is highly recommended that Fun3D is configured to use a parallel execution (MPI and ParMETIS) if you plan to perform any advanced calculations. SUGGAR++ and DiRTlib are only required if overset (chimera) meshes will be used. The 6-DOF library is only required if six degrees of freedom simulations will be performed (trajectories determined by integrating the equation of motion). KSOPT, PORT, SNOPT™, NPSOL™, and DOT/BIGDOT™ are optimization libraries. At least one of these optimization libraries is required for performing design optimization.

A.7.1 MPI

MPI provides Fun3D’s capability to communicate between processors. The partitioning library ParMETIS is also required for parallel execution, and it is critical that Fun3D and ParMETIS are compiled with exactly the same MPI installation. In some cases, MPI may already be installed on the target machine. If it is not, OpenMPI or MPICH can be used and the option of static MPI libraries is recommended.

Configure with the option

```
--with-mpi=/path/to/MPI
```

where /path/to/MPI is the directory where MPI is installed.
Some high performance computing environments use a proprietary MPI implementation that does not provide `mpif90`. In that situation, the configure option `--without-mpif90` may be required in combination with the FC environment variable to explicitly set the compiler.

### Verifying the MPI Implementation Functionality

A simple Fortran program is included in the FUN3D distribution to verify that the MPI implementation is functional. This is very helpful for quickly troubleshooting issues with the MPI implementation. It is located in `utils/MPIcheck`. From within that directory you should be able to

```
mpif90 -o mpi_hello_world mpi_hello_world.F90
```

and execute on two processors

```
mpiexec -np 2 ./mpi_hello_world
```

0 says, "Hello World!" 5 = 5
1 says, "Hello World!" 5 = 5

To verify the Fortran compiler that MPI is built with, try

```
mpif90 -show
```

if the MPI implementation supports it.

### A.7.2 ParMETIS

Website: [http://glaros.dtc.umn.edu/gkhome/metis/parmetis/overview](http://glaros.dtc.umn.edu/gkhome/metis/parmetis/overview)

ParMETIS is a parallel graph partitioner that is used to perform domain decomposition for all parallel FUN3D jobs. It is critical that FUN3D and ParMETIS are compiled with exactly the same MPI installation and compilers. This includes the C compiler used to compile ParMETIS, MPI, and FUN3D.

When configuring FUN3D, use

```
--with-parmetis=/path/to/ParMETIS
```

where `/path/to/ParMETIS` is the directory of the ParMETIS installation. FUN3D expects the `/path/to/ParMETIS` directory to contain the following files in `lib` and `install` subdirectories,

```
/path/to/ParMETIS/lib/libmetis.a
/path/to/ParMETIS/lib/libparmetis.a
/path/to/ParMETIS/include/metis.h
/path/to/ParMETIS/include/parmetis.h
```

See the `Install.txt` instructions in the ParMETIS distribution for build instructions. FUN3D requires both `libmetis.a` and `libparmetis.a` libraries and their accompanying header files. There is an example of commands to build both libraries,
cd parmetis-4.*
make config prefix=/path/to/ParMETIS
make install
cd metis
make config prefix=/path/to/ParMETIS
make install

where /path/to/ParMETIS matches the Fun3D configure argument.

A.7.3 SUGGAR++-1.0.10 or Higher

Website: http://celeritassimtech.com

SUGGAR++ is used for overset (chimera) applications and assembles composite meshes, cuts holes, determines interpolation coefficients, etc. If configuring with SUGGAR++, Fun3D must also be configured with DiRTlib v1.40 or higher.

SUGGAR++ may be compiled as a stand-alone executable and/or as a library. For static overset meshes you will need the stand-alone compilation; for moving body simulations you will need to compile both the stand-alone executable and the library. See the documentation that comes with SUGGAR++ for more information on how to compile the software.

When configuring Fun3D, use

--with-suggar=/path/to/SUGGAR++

where /path/to/SUGGAR++ is the directory where SUGGAR++ library archive files (.a files) reside. In this directory, there must be an archive file called libsuggar.a, which is the serial compilation of SUGGAR++, and there must also be an archive file called libsuggar_mpi.a, which is the MPI compilation of SUGGAR++.

A.7.4 DiRTlib v1.40 or higher

Website: http://celeritassimtech.com

The DiRTlib library must be linked to Fun3D in order to use the overset connectivity data computed by SUGGAR++-1.0.10 or Higher. See the documentation that comes with DiRTlib for more information on how to compile the software.

When configuring Fun3D, use

--with-dirtlib=/path/to/DiRTlib

where /path/to/DiRTlib is the directory where DiRTlib library archive files (.a files) reside. In this directory, there must be an archive file called libdirt.a, which is the serial compilation of DiRTlib, and there must also be an archive file called libdirt_mpich.a, which is the MPI compilation of DiRTlib.
A.7.5 6-DOF

Contact: Nathan.C.Prewitt@usace.army.mil

The 6-DOF libraries provide trajectory tracing. When configuring Fun3D, use

```
--with-sixdof=/path/to/sixdof
```

where /path/to/sixdof is the directory where your 6-DOF installation resides.

A.7.6 KSOPT

Contact: Gregory.A.Wrenn@nasa.gov

The KSOPT [19] library is used for multi-objective and constrained Fun3D-based design optimization. If you configure Fun3D to link to KSOPT, you must use the Fortran 90 implementation of KSOPT with its object files gathered into a library called libksopt.a.

When configuring Fun3D, use

```
--with-KSOPT=/path/to/ksopt
```

where /path/to/ksopt is the directory where your KSOPT installation resides.

A.7.7 PORT

Website: http://www.netlib.org/port

The PORT library is used for unconstrained Fun3D-based design optimization. The Netlib site offers a tarball of the PORT library with a Makefile. Download the tarball from Netlib, but replace the original Makefile with the file included inside the Fun3D distribution as Design/PORT.Makefile. If you install both the PORT and NPSOL™ libraries, you may have to comment out low-level BLAS routines in one of the two packages because the linker will report the duplicate versions of these routines.

When configuring Fun3D, use

```
--with-PORT=/path/to/port
```

where /path/to/port is the directory where your PORT installation resides.

A.7.8 SNOPT™

Website: http://www.sbsi-sol-optimize.com

The SNOPT™ library is used for Fun3D-based design optimization. By default the SNOPT™ package builds a shared library. Either build SNOPT™ with the --disable-shared option, or add the the SNOPT™ install directory
to your LD_LIBRARY_PATH environment variable to ensure Fun3D can find the shared library at run time.

When configuring Fun3D, use

```
--with-SNOPT=/path/to/snopt
```

where /path/to/snopt is the directory where your SNOPT™ installation resides.

### A.7.9 NPSOL™

Website: [http://www.sbsi-sol-optimize.com](http://www.sbsi-sol-optimize.com)

The NPSOL™ library is used for constrained Fun3D-based design optimization. If you install both the PORT and NPSOL™ libraries, you may have to comment out low-level BLAS routines in one of the two packages because the linker will report the duplicate versions of these routines.

When configuring Fun3D, use

```
--with-NPSOL=/path/to/npsol
```

where /path/to/npsol is the directory where your NPSOL™ installation resides.

### A.7.10 DOT/BIGDOT™

Website: [http://www.vrand.com/products.html](http://www.vrand.com/products.html)

The DOT/BIGDOT™ library is used for unconstrained or constrained Fun3D-based design optimization. When configuring Fun3D, use

```
--with-DOT=/path/to/dot
```

where /path/to/dot is the directory where your DOT/BIGDOT™ installation resides.

### A.7.11 Tecplot™

Website: [http://www.tecplot.com](http://www.tecplot.com)

By default, any Tecplot™ output generated from within the flow solver itself is written as a text file. If you have a copy of Tecplot™, you were provided with a library archive tecio.a (or tecio64.a for 64-bit versions) that allows for binary output.A2 You may configure the Fun3D suite to use the library via:

```
--with-tecio=/path/to/tecio
```

A2 The tecio library that was shipped with TECIPLT360-2008 had a bug that will result in error messages when the binary files are written. You must get an updated version of the library.
With this option, Tecplot™ solution data written out from the flow solver will be in binary form. This results in smaller file sizes and faster importation into Tecplot™.

If you have compiled against the Tecplot™ tecio library, you can still request text output via the `--ascii_tecplot_output` command line option.

**A.7.12 CGNS**

Website: [http://www.cgns.org](http://www.cgns.org)

The CGNS library is used for working with files written in CGNS format. CGNS is a convention for writing machine-independent, self-descriptive data files for CFD and includes implementation software. FUN3D has the capability to translate and write CGNS files. The translation utilities are only compiled when CGNS is configured. Version 2.5 or greater of the CGNS library is required. To include CGNS, use

```
--with-CGNS=/path/to/cgns
```

where `/path/to/cgns` is the directory where the CGNS installation resides.

**A.7.13 sBOOM**

Contact: Sriram.Rallabhandi@NASA.gov

This package propagates a computed pressure signature to the ground for sonic boom simulations. Atmospheric variations are included, and an adjoint version is available for coupling into design and grid adaptation. sBOOM is distributed as a standalone executable or a static library. FUN3D is not able to interact with the standalone executable; the static library must be linked.

You may configure the FUN3D suite to use the library via:

```
--with-SBOOM=/path/to/sBOOM
```

where `/path/to/sBOOM` is the directory where the sBOOM installation resides.
Appendix B

Fun3D Input Files

There are a variety of input files necessary for the various codes that make up the Fun3D suite. Table B1 lists frequently used input files with a short description. This chapter will describe the basic formats of each of these files and meaning of the specific inputs they contain.

Table B1: Fun3D input files.

<table>
<thead>
<tr>
<th>File</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>stop.dat</td>
<td>signals an immediate stop during execution</td>
</tr>
<tr>
<td>remove_boundaries_from_force_totals</td>
<td>omiss boundary faces from total force integration</td>
</tr>
<tr>
<td>[project_rootname].flow*</td>
<td>flow field solution</td>
</tr>
<tr>
<td>fun3d.nml</td>
<td>primary Fortran namelist (required)</td>
</tr>
<tr>
<td>moving_body.input</td>
<td>body motion Fortran namelist</td>
</tr>
<tr>
<td>rotor.data</td>
<td>describes the rotor actuator disk model</td>
</tr>
<tr>
<td>tdata</td>
<td>specifies the generic gas model</td>
</tr>
<tr>
<td>kinetic_data</td>
<td>specifies the possible chemical reactions in the generic gas model</td>
</tr>
<tr>
<td>species_transp_data</td>
<td>specifies generic gas model species collision cross sections</td>
</tr>
<tr>
<td>species_transp_data_0</td>
<td>specifies a higher-order generic gas model species collision cross sections</td>
</tr>
<tr>
<td>hara_namelist_data</td>
<td>controls the radiation models used by the HARA library</td>
</tr>
</tbody>
</table>

* The [project_rootname] is a &project namelist variable, see section B.4.1.

Fun3D utilizes Fortran namelists for a large portion of input specification because it is defined in the Fortran 90 standard. With all Fortran namelists, leaving out or misspelling any namelist (defined with an ampersand preceding its name) will result in default values being used for all of the parameters within that namelist. For example, if the namelist name linear_solver_parameters were to be misspelled as linear_solver_parameter (missing s), then all parameters within that namelist would be ignored and retain their default values. Leaving out any parameter within a namelist results in the default value for that parameter being used. Misspelling or misusing any particular parameter will typically cause Fun3D to issue an error and stop.

B.1 stop.dat

This optional file halts the solver during execution. The stop.dat plain text file contains a single integer. After every iteration, the solver will check to
see if this file exists. If the file is found in the directory that the solver was
invoked, the integer is read. If the integer is greater than zero and less than or
equal to the current iteration, the solver will write the current solution, delete
the stop.dat file, and halt execution. If the integer is greater than the current
iteration, code execution will proceed until the iteration matches the integer.

Some environments, especially ones with network-mounted filesystems (e.g.,
NFS), may result in a delay with the stop.dat file being read or being deleted.

B.2  [project_rootname].flow

The optional [project_rootname].flow binary file contains flow solution and
checkpoint information. The [project_rootname] is a &project namelist
variable, see section B.4.1. This file is read by the solver to restart computa-
tions from a previously computed flow solution. The contents vary due to
the checkpoint requirements of the simulation. The file contains a minimum
of the current solution and convergence history. It can also contain working
variables for the turbulence model, solutions from previous iterations for time
accurate cases, or previous grid positions and velocities for deforming grids.

B.3  remove_boundaries_from_force_totals

The optional remove_boundaries_from_force_totals file is for specifying bound-
daries that are not to be included in the calculation of force and moment totals.
This file is useful, for example, in situations where there may be a mounting
sting on a wind tunnel model, but only the forces on the model are actually of
interest. The forces on the specified boundaries are still computed and appear
in the [project_rootname].forces file. However, they are not included in
the totals. The position of the text lines in this file is significant. So, follow
this template carefully:

Remove selected boundaries from the total forces
Number of boundaries to turn off
2
Boundaries to turn off (boundary lumping changes indexes)
12
15

The third line is the number of boundaries to exclude. The fifth and subsequent
lines are the patch indexes of the excluded boundaries.

B.4  fun3d.nml

The main input namelist file, fun3d.nml, is described in detail below, with de-
faults listed before the descriptions. The namelist file contains a large number
of input variables. In general, it is not necessary to specify them all because they have suitable default values. Only those variables that are different from the defaults need to be specified.
B.4.1 &project

This namelist allows the user to specify the rootname of the project, which forms the majority of input and output filenames.

```fortran
&project
  project_rootname = 'default_project'
/

  project_rootname = 'default_project'
```

The project rootname is the root for the grid, restart, and visualization files. The manual refers to it as `[project_rootname]`. The `default_project` can be replaced with any filename allowed by the file system.
B.4.2 &raw_grid

This namelist specifies details of the grid format.

&raw_grid

    grid_format = 'vgrid'
data_format = 'default'
twod_mode = .false.
swap_yz_axes = .false.
fieldview_coordinate_precision = 'double'
patch_lumping = 'none'
ignore_euler_number = .false.
/

    grid_format = 'vgrid'

This specifies the grid file format. The currently supported values are:

- 'fast' for FAST .fgrid/.mapbc files.
- 'fun2d' for Fun2D.faces files.
- 'aflr3' for AFLR3 formatted, unformatted, or C-binary/Fortran-stream .ugrid/.r8.ugrid/.b8.ugrid/.mapbc files,
- 'felisa' for Felisa grid files.
- 'fieldview' for FieldView formatted or unformatted .fvgrid_fmt/.fvgrid_unf/.mapbc files.

data_format = 'default'

This provides the encoding of the grid file. A particular grid_format may only support a subset of encodings. Fun3D will stop with an error message if the data_format is inconsistent with the grid_format. The 'default' value is changed to an admissible value based on grid_format as noted next to each value,

- 'ascii' ASCII text grid file. It is the default for 'felisa' and 'fun2d' grids.
- 'unformatted' Fortran unformatted grid file. It is the default for 'fast', 'vgrid', and 'fieldview' grids.
- 'stream' C-binary/Fortran-stream grid file. It is the default for 'aflr3' grids.
- 'stream64' 64 bit integer C-binary/Fortran-stream grid file.

twod_mode = .false.

Turns on two-dimensional mode for a single layer prism or hex grid. If grid_format = 'fun2d', twod_mode is automatically .true.
**swap.yz_axes = .false.**

When `.true.` this swaps the $y$- and $z$-axes for the grid. This option can be used to rotate the grid so the $z$-axes is in the FUN3D convention for angle of attack and lift.

**fieldview.coordinate_precision = 'double'**

This specifies floating point precision of reals for FieldView meshes only.

- `'double'` for double precision coordinates.
- `'single'` for single precision coordinates.

**patch.lumping = 'none'**

This enables boundary patch lumping. It combines the grid patches into fewer patches to ease the bookkeeping of patch groups, but will effect all features that reference boundary patch numbers (e.g., `&boundary_conditions`). The .mapbc files for any of the supported grid formats may contain an optional third column of data, which specifies a family name. The exception is the VGRID.mapbc file, where the family name is mandatory and appears in the sixth column. If family names are not present in the .mapbc file, patch.lumping can not be family.

- `'none'` for no patch lumping.
- `'bc'` for physical boundary condition lumping.
- `'family'` for family name lumping

**ignore.euler_number = .false.**

This will permit the use of grids with a failing Euler number check. See section C.7 for a description of the Euler number and its implications. Ignoring the Euler number check is not recommended.
B.4.3 &force_moment_integ_properties

Reference lengths and area are defined in this namelist to scale aerodynamic force and moment data.

```
&force_moment_integ_properties
  area_reference = 1.0
  x_moment_length = 1.0
  y_moment_length = 1.0
  x_moment_center = 0.0
  y_moment_center = 0.0
  z_moment_center = 0.0
/

  area_reference = 1.0
  This area is used for non-dimensionalization of forces and moments, specified in grid units squared.

  x_moment_length = 1.0
  This length in x-direction is used to nondimensionalize moments about y (pitching moment), specified in grid units.

  y_moment_length = 1.0
  This length in y-direction is used to nondimensionalize moments about x (rolling moment) and z (yawing moment), specified in grid units.

  x_moment_center = 0.0
  This specifies the x-coordinate location of moment center, in grid units.

  y_moment_center = 0.0
  This specifies the y-coordinate location of moment center, in grid units.

  z_moment_center = 0.0
  This specifies the z-coordinate location of moment center, in grid units.
```
B.4.4 **governing equations**

This namelist specifies the equation set that describes underlying physics of the problem.

```plaintext
&governing_equations
  eqn_type               = 'compressible'
  artificial_compress    = 15.0
  viscous_terms          = 'turbulent'
  chemical_kinetics      = 'finite-rate'
  thermal_energy_model   = 'non-equilib'
  prandtl_number_molecular  = 0.72
  schmidt_number         = -1.
  gas_radiation          = 'off'
  rad_use_impl_lines     = .false.
  multi_component_diff   = .false.
  cpiv_min_factor        = 0.0001
  augment_kinetics_limiting = .false.
  implicit_rate_limiting = .true.
/

  eqn_type = 'compressible'
```

This specifies the set of governing equations to be solved.

- **compressible** for compressible, calorically perfect gas. See section 2.1 for equations and nondimensionalization.
- **incompressible** for incompressible, calorically perfect gas. [23] See section 2.2 for equations and nondimensionalization. The incompressible solution is affected by the choice of *artificial_compress*, see the description of this parameter for details.
- **generic** for multispecies, reacting gas simulations. See section 2.3 for nondimensionalization. The *tdata* input file is required. FUN3D is usually distributed with the **generic** option disabled. If it is required, see section 1.4 for information on obtaining the version of FUN3D with this capability.

**artificial_compress = 15.0**

This is the artificial compressibility factor, $\beta$, which is only used by the *eqn_type = 'incompressible'*. This parameter must be in the range of $(100, 1)$. See Anderson, Rausch, and Bonhaus [23] for details. The sensitivity of the solution to this parameter will decrease with mesh refinement, so consider a refined grid if an unacceptable amount of sensitivity is experienced. A high sensitivity to this parameter can also indicate that the problem is actually compressible and the user is encouraged to check the incompressible solution by performing a low Mach compressible simulation.
viscous_terms = 'turbulent'
This describes the modeling of the viscosity term in the governing equations.

'inviscid' no viscosity, for inviscid flow.

'laminar' apply laminar viscosity, to model laminar flow.

'turbulent' include laminar viscosity and model turbulent flow with a turbulent diffusion models.

chemical_kinetics = 'finite-rate'
This describes the chemical kinetics, only used when eqn_type = 'generic'.

'frozen' for frozen chemical compositions.

'finite-rate' for finite-rate reacting gases.

thermal_energy_model = 'non-equilib'
This describes the thermal energy model, only used when eqn_type = 'generic'.

'frozen' for frozen chemical compositions.

'non-equilib' for non-equilibrium gases.

prandtl_number_molecular = 0.72
This is the molecular Prandtl number. It must be greater than zero.

schmidt_number = -1.
This is the Schmidt number used in the generic gas path. If the user wants to override the default path of computing a variable Schmidt number from collision cross sections then use this parameter to specify the constant Schmidt number.

gas_radiation = 'off'
This controls flow field radiation coupling. When active, this option will compute radiation source terms and surface heat fluxes after the first time step. Radiation source terms are not further updated during the rest of the time steps. Radiation source terms are not stored in the restart file, so they need to be recalculated when restarting simulations that include flow field radiation. Only for eqn_type = 'generic'.

'off' no radiation calculations.

'uncoupled' will use the HARA program to compute radiative surface heat fluxes, but radiation source terms would not be included in the flow-field governing equations.

'coupled' will include radiation source terms in the governing equations, with the divergence of the radiative flux being computed by the
HARA program. Requires `rad_use_impl_lines = .true.` only valid if all of the domain nodes are included in one and only one line as defined by the implicit lines file.

`rad_use_impl_lines = .false.`

For `gas_radiation`, the mesh nodes in the lines of sight are read from the implicit lines file. Coupled radiation must use this option. For uncoupled radiation, the lines of sight can either be read from the implicit lines file when `.true.`, or the lines of sight will be generated during the FUN3D run when `.false. Only for `eqn_type = 'generic'`.

`multi_component_diff = .false.`


`cpiv_min_factor = 0.0001`

This variable sets the minimum value of the vibrational-electronic heat capacity as a fraction of the translational-rotational heat capacity for each species `i`. In some cases, ramping this value up to 0.01 can help suppress undershoot of vibrational temperature upstream of a strong shock. The vibrational-electronic heat capacity must be positive for stability. Only for `eqn_type = 'generic'`.

`augment_kinetics_limiting = .false.`

When `.true.`, augment chemical kinetic source term limiting. Only for `eqn_type = 'generic'`.

`implicit_rate_limiting = .true.`

When `.true.`, limit chemical rates if extrema exist in formulation Only for `eqn_type = 'generic'`.

103
B.4.5 &reference_physical_properties

This namelist is used to specify reference conditions and nominal freestream flow conditions in a user-defined unit system. It is also used to convert between grid units and flow solver units.

&reference_physical_properties
  dim_input_type = 'nondimensional'
  gridlength_conversion = 1.0
  mach_number = 0.0
  reynolds_number = 0.0
  velocity = 0.0
  density = 0.0
  temperature = 273.0
  temperature_units = 'Kelvin'
  angle_of_attack = 0.0
  angle_of_yaw = 0.0
/

  dim_input_type = 'nondimensional'

  This is the system of measurement for the reference conditions. Currently, it must be 'dimensional-SI' for eqn_type = 'generic' and 'nondimensional' otherwise. This input is intended for future expansion. The temperature is always input as a dimensional quantity.

  'nondimensional' requires mach_number and reynolds_number to be defined.

  'dimensional-SI' requires dimensional velocity and density to be defined.

  gridlength_conversion = 1.0

  For dim_input_type = 'dimensional-SI', this is the conversion factor to scale the grid and it should be set to meters per grid unit. It is used for providing heat flux in proper units and other tasks. For dim_input_type = 'nondimensional', this should be set to 1.0, because the grid is already in nondimensional grid units.

  mach_number = 0.0

  This is the reference Mach number defined as velocity/speed-of-sound. It is only allowed for dim_input_type = 'nondimensional' and eqn_type = 'compressible'. It must be set to a positive value.

  reynolds_number = 0.0

  This is the reference Reynolds number, per one unit of the grid. Not correctly accounting for the unit of the grid has been a point of confusion
in the past. For example, when the grid units are feet, Reynolds number should be specified per foot. This input is only used if `dim_input_type = 'nondimensional'` and is ignored by `eqn_type = 'generic'`. It must be set to a positive value.

velocity = 0.0
This is the reference velocity, in m/s. Only used for `dim_input_type = 'dimensional-SI'` and `eqn_type = 'generic'`.

density = 0.0
This is the reference density, in kg/m$^3$. Only used for `dim_input_type = 'dimensional-SI'` and `eqn_type = 'generic'`.

temperature = 273.0
This is the reference temperature, in units of `temperature_units`.

`temperature_units = 'Kelvin'
 The units used to specify `temperature`.
‘Kelvin’ for SI units.
‘Rankine’ for the English system.

angle_of_attack = 0.0
This is the freestream angle of attack in degrees.

angle_of_yaw = 0.0
This is the freestream angle of yaw (side-slip) in degrees.
B.4.6 &inviscid_flux_method

This namelist controls the construction of the inviscid fluxes and flux Jacobians.

```
&inviscid_flux_method
  flux_construction = 'roe'
  flux_construction_lhs = 'vanleer'
  kappa_umuscl = -1.0
  flux_limiter = 'none'
  first_order_iterations = 0
  multidm_option = 1
  fixed_direction = .true.
  recalc_dir_freq = 1
  adptv_entropy_fix = .false.
  rhs_u_eigenvalue_coef = 0.
  lhs_u_eigenvalue_coef = 0.
  rhs_a_eigenvalue_coef = 0.
  lhs_a_eigenvalue_coef = 0.
  entropy_fix = .false.
  re_min_vswch = 50.
  re_max_vswch = 500.
/
  flux_construction = 'roe'

This specifies the inviscid flux residual construction method.
'roe' for Roe flux difference splitting.
'vanleer' for van Leer flux vector splitting.
'hllc' for HLLC.
'aufs' for AUFS.
'ldfss' for LDFSS.
'dldfss' for dissipative LDFSS.
'aldfss' for LDFSS with an adaptive entropy fix.
'roe_ec' for entropy-consistent Roe scheme.
'stvd' for Yee's symmetric total variation diminishing scheme.
'stvd_modified' for a modified version of STVD.
'multidm' for Gnoffo's multidimensional scheme.
 flux_construction_lhs = 'vanleer'

This specifies the inviscid flux Jacobian construction method. A 'consistent' method yields the best asymptotic iterative convergence rate of the nonlinear residual, but a more diffusive flux may stabilize a poorly converging or diverging linear system iterative scheme.
```
'consistent' for a consistent linearization with the residual construction method.

'vanleer' for Van Leer.

'roe' for Roe linearization.

'hllc' for HLLC.

'aufs' for AUFS.

'ldfss' for LDFSS.

\texttt{\texttt{kappa}\_umuscl = -1.0}  
Controls the amount of upwinding in the unstructured-grid MUSCL reconstruction scheme. The default will adjust \texttt{kappa}\_umuscl internally to 0.5 for 3D mixed-element grids or 0.0 for all other grid types. 0.0 is the upwind-biased (Fromm) discretization, 1.0 is the (unstable) central-difference discretization, and the range [0,1] is a blend of the two.

\texttt{\texttt{flux}\_limiter = 'none'}  
This selects the flux limiter. The limiters that begin with the letter h, 'barth', and 'venkat' are stencil-based limiters (they apply a limiter to each edge in a node’s the reconstruction stencil and store the most restrictive edge limiter value at the node). Other limiters are evaluated in a strictly edge-based manner. The h-series of limiters automatically turns on a heuristic pressure based limiter that is used to augment the selected flux limiter. [25] The node-based limiters can be frozen with the \texttt{--freeze\_limiter \{freeze after this iteration\}} command line option to possibly improve “ringing” nonlinear iterative convergence. When restarting a solution with a frozen limiter, \texttt{--freeze\_limiter 0} retains the same limiter field. The adjoint solver is only compatible with a frozen limiter. For hypersonic flows computed using the calorically perfect gas path, the \texttt{hvanleer} or \texttt{hvanalbada} flux limiters are recommended.

'none' for no limiter.

'barth' for the Barth limiter.

'venkat' for the Venkatakrishnan [26] limiter. This limiter is dimensional and should be scaled if the grid is not normalized to a characteristic length of your model. The \texttt{--smooth\_limiter\_coeff} command line option should be set to a reciprocal of a characteristic length, i.e., 1/(Mean Aerodynamic Chord).

'hminmod' for the stencil-based min-mod limiter augmented with a heuristic pressure limiter.
'hvanleer' for the stencil-based van Leer limiter augmented with a heuristic pressure limiter.

'hvanalbada' for the stencil-based van Albada limiter augmented with a heuristic pressure limiter.

'hvenkat' for the Venkatakrishnan limiter augmented with a heuristic pressure limiter.

'minmod' for the min-mod limiter.

'venleer' for the van Leer limiter.

'venleer gg' for van Leer limiter that also turns on Green-Gauss gradients for inviscid reconstruction.

'venalbada' for the van Albada limiter.

**first_order_iterations = 0**

This is the number of iterations to use first-order spatial accuracy prior to using second-order spatial accuracy. If second-order spatial accuracy is not required, set this to a value larger than the number of steps. This option is useful for starting difficult supersonic flow simulations. For time accurate cases (**time_accuracy** not equal to 'steady'), this is the number of first-order accurate sub-iterations to run for each time step.

**multidm_option = 1**

This controls the multidm reconstruction weighting.

'1' virtual node averaging.

'2' weighted average of edges.

**fixed_direction = .true.**

This specifies the use of Cartesian directions in multidm reconstruction.

**recale_dir_freq = 1**

This sets the frequency of direction recalculation in the multidm scheme.

**adptv_entropy_fix = .false.**

This activates the adaptive entropy fix for Roe’s scheme.

**rhs_u_eigenvalue_coef = 0.**

This is the contact/shear eigenvalue smoothing coefficient for the adaptive entropy fix and the roe residual.

**lhs_u_eigenvalue_coef = 0.**

This is the contact/shear eigenvalue smoothing coefficient for the adaptive entropy fix and the roe Jacobian.
rhs_a_eigenvalue_coef = 0.
This is the acoustic eigenvalue smoothing coefficient for the adaptive entropy fix and the roe residual.

lhs_a_eigenvalue_coef = 0.
This is the acoustic eigenvalue smoothing coefficient for the adaptive entropy fix and the roe Jacobian.

entropy_fix = .false.
This activates the entropy fix for the stvd flux.

re_min_vswch = 50.
For the stvd flux, eigenvalue limiting is turned off below this cell Reynolds number.

re_max_vswch = 500.
For the stvd flux, eigenvalue limiting is fully engaged above this cell Reynolds number.
B.4.7 &turbulent_diffusion_models

When viscous_terms = 'turbulent', this namelist is used to set the form of the turbulence model.

```plaintext
&turbulent_diffusion_models
  turbulence_model = 'sa'
  turb_model = 'deprecated-use-turbulence_model'
  turb_intensity = 0.001
  turb_viscosity_ratio = 0.001
  turb_compress_model = 'off'
  turb_conductivity_model = 'off'
  prandtl-number_turbulent = 0.9
  schmidt-number_turbulent = 1.
/

turbulence_model = 'sa'
```

This selects the form of the turbulence model. The naming convention of http://turbmodels.larc.nasa.gov/ is used for the models described on the website.

'sa' for Spalart-Allmaras model.

'sa-catris' for Spalart-Allmaras Catris-Aupoix model.

'des' for Spalart-Allmaras based DES model. Not available for eqn_type='generic'.

'sa-neg' for Spalart-Allmaras model with negative turbulence variable provisions. Not available for eqn_type='generic'.

'des-neg' for Spalart-Allmaras based DES model with negative turbulence variable provisions. Not available for eqn_type='generic'.

'menter-sst' option is no longer valid, use sst or sst-v

'sst' Menter SST Two-Equation Model with strain source term.

'sst-v' Menter SST Two-Equation Model with vorticity source term. Not available for eqn_type='generic'.


110
‘gamma-ret-sst’ for gamma-ret SST. Not available for \texttt{eqn\_type=’generic’}.

‘baldwin-lomax’ Baldwin-Lomax algebraic model. Available only for \texttt{eqn\_type=’generic’}.

‘cebeci-smith’ Cebeci-Smith algebraic model. Available only for \texttt{eqn\_type=’generic’}.

\texttt{turb\_model = ‘deprecated-use-turbulence\_model’}

This is a deprecated namelist variable for \texttt{turbulence\_model}. It is included for backwards compatibility with fun3d.nml files, but may be removed in a future version.

\texttt{turb\_intensity = 0.001}

This sets the freestream turbulence intensity, $\sqrt{\frac{2k}{\rho U^2}}$, where $k$ is the turbulent kinetic energy. Only applies to \texttt{eqn\_type=’generic’}.

\texttt{turb\_viscosity\_ratio = 0.001}

This sets the freestream ratio of turbulent viscosity to molecular viscosity. Only applies to \texttt{eqn\_type=’generic’}.

\texttt{turb\_compress\_model = ‘off’}

This controls the turbulence compressibility model. Only applies to \texttt{eqn\_type=’generic’}.

‘off’ for no correction.

‘ssz’ for SSZ (use with Spalart-Allmaras models).

‘zeman’ for Zeman (use with $k-\epsilon$ models).

‘wilcox’ for Wilcox (use with SST-based models).

‘sarkar’ for Sarkar (use with $k-\epsilon$ models).

\texttt{turb\_conductivity\_model = ‘off’}

This controls whether a turbulence conductivity model is employed. Only applies to \texttt{eqn\_type=’generic’}.

‘off’ to turn off a turbulence conductivity model.

‘on’ to turn on a turbulence conductivity model.

\texttt{prandtl\_number\_turbulent = 0.9}

This is the turbulent Prandtl number.

\texttt{schmidt\_number\_turbulent = 1.}

This is the turbulent Schmidt number. Only applies to \texttt{eqn\_type=’generic’}.
B.4.8 &spalart

This namelist is used to modify details of the SA and SA based DES turbulence models.

```
&spalart
  turbinf    = 3.0
  dacles_mariani = .false.
  src = .false.
  ddes = .false.
/
```

```
turbinf = 3.0
This is the freestream turbulence value for the SA model.
```

```
dacles_mariani = .false.
This activates the Dacles-Mariani [27,28] rotation correction (denoted SA-R by http://turbmodels.larc.nasa.gov/).
```

```
sarc = .false.
This activates the rotation/curvature correction [29] (denoted SA-RC by http://turbmodels.larc.nasa.gov/).
```

```
ddes = .false.
This changes the turbulence_model='des' into Delayed DES [30] (DDES).
```
B.4.9 &code_run_control

This namelist controls the length of the simulation. Restart options, Jacobian
update strategy, and angle of attack continuation can also be specified.

&code_run_control
steps = 500
stopping_tolerance = 1.e-15
duration_limit_in_minutes = -1.0
no_restart = .false.
restart_write_freq = 250
restart_read = 'on'
smart_jupdate = .true.
jacobian_eval_freq = 0
jupdate_startup_steps = 10
ufduc3_jacobians = .false.
alpha_sweep = .false.
cycle_increment = 50
alpha_increment = 0.25
alpha_max = 180.0
alpha_min = -180.0
alpha_switchbacks = 0
/

steps = 500
This is the number of time steps or steady iterations to perform.

stopping_tolerance = 1.e-15
This instructs the solver to terminate before all steps are complete when
root mean square (RMS) of the residual is less than this tolerance. For
Euler or laminar perfect gas simulations, only the continuity equation
residual is examined. In all other simulations, each equation RMS (con-
tinuity, energy, etc.) must meet this tolerance.

duration_limit_in_minutes = -1.0
This is the maximum run duration limit in minutes (a negative value
is unlimited). This limit can terminate the solver before all steps are
complete, which may be helpful if the solver is run as a batch system job
with a time limit. Additional time is required to complete the current
iteration and write restart file. So, allow an extra time margin for code
shutdown. MPI required.

no_restart = .false.
When this is .true., no restart checkpoint file is written.
restart_write_freq = 250

The restart checkpoint and convergence history files will be written to disk every restart_write_freq time steps.

restart_read = 'on'

This defines the solution at the first time step.

'on' to initialize the simulation with a solution read from the restart file. The current convergence history will be concatenated with the prior solution history.

'on_nohistorykept' to initialize the simulation with a solution read from the restart file. The previous history (e.g., residuals, forces, moments) will be discarded.

'off' for no restart file read. The solution will be initialized as freestream or as specified in the &flow_initialization namelist.

smart_jupdate = .true.

This option allows the code to automatically adjust the Jacobian update frequency based on residual reduction.

jacobian_eval_freq = 0

This is the frequency of Jacobian evaluation based on time steps. It should be set to zero when smart_jupdate = .true.

jupdate_startup_steps = 10

The Jacobians are evaluated at every time step for the first jupdate_startup_steps, which aids robustness during initial start transients.

dfduc3_jacobians = .false.

This option only affects eqn_type = 'incompressible'. When .true., approximate Jacobians are computed that may improve the convergence of some cases.

alpha_sweep = .false.

This option activates a procedure to adjust angle_of_attack during a simulation. It can be used to calculate a drag polar in a single execution or explore a hysteresis loop. It is controlled by the following options.

cycle_increment = 50

When alpha_sweep=.true,

cycle_increment < 0 increments angle_of_attack after residuals have reached the stopping_tolerance.
cycle_increment > 0 is the number of iterations between increments to alpha.

cycle_increment = 0 is an inadmissible value.

alpha_increment = 0.25

When alpha_sweep=.true., increment angle_of_attack by these many degrees at a point controlled by cycle_increment.

alpha_max = 180.0

When alpha_sweep=.true., this is the maximum value of angle_of_attack.

alpha_min = -180.0

When alpha_sweep=.true., this is the minimum value of angle_of_attack.

alpha_switchbacks = 0

When alpha_sweep=.true., this is the number of directional changes in the angle_of_attack sweep. When alpha_switchbacks > 0, alpha_increment changed in sign after reaching alpha_max or alpha_min. This allows exploration of hysteresis loops.
B.4.10 &nonlinear_solver_parameters

This namelist defines the temporal accuracy of the solution advancement scheme. The subiterations and time step size of time accurate simulations can also be specified. The ramping of the pseudo time advancement CFL number is also set. Density and pressure floors on the update and relation factors are available.

&nonlinear_solver_parameters
  time_accuracy = 'steady'
  time_step_nondim = 0.0
  subiterations = 0
  temporal_err_control = .false.
  temporal_err_floor = 0.1
  schedule_number = -1
  schedule_iteration(1:2) = 1, 50
  schedule_cfl(1:2) = 200.0, 200.0
  schedule_cflturb(1:2) = 50.0, 50.0
  f_allow_minimum_m = 0.01
  invis_relax_factor = 1.0
  visc_relax_factor = 1.0
/

time_accuracy = 'steady'

This defines the temporal scheme.

'steady' for steady state calculations. This is a local time step pseudo-time advancement scheme that is not time accurate.

'1storder' is a first-order backward differencing scheme (backward Euler) for time-accurate temporal time integration.

'2ndorder' is a second-order backward differencing scheme (BDF2 in [31]) for time-accurate temporal time integration.

'2ndorderOPT' is an optimized second-order backward differencing (BDF2opt in [32]) for time-accurate temporal time integration. This scheme is second-order accurate in time but has an order-of-magnitude lower leading coefficient than standard BDF2.

'3rdorder' is a third-order backward differencing scheme (BDF3 in [31]) for time-accurate temporal integration.

'4thorderMEBDF4' is a fourth-order modified extended backward differencing scheme (MEBDF4 in [31]) for time-accurate temporal integration.

'4thorderESDIRK4' is a fourth-order explicit, singly diagonally implicit Runge-Kutta (ESDIRK4 in [31]) for time-accurate temporal integration.
time_step_nondim = 0.0
This is the nondimensional time step for time accurate simulations. It is ignored when time_accuracy = 'steady'. The nondimensionalization of this parameter depends on eqn_type. When eqn_type = 'compressible', it is $\frac{a_{ref} dt}{L}$, where $a_{ref}$ is the reference speed of sound, and $L$ is unit 1 of the grid. When eqn_type = 'incompressible' or 'generic', it is $\frac{u_{ref} dt}{L}$, where $u_{ref}$ is the reference velocity. See section 2.4 for more details and guidance on appropriate values.

subiterations = 0
Number of subiterations applied to solve the implicit time integration. It is ignored when time_accuracy = 'steady'. A constant CFL time step is used in each subiteration. By the end of a convergent subiteration process the pseudo time term drops out, giving the correct temporal discretization.

temporal_err_control = .false.
This governs whether the specified number of subiterations are run for each time step (.false.), or, if the temporal error is monitored and the subiterations are stopped when a specified tolerance is reached (.true.). It is ignored when time_accuracy = 'steady'.

temporal_err_floor = 0.1
This sets the tolerance for which time-accurate subiterations are stopped. The tolerance is given as a multiplicative factor of the flow residuals (mean and turbulence). It is ignored when time_accuracy = 'steady'.

schedule_number = -1
This variable is deprecated and will be removed. Please remove it from your namelist. A warning will be provided if this variable is set.

schedule_iteration(1:2) = 1, 50
These are the iteration or subiteration numbers at which CFL numbers are specified. When time_accuracy = 'steady', this controls the CFL number of the pseudo-time terms over iterations. When running time-accurately, this controls the CFL number of the pseudo-time terms of the linear system over subiterations. The parameter schedule_iteration(1) must be one, because it defines the starting CFL number at the first iteration or subiteration. The actual CFL number is determined by a linear ramp from schedule_cfl(1) at schedule_iteration(1) to schedule_cfl(2) at schedule_iteration(2). The CFL number is held constant at schedule_cfl(2) after schedule_iteration(2).
\texttt{schedule\_cfl(1:2) = 200.0, 200.0}

This controls the ramping and final CFL number of the meanflow equations. See the description for \texttt{schedule\_iteration}.

\texttt{schedule\_cfl\_turb(1:2) = 50.0, 50.0}

This controls the ramping and final CFL number of the turbulence model equations. See the description for \texttt{schedule\_cfl in schedule\_iteration}.

\texttt{f\_allow\_minimum\_m = 0.01}

This limits the solution update to prevent pressure and density from dropping below this fraction of their freestream values. Applied to eqn\_type = 'compressible' only:

\texttt{invis\_relax\_factor = 1.0}

This is the relaxation factor of inviscid terms. It scales the nonlinear update of the inviscid terms by this fraction and is only used for eqn\_type = 'generic'.

\texttt{visc\_relax\_factor = 1.0}

This is the relaxation factor of viscous terms. It scales the nonlinear update of the viscous terms by this fraction and is only used for eqn\_type = 'generic'.
B.4.11 &linear_solver_parameters

The FUN3D solution process involves constructing a linearization of the residual with appropriate time terms and then solving this linear system to compute the solution update. This namelist controls the solution process of this linear system. The linearization is grouped into the meanflow equations and the turbulence model equations.

&linear_solver_parameters
  meanflow_sweeps = 15
  turbulence_sweeps = 10
  linear_projection = .false.
  line_implicit = 'off'
/

  meanflow_sweeps = 15

This is the number of the linear system red-black relaxations at each steady iteration or time step of the meanflow equations when there is no turbulence model or a loosely coupled turbulence model. For eqn_type = 'generic' or fully coupled meanflow and turbulence relaxation, this refers to all equations (meanflow and turbulence).

  turbulence_sweeps = 10

This is the number of the linear system red-black relaxations at each steady iteration or time step of the turbulence equations, when the turbulence equations are loosely coupled. It has no effect for fully coupled meanflow and turbulence relaxation or a simulation without a turbulence model.

  linear_projection = .false.

This option uses a Krylov projection method generalized conjugate gradient (GCR) to stabilize and improve convergence of linear system. This will execute multiple sets of red-black relaxation to form the CGR search directions, until a convergence criteria is met.

  line_implicit = 'off'

This option selects the relaxation scheme.

‘off’ uses point implicit relaxation.

‘on’ uses line implicit relaxation where lines are defined and point relaxation elsewhere. The line implicit feature requires construction of these lines prior to running FUN3D. The lines are stored in the a file named [project_rootname].lines_fmt. The aflr3_line_extraction utility is distributed with FUN3D to generate these lines.
B.4.12 &boundary_conditions

This namelist provides auxiliary information to the boundary conditions. Refer to section 3 for the definition of boundary numbers and other information.

&boundary_conditions
    total_pressure_ratio(:) = 1.0
    total_temperature_ratio(:) = 1.0
    subsonic_inflow_velocity(:) = 'normal'
    alpha_bc(:) = 0.0
    beta_bc(:) = 0.0
    theta1(:) = 0.0
    theta2(:) = 0.0
    theta3(:) = 0.0
    ampl(:) = 0.0
    freq(:) = 0.0
    phase(:) = 0.0
    random(:) = .false.
    ramp_constant(:) = 1.0
    pressure_relaxation(:) = 1.0
    nozzle_symmetry = 1.0
    inlet_symmetry = 1.0
    npr_set = 0.0
    massflow_set_in = 0.0
    static_pressure_ratio(:) = 1.0
    inlet_solution_method(:) = 'mach'
    mach_bc(:) = 0.0
    massflow(:) = 0.0
    massflow_set_out = 0.0
    massflow_dimensions = 'nondim'
    grid_units = 'nondim'
    q_set(:,1:5) = 0.0
    q_set_ramp(:) = 0
    profile_type(:) = 'radial_polynomial'
    patch_center(:,1:3) = 0.0
    patch_scale(:) = 1.0
    profile_rho_coef(:,0:6) = 0.0
    profile_u_coef(:,0:6) = 0.0
    profile_p_coef(:,0:6) = 0.0
    profile_coef(:,1:3) = 0.0
    wall_velocity(:,1:3) = 0.0
    rotation_center(:,1:3) = 0.0
    rotation_vector(:,1:3) = 0.0
    rotation_rate(:) = 0.0
    unorm_bc(:) = 0.0
    wall_temperature(:) = 1.0
wall_temp_flag(:) = .false.
wall_radeq_flag(:) = .false.
wall_emissivity(:) = 0.8
wall_emissivity_b(:) = 0.
wall_emissivity_c(:) = 0.
wall_emissivity_d(:) = 0.
wall_temp_relax(:) = 0.001
wall_catalysis_model(:) = 'super-catalytic'
catalytic_efficiency_o(:) = 0.
catalytic_efficiency_n(:) = 0.
ablation_option_map(:) = 0
bprime_flag_map(:) = 1
compute_mdot_initial_map(:) = 1
freq_mdot_map(:) = 5000
freq_wall_map(:) = 50
uncoupled_ablation_flag_map(:) = 0
wall_blowing_model(:) = 'none'
virgin_density_wall(:) = 1.
char_density_wall(:) = 1.
CHONSi_frac_char_map(:,1) = 1.
CHONSi_frac_pyrolysis_map(:,1) = 1.
h_ablation_map(:,:) = 0.
mdot_pressure_map(:,:) = 0.
t_sublimation_map(:,:) = 0.
plenum_t0(:) = 1000.
plenum_p0(:) = 1000.
plenum_id(:) = 0
fixed_in_id(:) = 0
fixed_in_rho(:) = 0.
fixed_in_uvw(:,1:3) = 0.
fixed_in_t(:) = 0.
fixed_in_tv(:) = 0.
fixed_in_turb(:,1:7) = 0.
specified_transition(:) = .false.
impose_pressure_gradient = .false.
presure_gradient = 0.0
solidity(:) = 0.0
porous_coefficient(:) = 0.0
x_constant_boundary(:) = .false.
y_constant_boundary(:) = .false.
z_constant_boundary(:) = .false.
tol_const_coord = 1.0e-6
filter_c0(:) = 0.0
filter_c1(:) = 0.0
filter_c2(:) = 0.0
total_pressure_ratio(:) = 1.0
This is the ratio of plenum total pressure to reference pressure used by 7011 boundary condition. Inflow Mach number must be less than one.

total_temperature_ratio(:) = 1.0
This is the ratio of plenum total temperature to reference temperature used by 7011 boundary condition. Inflow Mach number must be less than one.

subsonic_inflow_velocity(:) = 'normal'
This sets the direction of the inflow velocity.
'normal' for inflow normal to each element face in the patch
'alpha,beta' the angle of the inflow is specified by alpha_bc and beta_bc as shown in Fig. 4.
'interior' the angle of the inflow is specified by the Euler angles theta1, theta2, and theta3 as shown in Fig. B4.
'offset' the angle of normal inflow to the patch boundary is rotated by the Euler angles theta1, theta2, and theta3 as shown in Fig. B4.

alpha_bc(:) = 0.0
When subsonic_inflow_velocity = 'alpha,beta', this is the angle of attack in radians for 7011 boundary condition inflow.

beta_bc(:) = 0.0
When subsonic_inflow_velocity = 'alpha,beta', this is the angle of sideslip in radians for 7011 boundary condition inflow.

theta1(:) = 0.0
When subsonic_inflow_velocity = 'interior' or 'offset', this is the Euler angle \( \theta \) in radians for 7011 boundary condition inflow.

theta2(:) = 0.0
When subsonic_inflow_velocity = 'interior' or 'offset', this is the Euler angle \( \phi \) in radians for 7011 boundary condition inflow.

theta3(:) = 0.0
When subsonic_inflow_velocity = 'interior' or 'offset', this is the Euler angle \( \psi \) in radians for 7011 boundary condition inflow.
ampl(:) = 0.0
For use with the 7103 boundary condition, this sets the amplitude of puls ed inflow conditions. Only the velocity is varied using the equation 
$[u,v,w] = \text{ampl} \cdot \sin(\text{freq} \cdot \text{simulation\_time} + \text{phase} \cdot \pi/180)$.

dfreq(:) = 0.0
For use with the 7103 boundary condition, this sets the frequency of pulsed inflow velocity values, see ampl.

dphase(:) = 0.0
For use with the 7103 boundary condition, this sets a phase shift (in degrees) of the inflow velocity values, see ampl.

drandom(:) = .false.
For use with the 7103 boundary condition, this creates random fluctuations in the magnitude of the supersonic inflow conditions in the range $[0, \text{ampl}]$ when .true..

dramp\_constant(:) = 1.0
For use with 7104 boundary condition, this ramps the velocity magnitude of supersonic inflow conditions with the exponential expression, $(1 - \exp(-\text{simulation\_time}/\text{ramp\_constant}))$.

dpressure\_relaxation(:) = 1.0
For use with the 5051 boundary condition, this weights the user specified back pressure with the solution pressure. $p_{\text{applied}} = \text{pressure\_relaxation} \cdot \text{static\_pressure\_ratio} + (1 - \text{pressure\_relaxation}) \cdot \text{pressure\_internal}$

\text{nozzle\_symmetry} = 1.0
Use this factor to scale the area and massflow of a nozzle face. For example, set to 2.0 to double a nozzle face on a symmetry plane or set to 1.0 for a full span model.

\text{inlet\_symmetry} = 1.0
Use this factor to scale the area and massflow of an inlet face. For example, set to 2.0 to double an inlet face on a symmetry plane or set to 1.0 for a full span model.

\text{npr\_set} = 0.0
This sets the nozzle pressure ratio for nozzle component performance.

\text{massflow\_set\_in} = 0.0
For use with 7036 boundary condition, this is the massflow into the computational domain in the units of grid squared. Inflow Mach number must be less than one.
**static_pressure_ratio:** 1.0

This is the ratio of specified boundary static pressure to reference pressure used by the 7012 and 5051 boundary conditions. Outflow Mach number must be less than one.

**inlet_solution_method:** 'mach'

This is the solution method for 7031 bc.

'mach' adjusts back pressure to attain a Mach number *mach_bc*. Do not use for a face that has boundary layer ingestion; use 'pressure' in that situation.

'massflux' adjusts back pressure to attain a specified mass flux *massflow*.

'pressure' adjusts back pressure to attain a specified mass flux *massflow*.

**mach_bc:** 0.0

This is Mach number on boundary face used by 5052 and 7031 boundary conditions. Outflow Mach number must be less than one.

**massflow:** 0.0

This is mass flow through boundary face in units of grid unit squared used by 7031 and 7036 boundary conditions. It is nondimensionalized according to \((\rho/\rho_\infty)(u/u_\infty)A_{boundarycondition}\).

**massflow_set_out:** 0.0

For use with 7031 boundary condition, this is massflow out of the computational domain in units of grid squared. Outflow Mach number must be less than one.

**massflow_dimensions:** 'nondim'

This is used for converting a user specified dimensional massflow to units of mesh units squared.

'nondim' for nondimensional massflow input (units of mesh squared)

'english' for lbm/s

'metric' for kg/s

**grid_units:** 'nondim'

This converts a user specified dimensional massflow to units of mesh unit squared. If massflow is input in units of mesh units squared, *grid_units* is not required.

'nondim' for nondimensional input

'm' for meters

'cm' for centimeters
‘mm’ for millimeters
‘feet’ for feet
‘inches’ for inches

q_set(:,1:5) = 0.0
This sets the primitive variables on boundary face used by boundary conditions 7100 and 7105.

q_set_ramp(:) = 0
When this is greater than zero, primitive variables on boundary face are ramped from zero to q_set over these iterations, for boundary conditions 7100 and 7105.

profile_type(:) = ‘radial Polynomial’
This switches between inflow profiles,
‘radial Polynomial’ defines a radial polynomial about patch_center
of size patch_scale with profile_rho_coef, profile_u_coef, and profile_p_coef.
‘power law’ defines a power-law velocity profile function with profile_coef.

patch_center(:,1:3) = 0.0
This is the center of a 7101 bc.

patch_scale(:) = 1.0
This scales the radius of a 7101 bc.

profile_rho_coef(:,0:6) = 0.0
This is the radius polynomial coefficients of density for 7101 bc.

profile_u_coef(:,0:6) = 0.0
This is the radius polynomial coefficients of u for 7101 bc.

profile_p_coef(:,0:6) = 0.0
This is the radius polynomial coefficients of pressure for 7101 bc.

profile_coef(:,1:3) = 0.0
These three coefficients are wind speed reference, reference height, and power law exponent,
profile_coef(:,1)*(z/profile_coef(:,2))**profile_coef(:,3)

wall_velocity(:,1:3) = 0.0
This is the 4000 solid wall specified translational velocity. It should be tangent to the boundary to ensure a well-posed problem.
rotation_center(:,1:3) = 0.0
This is the 4000 solid wall specified rotational velocity center point, see rotation_vector and rotation_rate.

rotation_vector(:,1:3) = 0.0
This is the 4000 solid wall specified rotational vector, see rotation_center and rotation_rate. Should be unit length.

rotation_rate(:) = 0.0
This is the 4000 solid wall specified rotational rate, see rotation_center and rotation_vector

unorm_bc(:) = 0.0
This is the specified velocity in the boundary normal direction, for 4000 solid walls.

wall_temperature(:) = 1.0
This is the ratio of wall temperature to reference temperature for eqn_type = 'compressible' or the wall temperature in degrees Kelvin for eqn_type = 'generic'. If set to -1, then the wall temperature is computed so that there is zero heat flux, i.e., adiabatic. The wall_temp_flag must be set to .true. for this boundary condition to take effect.

wall_temp_flag(:) = .false.
This must be .true. when specifying the wall temperature via wall_temperature.

wall_radeq_flag(:) = .false.
Compute the wall temperature via the Stefan-Boltzmann Law. The radiative equilibrium wall temperature is computed from the heating rate \( q_{\text{wall}} \) using \( q_{\text{wall}} = \epsilon \sigma T^4_{\text{radeq}} \) where the surface emissivity \( \epsilon \) is entered as wall_emissivity and \( \sigma \) is the Stefan-Boltzmann constant.

wall_emissivity(:) = 0.8
This is \( \epsilon_0 \), where emissivity is specified as a function of wall temperature with the expression \( \epsilon = \epsilon_0 + T(\epsilon_b + T(\epsilon_c + T\epsilon_d)) \). The other coefficients are entered via the following three variables.

wall_emissivity_b(:) = 0.
This is \( \epsilon_b \) in the above equation.

wall_emissivity_c(:) = 0.
This is \( \epsilon_c \) in the above equation.
wall_emissivity.d(:) = 0.
This is $\epsilon_d$ in the above equation.

wall_temp_relax(:) = 0.001
This is the relaxation factor $\eta$ used for wall_radeq_flag wall temperature boundary condition. The wall temperature is updated as $T^{\text{new}} = T^{\text{old}} + \eta * (T_{\text{radeq}} - T^{\text{old}})$

wall_catalysis_model(:) = 'super-catalytic'
This defines the catalytic efficiency of the wall to promote recombination of atoms to molecules. Allowable options are:

- **super-catalytic** Forces the mass fraction of species at the wall to equal the mass fractions specified for the free stream in the tdata file.
- **fully-catalytic** Specifies a catalytic efficiency of 1 thereby forcing homogeneous recombination of atoms diffusing to the wall.
- **non-catalytic** Specifies a zero mole fraction gradient at the wall - signifying zero catalytic efficiency.
- **equilibrium-catalytic** Computes the equilibrium chemical composition of species at the wall temperature and pressure.
- **constant-catalytic** Catalytic efficiency is user specified constants
- **Stewart-RCG** Reaction cured glass from Stewart
- **Zoby-RCG** Reaction cured glass from Zoby
- **Scott-RCG** Reaction cured glass from Scott
- **CSiC** Experimental CSiC from JSC for X-38
- **RCC-LVP** Stewart NASA TM 112206
- **CCAT-ACC** Shuttle RCC from Stewart NASA TM 112206
- **CSiC-SNECMA** Derived from Stewart RCC

\[ \text{catalytic_efficiency}_o(\cdot) = 0. \]
This is the fraction of diffusion flux of atomic oxygen striking wall that is converted to molecular oxygen, when \text{wall_catalysis_model} = 'constant-catalytic'.

\[ \text{catalytic_efficiency}_n(\cdot) = 0. \]
This is the fraction of diffusion flux of atomic nitrogen striking wall that is converted to molecular nitrogen, when \text{wall_catalysis_model} = 'constant-catalytic'.
**ablation_option_map(:) = 0**

This is an integer that specifies whether the pyrolysis ablation rate and wall temperature are computed in addition to the char ablation rate. This option only affects cases with `bprime_flag_map` equal to 0 or 1.

`0` The pyrolysis ablation rate and wall temperature are computed, in addition to the char ablation rate, assuming steady-state ablation.

`1` The pyrolysis ablation rate and wall temperature are held constant (they are set to the values present in `ablation_from_laura.m`) while the char ablation rate is computed.

**bprime_flag_map(:) = 1**

This is an integer defining if the b-prime approach is applied. Applicable only for blowing model `equil_char_quasi_steady = 0`.

`0` Do not use bprime approach, and instead use a rigorous diffusion model. This option is consistent with the ”Fully-Coupled” approach defined in Ref. [2].

`1` Use b-prime approach. This option is consistent with the ”Partially-Coupled” approach defined in Ref. [2].

`2` Hold the ablation rate and wall temperature constant from the restart file, while applying the rigorous diffusion model (thus, the surface energy balance and char equilibrium constraint are not satisfied). This option is sometimes useful when transitioning from a bprime flag = 1 computation to a bprime flag = 0 computation.

**compute_mdot_initial_map(:) = 1**

This is an integer defining if the ablation rates are computed before the first flowfield iteration.

`0` Applies the ablation rates and wall temperatures present in the `ablation_from_laura.m` file.

`1` Computes the ablation rates and wall temperatures before the first flowfield iteration.

**freq_mdot_map(:) = 5000**

For `bprime_flag_map = 1`, this is an integer defining frequency of updating ablation rates and wall temperature.

**freq_wall_map(:) = 50**

For `bprime_flag = 1`, this is an integer defining frequency of update to ablation wall boundary conditions. For `bprime_flag = 0`, an integer defining frequency of update to the surface energy balance solution, which defines the wall temperature.
uncoupled_ablation_flag_map(:) = 0
This is an integer defining if an uncoupled ablation analysis is applied. The uncoupled ablation option is included to provide a baseline solution for the coupled ablation analysis.

‘0’ Do not apply an uncoupled ablation analysis.

‘1’ Apply an uncoupled ablation analysis to a converged non-ablating flowfield.

wall_blowing_model(:) = ‘none’
This is the blowing or ablation model.

‘none’ No wall blowing

‘specified’ blowing rate is user specified function of pressure (see also mdot_press)

‘porous_chamber’ Special options for simulation of buoyancy driven flow in pressurized rig for BNNT production

‘quasi_steady’ Compute ablation rate as function of surface energy balance and equilibrium catalytic bc

‘equil_char_quasi_steady’ Include equilibrium char approximation

‘FIAT’ Couple to material response code FIAT (not active)

virgin_density_wall(:) = 1.
This is the density (kg/m$^3$) of thermal protection system ablator in virgin state (prior to heating level sufficient to cause any reactions).

char_density_wall(:) = 1.
This is the density (kg/m$^3$) of remaining char in ablator after binding resins have pyrolized.

CHONSi_frac_char_map(:,1) = 1.
See definition below for CHONSi_frac_pyrolysis_map

CHONSi_frac_pyrolysis_map(:,1) = 1.
These arrays set elemental mass fraction (second index) of C, H, O, N, Si, Fe, Mg, Na, B species for char and pyrolysis gas. The fractions in each array should sum to 1.

h_ablation_map(:,:) = 0.
This is a vector of extent 3 used to compute the heat of ablation in MJ/kg for quasi steady blowing option as $h_{\text{ablation}}_0(1) + (h_{\text{ablation}}_0(2) \log pw + (h_{\text{ablation}}_0(3))(\log pw)**2$ where $pw$ is the local pressure, in atmospheres.
\texttt{mdot\_pressure\_map(\text{:,}) = 0.}

This is a vector of extent 2 used to set the blowing or suction distribution defined as \texttt{mdot\_pressure\_0(1) + (mdot\_pressure\_0(2))*p/(rho\_inf*V\_inf**2)} where \(p\) is the local pressure, \(\rho_{\text{inf}}\) is the reference density, and \(V_{\text{inf}}\) is the reference velocity. Positive value produces blowing distribution, while negative value produces suction distribution.

\texttt{t\_sublimation\_map(\text{:,}) = 0.}

This is a vector of extent 3 used to compute the sublimation temperature in degrees Kelvin for quasi steady blowing option as \texttt{t\_sublimation\_0(1) + (t\_sublimation\_0(2))*log\ pw + (t\_sublimation\_0(3))*log\ pw**2} where \(pw\) is the local pressure, in atmospheres.

\texttt{plenum\_t0(\text{:) = 1000.}}

For use with the 7021 boundary condition, this is the total plenum temperature in Kelvin.

\texttt{plenum\_p0(\text{:) = 1000.}}

For use with the 7021 boundary condition, The total plenum pressure in \(N/m^2\) (Pascals) feeding this boundary.

\texttt{plenum\_id(\text{:) = 0}}

For use with the 7021 boundary condition (one or more \texttt{rcs\_jet plenum bcs}), the jet plenum contains this species set from the file \texttt{tdata}. For example, if an RCS jet is firing \(H_2\) and \(O_2\) into an air stream, the \texttt{tdata} file may look like:

\begin{verbatim}
One Temperature
N
O
N2 0.76
O2 0.24
NO

H2 0.5
O2 0.5
OH
H
O
\end{verbatim}

In this case, if the boundary to the plenum is surface number 5 then \texttt{plenum\_id(5)=2}, the second grouping of species in the \texttt{tdata} file. The numbers following the species name define the mass fraction of that species at the inflow boundary. The sum of the mass fraction in each
group must equal one. Species groups are separated by blank lines and multiple RCS jets may be defined in this manner.

\texttt{fixed_in_id(:) = 0}

For use with the 70XX boundary condition (one or more supersonic inflow bcs) the supersonic inflow boundary condition contains the species set in the same way as the \texttt{plenum_id} for the \texttt{rcs_jet} plenum boundary condition described above.

\texttt{fixed_in_rho(:) = 0.}

For use with the 70XX boundary condition (one or more supersonic inflow bcs) the dimensional inflow mixture density in kg/m**3.

\texttt{fixed_in_uvw(:,1:3) = 0.}

For use with the 70XX boundary condition (one or more supersonic inflow bcs) the dimensional inflow Cartesian velocity components in m/sec.

\texttt{fixed_in_t(:) = 0.}

For use with the 70XX boundary condition (one or more supersonic inflow bcs) the dimensional inflow translational rotational temperature in Kelvin.

\texttt{fixed_in_tv(:) = 0.}

For use with the 70XX boundary condition (one or more supersonic inflow bcs) the dimensional inflow vibrational-electronic temperature in Kelvin.

\texttt{fixed_in_turb(:,1:7) = 0.}

For use with the 70XX boundary condition (one or more supersonic inflow bcs) This is for the turbulence models for a one-equation model this is the ratio of the inflow eddy viscosity to the inflow molecular viscosity, for a two-equation model this is: the inflow turbulence intensity and the ratio of the inflow eddy viscosity to the inflow molecular viscosity. Full Reynolds stress models are not currently supported.

\texttt{specified_transition(:) = .false.}

When \texttt{.true.}, a turbulent transition point will be imposed on the solution.

\texttt{impose_pressure_gradient = .false.}

When \texttt{.true.}, a global pressure gradient in the x-direction of \texttt{pressure_gradient} will be imposed as a source term to the residual.
pressure\_gradient = 0.0
The nondimensional pressure gradient in the $x$-direction, imposed when impose\_pressure\_gradient = .true.

solidity(:) = 0.0
This is the percent solidity to be applied to the passive porosity boundary conditions.

porous\_coefficient(:) = 0.0
This is the Darcy law equation coefficient.

x\_constant\_boundary(:) = .false.
This specifies that a boundary is an $x$ constant face for mesh movement, constraining motion the to be tangent to face. Set automatically for $x$-symmetry boundaries.

y\_constant\_boundary(:) = .false.
This specifies that a boundary is an $y$ constant face for mesh movement, constraining motion the to be tangent to face. Set automatically for $y$-symmetry boundaries.

z\_constant\_boundary(:) = .false.
This specifies that a boundary is an $z$ constant face for mesh movement, constraining motion the to be tangent to face. Set automatically for $z$-symmetry boundaries.

tol\_const\_coord = 1.0e-6
This is the tolerance for verifying that a boundary surface is a planar boundary for mesh movement by restricting the minimum and maximum value in the “constant” direction.

filter\_c0(:) = 0.0
This is the constant for pressure loss through a filter, where $\Delta p = filter\_c2 v^2 + filter\_c1 v + filter\_c0$.

filter\_c1(:) = 0.0
This is the linear coefficient for pressure loss through a filter, where $\Delta p = filter\_c2 v^2 + filter\_c1 v + filter\_c0$.

filter\_c2(:) = 0.0
This is the quadratic coefficient for pressure loss through a filter, where $\Delta p = filter\_c2 v^2 + filter\_c1 v + filter\_c0$. 
B.4.13 &component_parameters

This namelist provides expanded ability to track forces, moments, and mass-flows according to user-specified groups of boundary patches that define a "component."

```
&component_parameters
  number_of_components   = 0
  component_count(:)    = 0
  component_moment_center(1,:) = x_moment_center
  component_moment_center(2,:) = y_moment_center
  component_moment_center(3,:) = z_moment_center
  component_x_moment_length(:) = x_moment_length
  component_y_moment_length(:) = y_moment_length
  component_input(:)    = ''
  component_name(:)     = ''
  allow_flow_through_forces = .false.
/

number_of_components = 0
This is the number of components (groups of boundary patches) to track.

component_count(:) = 0
This is the number of boundary patches assigned to a component. If -1 is given, this number is computed implicitly from component_input.

component_moment_center(1,:) = x_moment_center
This is x-coordinate of the moment center assigned to a component. The default value comes from the force and moment namelist, &force_moment_integ_properties.

component_moment_center(2,:) = y_moment_center
This is y-coordinate of the moment center assigned to a component. The default value comes from the force and moment namelist, &force_moment_integ_properties.

component_moment_center(3,:) = z_moment_center
This is z-coordinate of the moment center assigned to a component. The default value comes from the force and moment namelist, &force_moment_integ_properties.

component_x_moment_length(:) = x_moment_length
This is the x-direction reference length assigned to a component used to non-dimensionalize moments about y (pitching moment). The default value comes from the force and moment namelist &force_moment_integ_properties.

component_y_moment_length(:) = y_moment_length
This is the y-direction reference length assigned to a component used to non-dimensionalize moments about x (rolling moment) and z (yawing
moment). The default value comes from the force and moment namelist &force_moment_integ_properties.

\[\text{component_input}(:) = \text{''}\]

This is the list of boundary patches to assigned to a component. Boundary indexes are separated with commas and dashes can be used to specify ranges, i.e., \'1,2,5-7\'.

\[\text{component_name}(:) = \text{''}\]

This is the component output filename, \([\text{project_rootname}]_{\text{fm}}[\text{component_name}]\).dat.

\[\text{allow_flow_through_forces} = \text{.false.}\]

Pressure drag and skin friction forces are by default calculated only on boundary faces designated as solid surfaces (viscous or inviscid). To include the pressure and momentum flux forces due to nozzles or inlets allow_flow_through_forces should be set to .true.. The flow-through faces to be tracked should be listed in the component_count and component_input lists accordingly.
B.4.14 &two_d_trans

This namelist is used to specify a 2D transition location. If the airfoil is split into upper and lower patches, the transition location can be specified independently on each patch. If there is only a single patch, it can be split with a z value to designate the upper and lower airfoil surfaces. This transition specification is limited to specifying transition on a single-element configuration such as an airfoil or a flat plate. Only a single transition location is supported for multi-element airfoils. Transition is modeled by turning off the turbulent production terms in “laminar” regions of the grid. This option is only valid for the perfect gas SA turbulence model. This is the same approach taken in CFL3D and NSU3D. FUN3D results from this approach for a DLR-F6 transonic cruise condition are shown in Lee-Rausch et al. [33]

```
&two_d_trans
  turb_transition = .false.
  use_2d_values = .false.
  upper_patch = 1
  lower_patch = 1
  use_z_value = .false.
  z_location = 0.0
  upper_x_location = 0.0
  lower_x_location = 0.0
/

  turb_transition = .false.
  This must be .true. to specify laminar regions of flow during a turbulent
  flow simulation.

  use_2d_values = .false.
  This enables 2D transition specification.

  upper_patch = 1
  This is the upper patch with specified transition.

  lower_patch = 1
  This is the lower patch with specified transition.

  use_z_value = .false.
  This allows a single patch to be split into upper and lower surfaces of an
  airfoil by a z plane.

  z_location = 0.0
  This is the z location to split the airfoil if use_z_value = .true.
```
upper_x_location = 0.0
This is the upper surface $x$ transition location.

lower_x_location = 0.0
This is the lower surface $x$ transition location.
This namelist is used to specify 3D boundary layer transition locations. The command line option \texttt{--turb\_transition} is required to activate. If you run the flow solver without the \texttt{--turb\_transition}, it will default to fully turbulent even though you have the laminar boundaries defined. Transition is modeled by the turning off the turbulent production terms in “laminar” regions of the grid. This option is only valid for the perfect gas SA turbulence model. This is the same approach taken in CFL3D and NSU3D. Fun3D results from this approach for a DLR-F6 transonic cruise condition are shown in Lee-Rausch et al. [33]

\begin{verbatim}
&three_d_trans
  use_3d_values = .false.
  n_transition_group = 1
  transition_group_patches(:) = '1'
  transition_x1(:) = 0.0
  transition_y1(:) = 0.0
  transition_x2(:) = 0.0
  transition_y2(:) = 1.0
/

  use_3d_values = .false.
  This turns 3D transition specification on.

  n_transition_group = 1
  This is the number of patch groups, limited to 100.

  transition_group_patches(:) = '1'
  This is the patch indexes for each group. Commas and dashes can be used to specify ranges, i.e., '1,2,5-7'.

  transition_x1(:) = 0.0
  This is the x value for determining the start of the transition line.

  transition_y1(:) = 0.0
  This is the y value for determining the start of the transition line.

  transition_x2(:) = 0.0
  This is the x value for determining the end of the transition line.

  transition_y2(:) = 1.0
  This is the y value for determining the end of the transition line.
\end{verbatim}
B.4.16  &special_parameters

This namelist specifies changes to the discretization to handle elements with large face angles.

&special_parameters
  large_angle_fix      = 'off'
  edge_averaging_tet_m = .false.
  edge_averaging_tet_t = .true.
  ebv_tets             = .false.
/

large_angle_fix = 'off'

Grids with elements that have adjacent face angles that approach 180 degrees may result in a sudden onset of not a number (NaN). Grids produced by VGRID may contain these elements.

‘off’ uses all elements in viscous flux evaluation. This is a consistent viscous discretization.

‘on’ neglects viscous fluxes in cells containing angles between adjacent faces of 178 degrees or greater. This is an inconsistent discretization, but may allow the calculation of a solution on a grid that is not suitable for the consistent viscous discretization.

edge_averaging_tet_m = .false.

Average viscosity at edge in meanflow viscous diffusion for element-based grids on tetrahedra. If false, enables efficient numerical integration (with constant viscosity over an element) and corresponds to a classical Galerkin finite-element scheme.

edge_averaging_tet_t = .true.

Average viscosity at edge in turbulent viscous diffusion for element-based grids on tetrahedra. If false, enables efficient numerical integration (with constant viscosity over an element) and corresponds to a classical Galerkin finite-element scheme.

ebv_tets = .false.

Use edge-based viscous discretization on tetrahedral elements.
B.4.17 &flow_initialization

This namelist allows the user to initialize regions of the meanflow solution with quantities other than freestream. A maximum of 100 volumes can be defined. The volumes may overlap each other as well as domain boundaries. In the event that a grid point is contained in more than one volume, a subsequent volume in this file will supersede the volumes listed before it. Boundary conditions supersede the flow initialization.

&flow_initialization
  number_of_volumes = 0
  type_of_volume(:) = 'none'
  center(1:3,:) = 0.0
  radius(:) = 0.0
  point1(1:3,:) = 0.0
  point2(1:3,:) = 0.0
  radius1(:) = 0.0
  radius2(:) = 0.0
  rho(:) = 1.0
  c(:) = 1.0
  u(:) = 0.0
  v(:) = 0.0
  w(:) = 0.0
/

number_of_volumes = 0
This is the number of initialization volumes.

type_of_volume(:) = 'none'
This is the type of initialization volume.

'box' for a box. The diagonal corners are specified by point1 and point2.

'sphere' for a sphere. The position and size is specified by center and radius.

'cylinder' for a cylinder with size radius. The center axis is defined between point1 and point2.

'cone' for a cone or frustum. The center axis is defined between point1 and point2. Two radii are required, radius1 at point1 and radius2 at point2. A frustum is specified with two nonzero radii.

center(1:3,:) = 0.0
This is the center of the 'sphere' volume type.
\begin{verbatim}
radius(:) = 0.0
This is the radius of the 'sphere' and 'cylinder' volume types.

point1(1:3,:) = 0.0
This is one end of the 'cone' or 'cylinder' volume types or one corner of a 'box' volume type.

point2(1:3,:) = 0.0
This is the other end of the 'cone' or 'cylinder' volume types or the opposite corner of a 'box' volume type.

radius1(:) = 0.0
This is the radius at point1 of 'cone' volume type.

radius2(:) = 0.0
This is the radius at point2 of 'cone' volume type.

rho(:) = 1.0
This is the nondimensional density in the volume.

c(:) = 1.0
This is the nondimensional speed of sound in the volume.

u(:) = 0.0
This is the nondimensional x-component of velocity in the volume.

v(:) = 0.0
This is the nondimensional y-component of velocity in the volume.

w(:) = 0.0
This is the nondimensional z-component of velocity in the volume.
\end{verbatim}
B.4.18  &time_avg_params

This namelist controls the bookkeeping of time average and root mean square statistics for every point in the domain. Tracking these statistics enables visualization of variables like \( p_{\text{tavg}} \), \( u_{\text{trms}} \), etc. See the visualization output variable namelists &volume_output_variables, &boundary_output_variables, and &sampling_output_variables for details. When these statistics are tracked, the file \([\text{project_rootname}]_{-}\text{TAVG.1}\) maintains information across restarts. This statistics file is similar to the \([\text{project_rootname}]_{-}\text{flow}\) restart file in that it is not intended for the user to interact with directly.

```plaintext
&time_avg_params
  itime_avg  = 0
  prior_time_avg  = 0
  use_prior_time_avg  = 0
  tavg_header_version = 1
/

  itime_avg = 0
  This controls collection of statistics.
  '0' does not compute time averaging statistics.
  '1' computes time averaging statistics.

  prior_time_avg = 0
  This option specifies if a statistics file \([\text{project_rootname}]_{-}\text{TAVG.1}\) exists.
  '0' when no time averaging file exists.
  '1' when time averaging file from previous run exists.

  use_prior_time_avg = 0
  If available, use prior statistics.
  '0' for discarding the prior statistics.
  '1' for using and appending to the prior statistics.

  tavg_header_version = 1
  This option controls the variables for which statistics are collected.
  '1' primitive variable averages and root mean squares
  '2' primitive variable averages and root mean squares and the averages of \( u'v', u'w', v'w', \mu_t, vort_mag \).
```
B.4.19 &global

This namelist controls the frequency of visualization output and the logging of command line options. It also serves to control some global options otherwise set by command-line input.

&global
    moving_grid = .false.
    grid_motion_only = .false.
    grid_motion_and_dci_only = .false.
    body_motion_only = .false.
    timing = .false.
    time_moving_grid = .false.
    boundary_animation_freq = 0
    volume_animation_freq = 0
    slice_freq = 0
    record_command_lines = .false.
/

moving_grid = .false.
This governs whether the grid is moving or stationary

grid_motion_only = .false.
This turns off the flow solve during a simulation for which moving_grid = .true.. If the simulation involves overset grids, this command overrides dci_on_the_fly and DCI files are not output.

grid_motion_and_dci_only = .false.
This turns off the flow solve during a simulation for which moving_grid = .true.. If the simulation involves overset grids, this command honors dci_on_the_fly and DCI files are output if dci_on_the_fly = .true..

body_motion_only = .false.
This turns off both the flow solve and the linear elasticity solve during a simulation for which moving_grid = .true. and grid_motion = 'deform'

timing = .false.
This triggers a timing of the execution of various sections of the flow solver.

time_moving_grid = .false.
This triggers timing of the execution of various sections of the flow solver, with emphasis on operations associated with grid motion. The timing occurs over larger sections of the code than the timing option. For
correct timing information, timing and time_moving_grid should not be used simultaneously.

**boundary_animation_freq = 0**

This is the visualization output frequency of the domain boundaries. Zero is no output, -1 is output at the end of run, and a positive integer is periodic output every boundary_animation_freq iterations. See the &boundary_output_variables namelist for more details.

**volume_animation_freq = 0**

This is the visualization output frequency of the domain volume. Zero is no output, -1 is output at the end of run, and a positive integer is periodic output every volume_animation_freq iterations. See &volume_output_variables namelist for more details.

**slice_freq = 0**

This is the output frequency of boundary slices for visualization and to obtain loads. Zero is no output, -1 is output at the end of run, and a positive integer is periodic output every slice_freq iterations. See &slice_data namelist for more details.

**record_command_lines = .false.**

This creates a file temp.commands that contains the command line arguments
B.4.20 \&volume_output_variables

This namelist controls volume variable output. Output frequency is controlled by \texttt{volume\_animation\_freq} in the \&global namelist. The resulting volume-data files have the following naming convention for \texttt{export\_to='tecplot'} output:

```
[project\_rootname]\_part\[P\]\_tec\_volume\_timestep\[T\](.dat|.plt) if freq > 0
[project\_rootname]\_Part\[P\]\_tec\_volume(.dat|.plt) if freq < 0
```

where \(P = 1,2,\ldots,\text{nproc}\) (number of processors) and \(T\) is the iteration number. The file extension is \texttt{.dat} for ASCII Tecplot™ format and \texttt{.plt} for binary Tecplot™ format. Within the files, a single zone is written, with the zone title “time 0.0000000E+00 processor 32” where the time value is the integer iteration number for steady-state cases, and the current (non-dimensional) time for time-dependent cases.

A request to output an undefined variable will be overruled, i.e., \texttt{turb1} will be forced to \texttt{.false} regardless of user input when there is no turbulence model in the simulation.

\begin{verbatim}
&volume_output_variables
  export_to       = 'tecplot'
x               = .true.
y               = .true.
z               = .true.
primitive_variables = .true.
rho             = .false.
u               = .false.
v               = .false.
w               = .false.
p               = .false.
entropy         = .false.
mach            = .false.
temperature     = .false.
iblank          = .false.
imesh           = .false.
vort_mag        = .false.
vort_x          = .false.
vort_y          = .false.
vort_z          = .false.
q_criterion     = .false.
div_vel         = .false.
turbulent_fluctuations = .false.
uuprime         = .false.
vvprime         = .false.
wwprime         = .false.
\end{verbatim}

144
uvprime = .false.
uwprime = .false.
vwprime = .false.
cp = .false.
dp_pinf = .false.
volume = .false.
residuals = .false.
res1 = .false.
res2 = .false.
res3 = .false.
res4 = .false.
res5 = .false.
res_gcl = .false.
primitive_tavg = .false.
rho_tavg = .false.
u_tavg = .false.
v_tavg = .false.
w_tavg = .false.
p_tavg = .false.
primitive_trms = .false.
rho_trms = .false.
u_trms = .false.
v_trms = .false.
w_trms = .false.
p_trms = .false.
lambda1 = .false.
lambda2 = .false.
lambda3 = .false.
lambda4 = .false.
lambda5 = .false.
lambda6 = .false.
lambda7 = .false.
htot = .false.
ttot = .false.
ptot = .false.
etot = .false.
processor_id = .false.
turb_ke = .false.
turb_diss = .false.
mu_t = .false.
turb1 = .false.
turb2 = .false.
turb3 = .false.
turb4 = .false.
turb5 = .false.
turb6 = .false.
turb7 = .false.
turres1 = .false.
turres2 = .false.
turres3 = .false.
turres4 = .false.
turres5 = .false.
turres6 = .false.
turres7 = .false.
slen = .false.
iflagslen = .false.
hrles_blend = .false.
reconstruction_limiter_phi1 = .false.
reconstruction_limiter_phi2 = .false.
reconstruction_limiter_phi3 = .false.
reconstruction_limiter_phi4 = .false.
reconstruction_limiter_phi5 = .false.
tt = .false.
tv = .false.
sonic = .false.
mixture_mol_weight = .false.
mixture_density = .false.
ev = .false.
rho_i(1:n_species) = .false.
mu = .false.
id_l2g = .false.
divided_residuals = .false.

/ export_to = 'tecplot'

file format of volume export

'tecplot' is Tecplot™ format (one file written for each processor)

cgns is CGNS format, requires FUN3D to be configured with a CGNS library. This format already includes x, y, and z. Set these variables to .false. to avoid duplication.

'fvuns' is FieldView C-binary (Fortran stream) format. This format already includes x, y, and z. Set these variables to .false. to avoid duplication.

'fv' is depreciated. Slated for removal, use 'fvuns'

'VTK' is legacy VTK format

csv is comma separated value format

'sol' is INRIA Metrix sol format
'tec' is a single image ASCII Tecplot™ format
‘raw_ascii’ is a single image ASCII space separated format
‘native’ is the most efficient way to export the entire flow field to disk at every time step when the grid is over a billion elements, but requires specialized visualization tools to read.

```
x = .true.
X-coordinate
y = .true.
Y-coordinate
z = .true.
Z-coordinate
primitive_variables = .true.
Output primitive variables: rho, u, v, w, and p
rho = .false.
Density
u = .false.
X-component of velocity
v = .false.
Y-component of velocity
w = .false.
Z-component of velocity
p = .false.
Pressure
entropy = .false.
Entropy
mach = .false.
Mach number
temperature = .false.
Temperature
iblank = .false.
I-blanking variable (default becomes .true. for overset mesh cases)
```
imesh = .false.
For overset mesh systems, index of associated component grid where 0 indicates background grid

vort_mag = .false.
Vorticity magnitude

vort_x = .false.
X-component of vorticity

vort_y = .false.
Y-component of vorticity

vort_z = .false.
Z-component of vorticity

q_criterion = .false.
Q Criterion, the second invariant of \( \nabla V \)

div_vel = .false.
Velocity divergence

turbulent_fluctuations = .false.
Activate all the following \( XY^\prime \) turbulent shear stresses normalized by \( u_{ref}^2 \); the definition of these variables depends on the turbulence model, see [http://turbmodels.larc.nasa.gov/noteonrunning.html](http://turbmodels.larc.nasa.gov/noteonrunning.html) for details

uuprime = .false.
Turbulence fluctuation, \( u'u' \)

vvprime = .false.
Turbulence fluctuation, \( v'v' \)

wwprime = .false.
Turbulence fluctuation, \( w'w' \)

uvprime = .false.
Turbulence fluctuation, \( u'v' \)

uwprime = .false.
Turbulence fluctuation, \( u'w' \)

vwprime = .false.
Turbulence fluctuation, \( v'w' \)
cp = .false.
Pressure coefficient
dp_pinf = .false.
Normalized delta pressure \( (p - p_\infty)/p_\infty\)
volume = .false.
Dual-cell volume size
residuals = .false.
Activate all resN variables
res1 = .false.
Residual of equation 1, density
res2 = .false.
Residual of equation 2, x-momentum
res3 = .false.
Residual of equation 3, y-momentum
res4 = .false.
Residual of equation 4, z-momentum
res5 = .false.
Residual of equation 5, energy
res_gcl = .false.
For moving meshes, residual of grid conservation law
primitive_tavg = .false.
Output time-averaged primitives (requires &time_avg_params namelist):
rho_tavg, u_tavg, v_tavg, w_tavg, and p_tavg
rho_tavg = .false.
Time-averaged density
u_tavg = .false.
Time-averaged x-component of velocity
v_tavg = .false.
Time-averaged y-component of velocity
w_tavg = .false.
Time-averaged z-component of velocity
p_tavg = .false.
Time-averaged pressure

primitive_trms = .false.
Output root mean squared primitives (requires &time_avg_params namelist):
 rho_trms, u_trms, v_trms, w_trms, and p_trms

rho_trms = .false.
RMS-average of density

u_trms = .false.
RMS-average of x-component of velocity

v_trms = .false.
RMS-average of y-component of velocity

w_trms = .false.
RMS-average of z-component of velocity

p_trms = .false.
RMS-average of pressure

lambda1 = .false.
Adjoint Lagrange multiplier for equation 1 (when running the adjoint, the primitive variables are turned off, and this is turned on)

lambda2 = .false.
Adjoint Lagrange multiplier for equation 2 (when running the adjoint, the primitive variables are turned off, and this is turned on)

lambda3 = .false.
Adjoint Lagrange multiplier for equation 3 (when running the adjoint, the primitive variables are turned off, and this is turned on)

lambda4 = .false.
Adjoint Lagrange multiplier for equation 4 (when running the adjoint, the primitive variables are turned off, and this is turned on)

lambda5 = .false.
Adjoint Lagrange multiplier for equation 5 (when running the adjoint, the primitive variables are turned off, and this is turned on)

lambda6 = .false.
Adjoint Lagrange multiplier for equation 6 (when running the adjoint, the primitive variables are turned off, and this is turned on)
\texttt{lambda7 = .false.}
Adjoint Lagrange multiplier for equation 7 (when running the adjoint, the primitive variables are turned off, and this is turned on)
\texttt{htot = .false.}
Total enthalpy per unit volume
\texttt{ttot = .false.}
Total temperature
\texttt{ptot = .false.}
Total pressure
\texttt{etot = .false.}
Total energy per unit volume
\texttt{processor\_id = .false.}
Processor on which a node resides
\texttt{turb\_ke = .false.}
Turbulence kinetic energy
\texttt{turb\_diss = .false.}
Turbulence dissipation rate
\texttt{mu\_t = .false.}
Turbulent eddy viscosity
\texttt{turb1 = .false.}
Turbulence variable 1 (model dependent)
\texttt{turb2 = .false.}
Turbulence variable 2 (model dependent)
\texttt{turb3 = .false.}
Turbulence variable 3 (model dependent)
\texttt{turb4 = .false.}
Turbulence variable 4 (model dependent)
\texttt{turb5 = .false.}
Turbulence variable 5 (model dependent)
\texttt{turb6 = .false.}
Turbulence variable 6 (model dependent)
turb7 = false.
Turbulence variable 7 (model dependent)
turres1 = false.
Residual of 1st turbulence equation
turres2 = false.
Residual of 2nd turbulence equation
turres3 = false.
Residual of 3rd turbulence equation
turres4 = false.
Residual of 4th turbulence equation
turres5 = false.
Residual of 5th turbulence equation
turres6 = false.
Residual of 6th turbulence equation
turres7 = false.
Residual of 7th turbulence equation
slen = false.
Length to the nearest solid wall boundary
iflagslen = false.
Turbulence model distance function closest boundary entity. (a negative sign indicates the node has been prescribed as laminar)
hrles_blend = false.
HRLES blending function
reconstruction_limiter_phi1 = false.
ϕ for the node-based reconstruction limiters (equation 1)
reconstruction_limiter_phi2 = false.
ϕ for the node-based reconstruction limiters (equation 2)
reconstruction_limiter_phi3 = false.
ϕ for the node-based reconstruction limiters (equation 3)
reconstruction_limiter_phi4 = false.
ϕ for the node-based reconstruction limiters (equation 4)
reconstruction_limiter_phi5 = .false.
φ for the node-based reconstruction limiters (equation 5)
tt = .false.
Translational temperature only for eqn_type = 'generic'
tv = .false.
Vibrational temperature only for eqn_type = 'generic'
sonic = .false.
Frozen speed of sound only for eqn_type = 'generic'
mixture_mol_weight = .false.
Mixture molecular weight only for eqn_type = 'generic'
mixture_density = .false.
Mixture density only for eqn_type = 'generic'
ev = .false.
Vibrational energy only for eqn_type = 'generic'
rho_i(1:n_species) = .false.
Species concentration only for eqn_type = 'generic'
mu = .false.
Total viscosity
id.l2g = .false.
Local-to-global node map
divided_residuals = .false.
adds a .vol suffix to the residual output variable names and divide by volume
This namelist controls the boundary variable output. Output frequency is controlled by boundary_animation_freq in the &global namelist. By default, the output of solution data for all solid surfaces in 3D and on one y-constant symmetry plane in 2D is included unless boundary_list is specified.

Each time boundary data output is triggered, all output boundaries are written to one file with the following naming convention:

```
[project_rootname]_tec_boundary_timestep[T](.dat|.plt) if N > 0
[project_rootname]_tec_boundary(.dat|.plt) if N < 0
```

where T is the iteration number. The file extension is .dat for ASCII Tecplot™ format and .plt for binary Tecplot™ format. Within the files, each boundary is written as a separate zone. The zones are identified with the title “time 0.0000000E+00 boundary 5” where the time value is the integer iteration number for steady-state cases, and the current (non-dimensional) time for time-dependent cases.

By default, output is in the inertial reference frame. For moving body problems, the &observer_motion namelist can be used to change the visualization reference system to a body reference system or a reference system with arbitrary motion.

A request to output an undefined variable will be overruled, i.e., turb1 will be forced to .false regardless of user input when there is no turbulence model in the simulation.

```
&boundary_output_variables
  number_of_boundaries = 0
  boundary_list =''
  x = .true.
  y = .true.
  z = .true.
  primitive_variables = .true.
  rho = .false.
  u = .false.
  v = .false.
  w = .false.
  p = .false.
  entropy = .false.
  mach = .false.
  temperature = .false.
  iblank = .false.
  imesh = .false.
  vort_mag = .false.
  vort_x = .false.
  vort_y = .false.
```

154
vort_z = .false.
qu_criterion = .false.
div_vel = .false.
turbulent_fluctuations = .false.
uu_prime = .false.
vv_prime = .false.
ww_prime = .false.
uw_prime = .false.
vw_prime = .false.
cp = .false.
dp_pinf = .false.
volume = .false.
residuals = .false.
res1 = .false.
res2 = .false.
res3 = .false.
res4 = .false.
res5 = .false.
res_gcl = .false.
primitive_tavg = .false.
rho_tavg = .false.
u_tavg = .false.
v_tavg = .false.
w_tavg = .false.
p_tavg = .false.
primitive_trms = .false.
rho_trms = .false.
u_trms = .false.
v_trms = .false.
w_trms = .false.
p_trms = .false.
lambda1 = .false.
lambda2 = .false.
lambda3 = .false.
lambda4 = .false.
lambda5 = .false.
lambda6 = .false.
lambda7 = .false.
htot = .false.
ttot = .false.
ptot = .false.
etot = .false.
processor_id = .false.
turb_ke = .false.
turb_diss = .false.
mu_t = .false.
turb1 = .false.
turb2 = .false.
turb3 = .false.
turb4 = .false.
turb5 = .false.
turb6 = .false.
turb7 = .false.
turres1 = .false.
turres2 = .false.
turres3 = .false.
turres4 = .false.
turres5 = .false.
turres6 = .false.
turres7 = .false.
slen = .false.
tt = .false.
tv = .false.
sonic = .false.
mixture_mol_weight = .false.
mixture_density = .false.
ev = .false.
rho_i(1:n_species) = .false.
u = .false.
mu = .false.
id_l2g = .false.
vort_x_rms = .false.
vort_y_rms = .false.
vort_z_rms = .false.
vort_mag_rms = .false.
vort_mag_tavg = .false.
yplus = .false.
mu_t_tavg = .false.
cmu_star = .false.
bird_breakdown = .false.
recovery_temperature = .false.
turb_mach = .false.
turbindex = .false.
average_velocity = .false.
uavg = .false.
vavg = .false.
wavg = .false.
cf_x = .false.
cf_y = .false.
cf_z = .false.
skinfr = .false.
cq = .false.
shear_x = .false.
shear_y = .false.
shear_z = .false.
heating = .false.
mdot = .false.
utau_wf = .false.
phi_wf = .false.
rey_turb = .false.
k_wallfunction_bc = .false.
omega_wallfunction_bc = .false.

number_of_boundaries = 0
Number of boundary patches given in boundary_list (if -1 is given, this
number is computed from boundary_list)

boundary_list = '',
List of boundary patch numbers. Commas and dashes can be used to
specify ranges, i.e., '1,2,5-7'. If nothing is specified, then all but flow-
through boundaries are output for 3D or a single symmetry plane in
2D.

x = .true.
X-coordinate

y = .true.
Y-coordinate

z = .true.
Z-coordinate

primitive_variables = .true.
Output primitive variables: rho, u, v, w, and p

rho = .false.
Density

u = .false.
X-component of velocity

v = .false.
Y-component of velocity
\texttt{w = .false.}

Z-component of velocity

\texttt{p = .false.}

Pressure

\texttt{entropy = .false.}

Entropy

\texttt{mach = .false.}

Mach number

\texttt{temperature = .false.}

Temperature

\texttt{iblank = .false.}

I-blanking variable (default becomes \texttt{true.} for overset mesh cases)

\texttt{imesh = .false.}

For overset mesh systems, index of associated component grid where 0
indicates background grid

\texttt{vort\_mag = .false.}

Vorticity magnitude

\texttt{vort\_x = .false.}

X-component of vorticity

\texttt{vort\_y = .false.}

Y-component of vorticity

\texttt{vort\_z = .false.}

Z-component of vorticity

\texttt{q\_criterion = .false.}

Q Criterion, the second invariant of $\nabla V$

\texttt{div\_vel = .false.}

Velocity divergence

\texttt{turbulent\_fluctuations = .false.}

Activate all the following \texttt{XYprime} turbulent shear stresses normalized by
$u^2_{ref}$; the definition of these variables depends on the turbulence model,
see \url{http://turbmodels.larc.nasa.gov/noteonrunning.html} for details
uuprime = .false.
Turbulence fluctuation, \( u'u' \)

vvprime = .false.
Turbulence fluctuation, \( v'v' \)

wwprime = .false.
Turbulence fluctuation, \( w'w' \)

uvprime = .false.
Turbulence fluctuation, \( u'v' \)

uwprime = .false.
Turbulence fluctuation, \( u'w' \)

vwprime = .false.
Turbulence fluctuation, \( v'w' \)

cp = .false.
Pressure coefficient

dp_pinf = .false.
Normalized delta pressure \( (p - p_\infty)/p_\infty \)

volume = .false.
Dual-cell volume size

residuals = .false.
Activate all resN variables.

res1 = .false.
Residual of equation 1, density

res2 = .false.
Residual of equation 2, \( x \)-momentum

res3 = .false.
Residual of equation 3, \( y \)-momentum

res4 = .false.
Residual of equation 4, \( z \)-momentum

res5 = .false.
Residual of equation 5, energy
res_gcl = .false.
For moving meshes, residual of grid conservation law

primitive_tavg = .false.
Output time-averaged primitives (requires &time_avg_params namelist):
rho_tavg, u_tavg, v_tavg, w_tavg, and p_tavg

rho_tavg = .false.
Time-averaged density

u_tavg = .false.
Time-averaged x-component of velocity

v_tavg = .false.
Time-averaged y-component of velocity

w_tavg = .false.
Time-averaged z-component of velocity

p_tavg = .false.
Time-averaged pressure

primitive_trms = .false.
Output root mean squared primitives (requires &time_avg_params namelist):
rho_trms, u_trms, v_trms, w_trms, and p_trms

rho_trms = .false.
RMS-average of density

u_trms = .false.
RMS-average of x-component of velocity

v_trms = .false.
RMS-average of y-component of velocity

w_trms = .false.
RMS-average of z-component of velocity

p_trms = .false.
RMS-average of pressure

lambda1 = .false.
Adjoint Lagrange multiplier for equation 1 (when running the adjoint,
the primitive variables are turned off, and this is turned on)
\( \lambda_2 = \text{false} \)
Adjoint Lagrange multiplier for equation 2 (when running the adjoint, the primitive variables are turned off, and this is turned on)

\( \lambda_3 = \text{false} \)
Adjoint Lagrange multiplier for equation 3 (when running the adjoint, the primitive variables are turned off, and this is turned on)

\( \lambda_4 = \text{false} \)
Adjoint Lagrange multiplier for equation 4 (when running the adjoint, the primitive variables are turned off, and this is turned on)

\( \lambda_5 = \text{false} \)
Adjoint Lagrange multiplier for equation 5 (when running the adjoint, the primitive variables are turned off, and this is turned on)

\( \lambda_6 = \text{false} \)
Adjoint Lagrange multiplier for equation 6 (when running the adjoint, the primitive variables are turned off, and this is turned on)

\( \lambda_7 = \text{false} \)
Adjoint Lagrange multiplier for equation 7 (when running the adjoint, the primitive variables are turned off, and this is turned on)

\( h_{\text{tot}} = \text{false} \)
Total enthalpy per unit volume

\( t_{\text{tot}} = \text{false} \)
Total temperature

\( p_{\text{tot}} = \text{false} \)
Total pressure

\( e_{\text{tot}} = \text{false} \)
Total energy per unit volume

\( \text{processor id} = \text{false} \)
Processor on which a node resides

\( \text{turb ke} = \text{false} \)
Turbulence kinetic energy

\( \text{turb diss} = \text{false} \)
Turbulence dissipation rate
\texttt{mu_t = .false.}
Turbulent eddy viscosity
\texttt{turb1 = .false.}
Turbulence variable 1 (model dependent)
\texttt{turb2 = .false.}
Turbulence variable 2 (model dependent)
\texttt{turb3 = .false.}
Turbulence variable 3 (model dependent)
\texttt{turb4 = .false.}
Turbulence variable 4 (model dependent)
\texttt{turb5 = .false.}
Turbulence variable 5 (model dependent)
\texttt{turb6 = .false.}
Turbulence variable 6 (model dependent)
\texttt{turb7 = .false.}
Turbulence variable 7 (model dependent)
\texttt{turres1 = .false.}
Residual of 1st turbulence equation
\texttt{turres2 = .false.}
Residual of 2nd turbulence equation
\texttt{turres3 = .false.}
Residual of 3rd turbulence equation
\texttt{turres4 = .false.}
Residual of 4th turbulence equation
\texttt{turres5 = .false.}
Residual of 5th turbulence equation
\texttt{turres6 = .false.}
Residual of 6th turbulence equation
\texttt{turres7 = .false.}
Residual of 7th turbulence equation
slen = .false.
Length to the nearest solid wall boundary

tt = .false.
Translational temperature, only for eqn_type = 'generic'

tv = .false.
Vibrational temperature, only for eqn_type = 'generic'

sonic = .false.
Frozen speed of sound, only for eqn_type = 'generic'

mixture_mol_weight = .false.
Mixture molecular weight, only for eqn_type = 'generic'

mixture_density = .false.
Mixture density, only for eqn_type = 'generic'

ev = .false.
Vibrational energy, only for eqn_type = 'generic'

rho_i(1:n_species) = .false.
Species concentration, only for eqn_type = 'generic'

mu = .false.
Total viscosity

id_l2g = .false.
Local-to-global node map

vort_x_rms = .false.
RMS-average of x-component of vorticity

vort_y_rms = .false.
RMS-average of y-component of vorticity

vort_z_rms = .false.
RMS-average of z-component of vorticity

vort_mag_rms = .false.
RMS-average of vorticity magnitude

vort_mag_tavg = .false.
Time-average of vorticity magnitude
yplus = .false.
Dimensionless wall distance, y^+

mu_t_tavg = .false.
Time-average turbulent eddy viscosity

cmu_star = .false.
\(k - \epsilon\) model turbulent length scale parameter

bird_breakdown = .false.
Bird continuum breakdown parameter

recovery_temperature = .false.
Recovery temperature

turb_mach = .false.
Turbulent Mach number

turbindex = .false.
Turbulent index

average_velocity = .false.
Turns on uavg, vavg, wavg

uavg = .false.
\(X\)-component of average velocity near a wall (used to plot surface streamlines)

vavg = .false.
\(Y\)-component of average velocity near a wall (used to plot surface streamlines)

wavg = .false.
\(Z\)-component of average velocity near a wall (used to plot surface streamlines)

cf_x = .false.
\(X\)-component of skin friction

cf_y = .false.
\(Y\)-component of skin friction

cf_z = .false.
\(Z\)-component of skin friction
skinfr = .false.
Skin friction magnitude with sign determined by the inner product of the skin friction vector and the freestream velocity vector

cq = .false.
Temperature gradient at the wall

shear_x = .false.
X-component of shear on the boundary, in MKS units

shear_y = .false.
Y-component of shear on the boundary, in MKS units

shear_z = .false.
Z-component of shear on the boundary, in MKS units

heating = .false.
Heating on the boundary in Watts per centimeter squared (for eqn_type = 'compressible', make sure the grid is in meters)

mdot = .false.
Dimensionless blowing rate non-dimensionalized by \( \rho_\infty V_\infty \)

utau_wf = .false.
Friction velocity calculated from a wall function model.

phi_wf = .false.
Pressure gradient term from a wall function model.

rey_turb = .false.
Turbulence Reynolds number.

k_wallfunction_bc = .false.
Turbulent kinetic energy wall function boundary condition.

omega_wallfunction_bc = .false.
Omega wall function boundary condition.
This namelist controls output of variables from user defined regions of the computational domain. To use sampling, the &sampling_parameters namelist must be used to define the sampling geometries and the sampling_frequency set for each geometry.

The resulting sampling data files will have the following naming convention:

\[
\begin{align*}
\text{[project_rootname]_tec_sampling_geom[G]_timestep[T](.dat|.plt)} & \quad \text{if } N > 0 \\
\text{[project_rootname]_tec_sampling_geom[G](.dat|.plt)} & \quad \text{if } N < 0
\end{align*}
\]

where G = 1, 2, ... number_of_geometries, and T is the iteration number. The file extension is .dat for ASCII Tecplot™ format and .plt for binary Tecplot™ format. A global image of the sampling surface is output with the zone title “time 0.0000000E+00 geom 3” where the time value is the integer iteration number for steady-state cases, and the current (nondimensional) time for time-dependent cases.

A request to output an undefined variable will be overruled, i.e., turb1 will be forced to false regardless of user input when there is no turbulence model in the simulation.

\[
\begin{align*}
&\text{&sampling_output_variables} \\
&x = \ .true. \\
y = \ .true. \\
z = \ .true. \\
&\text{primitive_variables} = \ .true. \\
rho = \ .false. \\
u = \ .false. \\
v = \ .false. \\
w = \ .false. \\
p = \ .false. \\
&\text{entropy} = \ .false. \\
mach = \ .false. \\
temperature = \ .false. \\
ilblank = \ .false. \\
imesh = \ .false. \\
vort_mag = \ .false. \\
vort_x = \ .false. \\
vort_y = \ .false. \\
vort_z = \ .false. \\
&q\_criterion = \ .false. \\
&\text{div}_\text{vel} = \ .false. \\
turbulent\_fluctuations = \ .false. \\
uuprime = \ .false. \\
vvp_rime = \ .false. \\
wwprime = \ .false.
\end{align*}
\]
uvprime = .false.
uwprime = .false.
vwprime = .false.
cp = .false.
dp_pinf = .false.
volume = .false.
residuals = .false.
res1 = .false.
res2 = .false.
res3 = .false.
res4 = .false.
res5 = .false.
res_gcl = .false.
rho_tavg = .false.
primitive_tavg = .false.
u_tavg = .false.
v_tavg = .false.
w_tavg = .false.
p_tavg = .false.
mu_t_tavg = .false.
vort_mag_tavg = .false.
vort_x_tavg = .false.
vort_y_tavg = .false.
vort_z_tavg = .false.
primitive_trms = .false.
rho_trms = .false.
u_trms = .false.
v_trms = .false.
w_trms = .false.
p_trms = .false.
lambda1 = .false.
lambda2 = .false.
lambda3 = .false.
lambda4 = .false.
lambda5 = .false.
lambda6 = .false.
lambda7 = .false.
htot = .false.
ttot = .false.
ptot = .false.
etot = .false.
processor_id = .false.
turb_ke = .false.
turb_diss = .false.
rey_turb = .false.

data


\[
\begin{align*}
\text{mu_t} & = .false. \\
\text{turb1} & = .false. \\
\text{turb2} & = .false. \\
\text{turb3} & = .false. \\
\text{turb4} & = .false. \\
\text{turb5} & = .false. \\
\text{turb6} & = .false. \\
\text{turb7} & = .false. \\
\text{turres1} & = .false. \\
\text{turres2} & = .false. \\
\text{turres3} & = .false. \\
\text{turres4} & = .false. \\
\text{turres5} & = .false. \\
\text{turres6} & = .false. \\
\text{turres7} & = .false. \\
\text{slen} & = .false. \\
\text{iflagslen} & = .false. \\
\text{hrles_blend} & = .false. \\
\text{vort_x_rms} & = .false. \\
\text{vort_y_rms} & = .false. \\
\text{vort_z_rms} & = .false. \\
\text{vort_mag_rms} & = .false. \\
\text{yplus} & = .false. \\
\text{cmu_star} & = .false. \\
\text{mu_t_ratio} & = .false. \\
\text{iib} & = .false. \\
\text{iiib} & = .false. \\
\text{uplus} & = .false. \\
\text{kplus} & = .false. \\
\text{yplusretau} & = .false. \\
\text{t11plus} & = .false. \\
\text{t12plus} & = .false. \\
\text{t13plus} & = .false. \\
\text{t22plus} & = .false. \\
\text{t23plus} & = .false. \\
\text{t33plus} & = .false. \\
\text{bird_breakdown} & = .false. \\
\text{vgradrho} & = .false. \\
\text{f_r1} & = .false. \\
\text{xi_k} & = .false. \\
\text{reconstruction_limiter_phi1} & = .false. \\
\text{reconstruction_limiter_phi2} & = .false. \\
\text{reconstruction_limiter_phi3} & = .false. \\
\text{reconstruction_limiter_phi4} & = .false. \\
\text{reconstruction_limiter_phi5} & = .false. \\
\end{align*}
\]

168
/

x = .true.
X-coordinate
y = .true.
Y-coordinate
z = .true.
Z-coordinate

primitive_variables = .true.
Output primitive variables: rho, u, v, w, and p

rho = .false.
Density
u = .false.
X-component of velocity
v = .false.
Y-component of velocity
w = .false.
Z-component of velocity
p = .false.
Pressure

entropy = .false.
Entropy

mach = .false.
Mach number

temperature = .false.
Temperature

iblank = .false.
I-blanking variable (default becomes .true. for overset mesh cases)

imesh = .false.
For overset mesh systems, index of associated component grid where 0 indicates background grid
vort_mag = .false.
Vorticity magnitude

vort_x = .false.
X-component of vorticity

vort_y = .false.
Y-component of vorticity

vort_z = .false.
Z-component of vorticity

q_criterion = .false.
Q Criterion, the second invariant of \( \nabla V \)

div_vel = .false.
Velocity divergence

turbulent_fluctuations = .false.
Activate all the following XYprime turbulent shear stresses normalized by 
\( u_{ref}^2 \); the definition of these variables depends on the turbulence model, 
see http://turbmodels.larc.nasa.gov/noteonrunning.html for details

uuprime = .false.
Turbulence fluctuation, \( u'u' \)

vvprime = .false.
Turbulence fluctuation, \( v'v' \)

wwprime = .false.
Turbulence fluctuation, \( w'w' \)

uvprime = .false.
Turbulence fluctuation, \( u'v' \)

uwprime = .false.
Turbulence fluctuation, \( u'w' \)

vwprime = .false.
Turbulence fluctuation, \( v'w' \)

cp = .false.
Pressure coefficient
\[
\text{dp}\_\text{pinf} = \text{false.}
\]
Normalized delta pressure \( (p - p_\infty)/p_\infty \)
\[
\text{volume} = \text{false.}
\]
Dual-cell volume size
\[
\text{residuals} = \text{false.}
\]
Activate all \( \text{resN} \) variables
\[
\text{res1} = \text{false.}
\]
Residual of equation 1, density
\[
\text{res2} = \text{false.}
\]
Residual of equation 2, x-momentum
\[
\text{res3} = \text{false.}
\]
Residual of equation 3, y-momentum
\[
\text{res4} = \text{false.}
\]
Residual of equation 4, z-momentum
\[
\text{res5} = \text{false.}
\]
Residual of equation 5, energy
\[
\text{res\_gcl} = \text{false.}
\]
For moving meshes, residual of grid conservation law
\[
\text{rho\_tavg} = \text{false.}
\]
Time-averaged density
\[
\text{primitive\_tavg} = \text{false.}
\]
Output time-averaged primitives (requires \&time\_avg\_params namelist):
\[
\text{rho\_tavg}, \text{u\_tavg}, \text{v\_tavg}, \text{w\_tavg}, \text{and p\_tavg}
\]
\[
\text{u\_tavg} = \text{false.}
\]
Time-averaged x-component of velocity
\[
\text{v\_tavg} = \text{false.}
\]
Time-averaged y-component of velocity
\[
\text{w\_tavg} = \text{false.}
\]
Time-averaged z-component of velocity
\[
\text{p\_tavg} = \text{false.}
\]
Time-averaged pressure
mu_t_tavg = .false.
Time-averaged turbulent eddy viscosity
vort_mag_tavg = .false.
Time-averaged vorticity magnitude
vort_x_tavg = .false.
Time-averaged x-component vorticity
vort_y_tavg = .false.
Time-averaged y-component vorticity
vort_z_tavg = .false.
Time-averaged z-component vorticity
primitive_trms = .false.
Output root mean squared primitives (requires &time_avg_params namelist):
rho_trms, u_trms, v_trms, w_trms, and p_trms
rho_trms = .false.
RMS-average of density
u_trms = .false.
RMS-average of x-component of velocity
v_trms = .false.
RMS-average of y-component of velocity
w_trms = .false.
RMS-average of z-component of velocity
p_trms = .false.
RMS-average of pressure
lambda1 = .false.
Adjoint Lagrange multiplier for equation 1 (when running the adjoint, the primitive variables are turned off, and this is turned on)
lambda2 = .false.
Adjoint Lagrange multiplier for equation 2 (when running the adjoint, the primitive variables are turned off, and this is turned on)
lambda3 = .false.
Adjoint Lagrange multiplier for equation 3 (when running the adjoint, the primitive variables are turned off, and this is turned on)
\[ \lambda_4 = \text{false}. \]
Adjoint Lagrange multiplier for equation 4 (when running the adjoint, the primitive variables are turned off, and this is turned on)

\[ \lambda_5 = \text{false}. \]
Adjoint Lagrange multiplier for equation 5 (when running the adjoint, the primitive variables are turned off, and this is turned on)

\[ \lambda_6 = \text{false}. \]
Adjoint Lagrange multiplier for equation 6 (when running the adjoint, the primitive variables are turned off, and this is turned on)

\[ \lambda_7 = \text{false}. \]
Adjoint Lagrange multiplier for equation 7 (when running the adjoint, the primitive variables are turned off, and this is turned on)

\[ h_{\text{tot}} = \text{false}. \]
Total enthalpy per unit volume

\[ t_{\text{tot}} = \text{false}. \]
Total temperature

\[ p_{\text{tot}} = \text{false}. \]
Total pressure

\[ e_{\text{tot}} = \text{false}. \]
Total energy per unit volume

\[ \text{processor}_\text{id} = \text{false}. \]
Processor on which a node resides

\[ \text{turb}_\text{ke} = \text{false}. \]
Turbulence kinetic energy

\[ \text{turb}_\text{diss} = \text{false}. \]
Turbulence dissipation rate

\[ \text{rey}_\text{turb} = \text{false}. \]
Turbulence Reynolds number

\[ \mu_t = \text{false}. \]
Turbulent eddy viscosity

\[ \text{turb}_1 = \text{false}. \]
Turbulence variable 1 (model dependent)
turb2 = true
Turbulence variable 2 (model dependent)

turb3 = true
Turbulence variable 3 (model dependent)

turb4 = true
Turbulence variable 4 (model dependent)

turb5 = true
Turbulence variable 5 (model dependent)

turb6 = true
Turbulence variable 6 (model dependent)

turb7 = true
Turbulence variable 7 (model dependent)

turres1 = true
Residual of 1st turbulence equation

turres2 = true
Residual of 2nd turbulence equation

turres3 = true
Residual of 3rd turbulence equation

turres4 = true
Residual of 4th turbulence equation

turres5 = true
Residual of 5th turbulence equation

turres6 = true
Residual of 6th turbulence equation

turres7 = true
Residual of 7th turbulence equation

slen = true
Length to the nearest solid wall boundary

iflagslen = true
Turbulence model distance function closest boundary entity. (a negative
sign indicates the node has been prescribed as laminar)
hrles_blend = .false.
HRLES blending function
vort_x_rms = .false.
RMS-average of $x$-component of vorticity
vort_y_rms = .false.
RMS-average of $y$-component of vorticity
vort_z_rms = .false.
RMS-average of $z$-component of vorticity
vort_mag_rms = .false.
RMS-average of vorticity magnitude
yplus = .false.
Dimensionless wall distance, $y^+$
cmu_star = .false.
$k - \epsilon$ model turbulent length scale parameter
mu_t_ratio = .false.
Ratio of turbulent eddy viscosity to laminar (bulk) viscosity
iib = .false.
$-\text{trace}(B_{ij} * B_{ij})/2$
iiib = .false.
$\text{trace}(B_{ij} * B_{ij} * B_{ij})/3$
uplus = .false.
Dimensionless velocity, $u^+$
kplus = .false.
$k/u_f$
yplusretau = .false.
$u^+ / \nu_{eq}$
t11plus = .false.
$\tau_{11}^+$
t12plus = .false.
$\tau_{12}^+$
t13plus = .false.
\tau_{13}^+

t22plus = .false.
\tau_{22}^+

t23plus = .false.
\tau_{23}^+

t33plus = .false.
\tau_{33}^+

bird_breakdown = .false.
Bird continuum breakdown parameter

vgradrho = .false.
\left[u, v, w\right] \cdot \nabla \rho

fr1 = .false.
Curvature correction model function

xi_k = .false.
Cross-diffusion term for Wilcox $k\omega$ 1998

reconstruction_limiter_phi1 = .false.
\phi for the node-based reconstruction limiters (equation 1)

reconstruction_limiter_phi2 = .false.
\phi for the node-based reconstruction limiters (equation 2)

reconstruction_limiter_phi3 = .false.
\phi for the node-based reconstruction limiters (equation 3)

reconstruction_limiter_phi4 = .false.
\phi for the node-based reconstruction limiters (equation 4)

reconstruction_limiter_phi5 = .false.
\phi for the node-based reconstruction limiters (equation 5)
This namelist specifies the types and frequency of sampling data to be exported for visualization. The output variables themselves are specified in the \&sampling_output_variables namelist. The last dimension of each array references the geometry index, which is one to number_of_geometries.

\begin{verbatim}
&sampling_parameters
   number_of_geometries = 0
   sampling_frequency(:) = 0
   label(:) = '
   type_of_geometry(:) = 'none'
   crinkle = .false.
   nodal = .false.
   plot(:) = 'tecplot'
   patch_list_count(:) = 0
   patch_list(:) = '
   type_of_data(:) = 'volume'
   move_with_body(:) = '
   boundary_list = '
   default_boundary = .true.
   plane_center(1:3,:) = 0.0
   plane_normal(1:3,:) = 0.0
   box_lower_corner(1:3,:) = 0.0
   box_upper_corner(1:3,:) = 0.0
   sphere_center(1:3,:) = 0.0
   sphere_radius(:) = 0.0
   circle_center(1:3,:) = 0.0
   circle_normal(1:3,:) = 0.0
   circle_radius(:) = 0.0
   cylinder_face1(1:3,:) = 0.0
   cylinder_face2(1:3,:) = 0.0
   cylinder_radius(:) = 0.0
   cone_face1(1:3,:) = 0.0
   cone_face2(1:3,:) = 0.0
   cone_radius1(:) = 0.0
   cone_radius2(:) = 0.0
   corner1(1:3,:) = 0.0
   corner2(1:3,:) = 0.0
   corner3(1:3,:) = 0.0
   corner4(1:3,:) = 0.0
   number_of_points(:) = 0
   points(1:3,:,:) = 0.0
   number_of_lines = 0
   p1_line(1:3,:) = 0.0
   p2_line(1:3,:) = 0.0
\end{verbatim}
schlieren_aspect = ''
window_height(:) = 0.0
window_width(:) = 0.0
window_center(1:3,:) = 0.0
number_of_rows(:) = 0
number_of_columns(:) = 0
model_center(1:3,:) = 0.0
plot_lines(:) = .false.
make_shadow = .false.
blanking_list_count(:) = 0
blanking_list(:) = ''
isosurf_variable(:) = 'p'
isosurf_value(:) = 0.0
isosurf_box(:) = .false.
x_range_lower(:) = -1.0
x_range_upper(:) = 1.0
y_range_lower(:) = -1.0
y_range_upper(:) = 1.0
z_range_lower(:) = -1.0
z_range_upper(:) = 1.0
isosurf_dist_threshold(:) = 0.0
variable_list(:) = ''
snap_output_xyz = .true.
dist_tolerance = 1.0e-3
fwh_formatted = .false.
append_history(:) = .false.
asynchronous_fwh = .false.
reference_length = 0.0
/

number_of_geometries = 0
This is the total number of sampling geometries.
sampling_frequency(:) = 0
This specifies the iteration interval at which sampling is performed. The special value of -1 means to only perform sampling at the end of a successful run.
label(:) = ''
This customizes the filename of sampling output. When it is blank, the file will be [project_rootname].tec_sampling_geomN.(dat,plt) where N is the sampling geometry number, .dat is ASCII format, and .plt is binary format.
type_of_geometry(:) = 'none'
This is the type of sampling geometry,
'streamsurface' is a stream surface, requires number_of_points and points.

'boundary_points' for boundary point sampling, requires number_of_points and points, modified by snap_output_xyz and dist_tolerance.

'volume_points' for point sampling in the domain, requires number_of_points and points.

'schlieren' is a schlieren image via an integral of the refractive index field, requires number_of_rows, number_of_columns, window_height, window_width, window_center, and schlieren_aspect. It is controlled by make_shadow and plot_lines.

'isosurface' is an isosurface that requires isosurf_variable and isosurf_value. It is controlled by *range_lower and *range_upper.

'box' samples the surface of a box. It requires box_lower_corner and box_upper_corner.

'sphere' samples a spherical surface. It requires sphere_center and sphere_radius.

'cylinder' samples a cylindrical surface. It requires cylinder_face1, cylinder_face2, and cylinder_radius.

'cone' samples a conic surface. It requires cone_face1, cone_face2, cone_radius1, and cone_radius2.

'plane' samples a plane. It requires plane_center and plane_normal.

'quad' samples a quadrilateral. It requires corner1, corner2, corner3, corner4, and window_normal.

'circle' samples a circle. It requires circle_center, circle_normal, and circle_radius.

'line' is line sampling, which requires number_of_lines, p1_line, and p2_line.

crinkle = .false.
This snaps the sampling surface to nearest grid faces instead of using linear interpolation.

nodal = .false.
This uses the nearest nodal values instead of interpolating.

plot(:) = 'tecplot'
This is the format of sampling output,

'tecplot' Tecplot™ format.
'fwh' format for Ffowcs Williams-Hawkings analysis.

'serial_history' custom low-overhead point sampling format where all locations listed once at the top and then just the requested values per sampling_frequency.

patch_list_count(:) = 0
This is the number of patches in patch_list.

patch_list(:) = ''
A string list of patch face IDs to limit boundary survey to a subset of the boundary faces. Commas and dashes can be used to specify ranges, i.e., '1,2,5-7'.

type_of_data(:) = 'volume'
The source of data for extracting the requested sampling variables for each type_of_geometry.

'volume' extract data from the computational volume.

'boundary' extract data from a boundary.

move_with_body(:) = ''
Move the sampling geometry with the body if body is in motion. Use the fixed inertial reference frame when blank.

boundary_list = ''
List of patches to include when sampling boundaries; Commas and dashes can be used to specify ranges, i.e., '1,2,5-7'.

default_boundary = .true.
Use FUN3D default solid-wall-only boundary patches when sampling boundary points, i.e., ignore symmetry, slip, and flow-through boundaries.

plane_center(1:3,:) = 0.0
This is a point on a requested sampling 'plane'; it fixes the location.

plane_normal(1:3,:) = 0.0
This is a normal vector of sampling 'plane'; it fixes the orientation.

box_lower_corner(1:3,:) = 0.0
This is the coordinate of the lower corner of a 'box'.

box_upper_corner(1:3,:) = 0.0
This is the coordinate of the upper corner of a 'box'.

180
sphere_center(1:3,:) = 0.0
This is the coordinate of 'sphere' center; it fixes the location.

sphere_radius(:) = 0.0
This is the radius for 'sphere'; it fixes the size.

circle_center(1:3,:) = 0.0
This is the coordinate of center of a 'circle'; it fixes the location.

circle_normal(1:3,:) = 0.0
This is the normal vector for a 'circle'; it fixes the orientation.

circle_radius(:) = 0.0
This is the radius for a 'circle'; it fixes the size.

cylinder_face1(1:3,:) = 0.0
This is the coordinate for the center of the first face of a 'cylinder'.

cylinder_face2(1:3,:) = 0.0
This is the coordinate for center of the second face of a 'cylinder'.

cylinder_radius(:) = 0.0
This is the radius of a 'cylinder'.

cone_face1(1:3,:) = 0.0
This is the coordinate for center of the first face of a 'cylinder'.

cone_face2(1:3,:) = 0.0
This is the coordinate for center of the second face of a 'cylinder'.

cone_radius1(:) = 0.0
This is the radius of the first face of a 'cone'.

cone_radius2(:) = 0.0
This is the radius of the second face of a 'cone'.

corner1(1:3,:) = 0.0
This is the coordinate of the first corner of a 'quad'; the corners proceed clockwise.

corner2(1:3,:) = 0.0
The coordinate of the second corner of a 'quad'.

181
corner3(1:3,:) = 0.0
The coordinate of the third corner of a 'quad'.

corner4(1:3,:) = 0.0
The coordinate of the fourth corner of a 'quad'.

number_of_points(:) = 0
This is the number of points to be sampled by 'boundary_point' or 'volume_point'.

points(1:3,:,:) = 0.0
These are the coordinates of boundary_point and volume_point sampling. The first index is the Cartesian direction, the second index is the geometry, and the last index is the point in this geometry.

number_of_lines = 0
This is the number of lines in 'line' sampling.

p1_line(1:3,:) = 0.0
This is the first end point of a line in line sampling.

p2_line(1:3,:) = 0.0
This is the second end point of a line in line sampling.

schlieren_aspect = ''
This is the Cartesian direction for 'schlieren' view,
'y' Schlieren viewing along y axis.
'z' Schlieren viewing along z axis.
'y1' Schlieren viewing along y axis.
'z1' Schlieren viewing along z axis.
'' Schlieren viewing along window_normal.

window_height(:) = 0.0
This is the window height for 'schlieren'.

window_width(:) = 0.0
This is the window width for 'schlieren'.

window_center(1:3,:) = 0.0
This is the window center for 'schlieren'.

182
number_of_rows(:) = 0
This is the vertical number of pixels in the 'schlieren' window.

number_of_columns(:) = 0
This is the horizontal number of pixels in the 'schlieren' window.

model_center(1:3,:) = 0.0
This is the model center for 'schlieren'.

plot_lines(:) = .false.
This plots lines for 'schlieren'.

make_shadow = .false.
The boundary will cast a shadow in schlieren output.

blanking_list_count(:) = 0
This is the number of boundaries to search for 'schlieren' boundary shadow.

blanking_list(:) = ''
This is a list of boundaries to search for 'schlieren' shadow. Commas and dashes can be used to specify ranges, i.e., '1,2,5-7'.

isosurf_variable(:) = 'p'
This is the variable used to define the geometry of an 'isosurface' and isocrinkle.

'p' Pressure.
'r' rho' Density.
'u' X-component of velocity.
'v' Y-component of velocity.
'w' Z-component of velocity.
'vort_x' X-component of vorticity.
'vort_y' Y-component of vorticity.
'vort_z' Z-component of vorticity.
'vort_mag' Total magnitude of vorticity vector.
'vort_mag_avg' Average total magnitude of vorticity vector.
'vort_mag_rms' RMS total magnitude of vorticity vector.
'q_criterion' Q-criterion.
'mach' Mach number.
‘temperature’ Temperature.
‘p_tavg’ Time average pressure.
‘rho_tavg’ Time average density.
‘u_tavg’ Time average x-component of velocity.
‘v_tavg’ Time average y-component of velocity.
‘w_tavg’ Time average z-component of velocity.
‘p_trms’ RMS of pressure.
‘rho_trms’ RMS of density.
‘u_trms’ RMS of the x-component of velocity.
‘v_trms’ RMS of the y-component of velocity.
‘w_trms’ RMS of the z-component of velocity.
‘critical_d’ critical d
‘s1a’ other option
‘s1b’ other option
‘s1’ other option
‘s2’ other option
‘lambda1’ Adjoint variable for the 1st governing equation.
‘lambda2’ Adjoint variable for the 2nd governing equation.
‘lambda3’ Adjoint variable for the 3rd governing equation.
‘lambda4’ Adjoint variable for the 4th governing equation.
‘lambda5’ Adjoint variable for the 5th governing equation.
‘lambda6’ Adjoint variable for the 6th governing equation.
‘lambda7’ Adjoint variable for the 7th governing equation.
‘processor_id’ The assigned processor ID.
‘bird_breakdown’ Bird breakdown factor.

isosurf_value(:) = 0.0
This is the value of isosurf_variable(:) to create the ’isosurface’
and isocrinkle geometry.
isosurf_box(:) = .false.
This clips the sampling geometry to be inside a box sized by *range_*
within isosurf_dist_threshold.
x_range_lower(:) = -1.0
This limits isosurface or isocrinkle when isosurf_box(:) = .true.
This limits isosurface or isocrinkle when isosurf_box(:) = .true.

This limits isosurface or isocrinkle when isosurf_box(:) = .true.

This limits isosurface or isocrinkle when isosurf_box(:) = .true.

This limits isosurface or isocrinkle when isosurf_box(:) = .true.

This trims portions of an isosurface or isocrinkle that have a distance to the surface less than this threshold. It requires isosurf_box(:) = .true.

These variables augment &sampling_output_variables for this sampling object.

This snaps the requested points to the nearest surface.

This is the tolerance used when snap_output_xyz is engaged.

Write Ffowcs Williams-Hawkings in Fortran unformatted format. The default is Fortran stream (C-binary).

This option removes the step number from the filename and opens it with append.

This uses asynchronous I/O for permeable FWH output.

This is the reference length for Re, used in &sampling_output_variables.
B.4.24 &slice_data

This namelist specifies boundary slices for visualization and to obtain loads. Output frequency is controlled by slice_freq in the &global namelist, where zero for no output, -1 for output at the end of run, and a positive integer for periodic output.

This is a limited ability to take slices through boundary surfaces. For example, spanwise cuts along a wing may be extracted, and then the resulting pressure and skin friction data may be plotted at each station. Slices can only be extracted in Cartesian planes (e.g., constant y). For moving-body cases, the slices may be taken at constant coordinate positions in the body-fixed coordinate system, in which case the slices will not generally be in Cartesian planes in inertial space.

The sliced data is written to an ASCII formatted Tecplot™ file with the naming convention:

[project_rootname]_slice.dat

The variables output to this file are: x, y, z, cp, cfx, cfy, and cfz at each output time step. The slicing output variables are not customizable by the user.

Slicing occurs in the inertial frame, unless an alternate reference frame is specified. For stationary geometries, the inertial frame is the only option. For moving body cases, either the frame of one of the moving bodies or an observer frame may specified.

When slicing boundary surfaces, a file called slice.info is output that echos much of the input data. When the slicing is successful, the file will also contain information about the number of points in the slice.

Below, namelist variables are defined. See section B.4.24 for some important considerations when using this capability.

&slice_data
nslices = 1
replicate_all_bodies = .false.
slice_x(:) = .false.
slice_y(:) = .true.
slice_z(:) = .false.
slice_location(:) = 0.0
slice_increment = 0.0
xx_box_max(:) = huge(1.0)
yy_box_max(:) = huge(1.0)
zz_box_max(:) = huge(1.0)
xx_box_min(:) = -huge(1.0)
yy_box_min(:) = -huge(1.0)
zz_box_min(:) = -huge(1.0)
slice_xmc(:) = huge(1.0)
slice_ymc(:) = huge(1.0)
slice_zmc(:) = huge(1.0)
n_bndrys_to_slice(:) = 0
bndrys_to_slice(:, :) = 0
slice_frame(:) =''
slice_group(:) = 1
chord_dir(:) = 1
te_def(:) = 1
le_def(:) = 30
corner_angle(:) = 120.0
use_local_chord = .true.
tecplot_slice_output = .true.
output_sectional_forces = .true.
slice_initial_coords = .false.
custom_transform(1,1,1:4) = 1.0, 0.0, 0.0, 0.0
custom_transform(1,2,1:4) = 0.0, 1.0, 0.0, 0.0
custom_transform(1,3,1:4) = 0.0, 0.0, 1.0, 0.0
custom_transform(1,4,1:4) = 0.0, 0.0, 0.0, 1.0
output_in_slice_coords(:) = .false.
/

nslices = 1
This is the number of slices to create. If negative, then data for only one slice station need be input, along with slice_increment, and all the data specified for the first station will be applied to subsequent stations, with the exception of the slice location, which will be set using the slice increment between stations.

replicate_all_bodies = .false.
This will set similar slice stations on multiple bodies with minimal input beyond that required for slicing the first body. This is particularly useful for rotorcraft applications where multiple blades are to be sliced. This variable duplicates the input slice info for all moving bodies, with the exception of the slice_frame and the bndrys_to_slice.

slice_x(:) = .false.
This extracts the slice at \( x = \text{slice\_location} \) in the specified reference frame.

slice_y(:) = .true.
This extracts the slice at \( y = \text{slice\_location} \) in the specified reference frame.
slice_z(:) = .false.
This extracts the slice at \( z = \text{slice}_\text{location} \) in the specified reference frame.

slice_location(:) = 0.0
This is the coordinate value at which slice is taken.

slice_increment = 0.0
When nslices is negative, this is the increment in slice coordinate between consecutive slice stations.

xx_box_max(:) = huge(1.0)
This is the maximum \( x \)-coordinate used to define a bounding box to constrain the slicing to filter unwanted intersections.

yy_box_max(:) = huge(1.0)
This is the maximum \( y \)-coordinate used to define a bounding box to constrain the slicing to filter unwanted intersections.

zz_box_max(:) = huge(1.0)
This is the maximum \( z \)-coordinate used to define a bounding box to constrain the slicing to filter unwanted intersections.

xx_box_min(:) = -huge(1.0)
This is the minimum \( x \)-coordinate used to define a bounding box to constrain the slicing to filter unwanted intersections.

yy_box_min(:) = -huge(1.0)
This is the minimum \( y \)-coordinate used to define a bounding box to constrain the slicing to filter unwanted intersections.

zz_box_min(:) = -huge(1.0)
This is the minimum \( z \)-coordinate used to define a bounding box to constrain the slicing to filter unwanted intersections.

slice_xmc(:) = huge(1.0)
This is the \( x \)-coordinate of the moment center, in the specified reference frame, for aerodynamic moments acting on the slice. The default value "huge" will result in the moment center being taken as the computed quarter chord of the slice.

slice_ymc(:) = huge(1.0)
This is the \( y \)-coordinate of the moment center, in the specified reference frame, for aerodynamic moments acting on the slice. The default value
"huge" will result in the moment center being taken as the computed quarter chord of the slice.

\[ \text{slice}_z\text{mc}(:) = \text{huge}(1.0) \]

This is the \( z \)-coordinate of the moment center, in the specified reference frame, for aerodynamic moments acting on the slice. The default value "huge" will result in the moment center being taken as the computed quarter chord of the slice.

\[ \text{n}_\text{bndrys}_{\text{to}\_\text{slice}}(:) = 0 \]

This is the number of candidate boundaries to search while computing slice-plane intersections. The index is the slice. By default, all solid boundaries will be searched. Specifying which boundaries are candidates for slicing may speed up the slicing process and can be used to filter out unwanted intersections or to slice non-solid boundaries.

\[ \text{bndrys}_{\text{to}\_\text{slice}}(:, :) = 0 \]

This is the list of \( \text{n}_\text{bndrys}_{\text{to}\_\text{slice}} \) boundaries, when the variable \( \text{n}_\text{bndrys}_{\text{to}\_\text{slice}} \) is greater than zero. The first index is the slice and the second index is the boundary.

\[ \text{slice}_\text{frame}(:) = '' \]

This is the name of the slice reference frame. Blank indicates the inertial frame. For moving geometries, output may be requested in either the reference frame of a particular body, or an “observer” frame. To specify the frame of a particular body, use the \text{body}_\text{name} entered in the &\text{body}\_\text{definitions} namelist. To specify the observer frame defined in the &\text{observer}\_\text{motion} namelist, use 'observer'.

\[ \text{slice}_\text{group}(:) = 1 \]

This assigns this slice to a particular group number. Within a group, slice locations must be given in ascending order.

\[ \text{chord}_\text{dir}(:) = 1 \]

This is the direction of local chord relative to the direction from leading edge to trailing edge, in the slice plane. The value 1 indicates local chord in direction from leading edge to trailing edge. The value \(-1\) indicates local chord in direction from trailing edge to leading edge. Determination of the leading and trailing edges is described below.

\[ \text{te}_\text{def}(:) = 1 \]

This is the number of points or line segments to consider when defining the trailing edge of the slice (see Fig. B2). A value of 1 defines the
trailing edge as the aft-most point. This is best for sharp trailing edges. A positive number greater than 1 initiates a search over the aft-most te_def segments for corners, after which the trailing edge is taken as the average coordinate over all the detected corners. Two corners are assumed to be the desired number, and warnings are output if only one or more than two are found. The value of te_def must be chosen judiciously. It should be large enough to allow both corners to be found but not so large as to cause excessive searching or for any non-trailing edge corners to be found. A positive value of te_def is best for and recommended only for squared-off trailing edges. A negative number indicates a parabolic fit of the aft-most abs(te_def) points, which is best for rounded or blunted trailing edges.

le_def(:) = 30

This is the number of points to consider when defining the leading edge of the slice (see Fig. B3). A value of 1 defines the leading edge as the forward-most point. Use this if nothing else works or for special cases. A positive number indicates a search over the forward-most le_def points for the one that has the maximum distance from the previously determined trailing edge. A positive number for le_def is generally the best choice provided that the trailing edge can be accurately located. A negative number indicates a parabolic fit over the forward-most abs(le_def) points.

corner_angle(:) = 120.0

This is used in conjunction with a te_def greater than 1. Angles between adjacent sliced segments that are less than corner_angle degrees will be considered a corner between the two segments. For squared-off trailing edges, two and only two corners should be detected; warnings are output if only one or more than two are found.

use_local_chord = .true.

Use the computed local (sectional) chord based on the computed leading edge and trailing edge locations to normalize the sectional force and moment data. When .false., the value of x_moment_length in &force_moment_integ_properties will be used instead of the locally computed chord.

tecplot_slice_output = .true.

This outputs the sliced data to a formatted Tecplot™ file that is named [project_rootname]_slice.dat. This file can become very large for unsteady flows with frequently written data at many slice locations.
output_sectional_forces = .true.

This outputs detailed force and moment data for each slice to a formatted file, [project_rootname].sectional_forces. This file contains force and moment data, like that in the [project_rootname].forces file, for each and every slice. In addition, it contains geometrical data for each slice (leading and trailing edge coordinates, moment center, etc.) This file can become very large for unsteady flows with frequently written data at many slice locations. The data in the file, especially the geometry data, can be useful to assess whether the slicing is working as expected.

slice_initial.coords = .false.

This allows faster slicing for some cases by only computing slice interpolation coefficients once.

custom_transform(1,1,1:4) = 1.0, 0.0, 0.0, 0.0

This is a user-specified transform matrix to allow slicing in a custom coordinate system.

output_in_slice.coords(:) = .false.

This outputs sliced data in the user-specified coordinate system. The default outputs the data in the slice_frame, which is only available if custom_transform is set.

Important Considerations for Determination of Leading And Trailing Edges  Determining the locations of airfoil leading and trailing edges is especially important for rotorcraft applications where airloads are usually examined (and provided to a CSD code, if applicable), in an airfoil section-aligned coordinate system. The leading and trailing edge points determine the orientation of this section aligned coordinate system. In the section-aligned system, the local x coordinate is aligned with the local chord, positive in the direction from the leading edge to the trailing edge. The local span direction is defined by the moment centers at the slice_location points, positive in the direction of increasing slice_location. The local normal direction is defined as the cross product of the local chord vector and local span vector. When slicing boundary data, the computed forces are computed in both the selected frame of reference (see slice_frame) and in an airfoil section aligned system. If the data in the section-aligned system is irrelevant to you, then you do not need to worry about choosing the detection parameters carefully; the default values should be reasonable. However, if resolution of forces and moments into a section-aligned system is important to you, then there are a number of things that should be considered:

1. Make sure the chord direction chord_dir is correct; the default is that going from the leading edge to the trailing edge is the same as traveling in
the positive “chordwise” coordinate direction. For most applications this is the usual situation; however, the convention for rotorcraft applications is the opposite, requiring \texttt{chord\_dir = -1}.

2. Since the best option for determining the leading edge uses the trailing edge location (\texttt{le\_def > 1}), care should be taken to get the trailing edge correct. For \textit{sharp} trailing edges, this is very simple since the default of \texttt{te\_def = 1} (i.e., use the aft-most point) is the best option. However, smoothly blunted or squared-off trailing edges are more difficult. When the boundary surface of an unstructured mesh is sliced, the resulting section will be comprised of line segments determined by the intersection of the specified plane and the edges of the surface triangles. These segments and the points that make up the segments will not usually be the same as the surface points; typically there are more segments and points arising from intersected triangles, as illustrated in Fig. B1. This greater point count should influence the selection of \texttt{te\_def} and \texttt{le\_def} values. You will need enough segments (\texttt{te\_def} and \texttt{le\_def}) to ensure that both corners are detected, but not so many that other, non trailing-edge corners (if present) are detected. Another parameter that may be of use to aid in the detection of corners is the \texttt{corner\_angle}; corners with angles larger than \texttt{corner\_angle} between adjacent segments will require a larger value of \texttt{corner\_angle} for detection.

The resulting section corresponding to the slice depicted in Fig. B1 is shown in Fig. B2, where the view is zoomed in to the trailing edge region. The aft-most 8 segments (of the approximately 30 segments in this view) are shown in red. The computed trailing edge locations using two different \texttt{te\_def} values are shown. The minimum \texttt{te\_def} value at this particular station to pick up both corners would be 8, but a value of 20 was used in case another slice required more segments. If the blade was pitched downward rather than upward, then the point chosen by \texttt{te\_def = 1} would be the lower corner, rather than the upper corner as shown. Thus, when pitching up and down, \texttt{te\_def = 1} with squared-off trailing edges can lead to jumps in the trailing edge position as the section transitions from pitch up to pitch down. Depending on the thickness of the trailing edge, this can lead to jumps in the geometric pitch angle of a few tenths of a degree. To avoid this, the option \texttt{slice\_initial\_coords = .true.} will reuse the leading and trailing edges determined from the initial grid definition, rather than the current, displaced grid location.

3. Smoothly-blunted (rounded) trailing edges should be done with either \texttt{te\_def = 1} (aft-most point) or via a parabolic fit of the aft-most \texttt{abs(te\_def)} points; the latter option is probably better in general but will require some experimentation for the particular case at hand to
choose the optimal number of points over which to fit the parabola.

4. The leading edge is typically easier to determine, if a good trailing edge position has already been found. The default value of `le_def = 30` (search the 30 forward-most points for the one with the greatest distance from the trailing edge location) should do a decent job for most cases.

Figure B3 shows a sliced section, zoomed in to the leading edge region. The forward-most 20 segments (of the approximately 30 segments in this view) are shown in red. The computed leading edge locations using two different `le_def` values are shown. In this case, both results are fairly close but `le_def = 30` has picked out the true leading edge (as judged from the leading edge geometry at zero pitch angle).

5. The leading edge and trailing edge detection schemes can be somewhat sensitive to the input choices. For cases that rely on accurate resolution of forces and moments into section-aligned coordinates (e.g., rotorcraft), it is wise to spend some time up front to make sure that things are

![Figure B1: View looking upstream from the trailing edge of a rotor blade mesh; the light-colored region is the squared-off trailing edge; the red line shows the location where an x=constant slice will be taken; black circles indicate surface grid points on the trailing edge.](image-url)
coming out as expected. To do this, inspect the [project_rootname].sectional_forces file for a particular slice station; at each station the computed leading and trailing edge coordinates will be output. Plot the corresponding station from the [project_rootname].slice.dat, as done above, and make sure the computed coordinates are the correct ones. If many stations are sliced, it is impractical to inspect all of them in this manner, but it is good practice to spot check at least a few stations. For moving-geometry cases, try first running the case with --body_motion_only. That will allow output of the [project_rootname].sectional_forces and [project_rootname].slice.dat files without the expense of a flow solve or mesh deformation; for spot checking you may want to have the slicing done infrequently, perhaps using fewer stations than ultimately desired, as these output files can be huge.

6. While the [project_rootname].sectional_forces can be useful for spot checking, the data in the file is not in a format that is amenable to plotting. The FUN3D distribution `utils/Rotorcraft` directory contains a utility code that will read in both the files `slice.info` and [project_rootname].sectional_forces to output Tecplot files, for each slice group, containing force and moment data in the section-aligned

![Image](image_url)

Figure B2: Sliced section corresponding to Fig. B1; zoomed in to the trailing edge region.
coordinate system, as well as geometry data (leading edge, trailing edge, quarter-chord coordinates, and pitch angle).

7. After making sure that the leading edge and trailing edge positions are being computed correctly, you may want to turn off one or both of the [project_rootname].sectional_forces and [project_rootname]_slice.dat files unless needed. For instance, in rotorcraft applications with coupling to external CSD codes, although the blade boundary surfaces must be sliced to generate the aerodynamic loads data for the CSD code, this information is actually passed to the CSD code by another file; the [project_rootname].sectional_forces and [project_rootname]_slice.dat files are not used.

8. Although the slicing process will work for multi-element airfoils, at this time the computation of the leading edge and trailing edge is only done for the entire section, not each element individually.

Figure B3: A sliced section, zoomed in to the leading edge region.
B.4.25 &massoud_output

This namelist controls the output of files for interaction with the MDO packages (e.g., design, aeroelastics). In a design setting, these files contain the information necessary to parameterize the surface grid(s). The command-line option --write_aero_loads_to_file is required to output the aeroelastics file [project_rootname]_ddfdrive_bodyN.dat and the command line option --write_massoud_file is required to output the design parameterization file [project_rootname]_massoud_bodyN.dat for each of the N body groups present.

&massoud_output
  n_bodies = 0
  nbndry(:) = 0
  boundary_list(:) = ''
  massoud_output_freq = -1
  massoud_file_format = 'ascii'
  massoud_use_initial_coords = .false.
  aero_loads_output_freq = -1
  aero_loads_file_format = 'ascii'
  include_time_info = .true.
  aero_loads_use_initial_coords = .false.
  aero_loads_dynamic_pressure = 1.0
  output_transform(1,1:4) = 1.0, 0.0, 0.0, 0.0
  output_transform(2,1:4) = 0.0, 1.0, 0.0, 0.0
  output_transform(3,1:4) = 0.0, 0.0, 1.0, 0.0
  output_transform(4,1:4) = 0.0, 0.0, 0.0, 1.0
  output_scale_factor = 1.0
  input_transform(1,1:4) = 1.0, 0.0, 0.0, 0.0
  input_transform(2,1:4) = 0.0, 1.0, 0.0, 0.0
  input_transform(3,1:4) = 0.0, 0.0, 1.0, 0.0
  input_transform(4,1:4) = 0.0, 0.0, 0.0, 1.0
  input_scale_factor = 1.0
/

n_bodies = 0

This is the number of user-defined bodies. For moving-grid cases, these bodies are typically the same as those defined as moving bodies, but that need not be the case.

nbndry(:) = 0

This is the number of boundary patches listed for a given body.

boundary_list(:) = ''

This is a list of boundary patch numbers for a given body. Commas and dashes can be used to specify ranges, i.e., '1,2,5-7'.
massoud_output_freq = -1
This is the iteration frequency of massoud output, where the special value -1 corresponds to once at the end of a successful run.

massoud_file_format = 'ascii'
This is the format of the massoud file; the alternate choice is 'stream' (C binary).
‘ascii’ is ASCII file format
‘stream’ is Fortran stream (C binary) format

massoud_use_initial_coords = .false.
Write the massoud file for the x, y, z surface coordinates at t=0. Otherwise, use current x, y, z surface coordinates.

aero_loads_output_freq = -1
This is the iteration frequency of aerodynamic loads output, where the special value -1 corresponds to once at the end of a successful run.

aero_loads_file_format = 'ascii'
This is the format of the aerodynamic loads file; the alternate choice is 'stream' (C binary).
‘ascii’ is ASCII file format
‘stream’ is Fortran stream (C binary) format

include_time_info = .true.
Write simulation time and strand info to ASCII Tecplot™ file(s). Including time info in the files makes animation within Tecplot™ very simple.

aero_loads_use_initial_coords = .false.
Write the current aerodynamic loads mapped to the x, y, z surface coordinates at t=0. Otherwise, use current x, y, z surface coordinates. This option is only relevant if the grid is moved or changed during the solution process.

aero_loads_dynamic_pressure = 1.0
The dynamic pressure used to convert force coefficients into forces; the default value leaves the output in coefficient form. Note that the input variable output_scale_factor separately handles scaling of coordinate values, so care must be exercised to insure appropriate dimensional forces if both are used in combination.
**output_transform(1,1:4) = 1.0, 0.0, 0.0, 0.0**

This is a user-specified transform matrix to allow output of aero loads in a custom coordinate system; typically used to output aero loads in an FEM/CSD coordinate system that differs from the CFD coordinate system. Note, at this point in time, this transform is applied to ALL bodies that are defined in this namelist. Note: the transform matrix should NOT include a scaling factor (e.g. inches to meters); any required scale factor is input separately.

**output_scale_factor = 1.0**

Allows a scaling of the output x,y,z coordinates (e.g. meters to inches). Scaling is applied as a multiplicative factor.

**input_transform(1,1:4) = 1.0, 0.0, 0.0, 0.0**

This is a user-specified transform matrix to allow input of a new surface mesh that is defined in a custom coordinate system; typically used to read in a displaced surface defined in an FEM/CSD coordinate for use in the CFD coordinate system. Note, at this point in time, this transform is applied to ALL bodies that are defined in this namelist. Note: the transform matrix should NOT include a scaling factor (e.g. inches to meters); any required scale factor is input separately.

**input_scale_factor = 1.0**

Allows a scaling of the input x,y,z coordinates (e.g. inches to meters). Scaling is applied as a multiplicative factor.
This namelist specifies information for overset grid simulations.

```plaintext
&overset_data
  overset_flag = .false.
dci_on_the_fly = .false.
dci_period = huge(1)
reset_dci_period = .false.
dci_freq = 1
dci_dir = './'
reuse_existing_dci = .false.
skip_dci_output = .false.
dci_io = .false.
dci_io_nproc = 1
suggar_nproc = 1
/

overset_flag = .false.
When .true., overset mesh capability is enabled.

dci_on_the_fly = .false.
This controls whether overset connectivity is computed as the grid moves, or whether overset connectivity has been pre-computed for each grid position and is available to read in. Ignored if overset_flag = .false. and &rotor_data's overset_rotor = .false..

dci_period = huge(1)
This controls the period (in term of timesteps) at which the dci counter is reset. At time step dci_period, the flow solver will read dci data from the dci file for time step 1. Ignored if overset_flag = .false. and &rotor_data's overset_rotor = .false..
eset_dci_period = .false.
When .true., allows dci_period to be reset to a different value for restarting with a different time step.

dci_freq = 1
This controls how frequently the dci data is updated, either by computation within the flow solver, or by reading a new dci file. Dci data is updated every dci_freq time steps.

dci_dir = './'
This is the directory where dci files are located. Note: A trailing forward slash (/) is automatically added and should not be included in the directory name.
```
**reuse_existing_dci = .false.**

When .true., allows the computation of dci data to be skipped if a dci file for the current time step already exists. This option is typically used in conjunction with dci_period, so that dci files are computed on the fly for the first dci_period time steps, and then files are reused for all subsequent periods of grid motion, without having to change dci_on_the_fly in between. Ignored if dci_on_the_fly = .false..

**skip_dci_output = .false.**

When .true., the solver will not save the dci data to a file. Ignored if dci_on_the_fly = .false..

**dci_io = .false.**

When .true., dci files are read from disk with a dedicated rank (processor) to help mask communication with computation.

**dci_io_nproc = 1**

When dci_io = .true., this specifies the number of ranks to use for loading of dci files.

**suggar_nproc = 1**

This specifies the number of ranks to use for running libSUGGAR++. Intended for future expansion; currently must be 1.
B.4.27 &rotor_data

This namelist controls high-level rotor simulation settings. Eventually, this namelist may subsume rotor.input.

&rotor_data
  comprehensive_rotor_coupling = 'none'
  overset_rotor = .false.
/

comprehensive_rotor_coupling = 'none'

This controls whether the code is to be coupled to a rotorcraft comprehensive code, and if so, which one.

'none' not coupled.
‘camrad’ coupled to CAMRAD-II.
‘rcas’ loosely coupled to RCAS.
‘rcas_tight’ tightly coupled to RCAS.
‘fsi’ tightly coupled to DYMORE.

overset_rotor = .false.

This controls whether overset meshes are used for moving rotor simulations. If '.true.', the rotor motion is governed by the rotor.input file.
B.4.28 &adapt_metric_construction

This namelist controls how the metric is formed for metric-based mesh adaptation. More details on grid adaptation can be obtained in section 7.

&adapt_metric_construction
  adapt_hessian_key = 'mach'
  adapt_hessian_method = 'lsq'
  adapt_max_anisotropy = 1.0e6
  adapt_max_edge_growth = 2.0
  adapt_max_edge_length = -1.0
  adapt_min_edge_length = -1.0
  adapt_output_tolerance = -0.5
  adapt_complexity = -1.0
  adapt_error_estimation = 'embed'
  adapt_exponent = 0.2
  adapt_feature_scalar_key = 'density'
  adapt_feature_scalar_form = 'delta'
  adapt_feature_length_exp = 0.5
  adapt_intersect_metric_in_time = .false.
  adapt_metric_from_file = ''
  adapt_export_metric = .false.
  adapt_twod = .false.
  adapt_verbose = .false.
  adapt_export_feature_scalar_key = 'none'
  adapt_visualize_metric = 'none'
  adapt_current_h_method = 'edge'
  adapt_current_h_gradation = 1.5
/

adapt_hessian_key = 'mach'

This variable is used to define anisotropic Hessian,

'mach' is Mach number.

'pressure' is pressure.

'entropy' is entropy.

'temp' is temperature.

'density' is density.

'vorticity-magnitude' is the magnitude of the vorticity vector.

adapt_hessian_method = 'lsq'

This is the mathematical method used to recover the Hessian,

'lsq' applies a least-square gradient calculation twice. First it computes gradients via least-squares. Then the Hessian is computed by a second application of least-squares to the reconstructed gradient.
‘green’ use a Green variational approach, see Loseille et al. [34] for details.

‘kexact’ reconstructs the Hessian with a k-exact approach. See Barth [35] for details.

‘grad’ is volume-averaged element-based gradients, applied twice.

‘mesh’ implies the metric of the current grid for use in testing grid adaptation mechanics or maintaining the current anisotropy.

\texttt{adapt\_max\_anisotropy = 1.0e6}

This is the upper limit of the largest to smallest spacing in the metric.

\texttt{adapt\_max\_edge\_growth = 2.0}

This is the amount of coarsening that will be allowed by the scalar term of feature-based adaptation. It is not used by adjoint-based adaptation.

\texttt{adapt\_max\_edge\_length = -1.0}

This sets a maximum allowable spacing of the metric. It is a grid and problem dependent value and should be expressed in grid units. A negative value is unlimited.

\texttt{adapt\_min\_edge\_length = -1.0}

This sets a minimum allowable spacing of the metric. It is a grid/problem dependent value and should be expressed in grid units. A negative value is unlimited.

\texttt{adapt\_output\_tolerance = -0.5}

This is the error request for output-based adaptation and the scaling of the scalar term for feature-based adaptation. Feature-based adaptation requires a positive number. Output-based adaptation can be negative to indicate a relative error reduction or positive to indicate an absolute error request. It is difficult to choose a good value for this tolerance, see \texttt{adapt\_complexity} for a more intuitive way to request the adapted grid size.

\texttt{adapt\_complexity = -1.0}

This is the target complexity for the metric. It is intended to allow a user specification of the number of nodes in the adapted grid. There is a difference between the requested complexity and the number of nodes in the adapted grid that is a function of grid size. This is because the requested complexity is a continuous measure, but the metric is discrete. Also, the adaptation mechanics produce a grid that is near but does not exactly match the metric. Adjust the requested complexity manually to obtain the desired grid size if the grid is smaller or larger than expected.
adapt_error_estimation = 'embed'

This selects the method used for error estimation for output-based adaptation,

‘embed’ uses a uniformly refined grid and interpolated solution to estimate the output error. [36]

‘single’ uses the current grid and reconstructed solution to estimate the output error. Uses much less memory, but does not provide an improved estimate of the functional. [25]

‘opt-goal’ is the optimal goal-oriented metric. [37] It requires the namelist option adapt_complexity to be set.

adapt_exponent = 0.2

This is the exponent on error estimate to map local error to a change in grid spacing. It is based on an a priori spatial error convergence estimate. [36]

adapt_feature_scalar_key = 'density'

This is the “key” flow variable (feature) on which to adapt for feature-based adaptation,

‘mach’ is Mach number.

‘pressure’ is pressure.

‘entropy’ is entropy.

‘temp’ is temperature.

‘density’ is density.

‘vorticity-magnitude’ is the magnitude of the vorticity vector.

adapt_feature_scalar_form = 'delta'

This is the method to calculate feature-based refinement indicator from the adapt_feature_scalar_key scalar field. The following terms are computed for each edge in the grid and the nodal adaptation intensity is the maximum for all incident edges. The edge terms are,

‘delta’ is the jump in the key across the edge.

‘delta-1’ is the jump the key across the edge times the edge length to the adapt_feature_length_exp power.

‘average-1’ is the average key of the two nodes of an edge times the edge length.

‘ratio’ is the ratio of the largest to the smallest key at the edge nodes.

‘max’ is the largest key at the nodes of the edge.

‘none’ will not use scalar term. It is uses the Hessian only.
adapt_feature_length_exp = 0.5
This is the exponent for use with adapt_feature_scalar_form = 'delta-1'.

adapt_intersect_metric_in_time = .false.
This will export a metric intersected over a window that includes each
time step of the current run. It is used for fixed-point adaptation of
time-accurate simulations. [37]

adapt_metric_from_file = ''
This reads the metric from this file instead of computing it when it is
blank.

adapt_export_metric = .false.
This exports the metric for external grid adaptation tools.

adapt_twod = .false.
When .true., compute a 2D metric for a one cell wide 3D grid. This
is required when a 2D adaptation method is selected but the grid is
actually a one cell wide 3D grid, because the adjoint does not have a 2D
specific mode.

adapt_verbose = .false.
When .true., this option reports more information during the error es-
timation process.

adapt_export_feature_scalar_key = 'none'
This is the format to export the feature scalar key for visualization,
‘none’ will not export.
‘cgnst’ is CGNS format, requires Fun3D to be configured with a CGNS
library. This format already includes x, y, and z. Set these variables to
.false. to avoid duplication.
‘fvuns’ is FieldView C-binary (Fortran stream) format. This format
already includes x, y, and z. Set these variables to .false. to avoid
duplication.
‘VTK’ is legacy VTK format.
‘csv’ is a comma separated value format.
‘tec’ is a single image ASCII tecplot format.
‘raw_ascii’ is a single image raw ASCII space separated format

adapt_visualize_metric = 'none'
This is the format to export the metric for visualization,
'none' will not export.
'cgns' is CGNS format, requires FUN3D to be configured with a CGNS library. This format already includes x, y, and z. Set these variables to .false. to avoid duplication.
'fvuns' is FieldView C-binary (Fortran stream) format. This format already includes x, y, and z. Set these variables to .false. to avoid duplication.
'VTK' is legacy VTK format.
'csv' is a comma separated value format.
'tec' is a single image ASCII tecplot format.
'raw_ascii' is a single image raw ASCII space separated format.

adapt_current_h_method = 'edge'

This is the method to estimate the current spacing of the grid,
'edge' will use the shortest incident edge at a node.
'implied' will use the largest eigenvalue of adjacent element implied metrics.

adapt_current_h_gradation = 1.5

This limits the gradation of the current spacing estimate by requiring it to be larger than this ratio of its neighbor’s spacing estimate.
B.4.29  &adapt_mechanics

This namelist contains variables that control how grid adaptation is performed. More details on general metric-based grid adaptation can be obtained in section 7. This namelist also contains variables to control specialized line adaptation adapt_library = 'line' and shock fitting line adaptation adapt_library = 'sfline'.

Variables with the ladapt_ prefix control line adaptation and variables with a sfline_ prefix control shock fitting line adaptation. These specialized 1D adaptation methods originated in the LAURA code and have a number of requirements that are described in the LAURA User's Manual. [38] The grid origin must be structured and all nodes assigned to a unique line. All lines must have the same number of nodes. The outer boundary (opposite solid walls) can be moved in or out to align with a developing bow shock and the distribution of points across the boundary layer can be adjusted to recover a target cell Reynolds number. If the grid has prisms grown off a solid surfaces then the distribution of prism heights can be adjusted to recover a target cell Reynolds number at the wall while retaining the the original spacing at the top of the prism stack.

Variable names beginning with sfline_ control how shock fitting meshes are adapted. Currently the shock fitting is only available with line adaptation which is engaged by specifying adapt_library = 'sfline'. The variables ladapt_re_cell, ladapt_ep0_grd, ladapt_fstr, and ladapt_g_limiter are also active with shock fitting.

&adapt_mechanics

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>adapt_library</td>
<td>'refine/one'</td>
</tr>
<tr>
<td>adapt_project</td>
<td>''</td>
</tr>
<tr>
<td>adapt_freezebl</td>
<td>-1.0</td>
</tr>
<tr>
<td>adapt_cycles</td>
<td>2</td>
</tr>
<tr>
<td>adapt_bamg_command</td>
<td>'bamg'</td>
</tr>
<tr>
<td>adapt_bamg_geometry_format</td>
<td>'amdba'</td>
</tr>
<tr>
<td>ladapt_fsh</td>
<td>0.8</td>
</tr>
<tr>
<td>ladapt_fstr</td>
<td>0.75</td>
</tr>
<tr>
<td>ladapt_fctrjmp</td>
<td>1.05</td>
</tr>
<tr>
<td>ladapt_re_cell</td>
<td>1.</td>
</tr>
<tr>
<td>ladapt_beta_grd</td>
<td>0.</td>
</tr>
<tr>
<td>ladapt_ep0_grd</td>
<td>0.</td>
</tr>
<tr>
<td>ladapt_max_distance</td>
<td>1.e+06</td>
</tr>
<tr>
<td>ladapt_jumpflag</td>
<td>2</td>
</tr>
<tr>
<td>ladapt_freq</td>
<td>0</td>
</tr>
<tr>
<td>ladapt_max</td>
<td>1000</td>
</tr>
<tr>
<td>ladapt_g_limiter</td>
<td>0.</td>
</tr>
<tr>
<td>sfadapt_fsbuffr</td>
<td>3</td>
</tr>
<tr>
<td>sfadapt_ceqinc</td>
<td>0.5</td>
</tr>
</tbody>
</table>
adapt_library = 'refine/one'

Adaptation library to call. The options are,
‘refine/one’ is the refine tetrahedral metric-based adaptation library. See Park [25] for a detailed description.
‘refine/two’ is a version of the refine adaptation library that is still undergoing development. It is based on original version of refine with some ideas from Michal and Krakos. [39]
‘meshsim’ is the Simmetrix MeshSim™ adaptation library.
‘bamg’ is the BAMG [40] 2D metric-based adaptation library. The metric and solution files will be exported and the BAMG executable will be run in the ../Flow directory.
‘line’ is line-based adaptation [38] for structured grids.
‘sfline’ is shock-fitting line-based adaptation for structured grids.
‘interpolate’ will linearly interpolate the project_rootname solution to an existing adapt_project grid without adaptation using the approach of Shenoy. [41]

adapt_project = ''

This is the project name for exporting the adapted grid and solution. An empty string appends _R to the project_rootname from the &project namelist.

adapt_freezebl = -1.0

This prevents modification of the grid within this distance of solid wall boundaries. It is used to to preserve an existing boundary layer grid structure. A negative value does not freeze. It is described by Park and Carlson. [42]

adapt_cycles = 2

This is the number of adaptation passes. It is only used for adapt_library = ‘\refine/one’. Choosing more cycles will produce a grid that better matches the metric, but can increase the time required for adaptation.

adapt_bamg_command = 'bamg'

This the the system command to execute BAMG. It may include the full path or command line arguments.
adapt_bamg_geometry_format = 'amdba'

BAMG geometry file format

'amdba' specifies -b [project_rootname].ambda as the BAMG geometry source. This will spline current boundary nodes to form the geometry of the domain boundary. It is approximate, but less likely to fail than .msh file boundary reconstruction.

'msh' specifies -b [project_rootname].msh as the BAMG geometry source. This will access the original geometry .msh file to define the domain boundary, but BAMG may have problems with boundary reconstruction.

ladapt_fsh = 0.8

This is the fraction of the distance between the body and the opposing boundary along a line of nodes where the captured shock is situated.

ladapt_fstr = 0.75

This is the fraction of edges along a line that are intended to resolve the boundary layer.

ladapt_fctrjmp = 1.05

This is the property ratio used to detect the shock when marching from the freestream toward the body. It is assumed the flow above the shock is uniform and the property ratios across edges along the line remain equal to one until the shock is encountered.

ladapt_re_cell = 1.

This is the target cell Reynolds number based on the speed of sound used to define the edge length $\Delta n$ of the first edge leaving the wall. $re_{cell} = \rho \Delta n c / \mu$.

ladapt_beta_grd = 0.

This is an exponential grid distribution parameter. Any value greater than 1 will override adaptation. If it is used to override adaptation to local flow, the recommended value is 1.15.

ladapt_ep0_grd = 0.

This is a grid clustering factor to pull nodes into the captured shock. A minimum value of 0 produces no clustering. A maximum value of 6.25 produces greatest clustering. Larger values can produce negative volumes.

ladapt_max_distance = 1.e+06

This is the maximum distance in grid units the outer boundary can be moved away from the body. This parameter is useful when adapting to
the shock in the wake, where the adapting grid may become excessively skewed. This value then effectively defines the maximum length of the wake domain.

\[ \text{ladapt_jumpflag} = 2 \]

This is an integer flag used to select the method of shock detection, '0' is no movement of outer boundary. Resolution in the boundary layer is adjusted to recover target \( \text{re}_{cell} \).

'1' uses pressure as sensing parameter.

'2' uses density as sensing parameter.

'3' uses temperature as sensing parameter.

'4' scales all edges along the line by a factor equal to \( \text{ladapt}_fctrj.mp \).

\[ \text{ladapt_freq} = 0 \]

This is the number of relaxation steps between calls to line adaptation. The value 0 prevents line adaptation.

\[ \text{ladapt_max} = 1000 \]

This is the maximum number of calls to line adaptation permitted.

\[ \text{ladapt_glimiter} = 0 \]

This parameter insures a minimum mesh size does not get too big on a line and cause local skewing. It must be a positive number to engage.

\[ \text{sfadapt_fsbuffr} = 3 \]

This is the number of buffer nodes between the freestream boundary and the fitted shock. Zero buffer nodes make the freestream boundary the shock fitting surface, three buffer nodes moves the shock fitting surface three nodes into the interior of the computational domain relative to the freestream boundary.

\[ \text{sfadapt_ceqinc} = 0.5 \]

This is the shock fitting compatibility equation influence coefficient. A value of 0.0 means that shock fitting is controlled by the continuity equal to the compatibility equation, a value of 1.0 means that the shock fitting is controlled by momentum equation compatibility equal to the compatibility equation, and a value of 0.5 means that shock fitting is equally controlled by the continuity and momentum compatibility equations.

\[ \text{sfadapt_shkdtct} = 1.0e-01 \]

This is the shock fitting shock-boundary interaction detector coefficient. This parameter controls when the shock is considered to be interacting
with the shock fitting boundary nodes and determines when the boundary begins to be fitted to the shock. The value is one minus the local relative density jump below which the shock is not considered to be interacting with the boundary. Increasing this parameter decreases the sensitivity of the sensor and decreasing this parameter increases the sensitivity of the sensor.

\[ \textsf{sfadapt.ffrac0} = 1.0 \times 10^0 \]

This is the shock fitting initial freestream velocity boundary velocity fraction. When the bow shock has not yet reached the freestream boundary, the shock fitting equations are not valid. However, the code allows the freestream boundary to initially move towards the body at some fraction of the freestream boundary velocity. A value of 0.0 freezes the freestream boundary until the shock reaches it, a value of 1.0 moves the freestream boundary at the freestream velocity until the shock reaches it, and a value in the range (0.0, 1.0) moves the freestream boundary towards the body at \( \textsf{sfadapt.ffrac0} \times \text{freestream total velocity} \).

\[ \textsf{sfadapt.ffrac1} = 1.0 \times 10^{-1} \]

This is the shock fitting interaction freestream velocity boundary velocity fraction. When the bow shock has been determined to be interacting with the freestream boundary, the shock fitting equations are not valid all along the shock. However, the code allows the initial interaction speed of the shock with the freestream boundary to be scaled back at some fraction of the freestream boundary and/or shock velocity. A value of 0.1 constrains the freestream boundary to move at a fraction of the freestream velocity and a value in the range (0.0, 1.0) moves the freestream boundary towards/away from the body at freestream total velocity*\( \textsf{sfadapt.ffrac1} \).
B.4.30 &sonic_boom

This namelist contains variables that specify the sonic boom cost functions \texttt{boom_targ} and \texttt{sboom}. The namelist requires the \texttt{../rubber.data} file for the flow and adjoint solvers. See section 6.3 for details on the minimum inputs required for specifying the adjoint cost function. When \texttt{nsignals} is greater than zero, the pressure signature is output as a Tecplot™ file. To compute the functional in the flow solver, the \texttt{--design_run} command line option is required.

The rays are rotated about (\texttt{x_cor},\texttt{z_cor}) by \texttt{angle_of_attack} when the variable \texttt{rotate_ray_by_angle_of_attack} is true. Each of the \texttt{npoints} interpolation points are linearly spaced between the lower and upper bound of \texttt{x}.

\begin{verbatim}
&sonic_boom
  nsignals = 0
  y_ray(:) = 0.001
  z_ray(:) = 0.0
  x_cor = 0.0
  z_cor = 0.0
  rotate_ray_by_angle_of_attack = .true.
  npoints = 1000
  ray_x_limit_method = 'local'
  x_lower_bound = -1.e20
  x_upper_bound = 1.e20
  dp_pinf = .true.
  p_pinf = .false.
  weight = .false.
/

  nsignals = 0

  This is the total number of signal rays.

  y_ray(:) = 0.001

  This is the \texttt{y} value of each ray before \texttt{angle_of_attack} rotation. It is dimensioned 1 to \texttt{nsignals}.

  z_ray(:) = 0.0

  This is the \texttt{z} value of each ray before \texttt{angle_of_attack} rotation. It is dimensioned 1 to \texttt{nsignals}.

  x_cor = 0.0

  This is the \texttt{x} center of \texttt{angle_of_attack} rotation.

  z_cor = 0.0

  This is the \texttt{z} center of \texttt{angle_of_attack} rotation.
\end{verbatim}
rotate_ray_by_angle_of_attack = .true.
When .true., this will rotate the rays by angle_of_attack about x_cor and z_cor.
npoints = 1000
This is the number of points in each ray.
ray_x_limit_method = 'local'
This is the method used to determine the x-direction start and end of the ray. If x_lower_bound and/or x_upper_bound is specified, then ray_x_limit_method must be set to 'explicit'.
'local' sets each ray limit independently by computing the x min and x max of all grid cells intersected by the ray.
'explicit' explicitly sets the x min and x max of all rays with the x_lower_bound and x_upper_bound namelist variables. Only valid for adjoint cost function boom_targ.

x_lower_bound = -1.e20
This is the explicit x lower bound for ray_x_limit_method='explicit'.
x_upper_bound = 1.e20
This is the explicit x upper bound for ray_x_limit_method='explicit'.
dp_pinf = .true.
When .true., this will include normalized delta pressure \( (p - p_\infty)/p_\infty \) in tecplot output.
p_pinf = .false.
When .true., this will include normalized pressure \( p/p_\infty \) in tecplot output.
weight = .false.
When .true., this will include a weight of one in tecplot output for use in setting up a near-body pressure design.
### B.4.31 &sboom

This namelist contains variables that specify the inputs required to execute the sBOOM library. See section 8.2.6 for details.

```plaintext
&sboom
  alt = 45000.0
  hg = 0.0
  headangle = 0.0
  climbangle = 0.0
  dmdt = 0.0
  turnrate = 0.0
  climbrate = 0.0
  rs = 500.0
  signum = 10000
  zeronum = 1200
  tol2 = 1.e-6
  nonlinear = 1
  thermoviscous = 1
  relaxation = 1
  initialtimestep = 0.01
  refl = 1.9
  outflag = 0
  numouts = 1000
  tol = 0.005
  inputininches = 0
  adjmode = 1
  runmode = 1
  objmode = 1
  nazimuths = 1
  phi(:) = 0.0
  targetdbas(:) = 56.0
  createtarget = 0
  lowerbound = -1000.0
  upperbound = -1000.0
  lappass = 500
  target_numpts(:) = 0
  target_xx(:,:,:) = 0.0
  target_dpress(:,:,:) = 0.0
  tflag = 0
  ntalt = 0
  ztalt(:) = 0.0
  talt(:) = 0.0
  windflag = 0
  nwindx = 0
  zwindx(:) = 0.0
```
alt = 45000.0
This is the cruise altitude of the full-scale vehicle in feet.

hg = 0.0
This is the height of the ground in feet.

headangle = 0.0
This is the vehicle heading angle in degrees. A 180 heading would mean
away from the x-axis, a 90 heading would mean away from the y-axis.
This option is only relevant when winds are specified.

climbangle = 0.0
The is the vehicle climb angle in degrees.

dmdt = 0.0
This is the acceleration of the vehicle in 1/second (Mach number per
second).

turnrate = 0.0
This is the turning rate of the vehicle in degrees per second.

climbrate = 0.0
This is the climb rate of the vehicle in degrees per second.

rs = 500.0
This is the off-body distance in full-scale feet. This full-scale distance
should match the location in grid units used to define y_ray and z_ray
in &sonic_boom.
\textbf{signum = 10000}
This is the number of points representing the off-body waveform for propagation. Increasing the number points reduces the discretization error of the Burgers equation and increase the execution time of sBOOM.

\textbf{zeronum = 1200}
This in the number of points used to zero pad the front part of the signature. These points are required to prevent the initial shock from propagating to the front of the Burgers equation domain and causing a numerical instability. The actual waveform will be sampled using \((\text{signum} - \text{zeronum})\) points. Typically, 10–20\% of \text{signum} is required.

\textbf{tol2 = 1.e-6}
This is the leading zero tolerance of the input waveform. It is used to truncate the initial portion of the off-body waveform before zero padding, where \(dp/p\) value are less than this number.

\textbf{nonlinear = 1}
This controls solution non-linearity,
‘1’ uses cumulative non-linearity.
‘0’ does not use cumulative non-linearity.

\textbf{thermoviscous = 1}
This controls the modeling of thermo-viscous absorption,
‘1’ uses cumulative thermo-viscous absorption.
‘0’ does not use thermo-viscous absorption.

\textbf{relaxation = 1}
This controls the modeling of molecular relaxation,
‘1’ uses cumulative molecular relaxation.
‘0’ does not use molecular relaxation.

\textbf{initial timestep = 0.01}
Nondimensional initial step size for propagation. A smaller number prevents a multi-valued function, but increases execution time. If you receive a discontinuity error, reduce this value.

\textbf{refl = 1.9}
This is the ground reflection factor used to scale ground signatures.

\textbf{outflag = 0}
This determines the format of the output,
'0' outputs delta pressure in psf as a function of time in ms.

'1' outputs delta pressure divided by freestream pressure as a function of x in feet.

\texttt{numouts = 1000}

This is the number of points requested in the ground signature.

\texttt{tol = 0.005}

This is the slope tolerance needed for removing zero paddings from the ground signatures. This allows signatures at different azimuthal angles to have the same time axis starting with the initial shock.

\texttt{inputinches = 0}

This is the units of the FUN3D grid and geometry to scale the near field signature \( x \) for propagation,

'0' for FUN3D grid units in feet.

'1' for FUN3D grid units in inches.

'2' for FUN3D grid units in meters.

\texttt{adjmode = 1}

This is the sBOOM simulation mode,

'1' for a primal and adjoint simulation.

'0' for primal simulation only.

\texttt{runmode = 1}

This is the type of propagation and the class of cost function,

'1' propagates near field \( dp/p_\infty \) to ground and adjoint sensitivities of ground based metrics defined by \texttt{objmode}. The target pressure is specified with \texttt{target_numpts}, \texttt{target_dpress}, and \texttt{target_xx}.

'0' reverse propagates near field \( dp/p_\infty \) to compute equivalent area (when \( rs < (alt-hg) \)) and directly converts off-body pressures to equivalent area (when \( rs > (alt-hg) \)). No ground signature or ground-based cost function is computed. See \texttt{bodylen}, \texttt{reg}, \texttt{regr}, \texttt{lbd}, and \texttt{ubd}. The target area distribution is specified with \texttt{target_numpts}, \texttt{target_dpress}, and \texttt{target_xx}.

\texttt{objmode = 1}

This is the cost function definition. The value of \texttt{runmode} changes its behavior as follows,
‘1’ for an A-weighted loudness target of \( I = (dBA - dBA_t)^2 \) when runmode=1 and an equivalent area target \( I = \sum_{i=1}^{N} \frac{1}{2} [Ae(i) - Ae_{target}(i)]^2 \) when runmode=0.

‘2’ for an inverse pressure design objective of \( I = \sum_{i=1}^{N} [p(i) - p(t, i)]^2 \) when runmode=1 and an equivalent area sum \( I = \sum_{i=1}^{N} Ae(i)^2 \) when runmode=0.

‘3’ for combined A-weighted loudness and inverse pressure design objective of \( I = (dBA - 56.0)^2 + \sum_{i=1}^{N} [p(i) - p(t, i)]^2 \) when runmode=1.

‘4’ for an A-weighted loudness objective of \( I = dBA \) when runmode=1.

nazimuths = 1
This is the number of azimuths to propagate. The nazimuths must match nsignals in &sonicBoom.

phi(:) = 0.0
This is a nazimuths length vector of azimuthal locations in degrees.

targetdbas(:) = 56.0
This is the target \((dBA_t)\) at each azimuthal location when objmode=1.

createtarget = 0
This is the source of a ground target signature,

‘0’ for no ground target signature.

‘1’ to internally create a ground target by Laplace smoothing. The smoothing is controlled by lowerbound, upperbound, and lappass.

‘2’ for a user specified target. The target is defined by the target_numpts, target_xx, and target_dpress variables.

lowerbound = -1000.0
When createtarget=1, this is the lower bound in time (milliseconds) after which the user wants to Laplace smooth the computed signature to form the target. When runmode=0, this defines the start of locations in X (feet) where any difference in equivalent area between actual and target equivalent areas will contribute to the cost functional. Outside these bounds, even if the equivalent area does not match the target, it does not contribute to the cost functional.

upperbound = -1000.0
When createtarget=1, this is the upper bound in time (milliseconds) before which the user wants to Laplace smooth the computed signature to form the target. When runmode=0, this defines the end of locations
in X (feet) where any difference in equivalent area between actual and target equivalent areas will contribute to the cost functional. Outside these bounds, even if the equivalent area does not match the target, it does not contribute to the cost functional.

\[ \text{lapass} = 500 \]

The number of Laplace smoothing passes to generate a target ground signature. A higher number increases the smoothness of the target. It is only used for \text{createtarget}=1.

\[ \text{target.numpts}(::) = 0 \]

This is the number of points in the \text{target.xx} and \text{target.dpress} at each azimuth. It is used for \text{runmode}=1 with \text{objmode}=2,3 or \text{runmode}=0 with \text{objmode}=1.

\[ \text{target.xx}(::,:) = 0.0 \]

This is the time (in milliseconds) of the target signature at each azimuth for \text{runmode}=1 or \( x \) in feet for \text{runmode}=0. It is only used for some objective functions, see \text{objmode}. The first index is azimuthal location and the second index is the target signature point.

\[ \text{target.dpress}(::,:) = 0.0 \]

This is the delta pressure (in psf) of the target signature at each azimuth for \text{runmode}=1 or the equivalent area target for \text{runmode}=0. It is only used for some objective functions, see \text{objmode}. The first index is azimuthal location and the second index is the target signature point.

\[ \text{tflag} = 0 \]

This controls the source for the atmospheric temperature distribution, ‘0’ for the 1976 U.S. Standard Atmosphere temperature profile. ‘1’ for a temperature profile specified by \text{ntalt}, \text{ztalt}, and \text{talt}.

\[ \text{ntalt} = 0 \]

This is the number of \text{ztalt} altitude and \text{talt} temperature pairs to define the temperature profile.

\[ \text{ztalt}(::) = 0.0 \]

This is \text{ntalt} length vector of altitudes (in meters) to specify an atmospheric temperature distribution.

\[ \text{talt}(::) = 0.0 \]

This is \text{ntalt} length vector of temperature (in Fahrenheit) to specify an atmospheric temperature distribution.
windflag = 0
This controls the source for winds,
‘0’ for no winds.
‘1’ for a wind profile specified by nwindx, windx, zwindx, nwindy, windy, and zwindy.

nwindx = 0
This is the number of zwindx altitude and windx x-wind pairs that define the wind profile.

zwindx(:) = 0.0
This is a vector of length nwindx of altitude (in meters) to specify a wind profile.

windx(:) = 0.0
This is a vector of length nwindx of x-wind speeds (in meters/sec) to specify a wind profile.

nwindy = 0
This is the number of zwindy altitude and windy y-wind pairs that define the wind profile.

zwindy(:) = 0.0
This is a vector of length nwindy of altitude (in meters) to specify a wind profile.

windy(:) = 0.0
This is a vector of length nwindy of y-wind speeds (in meters/sec) to specify a wind profile.

rflag = 0
This controls the source for the atmospheric humidity distribution,
‘0’ for the 1976 U.S. Standard Atmosphere relative humidity profile.
‘1’ for a relative humidity profile specified by nhalt, zrh, and rh.

nhalt = 0
This is the number of zrh altitude and rh relative humidity pairs.

zrh(:) = 0.0
This is a vector of length nhalt of altitude (in meters) to specify a relative humidity profile.
\textbf{\texttt{rh(:)} = 0.0}

This is a vector of length \texttt{nhalt} of relative humidity (in percent) to specify a relative humidity profile.

\textbf{\texttt{bodylen = 127.0}}

This is the aircraft body length in feet. It is only used for the adjoint of equivalent area matching, \texttt{runmode=0} and \texttt{adjmode=1}.

\textbf{\texttt{reg = 1.e-4}}

This is the thermo-viscous absorption regularization parameter. A smaller value could lead to an ill-posed reverse diffusion problem. A higher value increases error. Applicable only when \texttt{rs<(alt-hg)}. It is only used for equivalent area, \texttt{runmode=0}.

\textbf{\texttt{regr = 1.e-4}}

This is the molecular relaxation regularization parameter. A smaller value could lead to an ill-posed reverse diffusion problem. A higher value increases error. Applicable only when \texttt{rs<(alt-hg)}. It is only used for equivalent area, \texttt{runmode=0}.

\textbf{\texttt{lbd = 0.99}}

This is the under-deviation parameter for reversed equivalent area matching. Equivalent area deviations below a target are generally favorable to deviations above a target. Equivalent area matching cost functions and their sensitivities are only computed when the equivalent area is within the \texttt{lbd} and \texttt{ubd} limits. It is only used for the adjoint of equivalent area matching, \texttt{runmode=0} and \texttt{adjmode=1}.

\textbf{\texttt{ubd = 1.0}}

This is the over-deviation parameter for reversed equivalent area matching. Equivalent area deviations below a target are generally favorable to deviations above a target. Equivalent area matching cost functions and their sensitivities are only computed when the equivalent area is within the \texttt{lbd} and \texttt{ubd} limits. It is only used for the adjoint of equivalent area matching, \texttt{runmode=0} and \texttt{adjmode=1}. 
B.4.32  &equivalent_area

This namelist contains variables that specify the inputs associated with equivalent area-based sonic boom cost functions. The number and order of these inputs should match the equivalent area (Ae) functions appearing in rubber.data.

&equivalent_area
  nfunctions = 0
  nplane(:) = 0
  global_scaling_factor(:) = 1.0
  lift_scaling_factor(:) = 1.0
  off_track_angle(:) = 0.0
/

nfunctions = 0
This is the total number of Ae functions, including functions used as objectives and constraints.
nplane(:) = 0
This is the total number of cutting planes along x-axis for each Ae function.
global_scaling_factor(:) = 1.0
This is the Ae(x) scaling factor for each Ae function.
lift_scaling_factor(:) = 1.0
This is the Lift L(x) scaling factor for each Ae function.
off_track_angle(:) = 0.0
This is the off-track angle in degrees for each Ae function.
B.4.33  &noninertial_reference_frame

FUN3D can perform simulations in noninertial reference frame rotating at a constant rate, Ω. The noninertial reference frame simulation can be run as a steady state problem if the freestream velocity crossed with the rotation vector is zero, $U_\infty \times \Omega = 0$. In a practical sense, freestream velocity should be zero or parallel to the axis of rotation. Using a standard inertial reference frame requires the same problem to be run as an unsteady simulation at a larger computational cost. Typical uses would be the simulation of an isolated rotor in hover (without forward motion) or an aircraft performing a steady-state pitching maneuver or constant roll about the wind axis.

```fortran
&noninertial_reference_frame
  noninertial = .false.
  rotation_center_x = 0.0
  rotation_center_y = 0.0
  rotation_center_z = 0.0
  rotation_rate_x = 0.0
  rotation_rate_y = 0.0
  rotation_rate_z = 0.0
/

  noninertial = .false.
When .true., use a noninertial reference frame. The default is the inertial reference frame.
rotation_center_x = 0.0
This is the x of the steady rotation rate center point.
rotation_center_y = 0.0
This is the y of the steady rotation rate center point.
rotation_center_z = 0.0
This is the z of the steady rotation rate center point.
rotation_rate_x = 0.0
This is the steady noninertial rotation rate about the rotation center x-axis in reference speed of sound per grid unit (eqn_type = 'compressible') or reference speed per grid unit (eqn_type = 'incompressible').
rotation_rate_y = 0.0
This is the steady noninertial rotation rate about the rotation center y-axis in reference speed of sound per grid unit (eqn_type = 'compressible') or reference speed per grid unit (eqn_type = 'incompressible').
```
rotation_rate.z = 0.0

This is the steady noninertial rotation rate about the rotation center $z$-axis in reference speed of sound per grid unit (eqn_type = 'compressible') or reference speed per grid unit (eqn_type = 'incompressible').
B.5  moving_body.input

This namelist file is only used with time-dependent, moving grid case to specify
grid motion as a function of time, and is used in conjunction with the command
line option --moving_grid. See the following sections for descriptions of the
namelists in this file.
B.5.1 &body_definitions

This namelist specifies which mesh surfaces define the moving bodies. In general, each body may have a different motion. However, there are some fundamental constraints. For example, in a mesh with multiple bodies undergoing different motions, either overset meshes or a deforming mesh would be required. Note that a deforming mesh might well support only small relative motions between the bodies before the mesh becomes invalid (negative cell volumes). For a single rigid mesh, all bodies within the mesh would need to have the same motion.

&body_definitions

\begin{verbatim}
  n_moving_bodies = 0
  body_frame_forces = .false.
  output_transform = .false.
  dimensional_output = .false.
  ref_velocity = 1117.0
  ref_density = 0.002378
  ref_length = 1.0
  body_name(:) = ''
  parent_name(:) = ''
  n_defining_bndry(:) = 0
  defining_bndry(:, :) = 0
  motion_driver(:) = 'none'
  mesh_movement(:) = 'static'
  x_mc(:) = xmc
  y_mc(:) = ymc
  z_mc(:) = zmc
  s_ref(:) = sref
  c_ref(:) = cref
  b_ref(:) = bref
  move_mc(:) = 1
  trim_control(:) = .false.
  baseline_psi(:) = 0.0
  steps_per_period(:) = 0
\end{verbatim}

/ 

n_moving_bodies = 0

This is the number of bodies in motion.

body_frame_forces = .false.

This outputs aero forces acting on the body in the frame of the body when .true., rather than in the inertial reference frame.

output_transform = .false.

This outputs the transform matrix to TransformMatrixBody_N.hst for body N when .true.
dimensional_output = .false.
This outputs the body state data (displacements, velocities, and aero forces) in dimensional form for forced or 6-DOF motions when .true.. Use with ref_velocity, ref_density, and ref_length.

ref_velocity = 1117.0
This is the reference velocity to make aerodynamic forces dimensional, when dimensional_output = .true. Note that this should correspond to the sound speed if compressible. The default corresponds to standard sea level speed of sound, in ft/sec.

ref_density = 0.002378
This is the reference density to make aerodynamic forces dimensional, when dimensional_output = .true. The default corresponds to standard sea level density, in slugs/ft^3.

ref_length = 1.0
This is the reference length to make aerodynamic forces dimensional, when dimensional_output = .true. The default corresponds to one unit consistent with the values of ref_velocity and ref_density (i.e., 1 ft for the default reference conditions).

body_name(:) = ''
This is the name used to identify the body; the array index corresponds to body number.

parent_name(:) = ''
This is the name of the parent body; the array index corresponds to body number. The motion of a body follows (i.e., is added to) the motion of the parent. When naming the parent, '' signifies that the parent is the inertial frame. For single or independently-moving bodies, the default parent_name should be used.

n_defining_bndry(:) = 0
This is the number of boundaries that define the body; the array index corresponds to body number.

defining_bndry(:, :) = 0
This is a list of n_defining_bndry boundaries that define the body; the array index 1 corresponds to the boundary (from 1 to n_defining_bndry) defining the body; the array index 2 corresponds to the body number. If boundary lumping is used (section B.4.2), the boundaries must correspond to lumped boundaries.
motion_driver(:) = 'none'

This is the body motion mechanism; the array index corresponds to body number:

'none' is for no motion.

'forced' uses forced motion prescribed in &forced_motion.

'surface_file' uses surface motion prescribed in &surface_motion_from_file.

'motion_file' uses motion prescribed in &motion_from_file.

'6dof' computes motion via 6-DOF library, which is governed by &sixdof_motion.

'aeroelastic' computes motion via &aeroelastic_modal.data, or by coupling with an external FEM.

'custom' uses custom_kinematics; the user supplies a custom transform matrix as a function of time and design variables.

mesh_movement(:) = 'static'

This is the type of grid movement associated with body motion; the array index corresponds to body number:

'static' no mesh movement.

'rigid' rigid mesh movement; all nodes of the mesh rotate/translate in unison with the body.

'deform' deforms the mesh locally to accommodate the motion of the solid body.

x_mc(:) = xmc

This is the x-coordinate of moment center at \( t = 0 \); the array index corresponds to body number.

y_mc(:) = ymc

This is the y-coordinate of moment center at \( t = 0 \); the array index corresponds to body number.

z_mc(:) = zmc

This is the z-coordinate of moment center at \( t = 0 \); the array index corresponds to body number.

s_ref(:) = sref

This is the nondimensional reference area for force and moment normalization; the array index corresponds to body number.

c_ref(:) = cref

This is the nondimensional reference chord length for force and moment normalization; the array index corresponds to body number.
\( b_{\text{ref}}(:) = \text{bref} \)

This is the nondimensional reference span length for force and moment normalization; the array index corresponds to body number.

\( \text{move} \_\text{mc}(:) = 1 \)

This controls the movement of the moment center; the array index corresponds to body number:

‘0’ leave moment center fixed in space

‘1’ move moment center with body

\( \text{trim} \_\text{control}(:) = .\text{false}. \)

The trim is used as a design variable when \( .\text{true}. \); the array index corresponds to body number.

\( \text{baseline} \_\text{psi}(:) = 0.0 \)

This is the starting azimuth for trim when trim is used as a design variable; the array index corresponds to body number.

\( \text{steps} \_\text{per} \_\text{period}(:) = 0 \)

This is the number of steps to define a trim period when trim is used as a design variable; the array index corresponds to body number.
B.5.2 &forced_motion

&forced_motion
  rotate(:) = 0
  rotation_rate(:) = 0.0
  rotation_freq(:) = 0.0
  rotation_phase(:) = 0.0
  rotation_tphase(:) = 0.0
  rotation_amplitude(:) = 0.0
  rotation_origin_x(:) = 0.0
  rotation_origin_y(:) = 0.0
  rotation_origin_z(:) = 0.0
  rotation_vector_x(:) = 0.0
  rotation_vector_y(:) = 1.0
  rotation_vector_z(:) = 0.0
  rotation_start(:) = 0.0
  rotation_duration(:) = 1.0e99
  translate(:) = 0
  translation_rate(:) = 0.0
  translation_freq(:) = 0.0
  translation_phase(:) = 0.0
  translation_tphase(:) = 0.0
  translation_amplitude(:) = 0.0
  translation_vector_x(:) = 0.0
  translation_vector_y(:) = 0.0
  translation_vector_z(:) = 1.0
  translation_start(:) = 0.0
  translation_duration(:) = 1.0e99
/

  rotate(:) = 0
  This is the type of rotational motion; the array index corresponds to
  body number:
  ‘0’ for no rotation.
  ‘1’ for constant rotation rate, rotation_rate.
  ‘2’ is sinusoidal rotation where \( \theta = \text{rotation_amplitude} \sin(2\pi \text{rotation_freq} t + \text{rotation_phase} \pi/180) \) and \( t \) is nondimensional.

  rotation_rate(:) = 0.0
  This is the nondimensional rotation rate associated with rotate=1; the
  array index corresponds to body number.

  rotation_freq(:) = 0.0
  This is the nondimensional rotation reduced frequency associated with
  rotate=2; the array index corresponds to body number.
rotation_phase(:) = 0.0
This is the rotation phase shift (in degrees) associated with rotate=2; the array index corresponds to body number.

rotation_tphase(:) = 0.0
This is the rotation phase shift (in degrees) applied to the transform matrix; the array index corresponds to body number.

rotation_amplitude(:) = 0.0
This is the rotation amplitude (in degrees) associated with rotate=2; the array index corresponds to body number.

rotation_origin.x(:) = 0.0
This is the $x$-coordinate of rotation center; the array index corresponds to body number.

rotation_origin.y(:) = 0.0
This is the $y$-coordinate of rotation center; the array index corresponds to body number.

rotation_origin.z(:) = 0.0
This is the $z$-coordinate of rotation center; the array index corresponds to body number.

rotation_vector.x(:) = 0.0
This is the $x$-component of a unit vector along the rotation axis; the array index corresponds to body number.

rotation_vector.y(:) = 1.0
This is the $y$-component of a unit vector along the rotation axis; the array index corresponds to body number.

rotation_vector.z(:) = 0.0
This is the $z$-component of a unit vector along the rotation axis; the array index corresponds to body number.

rotation_start(:) = 0.0
This is the nondimensional time at which the rotational motion begins; the array index corresponds to body number.

rotation_duration(:) = 1.0e99
This is the nondimensional time over which the rotational motion is active; the array index corresponds to body number. After this time the rotational motion is zeroed out.
translate($) = 0$
This is the type of translational motion; the array index corresponds to body number:

‘0’ for no translation.

‘1’ for a constant translation rate, translation_rate.

‘2’ is sinusoidal translation where displacement = translation_amplitude sin($2\pi$ translation_freq t + translation_phase $\pi/180$) and t is nondimensional.

translation_rate($) = 0.0$
This is the nondimensional translation rate associated with translate=1; the array index corresponds to body number.

translation_freq($) = 0.0$
This is the nondimensional translation reduced frequency associated with translate=2; the array index corresponds to body number.

translation_phase($) = 0.0$
This is the translation phase shift (in degrees) associated with translate=2; the array index corresponds to body number.

translation_tphase($) = 0.0$
This is the translation phase shift (in degrees) applied to transform matrix; the array index corresponds to body number.

translation_amplitude($) = 0.0$
This is the translation amplitude (in grid units) associated with translate=2; the array index corresponds to body number.

translation_vector_x($) = 0.0$
This is the $x$-component of a unit vector along the translation axis; the array index corresponds to body number.

translation_vector_y($) = 0.0$
This is the $y$-component of a unit vector along the translation axis; the array index corresponds to body number.

translation_vector_z($) = 1.0$
This is the $z$-component of a unit vector along the translation axis; the array index corresponds to body number.
\text{translation\_start(:)} = 0.0

This is the nondimensional start time of the translational motion; the array index corresponds to body number.

\text{translation\_duration(:)} = 1.0e99

This is the nondimensional duration of the translational motion; the array index corresponds to body number.
B.5.3 &observer_motion

This namelist specifies motion of an observer as a function of time for boundary animation purposes (see the &boundary_output_variables namelist for more details); the animation is output from the point of view of the observer.

&observer_motion
  ob_parent_name = ' '
  ob_rotate = 0
  ob_rotation_rate = 0.0
  ob_rotation_freq = 0.0
  ob_rotation_phase = 0.0
  ob_rotation_tphase = 0.0
  ob_rotation_amplitude = 0.0
  ob_rotation_origin_x = 0.0
  ob_rotation_origin_y = 0.0
  ob_rotation_origin_z = 0.0
  ob_rotation_vector_x = 0.0
  ob_rotation_vector_y = 1.0
  ob_rotation_vector_z = 0.0
  ob_translate = 0
  ob_translation_rate = 0.0
  ob_translation_freq = 0.0
  ob_translation_phase = 0.0
  ob_translation_tphase = 0.0
  ob_translation_amplitude = 0.0
  ob_translation_vector_x = 0.0
  ob_translation_vector_y = 0.0
  ob_translation_vector_z = 1.0
/

ob_parent_name = ''
This is the observer parent reference frame. The default indicates the inertial reference frame - the same as when observer motion is not explicitly specified.

ob_rotate = 0
This is the type of rotational motion:
‘0’ for no rotation.
‘1’ for a constant rotation rate, ob_rotation_rate.
‘2’ is sinusoidal rotation where \( \theta = ob\_rotation\_amplitude \sin(2\pi \ ob\_rotation\_freq \ t + ob\_rotation\_phase \ \pi/180) \) and \( t \) is nondimensional.

ob_rotation_rate = 0.0
This is the nondimensional rotation rate associated with rotate=1.
**ob_rotation_freq = 0.0**
This is the nondimensional rotation reduced frequency associated with rotate=2.

**ob_rotation_phase = 0.0**
This is the rotation phase shift (in degrees) associated with rotate=2.

**ob_rotation_tphase = 0.0**
This is the rotation phase shift (in degrees) applied to transform matrix.

**ob_rotation_amplitude = 0.0**
This is the rotation amplitude (in degrees) associated with rotate=2.

**ob_rotation_origin_x = 0.0**
This is the x-coordinate of rotation center.

**ob_rotation_origin_y = 0.0**
This is the y-coordinate of rotation center.

**ob_rotation_origin_z = 0.0**
This is the z-coordinate of rotation center.

**ob_rotation_vector_x = 0.0**
This is the x-component of a unit vector along the rotation axis.

**ob_rotation_vector_y = 1.0**
This is the y-component of a unit vector along the rotation axis.

**ob_rotation_vector_z = 0.0**
This is the z-component of a unit vector along the rotation axis.

**ob_translate = 0**
This is the type of translational motion:

‘0’ for no translation

‘1’ for constant translation rate, ob_translate_rate.

‘2’ is sinusoidal translation where displacement = ob_translation_amplitude sin(2\pi ob_translation_freq t + ob_translation_phase pi/180) and t is nondimensional.

**ob_translate_rate = 0.0**
This is the nondimensional translation rate associated with translate=1.
\texttt{ob\_translation\_freq = 0.0}  
This is the nondimensional translation reduced frequency associated with \texttt{translate=2}.

\texttt{ob\_translation\_phase = 0.0}  
This is the translation phase shift (in degrees) associated with \texttt{translate=2}.

\texttt{ob\_translation\_tphase = 0.0}  
This is the translation phase shift (in degrees) applied to transform matrix.

\texttt{ob\_translation\_amplitude = 0.0}  
This is the translation amplitude (in grid units) associated with \texttt{translate=2}.

\texttt{ob\_translation\_vector\_x = 0.0}  
This is the $x$-component of a unit vector along the translation axis.

\texttt{ob\_translation\_vector\_y = 0.0}  
This is the $y$-component of a unit vector along the translation axis.

\texttt{ob\_translation\_vector\_z = 1.0}  
This is the $z$-component of a unit vector along the translation axis.
B.5.4 &motion_from_file

This namelist specifies rigid body (and grid) motion via a file containing a 4x4 transform matrix as a function of time. It allows user-defined motion that is more general than the motions available in the &forced_motion namelist.

```
&motion_from_file
  n_time_slices_file(:) = 0
  repeat_time_file(:) = 1.0e+99
  motion_file(:) = ''
  motion_file_type(:) = 'transform_matrix'
/
```

```
n_time_slices_file(:) = 0
```
This is the number of transforms (at selected points in time) defining the motion of the body; the array index corresponds to body number. All the transforms for a particular body are contained in a single file.

```
repeat_time_file(:) = 1.0e+99
```
This is the nondimensional time when the motion will repeat; the array index corresponds to body number.

```
motion_file(:) = ''
```
This is the name of the ASCII file that contains the transform matrices used to set the grid position and orientation of the body for each of the specified instants in time; the array index corresponds to body number. The following pseudo code illustrates how such a motion file might be created:

```
loop over time steps
  write() simulation_time
  write() xcg, ycg, zcg
  do i=1,4
    write() transform_matrix(i,j), j=1,4
  end do
end time step loop
```
where `simulation_time` is the nondimensional time, and where `xcg, ycg, zcg` are the coordinates of the center of gravity of the body in grid units.

```
motion_file_type(:) = 'transform_matrix'
```
This is the type of transform matrix specified; the array index corresponds to body number:

`'transform_matrix'` specifies the transform from inertial coordinates to body (moving) coordinates as a matrix.
‘inverse_transform’ specifies the transform from body (moving) coordinates to inertial coordinates as a matrix.
This namelist allows the motion of surface grids to be defined in files. Since only the surface is specified, `mesh_movement = 'deform'` must be used to deform the volume mesh to conform to the specified surface. These files must be named `[project_rootname]_bodyN_timeM.dat`, where N is the body number and M is the time slice index (1 to `n_time_slices`). The files are ASCII Tecplot files with a zone title line that contains “TIME simulation_time”, where `simulation_time` is nondimensional time. The variables in the file are the values of `x`, `y`, `z`, as well as `id`, where `id` is the global grid number of the surface point.

```verbatim
&surface_motion_from_file
  n_time_slices(:) = 0
  repeat_time(:) = 1.0e+99
/

  n_time_slices(:) = 0

  This is the number of equally spaced instants in time (and files describing the shape at those times) defining the motion of the body; the array index corresponds to body number. Each file contains the surface shape at a point in time.

  repeat_time(:) = 1.0e+99

  This is the nondimensional time when the motion will repeat; the array index corresponds to body number.
```
B.5.6 &sixdof_motion

This namelist provides details of 6-DOF motion simulations. It requires linking to the 6-DOF library, see section A.7.5. NOTE: most data in this namelist is input as *dimensional* data. For 6-DOF motion, the variables *ref velocity*, *ref density* and *ref length* in the namelist &body definitions must also be set, in units consistent with those used in &sixdof_motion.

&sixdof_motion

mass(:) = 1.0

cg_x(:) = 0.0

cg_y(:) = 0.0

cg_z(:) = 0.0

i_xx(:) = 1.0

i_yy(:) = 1.0

i_zz(:) = 1.0

i_xy(:) = 0.0

i_xz(:) = 0.0

i_yz(:) = 0.0

body_lin_vel(:,:) = 0.0

body_ang_vel(:,:) = 0.0

euler_ang(:,:) = 0.0

gravity_dir(1:3) = 0.0, 0.0, -1.0

gravity_mag = 32.2

n_extforce(:) = 0

n_extmoment(:) = 0

file_extforce(:,:) = ''

file_extmoment(:,:) = ''

/  

mass(:) = 1.0

This is the mass of the body; the array index corresponds to the body number.

cg_x(:) = 0.0

This is the x-coordinate of the center of gravity; the array index corresponds to the body number.

cg_y(:) = 0.0

This is the y-coordinate of the center of gravity; the array index corresponds to the body number.

cg_z(:) = 0.0

This is the z-coordinate of the center of gravity; the array index corresponds to the body number.
\( i_{xx}(:) = 1.0 \)
This is the moment of inertia about the \( x \) axis as the body rotates about the \( x \) axis; the array index corresponds to the body number.

\( i_{yy}(:) = 1.0 \)
This is the moment of inertia about the \( y \) axis as the body rotates about the \( y \) axis; the array index corresponds to the body number.

\( i_{zz}(:) = 1.0 \)
This is the moment of inertia about the \( z \) axis as the body rotates about the \( z \) axis; the array index corresponds to the body number.

\( i_{xy}(:) = 0.0 \)
This is the moment of inertia about the \( x \) axis as the body rotates about the \( y \) axis; the array index corresponds to the body number.

\( i_{xz}(:) = 0.0 \)
This is the moment of inertia about the \( x \) axis as the body rotates about the \( z \) axis; the array index corresponds to the body number.

\( i_{yz}(:) = 0.0 \)
This is the moment of inertia about the \( y \) axis as the body rotates about the \( z \) axis; the array index corresponds to the body number.

\( \text{body_lin_vel}(::,:) = 0.0 \)
These are the components of linear velocity; The first array index (ranging from 1 to 3) corresponds to the \( x \), \( y \), and \( z \) components; The second array index corresponds to the body number.

\( \text{body_ang_vel}(::,:) = 0.0 \)
These are the components of angular velocity (degrees/sec); The first array index (ranging from 1 to 3) corresponds to the \( x \), \( y \), and \( z \) components; The second array index corresponds to the body number.

\( \text{euler_ang}(::,:) = 0.0 \)
These are the Euler angles (degrees); The first array (ranging from 1 to 3) corresponds to the rotation angle components; the second array index corresponds to the body number.

\( \text{gravity_dir}(1:3) = 0.0, 0.0, -1.0 \)
This is a unit length gravity vector.

\( \text{gravity_mag} = 32.2 \)
This is the magnitude of the gravity vector (default units: \( ft/sec^2 \)).
\texttt{n\_extforce(,:) = 0}

This is the number of imposed external forces, excluding gravity. The array index corresponds to the body number.

\texttt{n\_extmoment(,:) = 0}

This is the number of imposed external moments; the array index corresponds to the body number.

\texttt{file\_extforce(,:,:) = ''}

This file specifies the external forces; the first array (ranging from 1 to \texttt{n\_extforce}) corresponds to the external force number. The second array index corresponds to the body number.

\texttt{file\_extmoment(,:,:) = ''}

This file specifies the external moments; the first array (ranging from 1 to \texttt{n\_extforce}) corresponds to the external force number. The second array index corresponds to the body number.
B.5.7 &aeroelastic_modal_data

This namelist specifies modal data for static and dynamic aeroelastic analysis via time integration of the structural dynamics equations (see for example [43]).

```plaintext
&aeroelastic_modal_data
  plot_modes = .false.
  nmode(:, ) = 0
  grefl(:, ) = 1.0
  uinf(:, ) = 0.0
  qinf(:, ) = 0.0
  gdisp0(:, :) = 0.0
  gvel0(:, :) = 0.0
  gforce0(:, :) = 0.0
  gmass(:, :) = 0.0
  freq(:, :) = 0.0
  damp(:, :) = 0.0
  moddfl(:, :) = 0
  moddfl_amp(:, :) = 0.0
  moddfl_freq(:, :) = 0.0
  moddfl_t0(:, :) = 0.0
  moddfl_add(:, :) = 0
  mode_file_format = 'ascii'
/
plot_modes = .false.
```

This generates tecplot files of each mode shape added to the body surface. These can be inspected these to insure the validity of input modal surface data.

```plaintext
nmode(:, ) = 0
```

This is the number of aeroelastic modes used to represent the structural deformation; the array index indicates the body number.

```plaintext
grefl(:, ) = 1.0
```

This is the scaling factor between CFD grid units and the structural dynamics equation units; the array index indicates the body number.

```plaintext
uinf(:, ) = 0.0
```

This is the freestream velocity, in structural dynamics equation units; the array index indicates the body number.

```plaintext
qinf(:, ) = 0.0
```

This is the freestream dynamic pressure, in structural dynamics equation units; the array index indicates the body number.
\( \text{gdisp0}(:, :) = 0.0 \)

This is the generalized displacement of a specified mode at the starting time step. It is used to perturb a mode to excite a dynamic response. The first array index indicates the mode number and the second array index indicates the body number.

\( \text{gvel0}(:, :) = 0.0 \)

This is the generalized velocity of a specified mode at the starting time step. It is used to perturb a mode to excite a dynamic response. The first array index indicates the mode number and the second array index indicates the body number.

\( \text{gforce0}(:, :) = 0.0 \)

This is the generalized force of a specified mode at the starting time step. It is used to perturb a mode to excite a dynamic response. The first array index indicates the mode number and the second array index indicates the body number.

\( \text{gmass}(:, :) = 0.0 \)

This is the generalized mass of a specified mode. The first array index indicates the mode number and the second array index indicates the body number.

\( \text{freq}(:, :) = 0.0 \)

This is the frequency of specified mode, in rad/sec. The first array index indicates the mode number and the second array index indicates the body number.

\( \text{damp}(:, :) = 0.0 \)

This is the damping ratio of specified mode. The first array index indicates the mode number and the second array index indicates the body number.

\( \text{moddf1}(:, :) = 0 \)

This is the type of time-varying mode perturbation. The first array index indicates the mode number and the second array index indicates the body number.

‘-1’ is the modal displacement and velocity set to zero.
‘0’ is no modal perturbation.
‘1’ is a harmonic modal oscillation.
‘2’ is a Gaussian pulse modal deflection.
‘3’ is a step pulse modal deflection.
‘5’ specifies simultaneous inputs for reduced order model.

\[ \text{moddfl\_amp}(::) = 0.0 \]

This is the amplitude of mode perturbation. The first array index indicates the mode number and the second array index indicates the body number.

\[ \text{moddfl\_freq}(::) = 0.0 \]

This is the frequency of mode perturbation. If \( \text{moddfl}=2 \), it is the Gaussian pulse half-life. The first array index indicates the mode number and the second array index indicates the body number.

\[ \text{moddfl\_t0}(::) = 0.0 \]

If \( \text{moddfl}=1 \), this is the dimensional time at which the sinusoidal perturbation starts. If \( \text{moddfl}=2 \), this is the dimensional time of the center of the Gaussian pulse. If \( \text{moddfl}=3 \), this is the start time of a step pulse. The first array index indicates the mode number and the second array index indicates the body number.

\[ \text{moddfl\_add}(::) = 0 \]

This determines how the modal perturbation is applied. The first array index indicates the mode number and the second array index indicates the body number.

‘0’ replaces the perturbation with the static aeroelastic solution.

‘1’ adds the perturbation to an existing static aeroelastic solution (if one exists).

\[ \text{mode\_file\_format} = \text{'ascii'} \]

This indicates the type and format of the mode-shape files that are read by the code. \( \text{mode\_file\_format} = \text{'ascii'} \) indicates the files are ASCII Tecplot files, with a file extension .dat; \( \text{mode\_file\_format} = \text{'stream'} \) indicates binary (stream) DDF files, with a file extension .ddfb.
B.5.8 &composite_overset_mesh

This namelist provides the input for SUGGAR++ (see the documentation supplied with SUGGAR++ for details).

&composite_overset_mesh
  input_xml_file = ''
/

  input_xml_file = ''

This is the file containing the XML commands for SUGGAR++. Specify the same Input.xml file that was used to generate the initial composite grid with the “stand-alone” SUGGAR++ code.
B.5.9 &grid_transform

This namelist defines a constant grid translation and rotation that is applied before the start of the flow solution. For example, original grid may be re-oriented to position the geometry at a different angle of attack. Using this namelist requires the command-line option --grid_transform.

```plaintext
&grid_transform
  ds    = 0.0
  sx    = 1.0
  sy    = 0.0
  sz    = 0.0
  theta = 0.0
  tx    = 1.0
  ty    = 0.0
  tz    = 0.0
  x0    = 0.0
  y0    = 0.0
  z0    = 0.0
  scale = 1.0
  transform(1,1:4) = 1.0, 0.0, 0.0, 0.0
  transform(2,1:4) = 0.0, 1.0, 0.0, 0.0
  transform(3,1:4) = 0.0, 0.0, 1.0, 0.0
  transform(4,1:4) = 0.0, 0.0, 0.0, 1.0
/
  ds = 0.0
  This is the translation distance.
  sx = 1.0
  This is the x-component of a unit vector in the translation direction.
  sy = 0.0
  This is the y-component of a unit vector in the translation direction.
  sz = 0.0
  This is the z-component of a unit vector in the translation direction.
  theta = 0.0
  This is the rotation angle (in degrees). A positive rotation is applied by the right hand rule with the thumb pointing in direction of rotation axis.
  tx = 1.0
  This is the x-component of the rotation axis unit vector.
  ty = 0.0
  This is the y-component of the rotation axis unit vector.
```

247
\[ tz = 0.0 \]
This is the \( z \)-component of the rotation axis unit vector.

\[ x0 = 0.0 \]
This is the \( x \)-coordinate of the rotation origin.

\[ y0 = 0.0 \]
This is the \( y \)-coordinate of the rotation origin.

\[ z0 = 0.0 \]
This is the \( z \)-coordinate of the rotation origin.

\[ \text{scale} = 1.0 \]
This a scale factor applied to all coordinate values.

\[ \text{transform(1,1:4)} = 1.0, 0.0, 0.0, 0.0 \]
This is a 4x4 transform matrix (see for example [44]).
B.6 *rotor.input*

FUN3D is capable of modeling a rotating blade system using different levels of approximation. In order of increasing complexity/fidelity/cost, rotor systems may be analyzed using either a time-averaged actuator disk, or via first principles modeling of the moving, articulated, rotor blades using overset, moving grids. The actuator method utilizes momentum/energy source terms to represent the influence of the rotating blade system. Use of the source terms simplifies grid generation, since the actuator surfaces do not need to be built into the computational grid. However, the computational grid should have some refinement in the vicinity of the actuator surfaces to obtain accurate results. The steady-state actuator disk capability was originally implemented by Dave O’Brien, at the time a PhD candidate at Georgia Tech. [45] O’Brien also initiated the overset capability in FUN3D which was later extended and coupled to a rotorcraft comprehensive code by Biedron et al. [46]

The rotor.input file is used primarily for specifying input quantities related to an actuator surface model for rotor/propeller combinations. When using overset, moving grids and/or coupling Fun3D to a rotorcraft comprehensive code for a more detailed simulation, a limited number of the input fields in the rotor.input file are also required. The fields required for coupled rotorcraft simulations include *(required for coupled simulation)* in the variable description. The command line option *--rotor* is required for both types of analysis.

Fun3D can also use the actuator disk library developed by Dave O’Brien for the Department of Defense HI-ARMS/CREATE/HELIOS project, Software Module for Engineering Methods of Rotor Dynamics (SMEMRD). The Fun3D team is unable to provide technical support for SMEMRD; please contact Dave O’Brien directly for assistance (David.ObrienJr@us.army.mil). SMEMRD adds the ability to trim to thrust values and use airfoil lookup tables. The *--hiarms_rotor* command line option activates the SMEMRD model.

The two parameters used to set the flight condition and force/moment coefficient normalization in compressible rotorcraft simulations are *mach_number* in *fun3d.nml* and *Vinf_Ratio* in *rotor.input*. To nondimensionalize the forces with the rotor tip velocity, set *mach_number* to the tip mach number and *Vinf_Ratio* to the ratio of freestream velocity to rotor tip velocity (the helicopter advance ratio). When *mach_number* is the tip mach number then *reynolds_number* should be set to the corresponding tip Reynolds number. To nondimensionalize the forces with the freestream velocity, set *mach_number* to the freestream mach number and *Vinf_Ratio* to one. The *Vinf_Ratio* will still affect the force nondimensionalization as described above, for incompressible solutions.

A sample *rotor.input* file is shown below for a conventional main rotor and tail rotor helicopter.
# Rotors Vinf_Ratio Write Soln Force_ref Moment_ref
2 0.1 50 1.0 1.0

=== Main Rotor =============================================

<table>
<thead>
<tr>
<th>Rotor Type</th>
<th>Load Type</th>
<th># Radial</th>
<th># Normal</th>
<th>Tip Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>X0_rotor</td>
<td>Y0_rotor</td>
<td>Z0_rotor</td>
<td>phi1</td>
<td>phi2</td>
</tr>
<tr>
<td>Vt_Ratio</td>
<td>ThrustCoff</td>
<td>PowerCoff</td>
<td>psi0</td>
<td>PitchHinge</td>
</tr>
<tr>
<td>LiftSlope</td>
<td>alpha, L=0</td>
<td>cd0</td>
<td>cd1</td>
<td>cd2</td>
</tr>
<tr>
<td>CL_max</td>
<td>CL_min</td>
<td>CD_max</td>
<td>CD_min</td>
<td>Swirl</td>
</tr>
<tr>
<td>1.50</td>
<td>-1.50</td>
<td>1.50</td>
<td>-1.50</td>
<td>0</td>
</tr>
<tr>
<td>Theta0</td>
<td>ThetaTwist</td>
<td>Theta1s</td>
<td>Theta1c</td>
<td>Pitch-Flap</td>
</tr>
<tr>
<td>Theta0</td>
<td>ThetaTwist</td>
<td>Theta1s</td>
<td>Theta1c</td>
<td>Pitch-Flap</td>
</tr>
</tbody>
</table>

### Tail Rotor =============================================

<table>
<thead>
<tr>
<th>Rotor Type</th>
<th>Load Type</th>
<th># Radial</th>
<th># Normal</th>
<th>Tip Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>X0_rotor</td>
<td>Y0_rotor</td>
<td>Z0_rotor</td>
<td>phi1</td>
<td>phi2</td>
</tr>
<tr>
<td>Vt_Ratio</td>
<td>ThrustCoff</td>
<td>PowerCoff</td>
<td>psi0</td>
<td>PitchHinge</td>
</tr>
<tr>
<td>LiftSlope</td>
<td>alpha, L=0</td>
<td>cd0</td>
<td>cd1</td>
<td>cd2</td>
</tr>
<tr>
<td>CL_max</td>
<td>CL_min</td>
<td>CD_max</td>
<td>CD_min</td>
<td>Swirl</td>
</tr>
<tr>
<td>1.50</td>
<td>-1.50</td>
<td>1.50</td>
<td>-1.50</td>
<td>1</td>
</tr>
<tr>
<td>Theta0</td>
<td>ThetaTwist</td>
<td>Theta1s</td>
<td>Theta1c</td>
<td>Pitch-Flap</td>
</tr>
</tbody>
</table>

The header line is where the user specifies the number of rotors, the rotor advance ratio, and how often to output the plot3d loading file. The remainder of the file is in a block structure, where each block represents the inputs for one rotor. The first line of each block is a text line that can be edited to keep the rotors organized for the user. The input values do not have to be in a fixed format (spaces and number of decimal points do not matter), but the input values do have to be in the correct order as noted by the header lines for the individual input parameters.
B.6.1 Header

# Rotors (*required for coupled simulation*)

This is the number of actuator surfaces to create. The number of rotor sections must match the number of rotors specified.

\textit{Vinf\_Ratio} (*required for coupled simulation*)

This is the ratio of the freestream velocity to the velocity used for force normalization. This velocity used for force normalization is typically the tip velocity for rotorcraft applications; in this case, \textit{Vinf\_Ratio} is the advance ratio and \texttt{mach\_number} is the freestream velocity.

\textit{Write\_Soln}

This is the frequency of Plot3D rotor loading data output, in iterations. To write once, set \textit{Write\_Soln} to \texttt{steps}.

\texttt{Force\_ref}

This is the conversion factor to obtain forces in alternate units,

1.0 will output the standard FUN3D nondimensionalization

\[
\frac{(L^2_{\text{ref}} a^2_{\text{ref}})}{(\pi R^2_{\text{rotor}} V^2_{\text{tip}})} \text{ will output standard rotorcraft nondimensionalization}
\]

\[
\rho_{\text{ref}} a^2_{\text{ref}} L^2_{\text{ref}} \text{ will output dimensional units}
\]

\texttt{Moment\_ref}

This is the conversion factor to obtain moments in alternate units.

1.0 will output the standard FUN3D nondimensionalization

\[
\frac{(L^3_{\text{ref}} a^3_{\text{ref}})}{(\pi R^3_{\text{rotor}} V^2_{\text{tip}})} \text{ will output standard rotorcraft nondimensionalization}
\]

\[
\rho_{\text{ref}} a^2_{\text{ref}} L^3_{\text{ref}} \text{ will output dimensional units}
\]

B.6.2 Actuator Surface Model

\texttt{Rotor\_Type}

Type of rotor model to apply,

1 models the rotor as an actuator disk.

2 models the rotor as actuator blades [Future Expansion].

\texttt{Load\_Type}

Type of loading to apply to the rotor model,

1 is constant pressure jump base on \texttt{ThrustCoeff}.
2 is linearly increasing pressure jump base on ThrustCoeff.

3 is blade element based loading based defined by the blade element parameters set in rotor.input file i.e. the blade lift/drag curves and motion.

# Radial
This is the number of sources to distribute along the blade radius. A suggested value is 100.

# Normal
This is the number of sources to distribute along the circumference. For Rotor Type=1, 720 is suggested for a source every 0.5 degrees. For Rotor Type=2, 20 is suggested.

Tip Weight
This is the hyperbolic weighting factor for distributing sources along the blade radius. A suggested value is 0.0, which yields uniform distribution. A value larger than 2.0 concentrates too many sources at the blade tip.

B.6.3 Rotor Reference System

X0_rotor (required for coupled simulation)
This is the x coordinate of the hub center of rotation, in grid units.

Y0_rotor (required for coupled simulation)
This is the y coordinate of the hub center of rotation, in grid units.

Z0_rotor (required for coupled simulation)
This is the z coordinate of the hub center of rotation, in grid units.

phi1
This is the first Euler angle describing a rotation about the x axis, in degrees.

phi2
This is the second Euler angle describing a rotation about the a2 axis, in degrees.

phi3
This is the third Euler angle describing a rotation about the b3 axis, in degrees.
The Euler angles must be input correctly to obtain the correct orientation of the source based actuator disk. The following example illustrates how to determine these angles. Figure B4 depicts the rotations $\phi_1 = 10$, $\phi_2 = -15$, and $\phi_3 = 15$. Initially, the thrust is assumed to be in the $z$ direction and the disk is located in the $x-y$ plane. The first rotation of $\phi_1$ about the $x$ axis takes the $x-y-z$ system to the $a_1-a_2-a_3$ system shown in red. The second rotation of $\phi_2$ about the $a_2$ axis takes the $a_1-a_2-a_3$ system to the $b_1-b_2-b_3$ system shown in green. The final rotation of $\phi_3$ about the $b_3$ axis takes the $b_1-b_2-b_3$ system to the rotor reference system shown in blue. The black circle represents the initial disk orientation and the blue circle represents the final disk orientation. In general $\phi_1$ and $\phi_2$ are sufficient to define the thrust orientation. The variable $\phi_3$ only changes the location of the zero azimuth angle definition for the rotor.

![Figure B4: Rotor disk Euler angles.](image)

### B.6.4 Rotor Loading

**Vt\_Ratio (required for coupled simulation)**

This is the ratio of the tip speed to the velocity used for force normalization. If the force normalization velocity is freestream, then $Vt\_Ratio = 1/(\text{advance ratio})$.

**ThrustCoff**

This is the rotor thrust coefficient defined as, $C_T = \text{Thrust} / (\pi \rho_{\text{ref}} R^2 (\Omega_{\text{Dim}} R)^2)$, when $\text{Load Type}=1$ or $\text{Load Type}=2$. The blade element model does not trim to specified thrust coefficient.

**PowerCoff**

This is the rotor power coefficient [Not implemented].

253
B.6.5 Blade Parameters

\textit{psi0 (required for coupled simulation)}

This is the initial azimuthal position of blade one, in degrees; the azimuth position is defined as zero when the blade is oriented along the \(x\)-axis with the tip at the most positive \(x\) location.

\textit{PitchHinge (required for coupled simulation)}

This is the radial position of the blade pitch hinge normalized by tip radius.

\textit{DirRot (required for coupled simulation)}

This is the direction of rotor rotation. Zero is counter-clockwise rotation and one is clockwise rotation.

\# Blades (required for coupled simulation)

This is the number of rotor blades. It is only used for Load Type=3 and overset rotor simulations.

\textit{TipRadius (required for coupled simulation)}

This is the radius of the blade, in grid units.

\textit{RootRadius (required for coupled simulation)}

This is the radius of the blade root, in grid units. It accounts for the cutout region.

\textit{BladeChord (required for coupled simulation)}

This is the chord length of the blade, in grid units. It can only handle rectangular blade planforms and is only valid for Load Type=3.

\textit{FlapHinge (required for coupled simulation)}

This is the radial position of the blade flap hinge normalized by tip radius.

\textit{LagHinge (required for coupled simulation)}

This is the radial position of the blade lag hinge normalized by tip radius.

B.6.6 Blade Element Parameters for Load Type=3

These inputs are used to set the blade element lift and drag curves according to:

\[ C_L = \text{LiftSlope}(\alpha - \alpha_{L=0}) \]  \hspace{1cm} (B1)

and

\[ C_D = cd0 + cd1 \alpha + cd2 \alpha^2 \]  \hspace{1cm} (B2)
**LiftSlope**
This is the lift curve slope per radian.

**alpha, L=0**
This is the zero lift angle of attack, in degrees.

**cd0, cd1, and cd2**
These are the quadratic drag polar coefficients; where cd1 is per radian and cd2 is per radian squared.

**CL\_max and CL\_min**
These limiters to control the lift coefficient beyond the linear region.

**CD\_max and CD\_min**
These limiters to control the drag coefficient.

**Swirl**
One includes the swirl inducing terms and zero neglects the sources terms that create rotor swirl.

**B.6.7 Pitch Control Parameters for Load Type=3**
These inputs are used to specify the pitch/flap controls according to:

\[
\theta = \text{Theta0} + \text{ThetaTwist} (r/R) + \text{Theta1c} \cos(\psi) + \text{Thetas} \sin(\psi) \quad (B3)
\]

**Theta0**
This is the collective pitch defined at \(r/R=0\), in degrees.

**ThetaTwist**
This is the total amount of linear blade twist from the origin to the blade tip, in degrees.

**Thetas**
This is the longitudinal cyclic pitch input, in degrees.

**Theta1c**
This is the lateral cyclic pitch input, in degrees.

**Pitch-Flap**
Pitch-Flap coupling parameter [not implemented].
B.6.8 Prescribed Flap Parameters

These inputs are used to specify the flap harmonics according to:

\[
\beta = \Beta0 + \Beta1s \sin(\psi) + \Beta1c \cos(\psi) \\
+ \Beta2s \sin(2\psi) + \Beta2c \cos(2\psi) \\
+ \Beta3s \sin(3\psi) + \Beta3c \cos(3\psi) \tag{B4}
\]

# FlapHar
This is the number of flap harmonics to include. The valid input range is zero to three.

**Beta0**
This is the coning angle, in degrees.

**Beta1s and Beta1c**
This is the first flap harmonics, in degrees.

**Beta2s and Beta2c**
This is the second flap harmonics, in degrees.

**Beta3s and Beta3c**
This is the third flap harmonics, in degrees.

B.6.9 Prescribed Lag Parameters

These inputs are used to specify the lag harmonics according to:

\[
\delta = \Delta0 + \Delta1s \sin(\psi) + \Delta1c \cos(\psi) \\
+ \Delta2s \sin(2\psi) + \Delta2c \cos(2\psi) \\
+ \Delta3s \sin(3\psi) + \Delta3c \cos(3\psi) \tag{B5}
\]

# LagHar
This is the number of lag harmonics to include. The valid input range is zero to three.

**Delta0**
This is the coning angle, in degrees.

**Delta1s and Delta1c**
This is the first lag harmonics, in degrees.

**Delta2s and Delta2c**
This is the second lag harmonics, in degrees.

**Delta3s and Delta3c**
This is the third lag harmonics, in degrees.
B.7 tdata

This file defines the gas model when eqn_type = 'generic'. A keyword is required on the first line of tdata. Many of these models require additional information as detailed in each keyword section.

Some keywords require a list the species. For these keywords, additional groups of species can be specified for boundary conditions after a blank line. If new species are introduced in subsequent instances their mass fractions are automatically initialized to zero at any previous inflow boundary. All the species entries in this file are available as reactants throughout the entire flow field.

B.7.1 perfect_gas Keyword

A perfect gas can be modeled with the perfect_gas keyword. The parameters can be explicitly defined in tdata by the namelist &species_properties. The namelist in tdata has different variables than the &species_properties in species thermo_data. Here is an example of the namelist with defaults that are all given in SI units,

```plaintext
perfect_gas
&species_properties
   gamma    = 1.4
   mol_wt   = 28.8
   suther1  = 0.1458205E-05
   suther2  = 110.333333
   prand    = 0.72
/
```

Where gamma is the gas specific heat ratio, mol_wt is the gas molecular weight, prand is the gas Prandtl number, and suther1 and suther2 are the first and second Sutherland’s viscosity coefficients, $s_1$ and $s_2$, in

$$
\mu = s_1 \frac{T^{3/2}}{T + s_2}
$$

(B6)

These defaults are used if the &species_properties namelist or any of its variables are omitted.

B.7.2 equilibrium_air_t Keyword

The equilibrium_air_t keyword engages the Tannehill curve fits for thermodynamic and transport properties of equilibrium air. [47] This keyword does not require additional lines.

```plaintext
equilibrium_air_t
```
B.7.3 **equilibrium_air** Keyword

The `equilibrium_air` keyword engages the Tannehill curve fits for transport properties and a table look-up for equilibrium gases [48]. This keyword does not require additional lines.

`equilibrium_air_r`

B.7.4 **one** Keyword

This one-temperature (1-T) model assumes that all the species are thermally in equilibrium state; the translational temperature $T$ and vibrational temperature $T_v$ are equal. This is a mixture of thermally perfect gases and multi-species transport. In this example, only molecular oxygen and nitrogen are present on the inflow boundary, but atomic nitrogen and oxygen and nitric oxide may be produced elsewhere in the flow field due to chemical reactions. The inflow boundary mass fraction of molecular oxygen and nitrogen is given next to their symbols. Mass fractions should sum to one.

```
one
N2 .767
N
O2 .233
O
NO
```

B.7.5 **two** Keyword

This two-temperature (2-T) model assumes that energy distribution in the translational and rotational modes of heavy particles (not electrons) are equilibrated at translational temperature $T$ and all other energy modes (vibrational, electronic, electron translational) are equilibrated at vibrational temperature $T_v$. In this example, the gas is assumed to be a mixture of 11 thermally perfect gases. The inflow boundary mass fraction of molecular oxygen and nitrogen is given next to their symbols. Mass fractions should sum to one. Other products are the results of chemical reactions flow field.

```
two
N2 .767
N
O2 .233
O
NO
O2+
O+
NO+
e-
```
B.7.6 FEM Keyword

This Free-Energy Minimization (FEM) model causes the species continuity equations to be replaced with elemental continuity equations and equilibrium relations for remaining species. In this example, the gas is assumed to be a mixture of 11 thermally perfect gases. The inflow boundary mass fraction of molecular oxygen and nitrogen is given next to their symbols. Mass fractions should sum to one. Other products are the results of chemical reactions flow field.

```
FEM
N2 .767
N
O2 .233
O
NO
O2+
O+
NO+
e-
```

B.8 species thermo data

The species thermo data file is the master file for species thermodynamic data. The majority of simulations do not require changes to this file. Investigating other sources of thermodynamic data is the only reason to edit this file. If the file is not found in the local run directory, it is assumed to be located in the 
[install-prefix]/share/physics_modules directory. See section A.3 for [install-prefix].

Each species record consists of the species name, a &species_properties namelist, the number of thermodynamic property curve fit ranges, and the curve fit coefficients for each range. [49] No blank line is allowed in this file. This &species_properties namelist has different variables than the &species_properties in tdata. The elements of the &species_properties namelist are:

- mol_wt
  This sets the molecular weight of the particle. It is always required.

- molecule = .false.
  This is denotes the the species a molecule (composed of more than one atom);

- ion = .false.
  This is denotes the the species a charged particle. Do not set it for neutrals and electrons. This will initializes electron-neutral energy exchange
cross section and sum of the charges.

**charge = 0**
This is an integer number to determine number of positive charges in particle. It should only be used with ion = .true.

**elec_impct_ion = -1_dp**
This sets the energy for neutrals ion=.false. that is required to liberate an electron when the neutral impact a free electron, in units of electron volts (eV).

**siga(:) = -1_dp**
This is an array of three real numbers, which correspond to the curve fit coefficients for electron-neutron energy exchange. The cross section is defined as

\[ \sigma_{en} = a + bT + cT^2 \]  \hspace{1cm} (B7)

where \( \sigma_{en} \) is the electron-neutron energy exchange collision cross section in \( m^2 \). The variables \( a, b, \) and \( c \) are the curve fit coefficients and \( T \) is the gas temperature. [50,51] For example, \( \text{siga}=7.5e-20, 0, 0 \).

**disoc_ener = 0_dp**
This is the dissociation energy of molecule in electron volts (eV).

**alantel = 0_dp**
This is the Landau-Teller constant to compute vibrational relaxation time for molecule. [52,53]. It is required when molecule=.true.

**cprt0 = 0_dp**
This non-dimensional real number defines translational-rotational heat capacity. It is normalized by the gas constant and is equal to

\[ cp(t) = \frac{f + 2}{2} \]  \hspace{1cm} (B8)

where \( f \) is the number of degrees of freedom in translation and rotation. The defaults for atoms and diatomic molecules are 2.5 and 3.5, respectively.

A portion of the `species thermo data` that provides thermodynamic properties of carbon is shown below.

```
C
&species_properties
molecule = .false.
ion = .false.
```
charge = 0
elec_impct_ion = 11.264
siga = 7.5e-20, 5.5e-24, -1.e-28
mol_wt = 12.01070

<p>| | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.64950315E+03</td>
<td>-0.96490109E+00</td>
<td>0.25046755E+01</td>
</tr>
<tr>
<td></td>
<td>0.19801337E-07</td>
<td>-0.16061440E-10</td>
<td>0.53144834E-14</td>
</tr>
<tr>
<td></td>
<td>0.8545761E+05</td>
<td>0.47479243E+01</td>
<td>200.000</td>
</tr>
<tr>
<td></td>
<td>-0.12891365E+06</td>
<td>0.17195286E+03</td>
<td>0.26460444E+01</td>
</tr>
<tr>
<td></td>
<td>0.19801337E-07</td>
<td>-0.16061440E-10</td>
<td>0.53144834E-14</td>
</tr>
<tr>
<td></td>
<td>0.8545761E+05</td>
<td>0.47479243E+01</td>
<td>200.000</td>
</tr>
<tr>
<td></td>
<td>-0.12891365E+06</td>
<td>0.17195286E+03</td>
<td>0.26460444E+01</td>
</tr>
<tr>
<td></td>
<td>0.19801337E-07</td>
<td>-0.16061440E-10</td>
<td>0.53144834E-14</td>
</tr>
<tr>
<td></td>
<td>0.8545761E+05</td>
<td>0.47479243E+01</td>
<td>200.000</td>
</tr>
<tr>
<td></td>
<td>-0.12891365E+06</td>
<td>0.17195286E+03</td>
<td>0.26460444E+01</td>
</tr>
<tr>
<td></td>
<td>0.19801337E-07</td>
<td>-0.16061440E-10</td>
<td>0.53144834E-14</td>
</tr>
<tr>
<td></td>
<td>0.8545761E+05</td>
<td>0.47479243E+01</td>
<td>200.000</td>
</tr>
<tr>
<td></td>
<td>-0.12891365E+06</td>
<td>0.17195286E+03</td>
<td>0.26460444E+01</td>
</tr>
<tr>
<td></td>
<td>0.19801337E-07</td>
<td>-0.16061440E-10</td>
<td>0.53144834E-14</td>
</tr>
<tr>
<td></td>
<td>0.8545761E+05</td>
<td>0.47479243E+01</td>
<td>200.000</td>
</tr>
<tr>
<td></td>
<td>-0.12891365E+06</td>
<td>0.17195286E+03</td>
<td>0.26460444E+01</td>
</tr>
</tbody>
</table>

The species name is defined in line 1. Between lines 2 and 9 species properties are defined. Line 10 shows that there are three thermodynamic property curve fits for temperature ranges of 200 K < T < 1,000 K, 1,000 K < T < 6,000 K, and 6,000 K < T < 20,000 K. Each data range consists of 12 real numbers. Four real numbers must be given on each line. The first 10 real numbers are the thermodynamic curve fit coefficients, and the last two real numbers identify the temperature range for the given curve fit coefficients. These coefficients are used to calculate the following thermodynamic properties

\[
c_{p}(T)/R = a_{1}T^{-2} + a_{2}T^{-1} + a_{3} + a_{4}T + a_{5}T^{2} + a_{6}T^{3} + a_{7}T^{4} \quad (B9)
\]

\[
h(T)/RT = -a_{1}T^{-2} + a_{2}T^{-1} \ln T + a_{3} + a_{4}T + a_{5}T^{2} + a_{6}T^{3} + \frac{a_{7}T^{4}}{4} + \frac{a_{9}T}{5} \quad (B10)
\]

\[
s(T)/R = -a_{1}\frac{T^{-2}}{2} - a_{2}T^{-1} + a_{3}\ln T + a_{4}T + a_{5}T^{2} + a_{6}T^{3} + \frac{a_{7}T^{4}}{4} + a_{10} \quad (B11)
\]

where \(T\) is the gas temperature, \(R\) is the universal gas constant, \(c_{p}\), \(h\), and \(s\) are the species specific heat, enthalpy and entropy, respectively, and \(a_{i}\) are the provided curve fit coefficients in species thermo data.

The following corrections will be applied if the gas temperature is out of the given range for the given curve fit coefficients:

\[
c_{p}(T) = c_{p}(T^{*}) \quad (B12)
\]

\[
h(T) = h(T^{*}) + (T - T^{*})c_{p}(T^{*}) \quad (B13)
\]

\[
s(T) = s(T^{*}) + \ln \frac{T}{T^{*}}c_{p}(T^{*}) \quad (B14)
\]

where

\[
T^{*} = \begin{cases} 
T_{\text{lower}} & \text{for } T < T_{\text{lower}} \\
T_{\text{upper}} & \text{for } T > T_{\text{upper}} 
\end{cases} \quad (B15)
\]
### B.9 kinetic_data

The kinetic_data file defines possible chemical reactions and is optional. If the file is not found in the local run directory, it is assumed to be located in the `[install-prefix]/share/physics_modules` directory. See section A.3 for [install-prefix]. Reactants and products can be any species defined in the `species thermo data` described in section B.8. A sample entry looks like

```
O2 + M <-> 20 + M
2.000e+21 -1.50 5.936e+04
teff1 = 2
exp1 = 0.7
t_eff_min = 1000.
t_eff_max = 50000.
C = 5.0
O = 5.0
N = 5.0
H = 5.0
Si = 5.0
e- = 0.
```

The first line specifies the reaction while line 2 provides three coefficients of an Arrhenius-like equation,

\[
K_f = c_f T_{\text{eff}}^\eta e^{-\epsilon_0/kT_{\text{eff}}}
\]  

(B16)

where \(c_f\) is the pre-exponential factor, \(\eta\) is the power of temperature dependence on the pre-exponential factor, \(\epsilon_0\) is the Arrhenius activation energy, and \(k\) is the Boltzmann constant. The arrowheads in line 1 signify the allowed directionality of the reaction. The symbol \(\Rightarrow\) denotes forward reaction only while \(\leftrightarrow\) denotes forward and backward rates are computed. The coefficients in line 2 correspond to \(c_f\), \(\eta\), and \(\epsilon_0/k\), respectively. For reactions with a generic collision partner, \(M\), such as this one, these coefficients correspond to Argon; and other collision partners and their efficiencies (multipliers of \(c_f\)) are specified on lines following line 5 and 6, which give the valid temperature range for the reaction. The effective temperature, \(T_{\text{eff}}\), is defined according to a given integer number in line 3.

\[
t_{\text{eff}1} = 1, t_{\text{eff}2} = 1
\]

This defines the formula to compute the effective temperature \(T_{\text{eff}}\) for the forward rate and backward rate, respectively. It is engaged for the case of thermal nonequilibrium. Options for \(t_{\text{eff}}\) are:

1: \(T_{\text{eff}} = T_{tr}\)

2: \(T_{\text{eff}} = T_{tr}^{\exp1} T_{v}^{1-\exp1}\)

262
3: \( T_{\text{eff}} = T_v \)

where \( T_{tr} \) and \( T_v \) are translational and vibrational temperatures, respectively.

\( \exp1 = 0.7 \)

This is the exponent used to define the effective temperature when \( \text{teff1} = 2 \) (forward rate) or \( \text{teff2} = 2 \) (backward rate).

\( r_{f\text{max}} = 1.e+20 \)

This is the upper limit on forward reaction rate in cgs units when \text{augment_kinetics_limiting} = .true. For unlimited rates as function of temperature, see the output file \text{kinetic_diagnostics_output}.

\( r_{f\text{min}} = 1.e-30 \)

This is the lower limit on forward reaction rate in cgs units when \text{augment_kinetics_limiting} = .true. For unlimited rates as function of temperature, see the output file \text{kinetic_diagnostics_output}.

\( r_{b\text{max}} = 1.e+30 \)

This is the upper limit on backward reaction rate in cgs units when \text{augment_kinetics_limiting} = .true. For unlimited rates as function of temperature, see the output file \text{kinetic_diagnostics_output}.

\( r_{b\text{min}} = 1.e-30 \)

This is the lower limit on backward reaction rate in cgs units when \text{augment_kinetics_limiting} = .true. For unlimited rates as function of temperature, see the output file \text{kinetic_diagnostics_output}.

\( t_{\text{eff}\text{min}} = 1000 \).

This is the minimum temperature for \( T_{\text{eff}} \). This may circumvent stiff source terms when computing reaction rates.

\( t_{\text{eff}\text{max}} = 50000 \).

The maximum temperature for \( T_{\text{eff}} \). This may circumvent stiff source terms when computing reaction rates.

\textbf{B.10 \ species\_transp\_data}

The \texttt{species\_transp\_data} file contains log-linear curve fit coefficients for species collision cross sections that are defined based on temperature range of 2,000–4,000 K. [51] This temperature range spans boundary-layer temperatures for typical hypersonic entry. The curve fit for the given coefficients is poor at temperatures below 1,000 K. Higher order curve fit data is available
in `species_transp_data_0`. The file should not be changed by the user unless there is a need to investigate other sources of collision cross-section data. If the file is not found in the local run directory, it is assumed to be located in the `[install-prefix]/share/physics_modules` directory. See section A.3 for `[install-prefix]`. An example of the entries in the file is

```
Ar Ar
Ar+ Ar+
-11.48 -12.08 -11.50 -12.10
Ar N2
Ar CO
```

### B.11 `species_transp_data_0`

The file `species_transp_data_0` provides collision cross section coefficients [54,55] for a higher order curve fit data than those that are in the `species_transp_data`. The data in `species_transp_data_0` supersedes the data in `species_transp_data` The file should not be changed by the user unless there is a need to investigate other sources of collision cross-section data. If the file is not found in the local run directory, it is assumed to be located in the `[install-prefix]/share/physics_modules` directory. See section A.3 for `[install-prefix]`. An example of the entries in the file is

```
O2 N
1 1 1 (c)
-1.1453028E-03 1.2654140E-02 -2.2435218E-01 7.7201588E+01
-1.0608832E-03 1.1782595E-02 -2.1246301E-01 8.4561598E+01
1.4943783E-04 -2.0389247E-03 1.8536165E-02 1.0476552E+00

NO N
1 1 1 (c)
-1.5770918E-03 1.9578381E-02 -2.7873624E-01 9.9547944E+01
-1.4719259E-03 1.8446968E-02 -2.6460411E-01 1.0911124E+02
2.1014557E-04 -3.0420763E-03 2.5736958E-02 1.0359598E+00

NO O
1 1 1 (c)
-1.0885815E-03 1.1883754E-02 -2.1844909E-01 7.5512560E+01
-1.0066279E-03 1.1029264E-02 -2.0671266E-01 8.2644384E+01
1.4145458E-04 -1.9249271E-03 1.7785767E-02 1.0482162E+00

END END 1 1 0
0. 0. 0. 0.
0. 0. 0. 0.
```
B.12  hara_namelist_data

The hara_namelist_data file controls the radiation models used by the HARA radiation module. [56,57] It is optional for coupled radiation simulations. If it is not present, then the code automatically chooses the radiative mechanisms associated with species present in the flowfield that have number densities greater than 1000 particles/cm$^2$) Other options are set to the defaults. For users not experienced in shock-layer radiation, the recommended default options should be used. this hara_namelist_data A default hara_namelist_data is available in the PHYSICS_MODULES directory of the Fun3D distribution.

specifying radiation mechanisms for atomic species: The treatment of radiation resulting from atomic lines, atomic bound-free, and free-free photoionization (referred to here as atomic continuum), and negative ion continuum is available for atomic carbon, hydrogen, oxygen, and nitrogen. These mechanisms are specified through the following binary flags (on=1/off=0). If any of these flags are not present in hara_namelist_data, then that flag is set to true only if the number density of the associated atomic species is greater than 1000 particles/cm$^2$ somewhere in the flowfield.

- **treat_c_lines**
  A binary flag to enable the treatment of atomic lines for species [c], where [c] can be c, h, n, and o, for atomic carbon, hydrogen, nitrogen and oxygen, respectively.

- **treat_h_cont**
  A binary flag to enable the treatment of atomic bound-free and free-free continuum for species [h], where [h] can be c, h, n, and o, for atomic carbon, hydrogen, nitrogen and oxygen, respectively.

- **treat_o_other**
  A binary flag to enable the treatment of the negative-ion continuum for species [o], where [o] can be c, h, n, and o, for atomic carbon, hydrogen, nitrogen and oxygen, respectively.

specifying radiation mechanisms for molecular species: The treatment of radiation resulting from numerous molecular band systems is available through the following flags (0 = off, 1 = SRB, 2 = LBL). The smeared rotational band (SRB) approach applies a simplified and efficient treatment of each molecular band system, which is accurate for optically thin conditions. The line-by-line (LBL) approach is a detailed but highly inefficient treatment of each molecular band system. The LBL option is not recommended for coupled radiation-flowfield computations, except for possibly the CO 4+ system.
which may be optically thick for Mars entry conditions. If any of these flags are not present in \textit{hara\_namelist\_data}, then that flag is set to the SRB option only if the number density of the associated molecular specie is greater than 1000 particles/cm$^2$ somewhere in the flowfield. Additional band systems are listed in the following paragraph. These additional band systems are generally considered negligible relative to those listed in this section. Therefore, for computational efficiency, they are not engaged by default. Definitions of each band system and the modeling data applied are discussed in Refs. [56,58].

\textbf{treat\_band\_c2\_swan}
A flag activating the C$_2$ Swan band system.

\textbf{treat\_band\_c2h}
A flag activating the C$_2$H band system.

\textbf{treat\_band\_c3}
A flag activating the C$_3$ and vacuum ultra-violet (VUV) band systems.

\textbf{treat\_band\_cn\_red}
A flag activating the CN red band system.

\textbf{treat\_band\_cn\_violet}
A flag activating the CN violet band system.

\textbf{treat\_band\_co4p}
A flag activating the CO 4+ band system.

\textbf{treat\_band\_co\_bx}
A flag activating the CO B-X band system.

\textbf{treat\_band\_co\_cx}
A flag activating the CO C-X band system.

\textbf{treat\_band\_co\_ex}
A flag activating the CO E-X band system.

\textbf{treat\_band\_co\_ir}
A flag activating the CO X-X band system.

\textbf{treat\_band\_h2\_lyman}
A flag activating the H$_2$ Lyman band system.

\textbf{treat\_band\_h2\_werner}
A flag activating the H$_2$ Werner band system.
treat_band_n2fp
A flag activating the N$_2$ 1+ band system.

treat_band_n2sp
A flag activating the N$_2$ 2+ band system.

treat_band_n2PFN
A flag activating the N$_2^+$ first-negative band system.

treat_band_n2_BH1
A flag activating the N$_2$ Birge-Hopfield I band system.

treat_band_n2_BH2
A flag activating the N$_2$ Birge-Hopfield II band system.

treat_band_NO_Beta
A flag activating the NO beta band system.

treat_band_NO_Delta
A flag activating the NO delta band system.

treat_band_NO_Epsilon
A flag activating the NO epsilon band system.

**additional molecular band systems:** This paragraph lists molecular band systems available in addition to those listed in the paragraph above. The band systems listed here are generally weak emitters and absorbers, and are therefore not engaged as a default. Therefore, for these band systems to be engaged, the following flags (0 = off, 1 = SRB, 2 = LBL) must be present in the hara_namelist_data file. The LBL treatment of these bands is not recommended.

treat_band_c2_br
A flag activating the C$_2$ Ballik-Ramsay band system.

treat_band_c2_da
A flag activating the C$_2$ Deslandres-d’Azambuja band system.

treat_band_c2_fh
A flag activating the C$_2$ Fox-Herzberg band system.

treat_band_c2_mulliken
A flag activating the C$_2$ Mulliken band system.
treat_band_c2_philip
A flag activating the C$_2$ Philips band system.

treat_band_co3p
A flag activating the CO 3+ band system.

treat_band_co_angstrom
A flag activating the CO angstrom band system.

treat_band_co_asundi
A flag activating the CO Asundi band system.

treat_band_co_triplet
A flag activating the CO triplet band system.

treat_band_co2
A flag activating the CO$_2$ band system. A value of two activates an approximate nonequilibrium model for UV emission, while a value of one assumes Boltzmann emission. The LBL treatment of this band is not available.

treat_band_n2_cy
A flag activating the N$_2$ Carrol-Yoshino band system.

treat_band_n2_wj
A flag activating the N$_2$ Worley-Jenkins band system.

treat_band_n2_worley
A flag activating the N$_2$ Worley band system.

treat_band_no_gamma
A flag activating the NO gamma band system.

treat_band_no_betap
A flag activating the NO beta-prime band system.

treat_band_no_gammap
A flag activating the NO gamma-prime band system.

treat_band_o2_sr
A flag activating the O$_2$ Schumann-Runge band system.

treat_[?]_photo_dis
A binary flag activating the molecular photo-dissociation mechanism [59] for [?] specie, where [?] can be O$_2$ or N$_2$. This mechanism is not technically a molecular band system.
treat[?].photo_ion
A binary flag activating the molecular photo-ionization mechanism [59] for [?] specie, where [?] can be O2 or N2. This mechanism is not technically a molecular band system.

treat_no_photo
A binary flag activating the molecular photo-ionization mechanism [59] for NO.

atomic line models: There are various models available for atomic line radiation, one of which must be chosen for each specie that engages atomic line radiation (as specified using treat[?].lines). This choice of atomic line model is made using the following flags. The listed defaults are applied if the individual flag is not present in hara_namelist.data, or if hara_namelist.data is not present in the working directory. All model types in this category must be surrounded by a quotation marks.

c_atomic_line_model, h_atomic_line_model
A character identifier for selecting the atomic line model for atomic carbon or hydrogen. Presently, the only available option is the model compiled in Ref. [58], which is referred to here as the Complete Line Model (CLM). Default: ‘clm’

n_atomic_line_model, o_atomic_line_model
A character identifier for selecting the atomic line model for atomic nitrogen or oxygen. The available models are compiled and compared in Ref. [56], which is referred to here as the Complete Line Model (CLM). Default: ‘clm’ Available models are:

‘all multiplets’
This model treats all lines as grouped multiplets. This significantly reduces the number of lines treated as well as the computational expense. However, this grouped multiplet approximation will lead to errors for non-optically-thin conditions.

‘clm’
This model, which stands for Complete Line Model, applies the individual treatment of strong atomic lines while applying multiplet averages for weak lines. This is the recommended model.

electronic state population models: These flags specify the model applied for predicting the electronic state populations of atoms and molecules. The listed defaults are applied if the individual flag is not present in
hara_namelist_data, or if hara_namelist_data is not present in the working directory. All model types in this category must be surrounded by a quotation marks, e.g. ‘ ’.

atomic electronic states

The electronic state populations for atoms are required for computing atomic line and photoionization emission and absorption. The compilation and comparison of the available models are presented in Ref. [57].

\texttt{c\_electronic\_state, h\_electronic\_state}

A character identifier for selecting the electronic state model for atomic carbon and hydrogen. Available models are (default : ‘boltzmann’):

‘boltzmann’
Applies Boltzmann population of electronic states.

‘Gally\_1st\_order\_LTNE’
Applies the Gally first-order local thermodynamic nonequilibrium method [60], which approximately accounts for the non-Boltzmann population of atomic states.

\texttt{n\_electronic\_state, o\_electronic\_state}

A character identifier for selecting the electronic state model for atomic nitrogen and oxygen. Available models are (default : ‘CR’):

‘boltzmann’
Same as for \texttt{c\_electronic\_state}

‘Gally\_1st\_order\_LTNE’
Same as for \texttt{c\_electronic\_state}

‘CR’
Applies the detailed collisional radiative (CR) model developed in Ref. [57].

‘AARC’
Applies the approximate atomic collisional radiative (AARC) model developed in Ref. [57]. This model is essentially a curve-fit based approximation of the CR model, which allows for improved computational efficiency with a slight loss in accuracy.
molecular electronic states

The electronic state populations for molecules are required for computing molecular band emission and absorption. The compilation and comparison of the available models are presented in Refs. [57,61].

molecular_electronic_state

A character identifier for selecting molecular electronic state for all molecular band systems. Available models are (default : ‘CR’):

‘boltzmann’
Applies Boltzmann population of electronic states.

‘CR’
Applies a detailed collisional radiative model considering both heavy-particle and electron impact transitions. Some molecular states are still assumed Boltzmann with this model because no data is presently available for the CR model.

other flags:

use_triangles
A logical flag specifying whether optically-thin atomic lines are modeled as triangles to reduce computational time. This option has shown to result in a negligible loss of accuracy while greatly reducing the computational time, [56] and is therefore recommended. Default : .true. Note: This flag is automatically set to .true. when n_or o_atomic_line_model= ‘clm’ — see atomic line models earlier in this section.

use_edge_shift
A logical flag to engage the photoionization edge shift [56] for atomic bound-free radiation. (on=1/off=0). Default : .true.
Appendix C

Troubleshooting

The goal of the Fun3D developers is to produce as robust a solver as possible. However, there are times when the code fails to produce a converged solution. For example, taking the square root of a negative quantity (due to a negative density or pressure) results in a NaN. We hope that these suggestions are helpful. If none of the suggestions listed here remedy your problem, please contact Fun3D-Support@lists.nasa.gov.

C.1 What if the solver has trouble starting or reports NaNs?

Check that the freestream, reference, and boundary conditions are specified correctly. Visualize the solution, especially near the location of the maximum residual. If the problem is widespread, run the simulation again and visualize the solution a few iterations before the problem happens. Look for extremely large Mach numbers, low pressures, low densities, or reversed flow at boundaries.

Examining the residual history may help to isolate the problem to the mean flow or turbulence model. Lowering the CFL numbers of the mean-flow and turbulence equations can aid linear and nonlinear convergence, see section B.4.10. If the linear system is diverging (the linear system can be examined with the --monitor_linear command line option), increase the number of sweeps or utilize the Krylov projection method linear_projection=true., see section B.4.11.

Try flow field initialization in the problematic region of the domain (e.g., engine plenum), see section B.4.17. Start with some first_order_iterations (section B.4.6) or try continuation from less challenging freestream condition (e.g., lower angle of attack, subsonic Mach number).

C.2 What if the forces and moments aren’t steady or residuals don’t converge to steady-state?

Try lowering the CFL numbers of the mean flow and turbulence model, see section B.4.10. Examining the residual history may help to isolate the problem to the mean flow or turbulence model. The problem may be unsteady; try restarting the solution with a time-accurate simulation.
C.3 What if the solver dies unexpectedly?

You may need to set your shell limits to unlimited,

```
$ ulimit unlimited  # for bash
$ unlimit           # for c shell
```

C.4 What if a segmentation fault occurs after “Calling ParMetis”?

Make sure that the the width (32 or 64 bits) of integers in Fun3D and ParMETIS are consistent. More recent implementations of MPI (e.g., Open-MPI) internally manage the handles (i.e., communicators) differently from previous versions of MPI. However, ParMETIS 3.* uses an older paradigm, which has been updated in ParMETIS 4.*. Upgrade to ParMETIS 4.*.

C.5 What if the solver dies with an error like “input statement requires too much data” after echoing the wrong number of elements or nodes?

The endianness (section 4.1) of the grid files (section 4) may be different than Fun3D expects.

C.6 What if the solver dies with an error like “input statement requires too much data” after echoing the number of tetrahedra and nodes for a VGRID mesh?

Single-segmented VGRID grids over 20 million nodes exceed the allowable record length. Use `postgrid` to save grid as a multi-segmented format (option 05 in batch mode).

C.7 What if the solver reports that the Euler numbers differ?

The Euler number is a global indicator of consistent node, element, and face connectivity. There is some limited evidence that suggests there may be times when it reports a problem that may not be an issue for the solver, but the failure of the Euler number check indicates a problem with the grid in a majority of cases. Instructions to determine if the Euler number will impact your solution follow this description of the Euler number.
C.7.1 Euler Number Description

A valid grid is composed of four elemental volume types (tetrahedra, pyramids, prisms, and hexahedra) face-connected either to each other or one of two boundary face types (triangles or quadrilaterals). Each boundary edge connects to precisely two boundary faces. Two neighboring boundary faces share exactly one boundary edge. For each boundary face connecting to a boundary node, every other boundary face connecting to the same boundary node can be found by a path through a connected-edge/connected-face traverse starting from that boundary face.

The above restrictions are meant to exclude certain topologies such as two spherical boundaries coming together at a point or two rectangular boundaries connecting along an edge. These restrictions are not checked explicitly but will cause the Euler number check described below to fail.

The Euler Number computed from boundary data ($EN_b$) is

$$EN_b = N_b - E_b + F_b$$  \hspace{1cm} (C1)

where

- $N_b \equiv$ boundary nodes (counted) \hspace{1cm} (C2)
- $E_b \equiv$ boundary edges (inferred from $N_{tri}$ and $N_{quad}$) \hspace{1cm} (C3)
- $F_b \equiv$ boundary faces (inferred from $N_{tri}$ and $N_{quad}$) \hspace{1cm} (C4)

The Euler number is a characteristic number for the topology of the boundary or boundaries. $N_{tri}$ and $N_{quad}$ are the number of triangular and boundary faces, respectively.

The Euler Number computed from volume data ($EN_v$) is

$$EN_v = 2(N - E + F - C')$$  \hspace{1cm} (C5)

where

- $N \equiv$ volume nodes (counted) \hspace{1cm} (C6)
- $E \equiv$ volume edges (counted) \hspace{1cm} (C7)
- $F \equiv$ volume faces (inferred from $C$ and $F_b$) \hspace{1cm} (C8)
- $C \equiv$ volume cells (counted) \hspace{1cm} (C9)

The formula that is checked is

$$EN_v - EN_b = 0$$  \hspace{1cm} (C10)

Barth [62] derived this formula for tetrahedra and noted the formula does not hold in certain cases, such as two simplices that share only one common edge or two simplices that share only one common node. Barth notes that the above formulas are specific forms of the general Dehn-Sommerville formula,
reported in Wikipedia to hold for simplicial polytopes and simple polytopes. The pyramid is not a simple polytope. It can be proved by induction [63] that the formula holds for every valid grid as defined above.

Try this checklist to diagnose the problem:

1. Check the $EN_b$ with your expectations for this case:
   - 2 for a spherical topology, simple 3D wing with symmetry, . . .
   - 0 for a torus, donut, . . .
   - -2 for a double torus, . . .
   - -4 for a triple torus, pretzel, . . .
   - 4 for a sphere within a sphere, . . .
   - 6 for two spheres within a sphere, . . .

2. Ensure the number of boundary nodes $N_b$ and faces $F_b$ reported match expected values.

3. Ensure the number of nodes $N$ and cells $C$ reported match expected values.

4. The difference between $EN_b$ and $EN_v$ points to inconsistencies in edge counts, i.e., $\delta(E) = 2(EN_b - EN_v) \neq 0$. The inequality $EN_b > EN_v$ implies you have more edges than expected. When this occurs, the reported face counts will differ from an actual count. An error of this type would arise when there are adjacent faces that are inconsistent, such as a quadrilateral face shared between two elements that is cut into two triangular faces by different edges.

C.7.2 Determining the Impact of an Euler Number Mismatch

A freestream residual problem localization technique is described. However, the best practice is to not proceed without repairing the grid to ensure $EN_b = EN_v$. The `ignore_euler_number` namelist variable does just what the name implies, and allows the solver to proceed. The `--test_freestream` option can be used as a secondary check on the mesh. On a valid mesh, the solver should preserve the freestream for an arbitrary number of iterations. You should run 20–50 iterations, which may require a lowering of the `stopping_tolerance` to 1e-20 so the solver does not automatically stop. All residuals should hover around machine zero, and not slowly increase (there will be iteration-to-iteration variation in the exact number, however).

If freestream is maintained by this test, you could proceed with your intended computation using only `ignore_euler_number`, but this is not recommended. If freestream is not maintained, this test confirms that there is a
problem with the mesh and the location of the max residual may give a clue as to where in the mesh to start looking for the problem.
**REPORT DOCUMENTATION PAGE**

The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.

**PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.**

<table>
<thead>
<tr>
<th>1. REPORT DATE (DD-MM-YYYY)</th>
<th>2. REPORT TYPE</th>
<th>3. DATES COVERED (From - To)</th>
</tr>
</thead>
<tbody>
<tr>
<td>01-03-2014</td>
<td>Technical Memorandum</td>
<td></td>
</tr>
</tbody>
</table>

**4. TITLE AND SUBTITLE**

FUN3D Manual: 12.4

**6. AUTHOR(S)**


**7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)**

NASA Langley Research Center
Hampton, VA 23681-2199

**9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)**

National Aeronautics and Space Administration
Washington, DC 20546-0001

**11. SPONSOR/MONITOR’S REPORT NUMBER(S)**

NASA/TM-2014-218179

**12. DISTRIBUTION/AVAILABILITY STATEMENT**

Unclassified - Unlimited
Subject Category 61
Availability: NASA CASI (443) 757-5802

**13. SUPPLEMENTARY NOTES**

**14. ABSTRACT**

This manual describes the installation and execution of FUN3D version 12.4, including optional dependent packages. FUN3D is a suite of computational fluid dynamics simulation and design tools that uses mixed-element unstructured grids in a large number of formats, including structured multiblock and overset grid systems. A discretely-exact adjoint solver enables efficient gradient-based design and grid adaptation to reduce estimated discretization error. FUN3D is available with and without a reacting, real-gas capability. This generic gas option is available only for those persons that qualify for its beta release status.

**15. SUBJECT TERMS**

Aerodynamics; Computer Programming Software; Fluid Mechanics; Mathematics; Propulsion Systems

**16. SECURITY CLASSIFICATION OF:**

<table>
<thead>
<tr>
<th>a. REPORT</th>
<th>b. ABSTRACT</th>
<th>c. THIS PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>U</td>
<td>U</td>
<td>U</td>
</tr>
</tbody>
</table>

**17. LIMITATION OF ABSTRACT**

UU

**18. NUMBER OF PAGES**

278

**19a. NAME OF RESPONSIBLE PERSON**

STI Help Desk (email: help@sti.nasa.gov)

(443) 757-5802

Prescribed by ANSI Std. Z38.18

Standard Form 298 (Rev. 8-98)